Universidad de Jaén
Escuela Politécnica Superior de Jaén
Ingeneria Mecanica y Minera

Don Ruben Dorado Vicente , tutor del Trabajo de Fin de Master titulado: Potential relations of Look-ahead and Spline Interpolation, que presenta Weronika Olczyk, autoriza su presentación para defensa y evaluación en la Escuela Politécnica Superior de Jaén.

Jaén, Julio de 2017

El alumno: Los tutores:

Weronika Olczyk Ruben Dorado Vicente

CONTENTS

1 ABSTRACT

The paper presents the effect of Look-Ahead on machining time on the Odisea Fagor 8055 MC. Three different geometries were specified with different acceleration parameters and a Look-Ahead error. The work piece geometry analysis was performed by recording the milling machine's motion camera and processed in Mathematica Wolfram's mathematical program to obtain information on the differences between the trajectories. Hausdorff distance was calculated as well as the real feed rate. Similarities between the real tool-path and B-Spline curves were pointed out through the weights manipulation. It was shown that Look-ahead function has and influence on the cycle time and on geometry of the work piece. When working with look-ahead results the smoothness of the feed rate changes. The work of the look-ahead can be represented and explain using the NURBS function as a similarities have been seen.

2 ANALYSIS OF CURRENT KNOWLEDGE

2.1.1 History of CNC machine tools

Globalization of the global economy forces the continual improvement of all production spheres by making more and more accurate products in less and less time. A large amount of competition on the market causes the need to modify and optimize the existing process chains. In order to increase the machining capacity and also meet the requirements for surface quality and dimensional accuracy, it is necessary to modify the machine tools in terms of construction and control systems. The first machine tools to automate production could only perform a fixed and predetermined sequence of movements without programming. The development of numerical control (NC) owns a lot to the Air Force. Numerical control concept was developed by John Parsons and his associate to improve production in the aerospace industry at Parsons Corporation in Traverse City, Michingan which was contractor for Air Force. Parsons had experimented with coordinate using data contained on punched cards to define the surface contours of airfoil shapers. His first milling machine which used punched cards was named Cardamatic. Next work on this systems required invention of prototype machine tool based on the Cardamatic principle. In 1952, with the development of electronic data processing at the Massachusetts Institute of Technology for the military purposes of the United States, the first numerically controlled machine tool was created, as shown in Figure 2.1, [7, 31, 10].

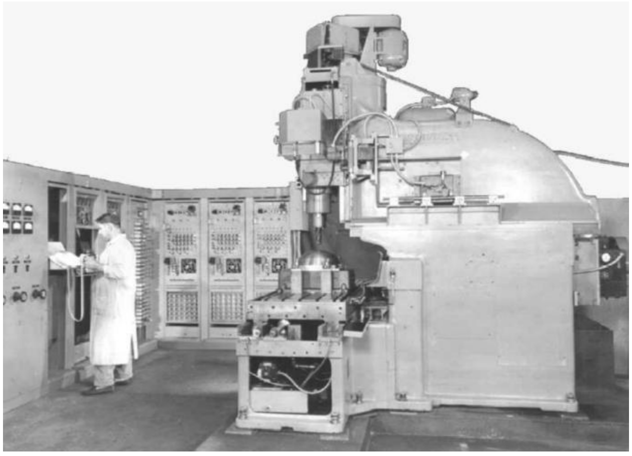


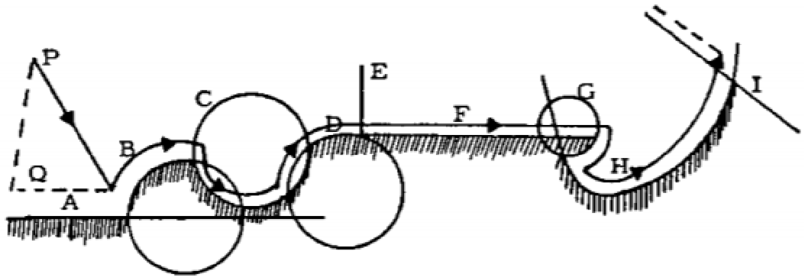
Figure 2.1 First machine with numerical control (NC), [26]

The researches of MIT showed that punched cards cannot be used to achieve data transfer rates between the controller and the machine tool. Cincinnati's NC Hydrotel had analog and digital components consisted of 292 vacuum tubes, a vertical spindle, linear 3D interpolation and binary coded bundle tape. The basis of control were the input values of the connections and the binary and pulsed circuit diagrams that process mathematical functions describing the shape of the workpiece. Punched tapes and cards were used for their programming. A string of information controlled in the form of letters and numbers is named NC (from English numerical control), [31].

Numerical control is one of the forms of programmable automation, where program containing coded alphanumeric data controls the mechanical actions of machine tool or any other equipment. The term 'numerical control' means that machine is controlled by numbers obtained from the description of the parts. Positions between a cutting tool and work part as well as other instructions are represented by the alphanumeric data. After current job is finished, the program can be changed to make a new job. That capability of NC to easily change the program makes it suitable for low and mediums production. Part programming was recognized as difficulty with the new technology. To disseminate the knowledge about the NC machine, the aircrafts companies across the country got a trial parts to learn about the operation features and economic advantages. Also the public demonstration was held for a machine tool builders, producers and potential interested partners. What is more Air Force sponsored the development of NC in several different companies. In the beginning machine tool companies were not optimistic. They were worry about the qualification of the workers to maintain machine as well as in general use of vacuum tubes, which soon were replaced with transistors and integrated circuits, [7].

The first NC machines were developed for purpose of machining parts with difficult and complex shape in a precise manner, which is the reason that the first NC machine was a milling machine and boring machines. After some years it started to be popular to apply NC to other kind of machine in order to increased productivity. It started a process of manufacturing NC machines such as turning machines and drill/tapping machines. The application of NC has extended to non-conventional machine tools such as wire laser cutting machines electro-discharge machines in addition to conventional metal-cutting machines, [3,10].

In 1956, as the first computer-aided part programming system was showed by MIT, the American Air Force accepted the proposal of them to create a programming language. In 1958 this researches resulted in creation Automatically Programmed Tool language (APT). As a father of this high-level computer programming language is considered Douglas T. Ross, a great mathematician. The APT design was a pioneering effort not only for NC technology but also in computer graphics, computer programming concepts and computer-aided design (CAD). In 1974, APT became a U.S. standard for programming NC metal cutting machine tools. In 1978 it became an international standard accepted by ISO. On the figure 2.2 is showed the example of APT program, [10, 7].




```
FED RT = +80. $$
FROM / P $$
DNT CT, GO TO / Q $$
TL LFT, DNT CT, GO LFT, NEAR / A,B $$
GO LFT / B,C $$
GO RGT / C,D $$
GO LFT / D,E $$
FAR, CROSS / F,G, $$
NEAR,GO CLW / G,H $$
GO CCW / H,I $$
TL RGT,TERM,GO LFT / I $$
STOP $$
END $$
```

```
NOTE: ANY INSTRUCTION
HERE CAN HAVE FEEDRATE
GIVEN BEFORE "$$"
IF ONLY ONE SYMBOL IS
USED IT IS THE DESTINATION
CURVE (EXCEPT FOR "TERM")
I. E. COULD HAVE
"- - - - - - - - - - - - - - - -
GO LFT / B,C $$
GO RGT / D $$
- - - - - - - - - - - - - - - - -"
INSTEAD OF
"- - - - - - - - - - - - - - - -
GO LFT / B,C $$
GO RGT / C,D $$
- - - - - - - - - - - - - - - - -"
```

Figure 2.2 Example of on of the first APT program, [14]

The development of microcomputer technology, which occurred in the 1970s, enabled the creation of machine control units (MCU) and going further computerized CNC control systems (Computer Numerical Control). Computerized CNC control is an automatic machine that controls a machine using a pre-programmed command sequence computer. This is the opposite of machines that are manually controlled by hand wheels, levers or mechanically automated by means of cams.

The first CNC machine that reduced production time from 8 hours to 15 minutes was designed by John Runyon. The first generation required the use of large vacuum valve valves that consumed a lot of energy and generated large amounts of heat. Second generation models replaced vacuum tubes with more reliable, less energy efficient and less space-efficient transistors. First and second generation machines did not have any memory controllers. The driver had to be fed with instructions one at a time from an external source, such as a tape reader. The driver accepts a single statement, executes the command, then accepts the command, executes it, and so on. Commands are coded on paper tape. When the tape passes through the tape reader, a single data block (command) is read, sent to the control device for execution. After the execution, the controller sends a signal to the tape reader, indicating that it is ready for another command. The reader reads the tape of the next block, and so on until all the tape is read, passed to the controller, and executed. The last command on the tape is the code to stop the reader and scroll the tape.

The use of CNC machines has saved production costs, so machines were quickly popularized among manufacturers. In order to promote the machines, the US Army purchased 120 numerically controlled machines and lent them to various manufacturers to get acquainted with them. G code (from G-code), a standard language developed in the MIT servo laboratory in 1958, was adopted by many machine manufacturers. Figure 2.3 shows the Milwaukee-Matic-II CNC milling machine, which was produced in 1959 as the first automatic gear milling machine, [1].

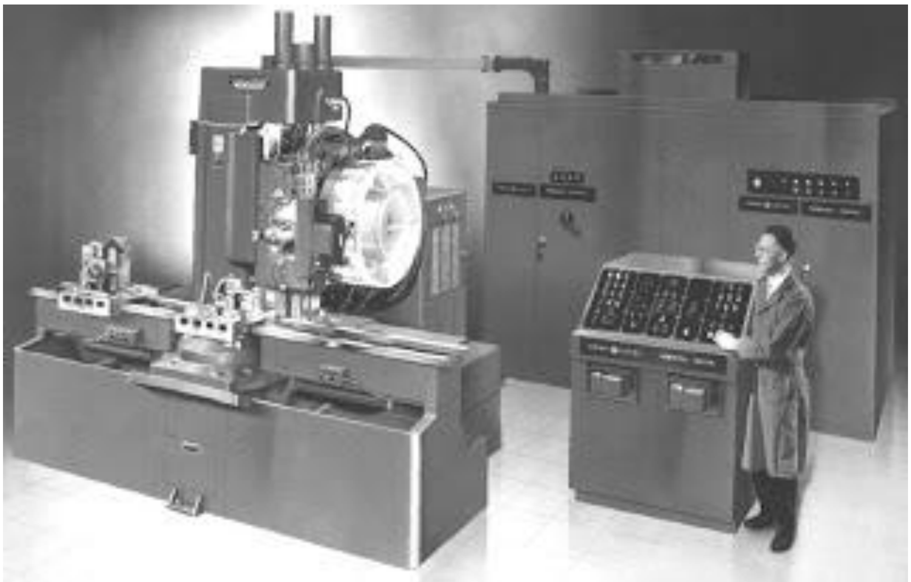


Figure 2.3 Milwaukee-Matic-II, [26]

The history of CNC system development can be pictured as is showed in the figure 2.4. In the 1950s NC systems were develop based on the hardware. Further researches in the area contributed to replacement of hardware NC with software CNC systems. They function with micro processor but the closed system did not allowed to add customized functions. In the 1990s, closed system replaced by Open Architecture Controller (OAC). OAC is a user orientated CNC containing PC and Virtual Machine Environment (VME). The PC-based open CNC have not still satisfied the numerous requirements of the user from the hardware and software, which is the reason the further researches are made to develop soft-NC, [3]

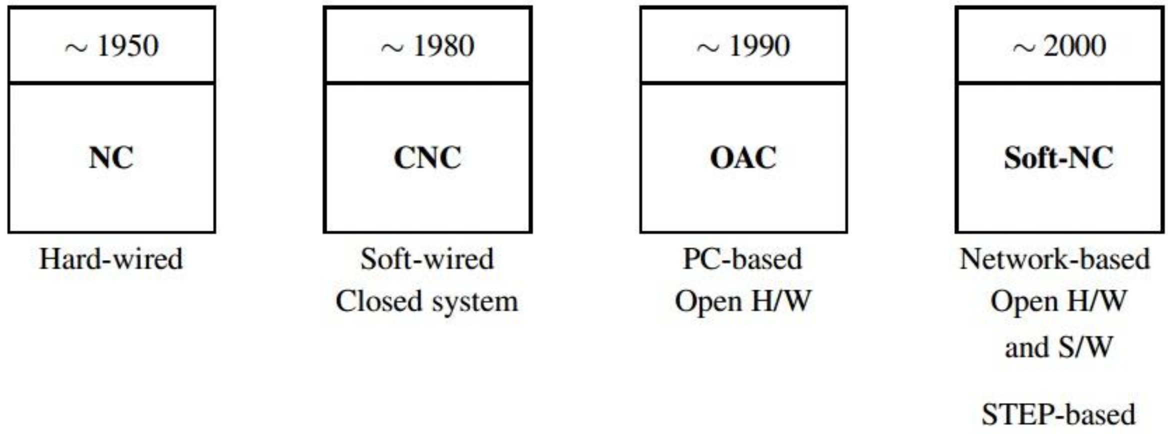


Figure 2.4 of CNC system over time, [3]

In modern CNC systems, the design of the mechanical part and the production program is highly automated. The mechanical dimensions of the parts are defined using software of computer-aided design (CAD) and then translated into production directives using computer-aided manufacturing (CAM) software. The resulting directives are transformed (by the "postprocessor" software) into the specific commands needed to execute the machine and then loaded into the CNC machine.

Computer numerical control can be used to a wide variety of processes:

- machine tooling applications: milling, turning, drilling,
- nonmachine tool applications: control, assembly, drafting

Every particular item may require the use of many different tools like drill bits, saw etc. Modern machines often combine multiple tools or use a variety of machines with an external controller, human operators, robots that transfer the elements from the machine to the machine. In each case the number of steps needed to produce any part is highly automated and produces part is closely matched the original CAD.

CNC machine tools have a great amount of characteristic. Some of them where listed and described below.

- Storage the numerous part programs - CNC controllers have a capacity to store wide number of programs. What's more use of MCU offer one or more memory expansion.

- Various forms of program input - CNC controllers have multiple data enter capabilities like punched tape, magnetic tape, RS-232 communicators, floppy diskettes, manual input data.
- Edition of program at the machine - program can be tested, edited and optimized at the machine. There is no need of returning the program to the programming office for the further correction.
- Fixed cycles and programming subroutines - thanks to big memory it is possible to make a part programs which instead of writing the full instructions for the particular cycle in every program can be call and executed in the right place.
- Positioning feature for a set up - positioning needed to be done to accomplish the machine axes and the workpart is establish with correct parameters. In CNC the operator is not needed to locate the fixture on the machine table with enormous accuracy.
- Cutter length and size compensation - compensation can be automatically made in the computed tool path and also by using the length sensor.
- Acceleration and deceleration calculations - is applicable when the cutter moves at high feed rate. It is made to avoid tool mark on work part surface which is made in the moment when the path changes quickly.
- Communication interface - most modern CNC controllers are linked with computer by being equipped with RS-232 or other communication interfaces. It founds the advantages in downloading part program from a data file, collecting operational data, interfacing with peripheral devices
- Automatic supervision and diagnostics - modern CNC system has monitor capabilities to detect malfunctions or diagnose system breakdowns.
- infinitely variable speed control and feed rates,
- the drive transmitted via ball screw,
- elimination of sliding guides for rolling,
- elimination of gears,
- compact closed-space design,
- modular design with flexible design elements,
- high power (sum of power of individual drives),
- achievement of significant values of processing parameters,
- parallel machining using multiple spindles or tool supports,

- complex kinematics of work,
- coding tools,
- tool magazines with automatic tool change,
- tooling systems with folding tools,
- automatic measurement of tools,
- automatic replacement of workpiece,
- automatic measurement of the workpiece,
- automatic chip removal, [7, 26].

2.2 Computer Numerical Control Fundamentals

CNC system is based on the 6 major elements, which are: input device, machine control units, machine tool, driving system, feedback devices and display unit. The principles of work are described below and showed on the Figure 2.5, [12].

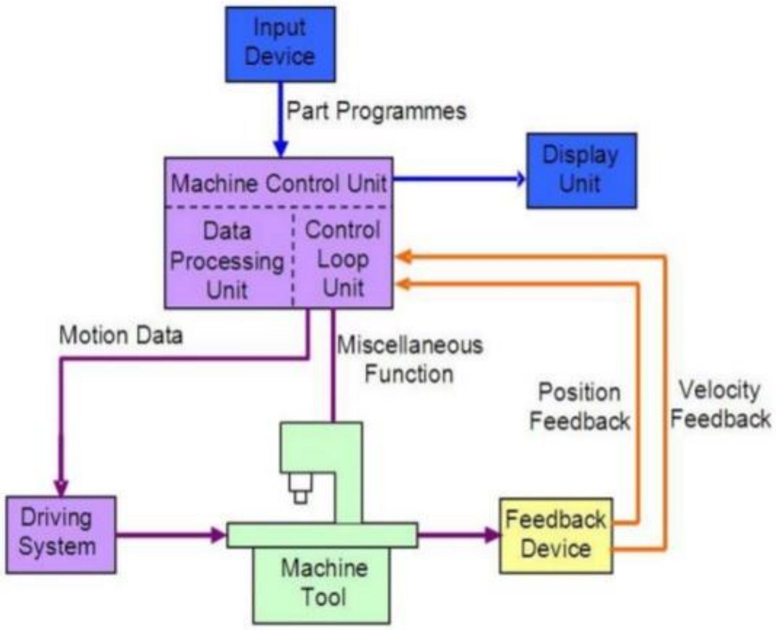


Figure 2.5 Work principle of CNC machine, [12]

2.2.1 Input Devices

Input devices are categorized:

1. Floppy Disk Drive

They are quite old storage devices for CNC data input. They were the most popular in the 1970s. They were easy to use. Data transfer was speed and reliable. The program describe on them was easy to edit. Although, this method is debatable for a long run, because floppy disk have high probably of fast degradation and are sensitive to magnetic field, dust, scratches. Also the data storage is incredible little, until 1,44 MB. The example of floppy disk drive in CNC has been shown on the Figure 2.6, [12].

Figure 2.6 Floppy Disk Drive in CNC machine, [16]

2. USB Flash Drive

USB flash drive, also known as thumb drive, pen drive or memory stick is a data storage removable and rewritable portable device. First USB drive appeared on the market at 2000 and until now is one of the most popular device for data storage. It can have even 1TB units. The example of USB Flash drive in CNC is shown on the Figure 2.7, [12].

Figure 2.7 USB Flash drive in CNC, [12]

3. Serial communication

Serial communication port transfers data from computer to CNC machine tool via cable. Because of ISO standards of communication ports the information is exchanged in orderly way. Most of PC and CNC machine tool have built ports and install standards with common interface EIA Standard RS 232. With serial communication is also referred a Direct (or Distributed) Numerical Control, which is a system connecting a set of NC machines to common memory for part program. Sometimes CNC machines have not enough available to contain the machining program (complex surfaces). In this case the program is stored on host computer, from where information is distributed to other computers and after sent directly to the machines, block by block at a time. If one of computers is connected to a few machines it can send programs to them. The schema of work is presented on the Figure 2.8, [12, 34].

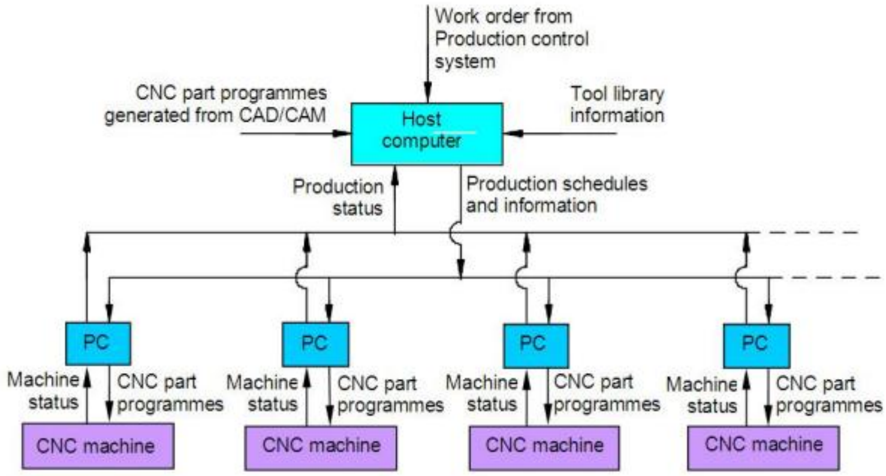


Figure 2.8 Distributed Numerical Control system, [12]

4. Ethernet communication

Ethernet communication is mostly use in the bigger companies, where Local Area Network (LAN) is built and CNC machine is provided in Ethernet Card for directly communication. Computer connects with machine via Ethernet communication cable. The schema of ethernet network in a DNC system is showed on figure 2.9, [12].

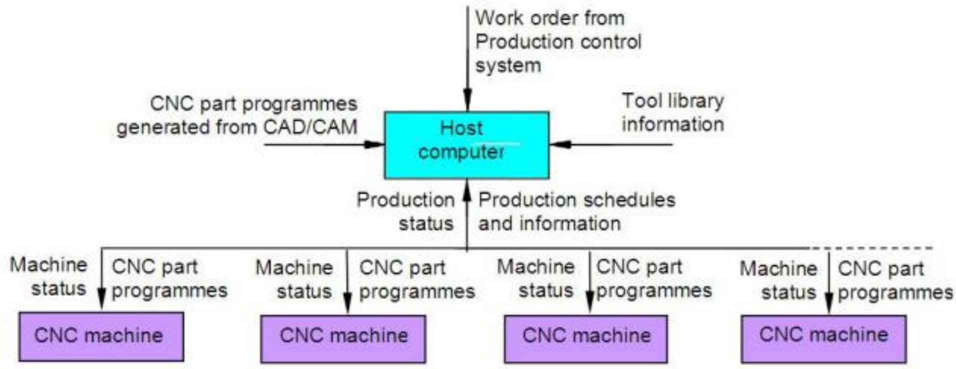


Figure 2.9 Ethernet network in a Distributed Numerical Control system, [12]

5. Conversational programming

Part program can be input to the machine via the keyboard. Software provides the possibility of writing the required data step by step by keyboard. This way is mostly use for very easy parts. Conversational Programming in CNC machine is showed on figure 2.10, [12].

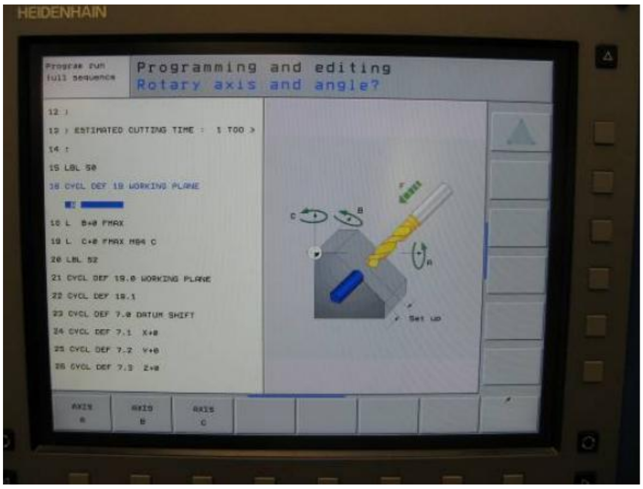


Figure 2.10 Conversational programming on the CNC machine, [12]

2.2.2 Means of Machine Control Units

Machine control unit for CNC is the hardware which identifies CNC from conventional NC. The machine control Unit is made of the five elements, which are represented on the figure 2.11 and described below.

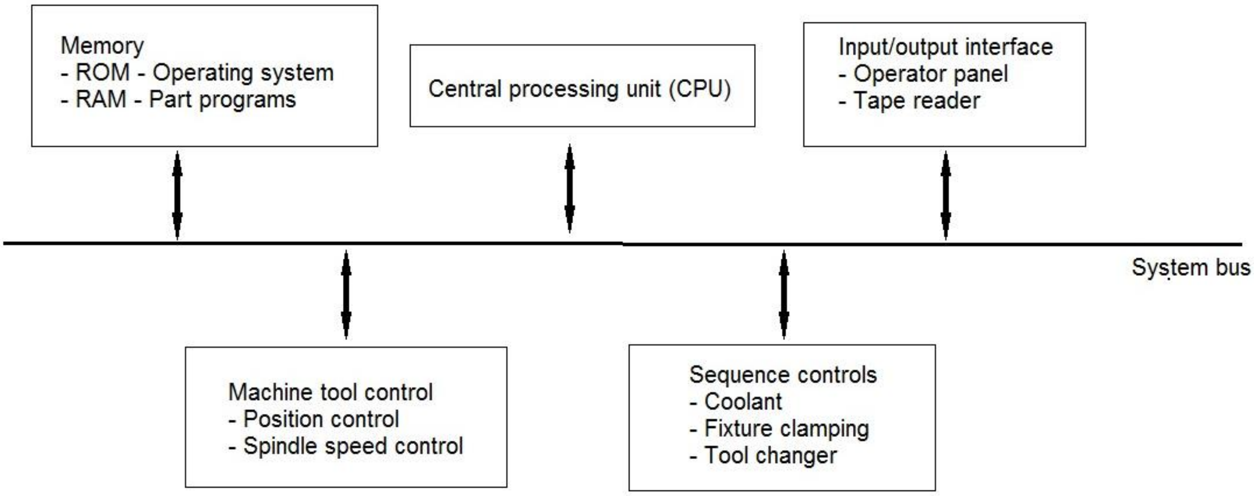


Figure 2.11 Configuration of CNC machine control units, [7]

1. Central Processing Unit (CPU) is the brain of the Machine Control Unit. It manages the other elements of MCU by main software memory. CPU consists:
   - control section, which download commands and data from memory and generate signal to activate other elements
   - arithmetic-logic unit (ALU) is responsible for counting and logical functions
   - immediate access memory is a temporary storage for data in process
2. Memory is necessary for the various programs to operate the CNC system. The purpose of CPU's memory, which was mention before, is intended for store CNC software. Memory is divided on:
   – main memory (primary storage) is made from read-only memory (ROM), which store operating system software and machine interfaces program, and random access memory (RAM), which store numerical part programs,
   – secondary memory (auxiliary storage/secondary storage) is intended to storage large programs and data files. The most popular secondary memory is hard disk and portable devices
3. Input/Output interfaces are communications providers between components of the CNC system, machine operators and computer. It main function is to transfer and collect data and signals to and from external devices like operation panel which is used to communicate with CNC system.

4. Controls for Machine Tool Axes and Spindle Speed are the components of hardware which are used to regulate the position and velocity. Positioning system is classified as open loop and close loop. Spindle speed regulation components have drive control circuit and a feedback sensor interface.
5. Sequence Controls for Other Machine Tool Functions is an auxiliary function for control of position, spindle speed, feed rate. Normally is this function is using binary actuations, interlocks and discrete numerical data, [7].

2.2.3 Machine tool

Very important part in the CNC machine is a driving leadscrew. It need to be designed and made properly in order to obtain high repeatability and high accuracy. Usually, leadscrew is made from polytetrafluoroethylene (PTFE) and Turcite (Ethylene ChloroTriFluoroEthylene ECTFE), which reduce the stick and slip. To eliminate the backlash and motion lost the screws with large size recirculation balls is applied in CNC. The examples of ball screws are showed on figure 2.12, [12].

a) b)

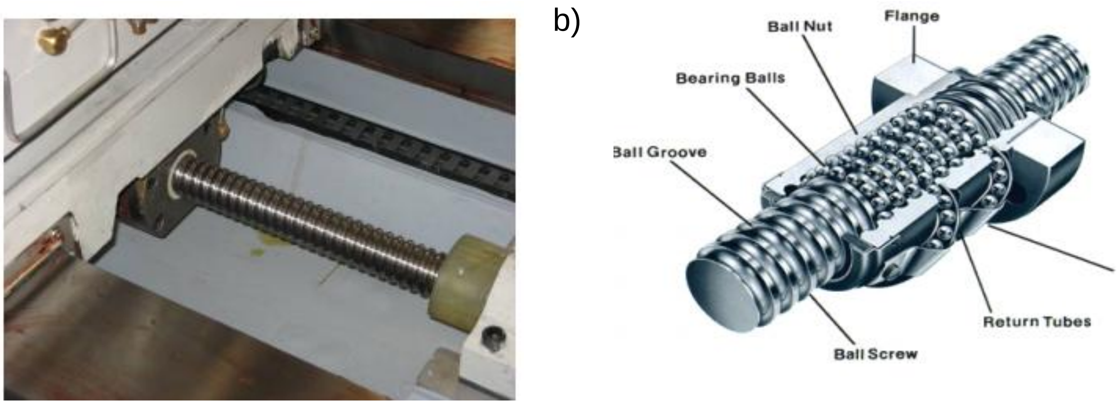


Figure 2.12 Ball screws: a) inside CNC b) the structure, [12]

Driving systems components in the NC machine are: servo driving mechanism and spindle driving mechanism. Both of them were represented on the Figure 2.13.

The driving systems convert the instruction from NC to machine movements. The servo driving mechanism is made from servo motor and power transmission device, which move the bed of work piece. The NC instruction makes that the servo motor rotate. Coupling transmits this rotation to a ball screw, which convert them into linear movement of nut. The bed of machine with a work piece moves in the lines. In addition servo driving device of each axis takes the instruction from NC and with motor controls the torque of bad and velocity, [3].

The spindle driving mechanism is made from spindle motor and power transmission device. The spindle device generates cutting speed and torque. Belt is in control of transmission the spindle motor rotation to spindle body. The velocity ratio depends on the pulley size ratio between the motor and spindle body. Induction motors, thanks to no brush, are recently replacing the spindle motors, which are needs of respecting to size, weight, efficiency, speed, maintenance, [3].

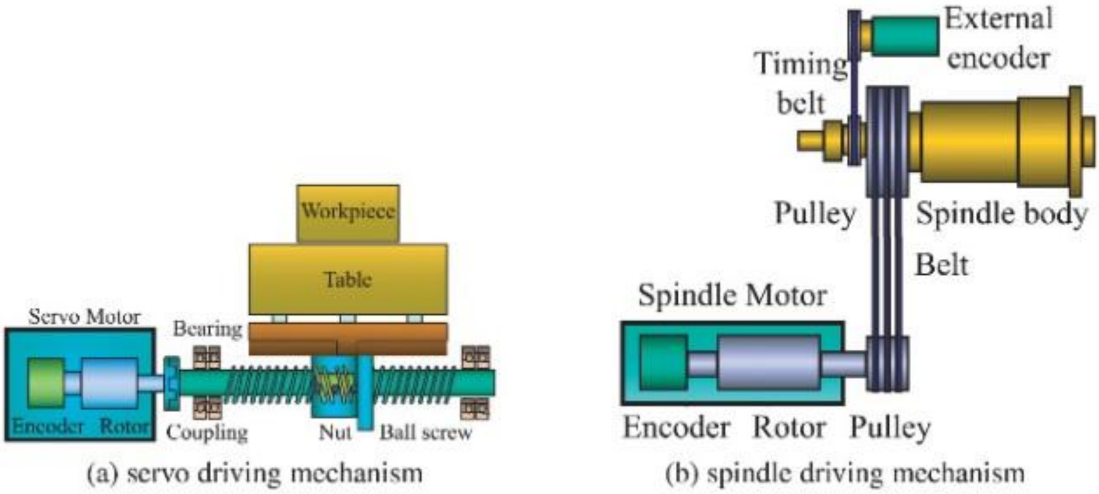


Figure 2.13 Driving mechanisms of machine tools, [3]

2.2.4 Driving system

Driving systems are playing important role in arranging the satisfactory accuracy and repeatability. Driving motors are both in the spindle and servo motor. High speed and high torque are very important for spindle motor. The servo motor needs to be describe by high torque and acceleration, fast reaction at low speed as well as good control of velocity and position. The servo motors have to be characterized by:

–quick reaction on the commands,

–capable power output for work load,

–capable for quick changes of velocities,

–capable to operate for a long time,

–have high resolution to generate torque in small block,

–high rotation accuracy,

–decent stopping torque,

–easy to maintain.

The following types of servo motors can be distinguish (Figure 2.14):

1. DC Servo Motor - is built from cylindrical frame, magnet attached to the frame from inside, rotor made from shaft and brushes, which are connected to the machine terminals, commutator, armature coil, detector, which detects the rotation speed. Principle of operation is based on rotation of armature in magnetic field which is connected to communicator. Carbon brushes pass DC current to commutator. Varying of armature voltage influence on motor speed.
2. Synchronous-type AC Servo Motors - similar like DC Servo motor is built from cylindrical frame, detector, armature coil, magnet, which is attached to rotor. Unlike DC servo motor is no need of brush, because of lack of velocity limitation due to rectification spark and stator code can be cool down from outside, which makes it resisted on rapidly change of temperature. Also, because lack of brushes maintenance is not so much often like in DC servo motors.
3. Induction (Asynchronous)-type AC Servo Motors - simply structure of the driving system consist of armature coil, rotor, magnet.

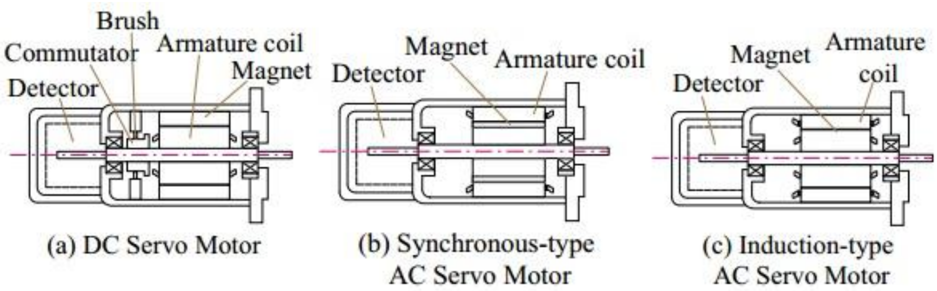


Figure 2.14 Servo motor's types, [3]

4. Stepping motor - converts the electrical pulses into discrete mechanical rotational motions of motor shaft. If CNC machine can convert digital

data into mechanical displacement. They are suited to open loop system. These kinds of motor are not so much popular in CNC machines. The schema of stepping motor is shown on figure 2.15, [12].

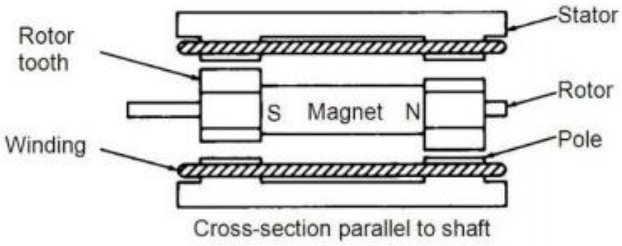
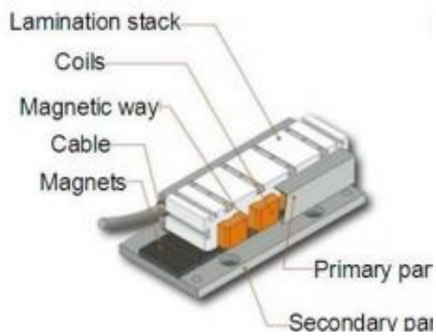


Figure 2.15 Stepping Motors, [12]

5. Linear motors - are direct drive servo motors, which consists magnetic track fixed to a base plate and moving coil. By putting together the magnet tracks can have unlimited length. There are two types of linear motors: Iron Core and Ironless core, which are showed on the figure 2.16. Although, linear motors have a good accuracy, high maximum speed, stable force constant, acceleration, repeatability, they are not common use in CNC machines, [2, 12].

a)



b)

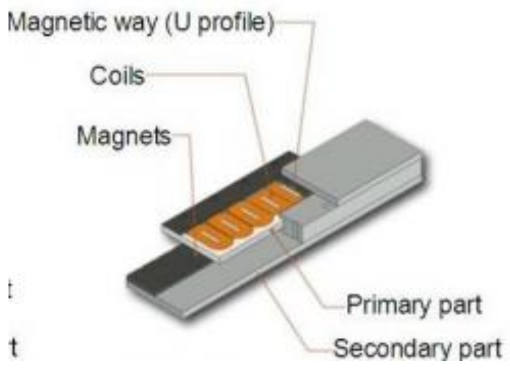


Figure 2.16 Ironcore and Ironless Linear Motor, [2]

The advantages and disadvantages of the DC and AC servo motors are shown in the Table 2.1, [3].

	DC Servo Motor	Synchronous Type AC Servo Motor	Induction Type AC Servo Motor
Strengths	Low price Broad velocity range Easy control	Brushless Easy stop	Simple structure No detector needed
Weaknesses	Heat Brush wear Noise Position-detection needed	Complex structure Torque ripple Vibration Position-detection needed	Dynamic braking impossible Loss of heating
Capacity	Small	Small or medium	Medium or large
Sensor	Unnecessary	Encoder, resolver	Unnecessary
Life length	Depends on brush life	Depends on bearing life	Depends on bearing life
High speed	Inadequate	Applicable	Optimized
Resistance	Poor	Good	Good
Permanent magnet	Exists	Exists	None

Tab. 2.1 Comparison of Servo Motor, [3]

2.2.5 Feedback devices

Feedback device controls the positional values and axes' speed. In CNC machine there are distinguished two types of feedback devices:

1. Positional Feed Back Devices, are divided into:
   - linear transducer - it is the highest accuracy measurement devices fix on machine table to analyze displacement of the slide to prevent error in the feedback data,
   - rotary encoders - device is fixed in the end of motor shaft or screw to analyze the angular displacement.
2. Velocity Feedback Device

   The voltage generated from tachometer fixed in the end of motor shaft can be base for calculation of actual speed. DC tachometer is small generator producing the voltage which can be compared with corresponding desired speed.

2.2.6 Display unit

The Display Units are the interactive communicators between operator and CNC machine. During the program run, display unit shows in what stage of work is the program. It also shows spindle RPM, feed rate, needed maintenance, machine parameter, logic diagram, error massages or graphic simulation of the tool path before part is machining. Example of CNC Display Unit is showed on figure 2.17, [3]

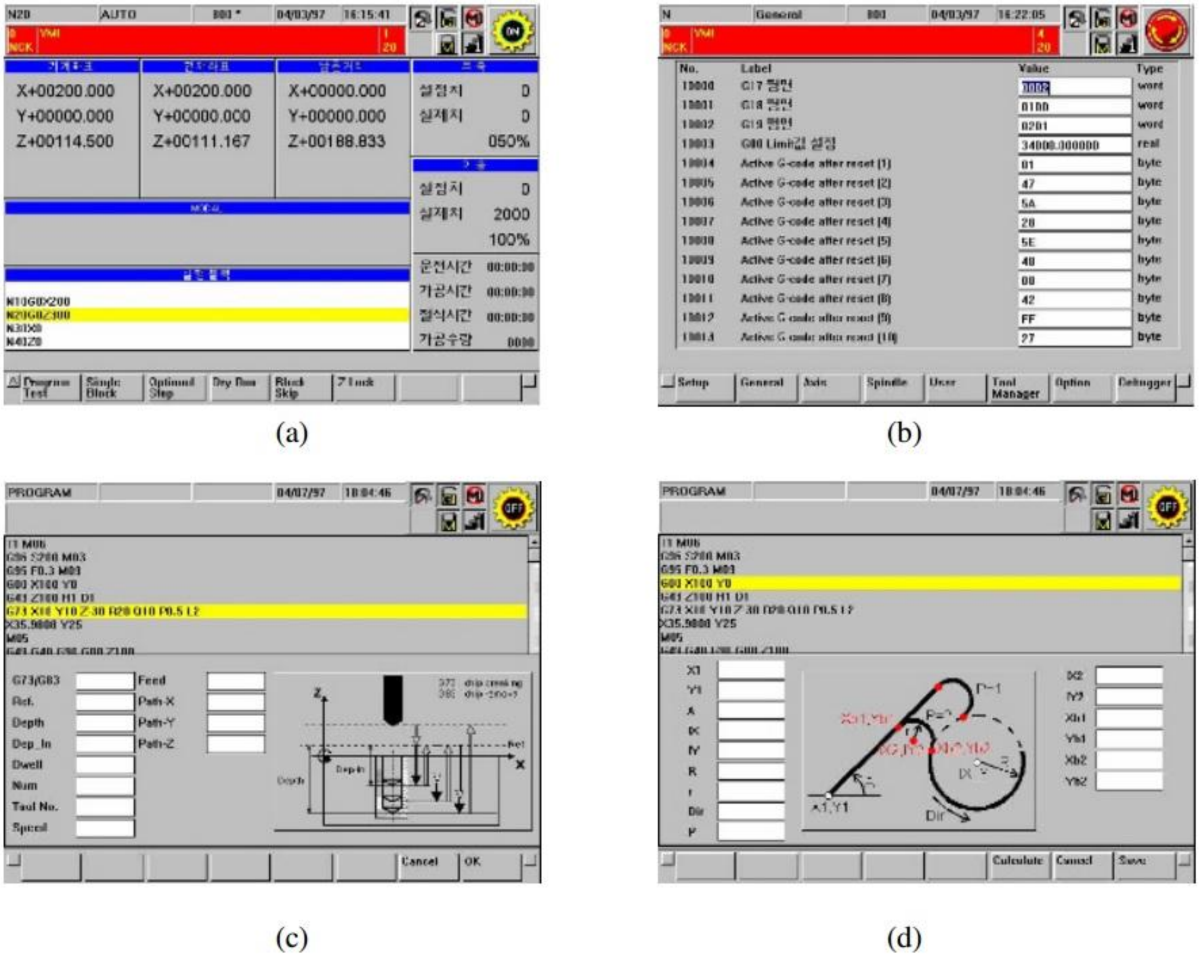


Figure 2.17 Example of Display Unit for HiTrol-M100: a) Operation functions, b) Parameter setting functions, c) Drilling editing functions, d) Geometric shape calculator, [3]

2.3 Classification of CNC control systems

Control devices, regulating and managing provide automatic, self-working machinery and equipment. Control is the management of machinery and equipment, by introducing or modifying parameters to achieve specific action and achieving a specific goal.

The control system is called the change of the physical size typical of the machine operation, eg: switching on and off the drive motor, control of main movements. The control system comprises: a control object and a control element so- controller. There are two types:

- manual control, that is realized with the help of man,

- automatic control using a specially designed device, [15]

Classification of CNC Machine tool systems is carried out according to different criteria.

2.3.1 According to type of Feedback systems

The servo motor used in the CNC machine is constantly controlled to reduce the error of and position. The feedback control system consists of three independent control loops, which are showed on the figure 2.18. The external loop is a position control loop; central loop is a velocity control loop and the inner loop is responsible for current loop.

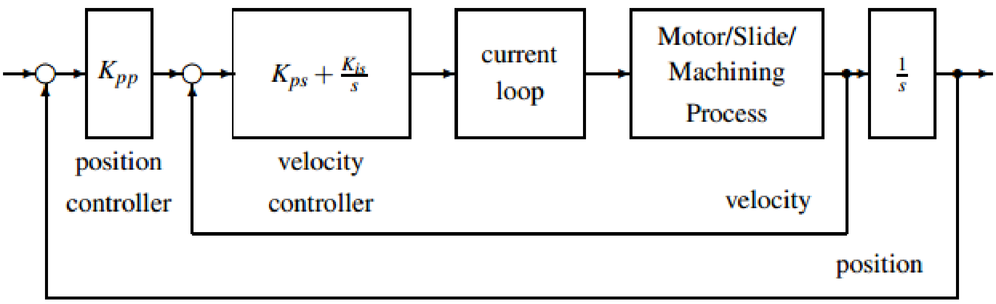


Figure 2.18 Three kinds of control loop in CNC, [3]

There are four types of feedback control systems according to localization of attached detector:

- open-loop control, which is without feedback, so elimination of the impact of interference is impossible
- closed-loop control, detector attached on the machine table
- semi-loop control, detector attached on the shaft
- hybrid control,

Open loop type CNC machine

In open loop system sent instructions are converted to the signals by the controller and sent to the servo amplifiers to energize the servo motors. Information about faults is the basis for the correction, which in the case of the open system is only possible after the machining. The use of an open loop control is justified only if the effects of the control can be foreseen for the signals concerned. This kind of systems finds applications in the point to point control systems where the accuracy doesn't matter. The schema of open-loop is showed on the Figure 2.19, [20, 16]

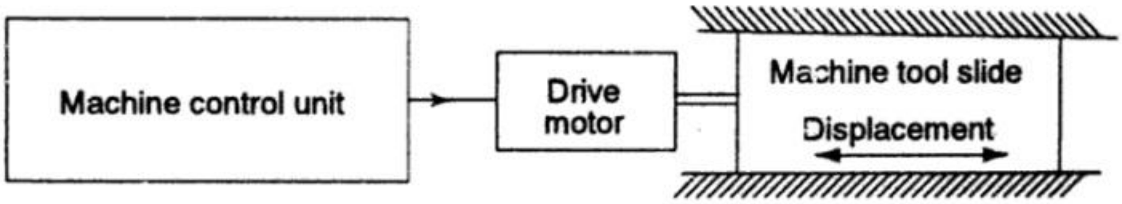


Figure 2.19 Schema of open-loop system, [20]

Closed loop type CNC Machine

The CNC systems works on servo mechanism. The detector of the position is attached to the machine table. The error is fed back to the control system. Actual displacement of machine slide is compared with input signal. In order to get a very high accuracy of the displacement the monitor device is used. The feedback can be measured in two ways: analog or digital systems. The analog systems measure the physical variables like position, velocity by voltage. The digital systems is monitoring electrical pulses to get operational condition through feedback systems and automatically adjusting the variations in real time. The schema of close-loop is showed on the Figure 2.20, [20, 16].

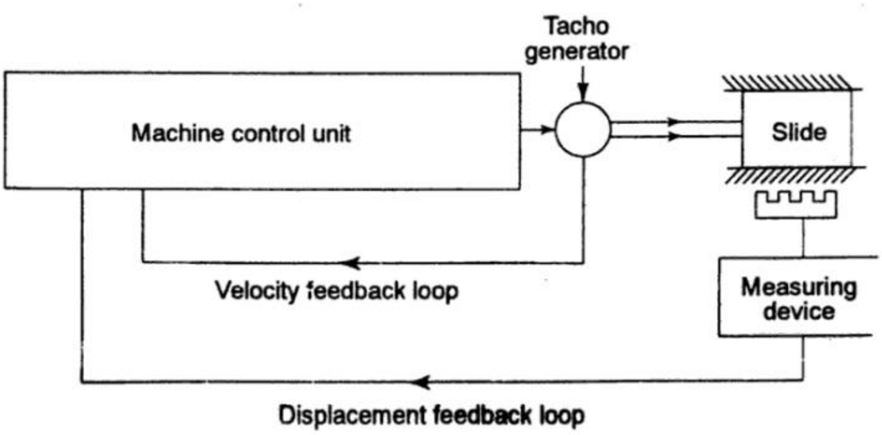


Figure 2.20 Schema of close-loop system, [16]

There are two types of closed system:

- closed-loop with negative feedback loop - initiates an action leading to a change in the value of the parameter that causes the disturbance / deviation to the opposite, thereby reducing the effect of such deviation, [32],

- closed system with positive feedback - initiate action which changes the value of the parameter that interferes with the work in the direction corresponding to the direction in which the deviation occurred, [32].

Semi-closed Loop

The semi-closed loop is one of the most popular control mechanisms in CNC. Position detector, which is attached to the shaft, detects the rotation angle. The schema of semi-closed-loop system is showed on the figure 2.21, [3].

To make the accuracy more precise there are designed the high precision ball screws. Thanks to precision of them the accuracy problem has been virtually defeated. Also to increase the position accuracy the two types of compensation can be used:

- pitch-error compensation, which at specific pitch the information about servo driver system are modified to remove the position error accumulation
- backlash compensation, which send to the servo driver system additional pulses corresponding with amount of backlash when the direction is changed.

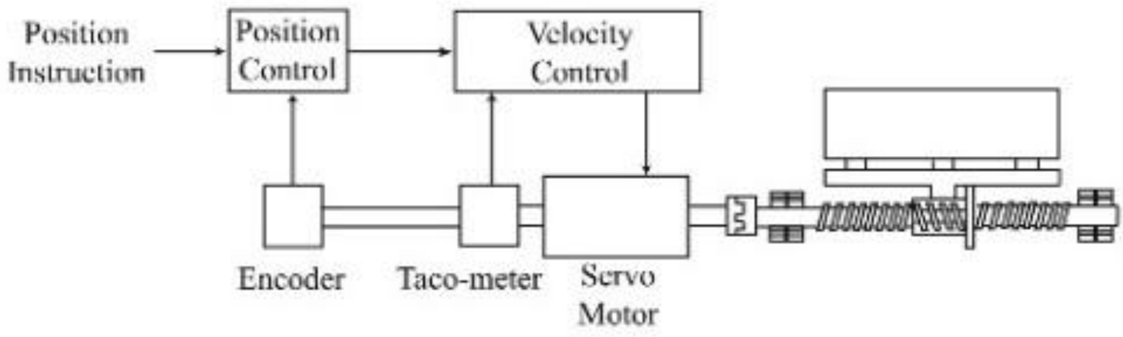


Figure 2.21 Schema of semi-close-loop system, [3]

Hybrid Loop

In the hybrid loop, there are two types of control loop working together. Semi-closed loop detects the position from the shaft of a motor, which give the advantage of control with high gain. It is possible control is possible because the machine is not connected to the control system. Closed loop, based on a linear scale, increases accuracy by compensating the error that the semi-closed loop cannot control. The schema of hybrid loop is showed on the figure 2.22.

By combining a closed loop and a semi-closed loop, it is possible to achieve high accuracy at high profit in a lightweight machine.

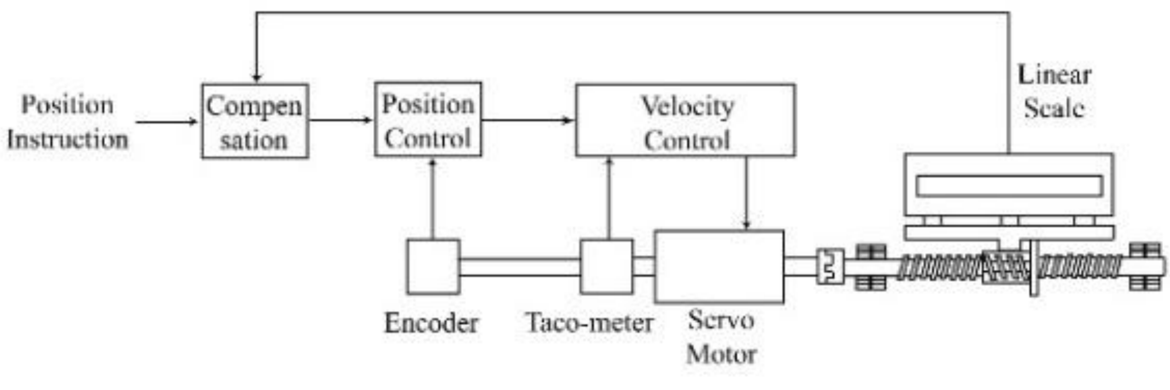


Figure 2.22 Schema of hybrid-loop system, [3]

2.3.2 According to type of tool motion control

Division of tool motion control is made due to tasks in technological process. There are three types of machine tools control system: point-to-point, straight line, contouring systems.

Point-to-point system

In point-to-point control system it is needed to determine the coordinates of the points, between which the traffic is going at any speed and along any road. Due to the shortening of the operation time it is advisable for the movement to take place at the maximum speed and in a straight line. During the movement of the object or tool, treatment does not take place. The point control principle is shown in Figure 2.23. The point control system is used in machines where it is important to accurately position the tool over the axis of the hole. Point control is used in measuring and assembly equipment, as well as drilling machines, boring machines, spot welding machines, [32].

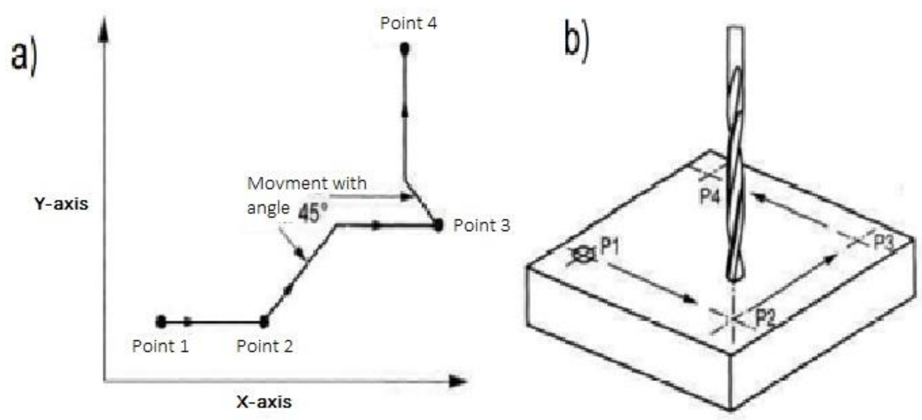


Figure 2.23 Point control principle: a) tool movement b) machining, [32]

Straight line control system

In the straight line control, the movement takes place at a predetermined speed and on a specific track, which is determined by the direction of the machine tool guidways. The tool path sections must be parallel to the axis of the coordinate system. The contour of the workpiece should be parallel to the control axes. Tool movement, on the other hand, can only take place in one of them. In the figure 2.24 is shown the examples of straight line control. This type of control is used in the planning and longitudinal turning of the shafts, in milling the rectangular outlines and milling flat-parallel with the milling heads, [32].

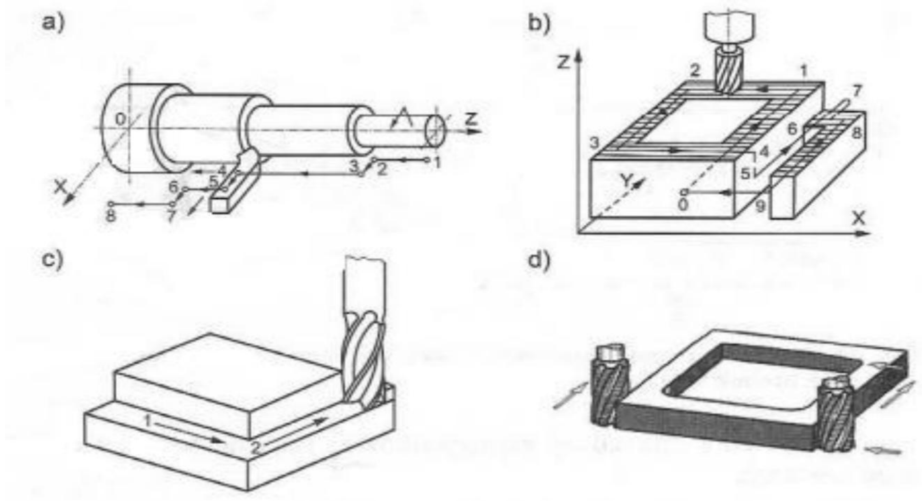


Figure 2.24 Examples of straight line control system: a) turning multistage shaft b) milling plane-parallel, c) counterboring, d) machining the sides contours, [32]

Continuous (contour) path control CNC machines

In continuous path control, the work piece path passes through the defined track points using the straight lines and curved lines sections. Machining of any surface is possible through the use of interpolation and the interaction of two or more motors of the motion motors. The interpolator calculates the intermediate points at which moves with variable speed and direction. Figure 2.25 is shown interpolation for two types of contour machining, [32].

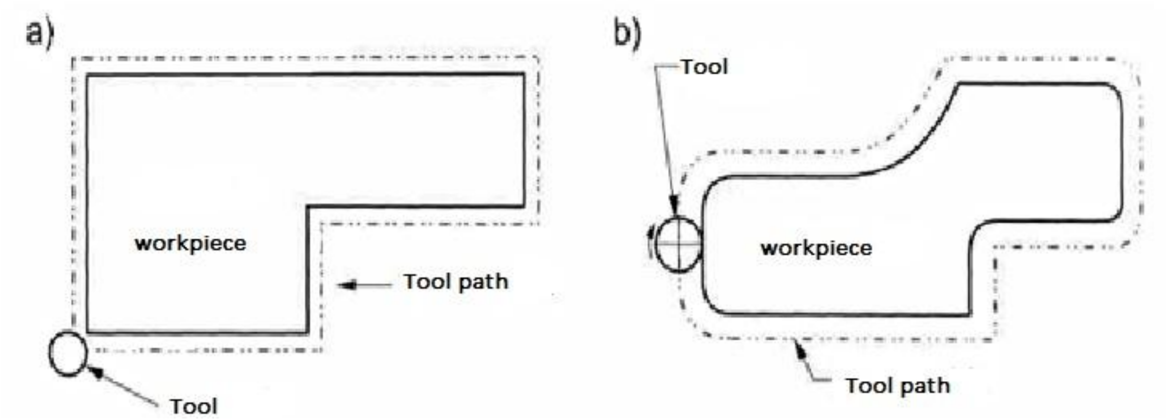


Figure 2.25 Continuous path : a) simple outline, b) complex lines [32]

Depending on the number of controlled axes are distinguished:

a) 2D contour control - the tool performs an arc, circle, or ellipse motion, allowing the work piece to have a symmetrical shape,
b) 2 1/2 D contour control - The tool moves on different planes of X / Y, X / Z, Y / Z coordinates and circular interpolation is performed only in one plane,
c) 3D contour control - allows to perform spatial movements of the tool, which are continuously controlled (3C - three feed motion, 4C - three feed motion and one rotary, 5C - three feed motion and two rotary movements).

Example of 2D contour control is represented on the Figure 2.26, [32, 4].

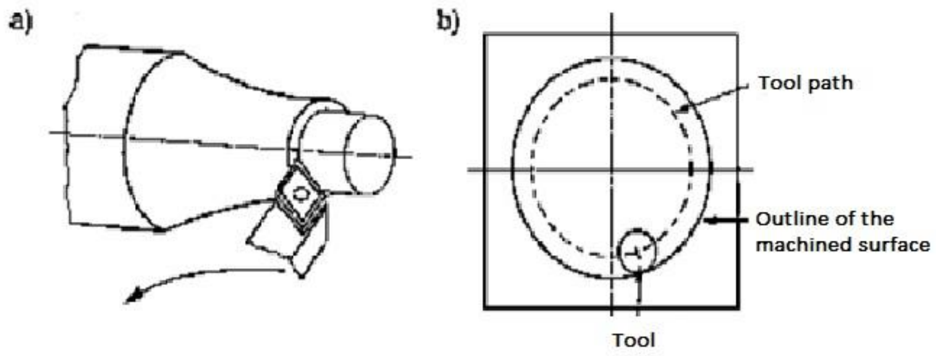


Figure 2.26 2D contour control: a) on the lathe, b) on the CNC milling machine, [9]

In practice, there are mixed control systems, i.e. point-to-point- straight line or continuous-straight line, [32]. The structure of the contour control system differs from the point-to-point- straight line control system.

2.3.3 According to program methods

Two types of co-ordinate systems are used in CNC machine to define and control the tool position in relation to the work piece. Absolute and incremental systems may be use independently or mixed within CNC part.

Absolute Programming CNC machine system

It is the most popular used form of dimensional programming format. The data of the points in absolute co-ordinate system always referrers to the same datum in the X-axis, Y-axis, Z-axis, which is called "Part Zero" or "Datum Zero". It is defined by operator before even the program is written. The major advantage of absolute system is facility of checking the code and correcting it. The error of one point will not affect the other program points. [20, 32, 16]

Incremental Programming CNC machine system

The data of the points in incremental co-ordinate system are referred to the previous point. If the mistake is made in the one of the point the further points also will have an error. This kind of programming is very helpful in the operations where some features change from job to job.

The differences between these two systems are better showed on the figure 2.27. In both programming methods is made the same contour. The points of absolute systems are made referring to point: X=0 and Y=0. The points of incremental system are made referring to calculation of distances between them.

Point	*Absolute system*	*Incremental System*
P1	1, 3	1, 3
P2	3, 2	2, –1
P3	4, 2	1, 0
P4	4, 3	0, 1

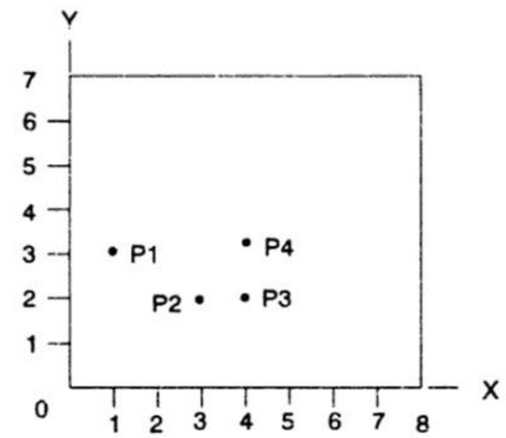


Figure 2.27 Difference in programming between absolute and incremental programming, [20]

2.3.4 According to type of controller

The two types of controller are distinguished in this kind of system: hybrid and straight controllers.

Hybrid controller CNC system

Hybrid defines the object made by combining two different elements; a mixture. In the case of CNC it means that control system is a mixture of hard weird logic circuits and soft weird computer. The hard weird logic circuits perform those functions for which they are suited like feed generation or interpolation. The soft weird computer performs all controls which are not associated with a conventional hard-weird controller. Schema of hybrid controller is showed on the figure 2.28, [27].

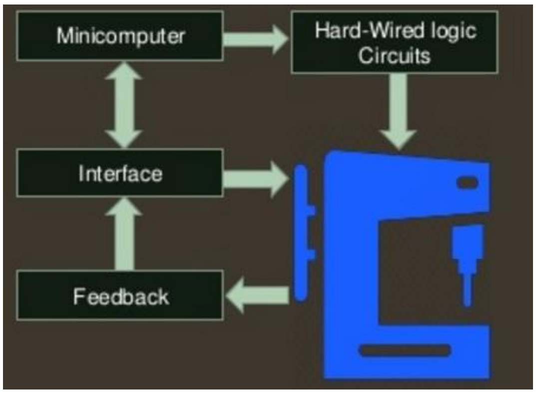


Figure 2.28 Hybrid controller CNC system schema, [27]

Straight controller CNC system

Straight controllers do not consist the hard wired logic circuits, which is the reason computer performs all the functions. The only hard-wired elements are those required to interface the computer with machine tool and display unit. The schema of straight controller is showed on the figure 2.29, [27].

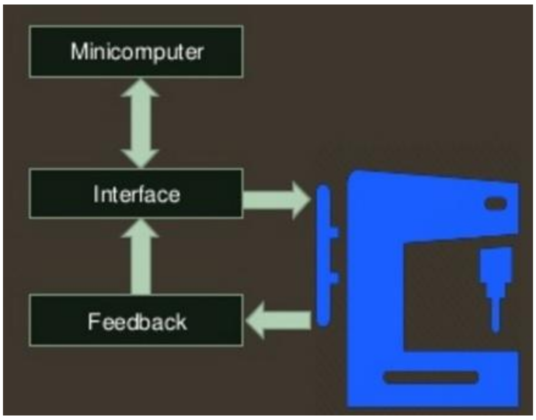


Figure 2.29 Straight controller CNC system schema, [27]

2.3.5 According to axis & type of operations

Two axis control system

The machine tool is classified as two axis machine if the machine tool can simultaneously control the tool along two axes. One of the examples is lathe machine, in which the job is rotating and the tool is moving in 2 axis: Z direction for depth and X direction for cut. The example is showed on the figure 2.30, [27, 16]

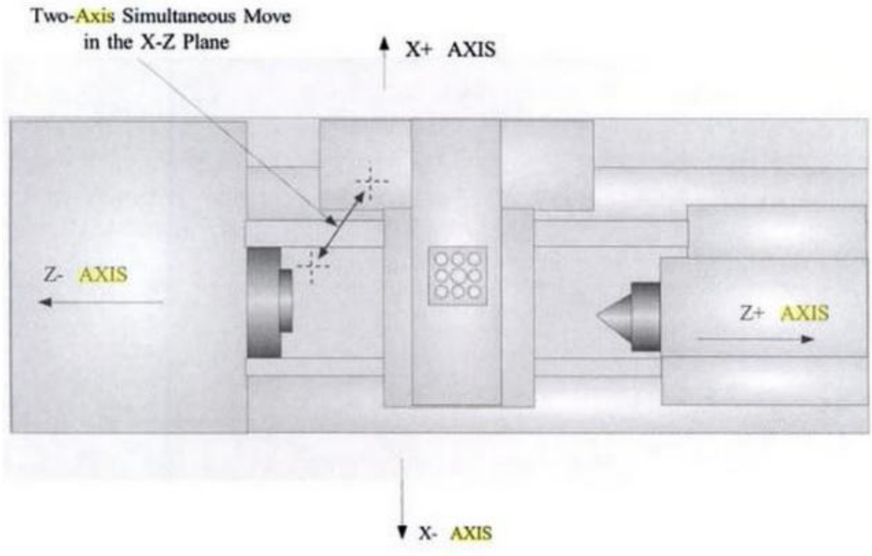


Figure 2.30 Two axis simultaneous machining (Lathe machine), [16]

Two and half axis control system

Two and half axis control system has its example in drilling and tapping machines, where XY position for the hole center and Z-axis completes a fixed cycle. It has the capability to translate in all three axes but only two of them can perform the cutting operation, [16].

Three axis control system

Three axis control systems are the most popular machines, which move in three dimensional. The servo motor controls the movements per the given instruction. The XYZ axis has verity of machines based on their bed lengths. Three axis machines more complex surfaces can be machine and parts can be produced with high accurate precision. The example is showed on the figure 2.31, [16].

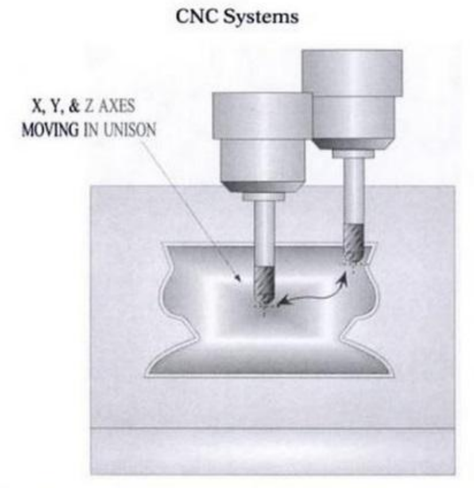


Figure 2.31 Three axis combination simultaneously on milling machine, [16]

Four axis control system

Four axis control system is a three axis machine with an extra rotation on B-axis. There are distinguished two types of four axis machines. One of them is vertical machine, in which the rotary head is added on the side of the machine bed. The other kind is horizontal 4-axis machine, in which the part is fixed on the rotation bed. Horizontal machines are more used for machining big parts in a single setup. The vertical 4-axis machines are limited only for the small jobs.

Five axis control system

Five axis control system is based on the three axis machines with an extra rotation along Y and Z directions which are called B-axis and A-axis. This rotations are possible thank to the bed and the spindle movement called PIVOT point. The biggest benefit of using the 5-axis machine is cycle time reduction even in the very complex parts. By use of single setup the accuracy on the positional errors improves. It also gives excellent access to the under cuts and deep pockets by tilting the tool. What is more the finished surface is very good and the tool life is longer. The example of 5-axis orientation on the milling and turning machine are presented respectively on the figure 2.32a and 2.32b, [16].

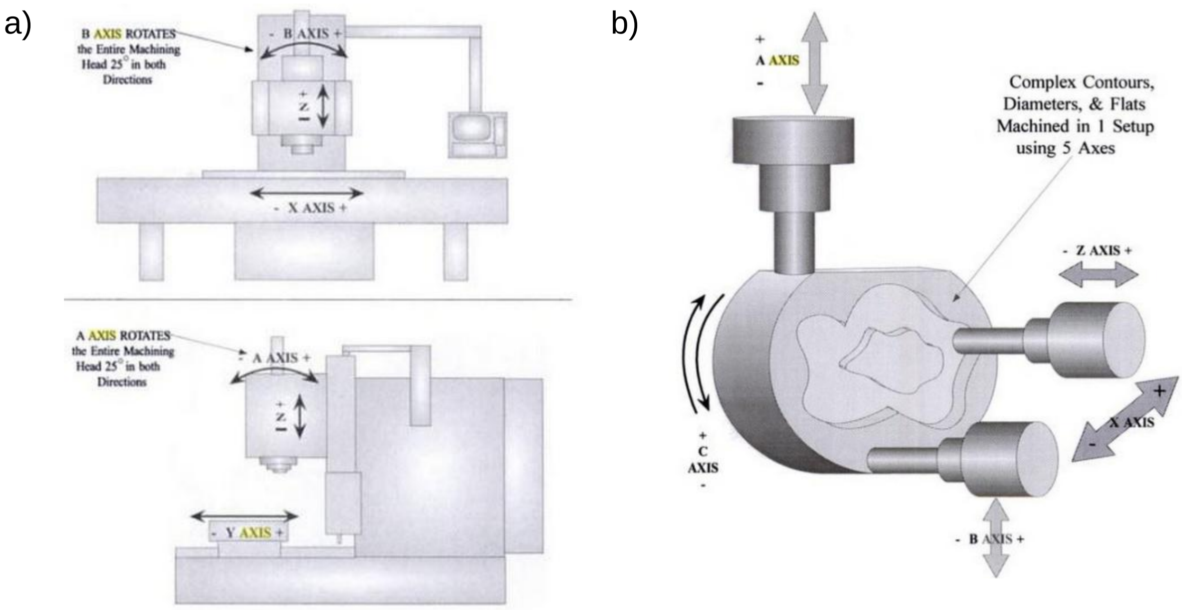


Figure 2.32 **Two axis simultaneous machining a) milling b) turning *, [16]***

2.4 CNC Programming Methods

CNC programming is a process which leads to generate a series of instruction for activating CNC machine tools in order to machine part. The programming consist such the processes as:

- analyzing the part drawing
- deciding the removal volume
- selecting machine
- decision of use chucks, setups, starting points, cut depths and machining sequences
- selecting tools and tool holders
- calculating the spindle speed, feed rate and coolant to be used
- writing/generating the part program
- verification of program
- machining

Programming process depends in majority on the on the machine's technical capabilities. Therefore, analyzing the data processing method can be distinguished several programming methods like manual, automatic and assisted computer programming.

2.4.1 Manual Part Programming

Manual programming is defined as direct editing the program with the commands provided by CNC equipment. The execution of part program can be both in CNC equipment and outside it. Mainly the efficiency and productivity depends on the knowledge of the programmer.

Manual programming is suitable for simple 2.5 and 3D shapes, where programming is very quickly made. It also gives opportunists for rapid modifying already existing program.

The disadvantages of this type of programming is that programmer needs to have a wide knowledge about machining theory, G-code programming, process planning and possible long training time. Programmer needs to learn new G-codes for each incompatible programming instruction in CNC systems. However, one of the biggest disadvantages of manual programming is that it is not adapted and suitable for complex 2.5D and 3D shape programming.

2.4.2 Automatic Part Programming

The automatic programming method can be divided into language-type and conversational programming method.

The tools and commands defined in language programming method are understandable by programmers, which after are changed into instruction understand by CNC machine. The most popular language-type programming are: APT, EXAPT, FAPT, KAPT, COMPACT II. The example of the EXAPT machining program is showed in the figure 2.33, [3, 31].

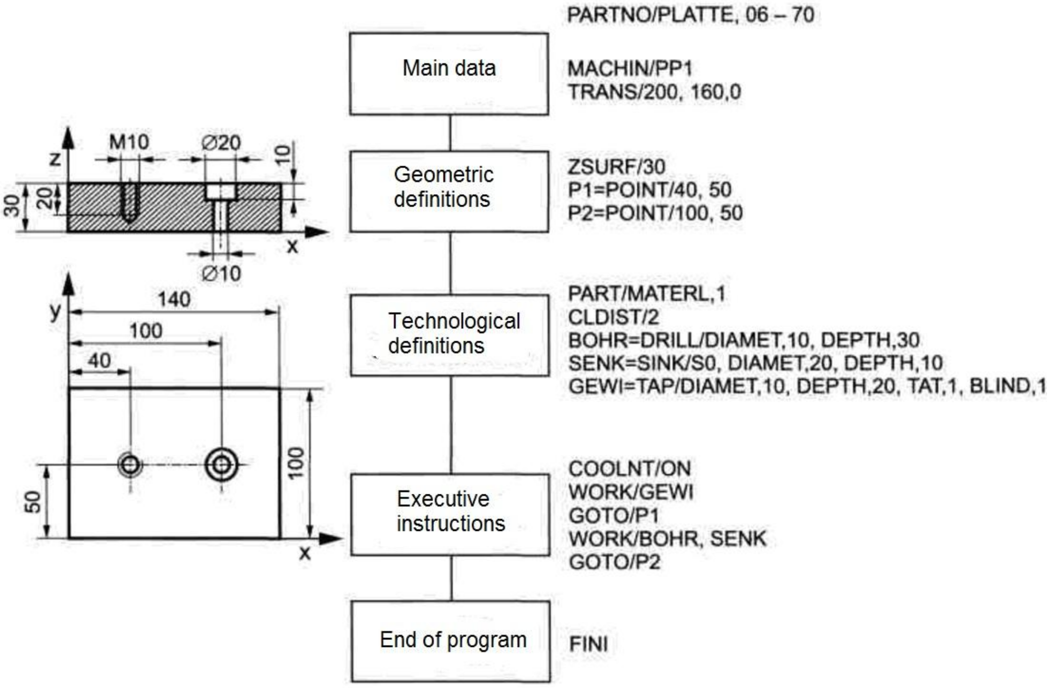


Figure 2.33 EXAPT machining program example, [3]

The conversational programming is also called an shop floor programming, because program is created during the machine is in setup .In this type of method, a programmer introduces data using the Graphical User Interface (GUI), which after are converted by CNC and is generated a part program. Conversational programming system is offline system; a part program is generated on an external computer rather than on the CNC system. Typically, conversational programming can be performed by an external CAM system. This kind of method is use when lot size is very small and the cycle time is shore.

2.4.3 Assisted Computer Programming

The assisted computer programming can be also called the CAD/CAM programming. It offers great part programming organization and flexibility. Work piece geometry is made in CAD system, which eliminates the need of define the work piece size and shape. After the part geometry is made, the programmer specifies how machining is to be done and all the data are sent to CAM systems. The CAM system generates a CNC program, which is the same type of program a manual programmer will write. This program is imported to the CNC machine and run, [31, 24, 26].

CAD/CAPP/CAM in CNC machine

The task flow, which is needed for manufacturing a part using CNC machine, is classified as the following three types:

- offline tasks: Computer Aided-Design (CAD), Computer Aided Process Planning (CAPP), Computer Aided Manufacturing (CAM),
- online tasks: CNC machining, monitoring, on-machine measurement,
- post-line tasks: Computer-Aided Inspection (CAI), post-operation.

The architecture of CNC processing is showed on the figure 2.34, [3].

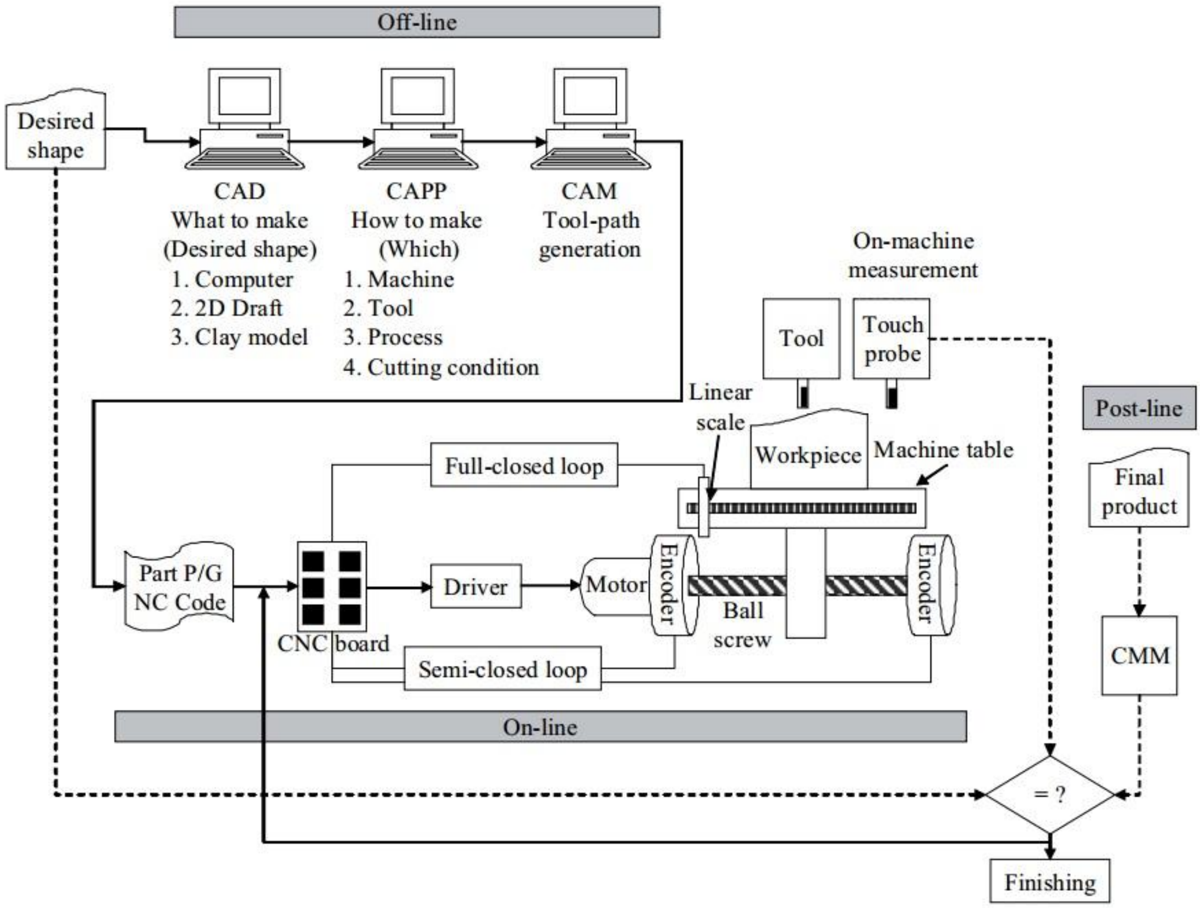


Figure 2.34 The task flow on CNC machine tools, [3]

The offline stage starts after the shape of the part has been defined. The geometry model of the part is designed in 2D or 3D CAD program. CAD can be characterized as any type of design activity use of the computer system to help with creation, modification, analysis and optimization of part manufacturing. CAD system software originally was made with computer language FORTRAN but through rapidly changes nowadays CAD system is built using the interaction of graphical user interface (GUI), NURBS geometry and, but not necessarily, boundary representation (B-rep) data via a geometric kernel modeling, [26, 25 3]

The use of CAD system helps in reduction the production time to even 70%. This reduction is the elimination effect of preparing the documentation, correction of it, and effective creative work, which occur in the conventional manufacturing. Thanks to CAD tools the full documentation of parts is written in the database, which helps in future modification of projects as well as in development of new projects on the already generated ones. The CAD system is used in all engineering process by providing parts specification or performing complex calculations. Another extraordinary benefit of using the CAD system is an accurate visualization of the object, superior accuracy of performed part, stress-strain analysis of components, heat transfer calculation and dynamic response of mechanisms. The most popular CAD programs include: Autodesk (AutoCAD, Inventro), CATIA, SolidWorks, Rhinoceros 3D, PTC Creo, SolidEdge, [9, 3, 25,26].

Computer Aided Process Planning comes next after geometric model is finished. It is a link between CAD and CAM. This stage includes the selection of machine tools, tools, jig, cutting conditions, creation of schedule and machine sequences. Process planning is generally very hard to automatisms because depends on the knowledge of process planner. CAPP systems are currently under development, which is the reason of small number of commercial solutions. One of the few CAM solutions supporting CAPP systems is the ADEM or CAMWorks system that help the design of a CAPP process and automatically generates it, [3].

Computer Aided Manufacturing generates a part program based on CAD information about geometry of part and CAPP machining information. CAM configures and controls the manufacturing process for CNC machines. Program based on the introduced model optimized tool wear, reduce machining time and avoid idle tool passages. The structure of CAM system is shown in the figure 2.35. The main advantages of using this type of programs are full instrumentation capacity and optimal production capacities. The most popular CAM systems include: SolidCAM, MasterCAM, GibbsCAM, EdgeCAM, CATIA, PTC Creo [9, 3, 25, 26].

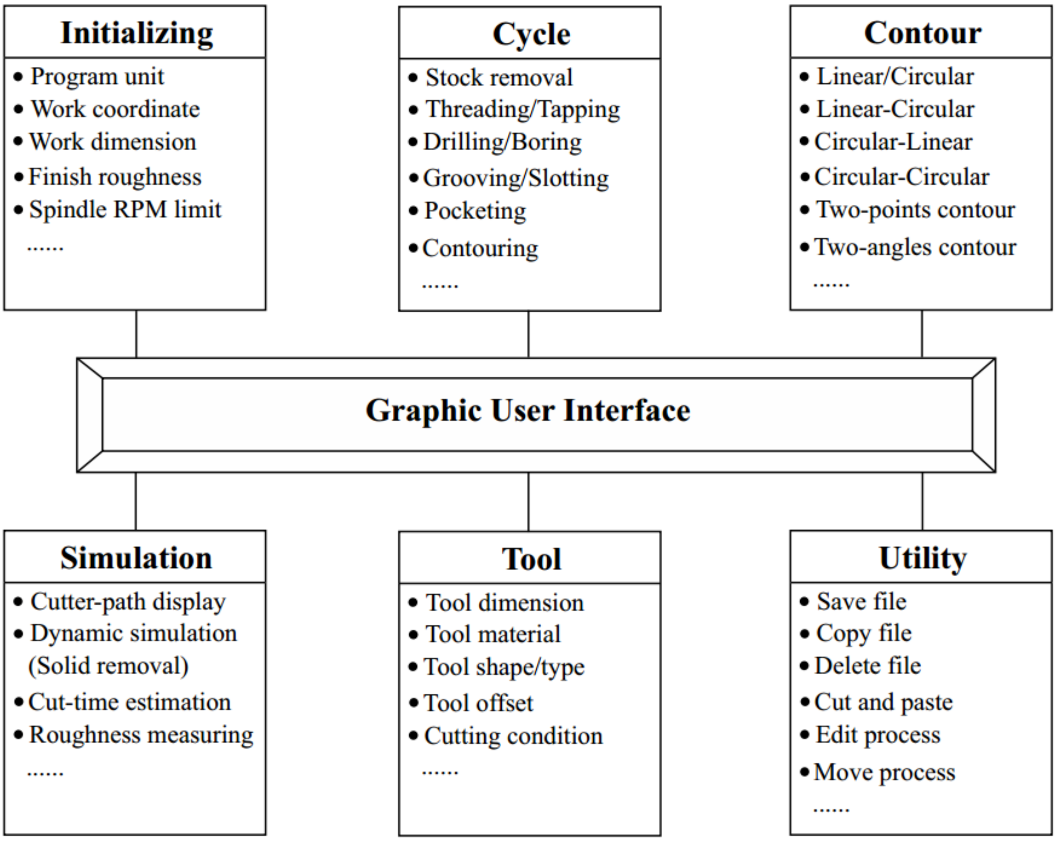


Figure 2.35 CAM system structure, [3]

The good collaboration between this 3 offline tasks is very important in improvement of production systems. The collaboration of CAD systems with CAPP and CAM systems provides the ability to use a common database containing product descriptions, which mostly are geometric model data (Figure 2.36). Systems generate technological data from one to other, which simplify a checking system. Unfortunately sometimes the CAD/CAPP/CAM is incompatible in means of differences in hardware and software. The interface needs to be configurable in order to their work together, [3, 9, 25].

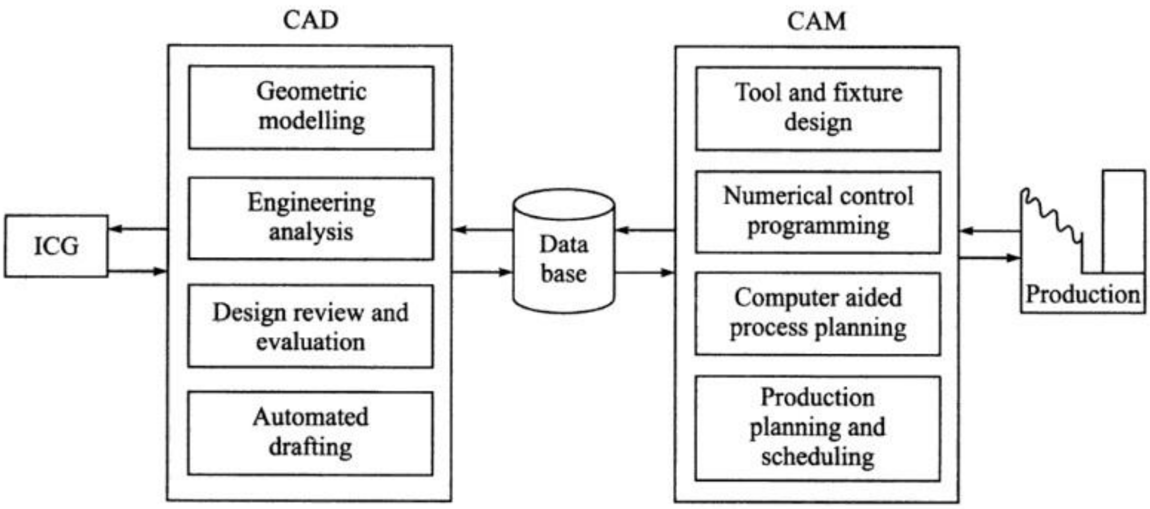


Figure 2.36 Relationship of CAD/CAM databases to CAD and CAM, [25]

Online tasks are the ones which use the NC machines to produce the part. The program can be uploaded using floppy disk, can be sent on the machine or can be written directly at the machine, where after can be edited. Program is machine-understandable instruction, which controls servo motor and gives idea about position, velocity, program movements. With the rotation of servo motor linear movements are transformed via ball-screw mechanism and the part is machined. Transformation of the instruction from NC to machine movements is going to be described further. Machine components should be designed to prevent high temperatures and vibrations. During machining the operator can observe the status of the machine and machining process on CNC board panel, [3].

The post-line task is responsible for inspecting the finished part. The supervision use Coordinate Measurement Machine (CMM), which measure the differences between the geometry model and already made part. The compensation is executed by modifying tool or by post-operates the part, [3].

2.5 *Structure of the control program*

A machine tool program is a sequence of instructions that encodes a tool's motion relative to a work piece by writing the coordinates together with complementary instructions. The motion record, the steps that must be made to perform the object of given dimensions, shape and roughness is called a block. Such actions include replacing tools or turning on cooling. Each block is written in a separate line and deals with one activity. The entire program consists of sequences of blocks that is read and analyzed wholeheartedly to ensure computational continuity. The program executes the corresponding blocks in a sequence. The NC program consists of components such as [13]:

- beginning of the program,
- sequence of treatment blocks,
- end of program.

The first of the blocks is called the program header. Its role is to store information about the data type and program name. The maximum name length and the use of accepted characters depend on the control system. The correctness of the name is important in the transmission of the names to the CNC control system, which creates a file with the name taken from the header. The file name may be different than the one that is stored on your computer. It is recommended to keep the headers compatible with the filename, [19].

The last block, contains information for the control system about the end of the program. M30 or M2 are the signs of program's end.

2.5.1 Block's functions

The block consists of words of any order that evoke control system functions. The last word of the block is LF, Line Feed, or *. Thanks to this element, the control system executes sequences from the beginning to the end of the block, which may occupy one or more lines. The word block consists of an address and a value, expressed by the word and number. The structure of CNC program is showed in the figure 2.37 and the block configuration is showed on the figure 2.38, [4, 12, 19].

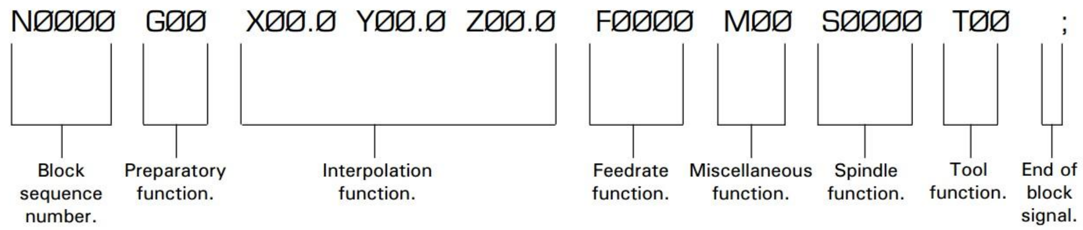


Figure 2.37 Block configuration, [4]

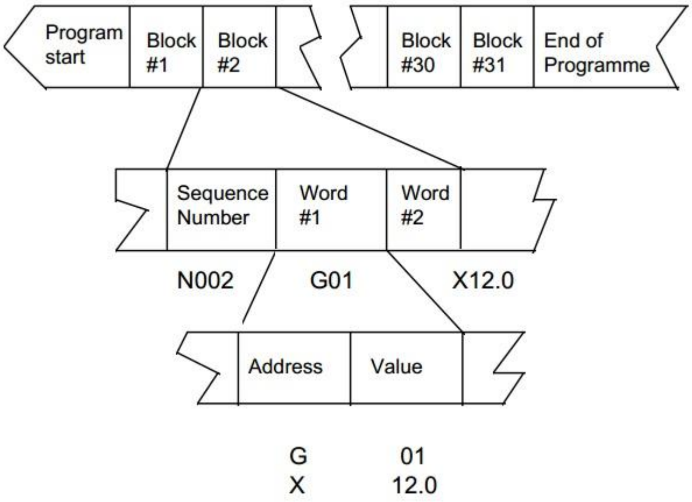


Figure 2.38 Structure of CNC program, [4]

There is a difference between the words:

- P - program number
- N - block's number,
- G - preparatory functions,
- F - feed rate
- S - spindle speed
- T - tool number
- D - tool registry number
- M - machine functions
- L - number of repetitions of subprogram
- A, B, C - coordinate values in rotary axes, in deg
- I, J, K - axis interpolation parameters
- X, Y, Z - values coordinate in axes, in mm and inch

a) N - block's number

Block number does not call any function, so it is not mandatory to use but it is recommended. It makes the work easier for the operator and the programmer by indicating where in the program is currently executed operation. This address is especially useful for searching for program errors. There are no set standards for block numbering. Mostly blocks are numbered ascending.

b) G - preparatory functions

Like the number of block N, the preparatory functions do not call up any machine tool activity, however, are defined as one of the most important addresses. The task of the preparatory function is to interpret other addresses. There are global addresses that are valid for the entire program and local addresses for the block in which they were invoked. It is important that only one function in the group is active. If a function from group G is called, the previous function in this group ceases to function and becomes inactive. Examples of preparatory functions are shown in figure 2.39.

Note - Not all G codes apply to each machine.

G Code.	Group.	Function.
GØØ	1	Positioning (Rapid Traverse)
GØ1	1	Linear Interpolation (Cutting Feed)
GØ2	1	Circular Interpolation CW
GØ3	1	Circular Interpolation CCW
GØ4	Ø	Dwell, Exact Stop
G2Ø	6	Imperial Data Input (Inches)
G21	6	Metric Data Input (Millimetres)
G28	Ø	Reference Point Return
G4Ø	7	Cutter Compensation Cancel
G41	7	Cutter Compensation Left
G42	7	Cutter Compensation Right
G73	9	High Speed Peck Drilling Cycle
G74	9	Counter Tapping Cycle
G76	*9*	*Fine Boring Cycle(not recommended on Denford Machines)*
G8Ø*	9	Canned Cycle Cancel
G81	9	Drilling Cycle, Spot Boring
G82	9	Drilling Cycle, Counter Boring
G83	9	Deep Hole Peck Drilling Cycle
G84	9	Tapping Cycle
G85	9	Boring Cycle
G86	9	Boring Cycle
G87	*9*	*Back Boring Cycle(not recommended on Denford Machines)*
G89	9	Boring Cycle
G9Ø*	3	Absolute Zero Command
G91	3	Incremental Command
G94*	5	Feed per Minute
G95	5	Feed per Revolution
G98*	1Ø	Return to Initial Level in Canned Cycle
G99	1Ø	Return to R Point Level in Canned Cycle
G17Ø	Ø	Circular Pocket Canned Cycle
G171	Ø	Circular Pocket Canned Cycle
G172	Ø	Rectangular Pocket Canned Cycle
G173	Ø	Rectangular Pocket Canned Cycle

Code listing full and correct at the time of printing.

Figure 2.39 G functions for FANUC milling machine, [4]

c) M - machine functions

Machine functions are used to control discrete axes. Table 1 explains the importance of the basic M codes functions. Code M02 and M30 call the function to stop the program even if there are still blocks in it after the code is executed. Similar action has an M17 function that is not included in the table below, [19].

M00	**Program Stop**
M01	Optional Stop
M02	Program End
M03	Spindle Clockwise
M04	Spindle Counter Clockwise
M05	Spindle Stop
M06	Tool Change
M07	Thru Spindle Coolant ON
M08	Flood Coolant ON
M09	Coolant Off
M10	Table Pallet Clamp
M11	Table Pallet Unclamp
M12	Shower Coolant On
M14	Spindle Air Blow On
M15	Spindle Air Blow Off
M16	Air Blast/Tool Change
M18	Air Blast Off
M19	Spindle Orientation
M29	Rigid Tapping
M30	End Program
M60	Pallet Change
M61	Load Pallet #1
M62	Load Pallet #2
M98	Call Sub-Program I
M99	Return to main Program from Sub-Program

Tab. 2.2 **M function, [33]**

d) F - Feed function

The feed rate is programmed on the beginning of the program under the F address except for rapid traverse. The following formula helps to calculate feed rate:

$$Feed\ Rate = \frac{Chip\ Load}{tooth} \times No\ of\ tooth \times Spindle\ Speed$$

Feedrate can be measure per minutes or per revolution.

e) S - Spindle Function

Before starting the S spindle speed values should be check in the documentation of machine to verify its acceptable range. The specified rotational speed is only achievable after the spindle has sped up, so it is necessary to program the spindle speed sufficiently early enough for the tool to reach the desired speed before cutting starts.

The speed value can be calculated from the given formula:

$$Spindle\ Speed = \frac{Surface\ Cutting\ Speed\ ({}^{m}/_{min}) \times 1000}{\pi \times Cutter\ Diameter\ (mm)}$$

f) T - Tool Number

Most modern numerically controlled machine tools are equipped with a tool magazine from which tools can be automatically taken during the program is running. Every individual tool has a specific number. A special board with all tool numbers and its characteristics (diameter, length et.) is stored in the control unit memory.

The T function usually causes the tool magazine to rotate or move in such a way that it is possible to retrieve and attach a tool in the spindle using a special machine tool. The tool table looks is presented on the figure 2.40.

The basic data tools are:

- tool type - marked by icon,
- tool name - the unique tool name should include dimensions such as diameter,
- length - tool length from spindle face to characteristic point e.g. 100 mm,
- diameter - tool diameter,
- other - depending on the type of tool, e.g. apex angle,

– the number of cutting edges,
– direction of spindle rotation,
– if the coolant is to work, [23].

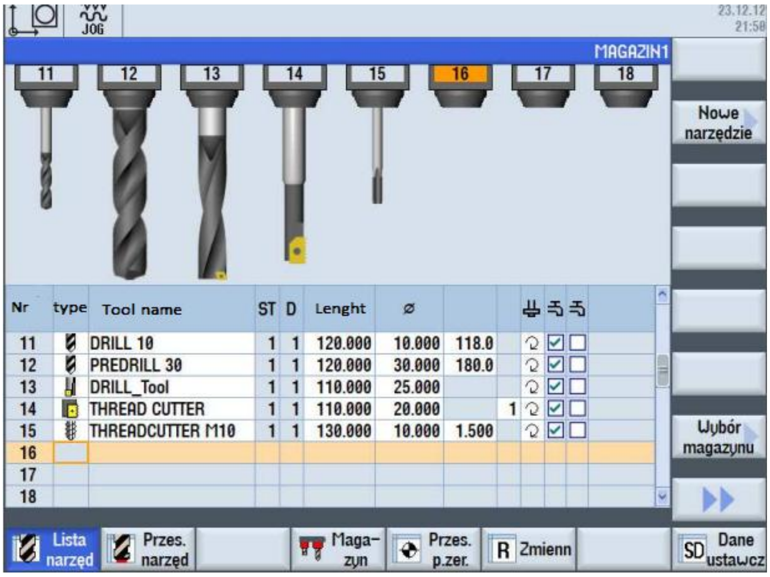


Figure 2.40 The tool table, [23]

g) Subprogram

Main program can have the instruction that is used for calling subprograms. After executing of subprogram, the instruction for returning to the main program is demand and the main program is continued at the block after the instruction that called the subprogram, as shown in figure 2.41. It is also possible to call other subprogram from the subprogram, [3].

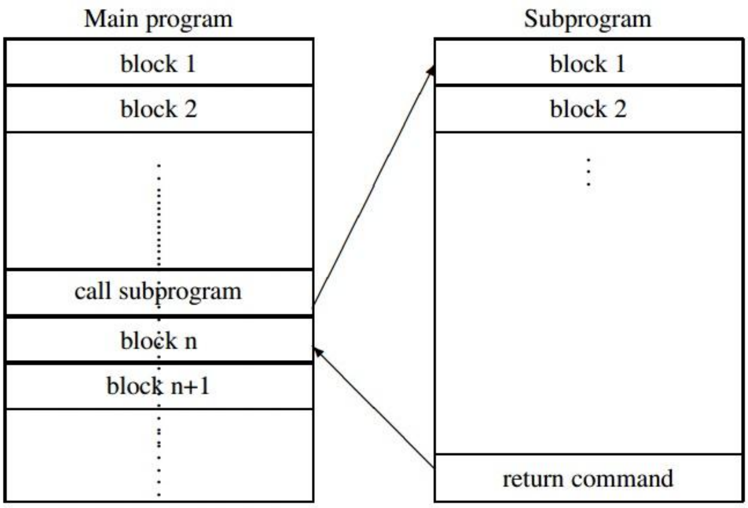


Figure 2.41 Main program* and *subprogram execution, [3]

2.5.2 Example of CNC Code

In the figure 2.42 - 2.43 an exemplary machining contour and used treatment solution are shown. In N5, the most important preparatory functions were recalled, starting with the interpretations of successive blocks:

G40 – end of correction,

G54 – recalling the first zero point offset register (selecting the object coordinate system),

G71 – programming in basic units,

G90 – absolute positioning,

G94 – programming of feed rate in [mm / min].

Block N10 includes:

T1- recall of tools,

M6 - insert tool into spindle,

D1 - set the tool registry for the tool,

S – spindle speed,

M3 - turning clockwise (right) rotation,

F – feed rate setting,

M8 – turn on the coolant.

The next blocks are the movement of the tool. In N15, positioning in the XY plane is made over the starting point of the contour with rapid movements G0. Block N25 includes setting in the Z axis with G1 workflow. Blocks N30 to N80 program the tool movement in the XY plane. Block N85 is the withdrawal of the tool in the Z-axis rapid traverse G0. In the N90 block, the coolant pump is switched off with the M9 function and the spindle speed is switched off with the M5 function. Then M6 indicates the retraction of the tool from the spindle to the tool magazine. While T0 ensures that no new tool will be introduced. The sequence of words G53 T0 and D0 is the transition to machine programming. The last block is the M30, which is the end of the program.

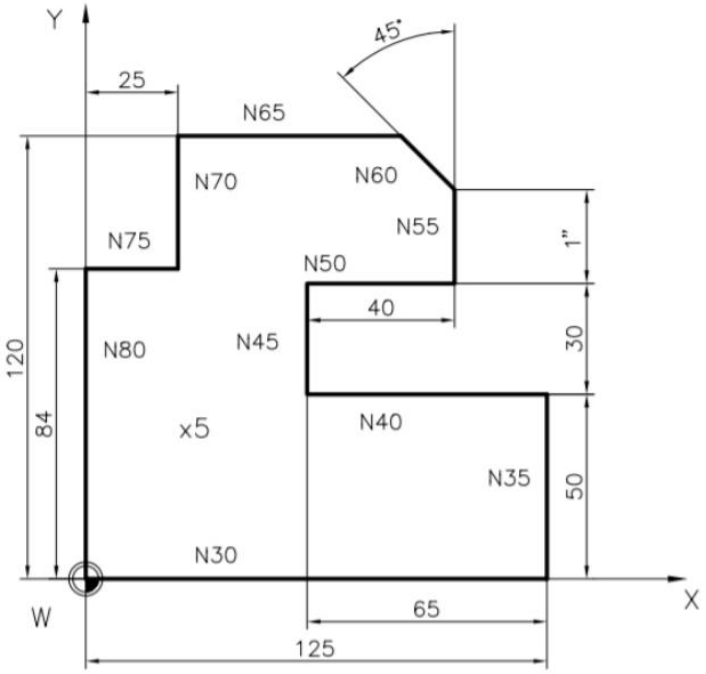


Figure 2.42 Sketch of an object for example of contour processing with linear interpolation, [19]

```
%_N_EX01_MPF
; 11-08-2003
N5 G40 G54 G71 G90 G94
N10 S800 F100 T1 D1 M3 M8 M6
N15 G0 X0 Y0
N20 Z3
N25 G1 Z-5
N30 X125
N35 Y50
N40 X60
N45 Y80
N50 X100
N55 Y105.4
N60 X85.4 Y120
N65 X25
N70 Y84
N75 X0
N80 Y0
N85 G0 Z100
N90 G53 T0 D0 G0 X300 Y300 Z200 M9 M6 M5
N95 M30
```

Figure 2.43 Code example, [19]

The block is used to program the tool path. For this purpose, the block must contain information about the start and end points of the movement, speed and trajectory. The continuity of motion is achieved by establishing the final point of one block as the starting point in the next block. By speed is meant the feed rate F, which is the work motion or rapid motion. The tool path is determined by the interpolation, [19].

2.5.3 Interpolation

Interpolation is a numerical method that describes the path of a tool movement between two points through a mathematical equation. In CNC programming, at least four types of interpolation are distinguished.

1. Linear interpolation

It is one of the basic and simplest interpolation. It is expressed by the function G1. Tool movement (requires feed rate F) takes place in a straight line between the two points. An example of linear interpolation is shown in Figure 2.44 ,[19].

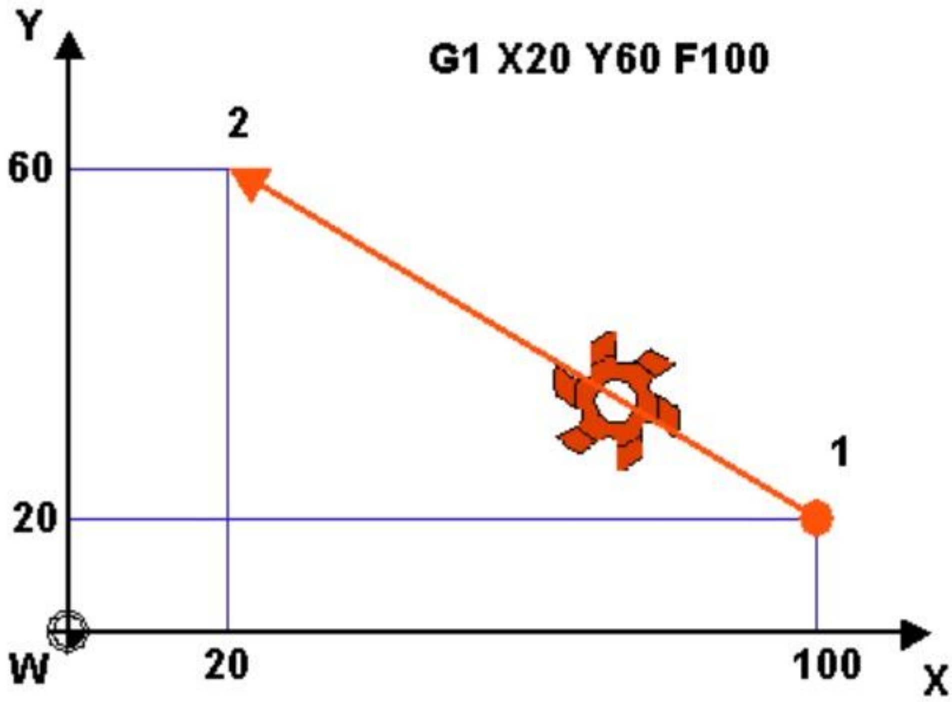


Figure 2.44 Example of linear interpolation G1, [19]

2. Point interpolation

Point interpolation involves the movement of a tool by quick motion to the endpoint of the program at high speed. Movement in the numerically controlled axis occurs on an unpredictable path, RTLIOF (Rapid Tool Linear Interpolation Off), or in a straight line, RTLION (Rapid Tool Linear Interpolation On). Point interpolation is used by entering G0 (Rapid Positioning/Rapid Traverse) into the machine tool program. As a result of the use of point interpolation, the tolerance of positioning accuracy at the target point is increased. Therefore, it is used for constant tool movements. An example of linear interpolation is shown in Figure 2.45, [19].

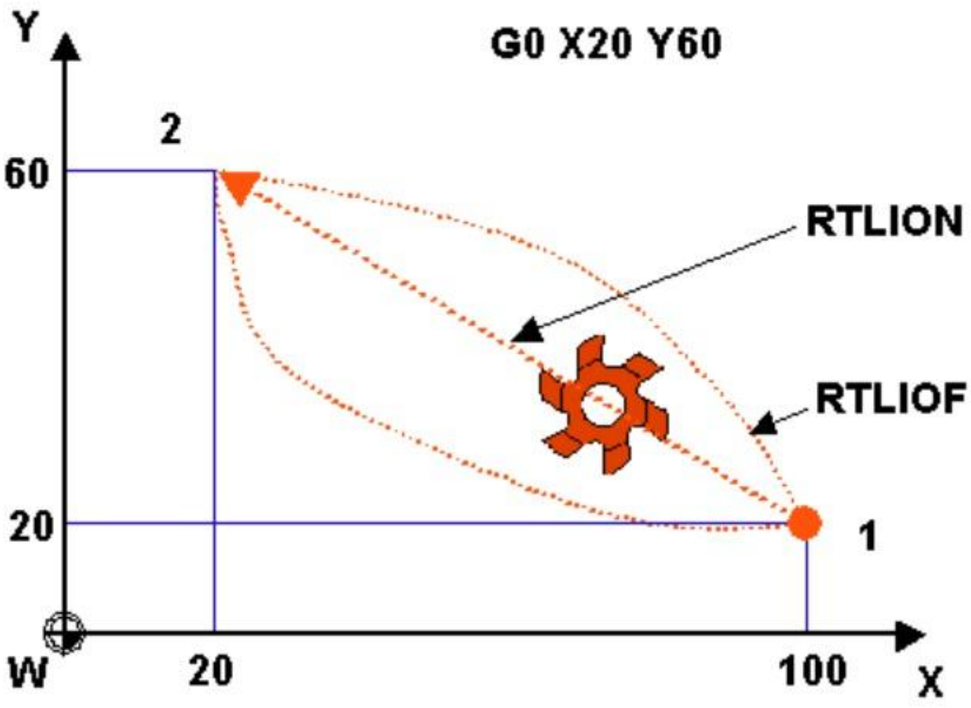


Figure 2.45 Example of point interpolation G1, [19]

3. Circular interpolation

Circle interpolation requires precise parameters of the circle such as radius, center of circle, and direction of movement of the tool. For clockwise directions G2 is used, while for directions opposite to clock G3. An example of circular interpolation with different directions of motion is shown in the figure 2.46

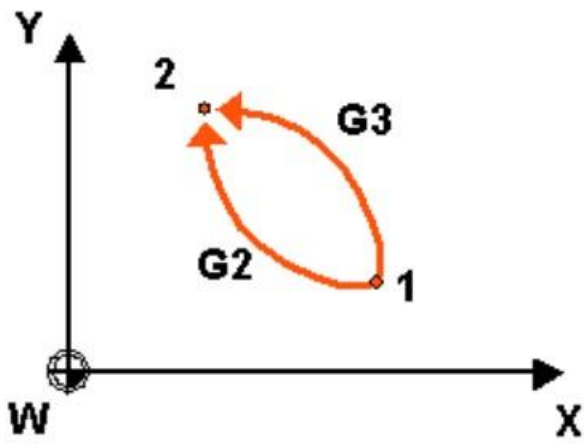


Figure 2.46 Example of applying different directions in circular interpolation, [19]

2.6 NURBS

There are various mathematical models, like cubic-spline, Bezier, B-spline to represent free-form curves. But the Non-Uniform Rational B-Splines (NURBS) is the most general model covering others as special cases. It finds their place in visual art industries like film, entertainment and art graphic. NURB-splines curves have a lot of to offer, that is why they are interest of mechanical industries and computer aided design community. These types of curves are especially useful for programming curved paths. As a rule, complex shapes require the creation of monstrous programs and use of speed tolerance. Due to the large amount of data, the program may stop working. The solution to this problem is to use the spline interpolation. Experience shows that a block with NURBS functionality can be exchanged between 5 and 10 linear blocks without loss of accuracy, [11, 22].

Non-Uniform Rational B-Splines is defined by the equitation:

$$C(u) = \frac{\sum_{i=0}^{n} N_{i,p}(u)\omega_i P_i}{\sum_{i=0}^{n} N_{i,p}(u)\omega_i} \qquad a \le u \le b$$

where:

P_i - control points,

ω_i - weights,

$\{N_{i,p}(u)\}$ - pth-degree B-spline basic function.

(*Formula 2.1*)

The pth-degree B-spline functions is defined on the nonperiodic knot vector, [22]. Example of the NURBS curve is showed on the figure 2.47

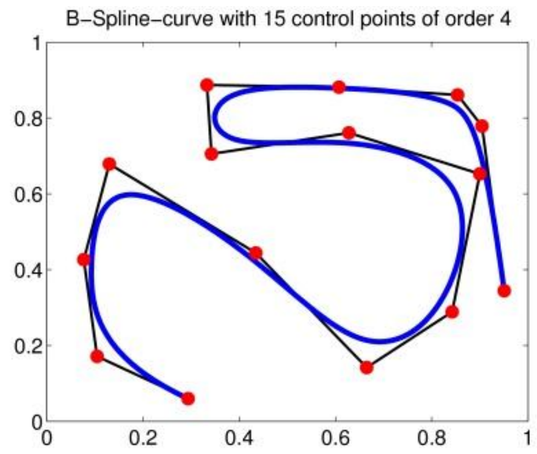


Figure 2.47 Example of work the NURB curves, [32]

2.6.1 NURBS interpolation

There are two methods of use the NURBS curves:

- indirect, which means work based on making a small line segments, (linear interpolation)
- direct, data (control points, weights, knot vector) are inputs for the CNC system (spline interpolation)

Difference between the interpolation methods based on line-segment and a NURBS curve is showed on the figure 2.48. The offline CAD software is used to create free-form curve geometry. In interpolation method based on the small line segments CAM is used to pre-define the maximum machining error (tolerance) by a set of line segments. In the direct NURBS interpolation CNC system generates interpolation points based on the NURBS curve data, the programmed feed rate and the tolerance to enable high-speed and high-accuracy machining performance.

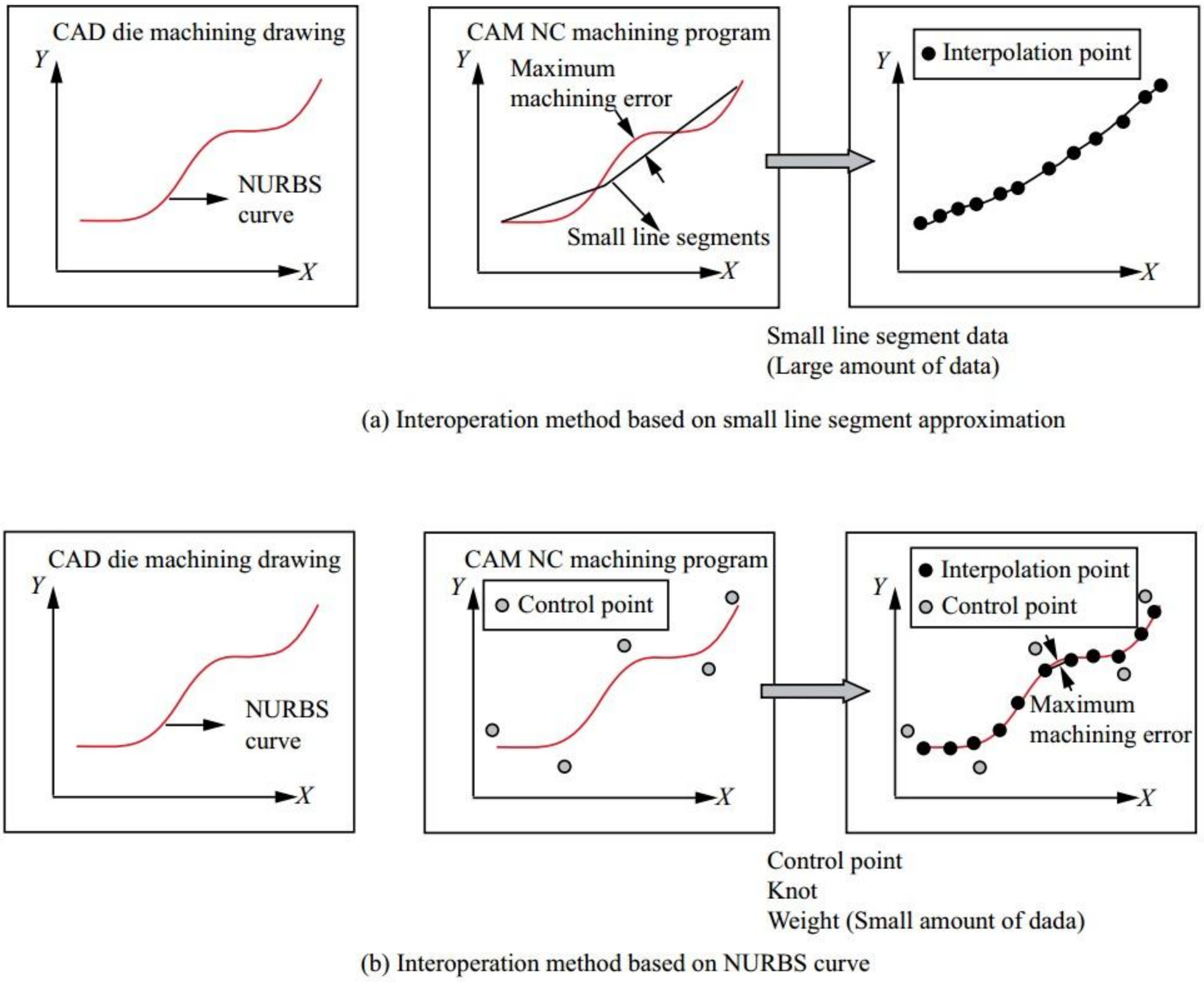


Figure 2.48 Direct and indirect NURBS interpolation, [3]

To use NURBS in the direct way is needed to input data like control points, weights, and knot vector into CNC G code. The example for Fanuc system is given below on the figure 2.49. In Fanuc system the NURBS interpolation is given by G06.2 function [3,6].

G06.2 P_X_Y_Z_R_K_F ;
G06.2 : NURBS interoperation
P : NURBS curve order
X, Y, Z : Control point
R : Weight
K : Knot
F : Feedrate

Figure 2.49 FANUC system NURBS G-code format, [3]

Despite of its advantages, spline interpolation requires of expensive controls, and it is not common in conventional machines. This is the reason for the further researches of replacing the spline interpolation by the easier method need to be done.

2.7 Look - ahead

CNC machine have become more accurate, faster and possible to machine more complicated parts. In order to fulfill the requirements, new advanced function of CNC machine needed to be developed. One of them is Look-ahead function.

The realization of part program consisting of block with short length and fast feed rate may tend to slow down. Program is read by each block and executed line by line. It provokes that feed rate is willing to decline and can occur even lower than a programmed feed rate. What is more the feed rate at the corners between one specific block and next becomes discontinuous.

Since the acceleration, deceleration and discontinuity of feed rate is bases for the good machining surface quality, the function which can read forward the program needed to be developed. This function is called Look-ahead.

Look ahead read forward numerous block and calculates the adequate, maximum allowable feed rate for each axis. It is applicable for free-form surfaces and complicated shape contours, where the movement between blocks should not be stopped at high speed. The idea of look-ahead is presented on the figure 2.50. The program without the look-ahead function cannot achieve the programmed feed rate, where with look-ahead the feed rate has all the time on the same value. With this function, reduction of machining time is more possible. The Look-ahead function requires much computing power. Improved CPU power gives higher number of reading forward blocks, even to the thousands of them, [3].

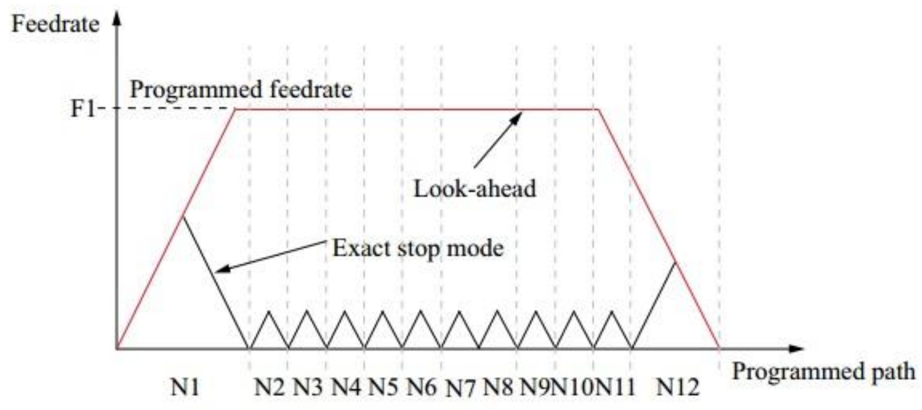


Figure 2.50 Differences in feed rates with a look-ahead and without look-ahead, [3]

2.7.1 Look-ahead velocity calculations

A Look-ahead algorithm calculates feed rate on the beginning and the end of each block based on the length of the successive blocks and the maximum allowable acceleration. The speed on the beginning of the current block is calculated as following equations

$$V_0 = \sqrt{V_f^2 + 2 \times A \times L}$$

where,

V_f - feasible entry feed rate to the next block,

V_0 - feasible entry feed rate of the current block,

A - maximum allowable acceleration

L - length of current block

(Formula 2.2)

If the feed rate on the entrance to the next block from Formula 2.2 is smaller than the programmed feed rate, the feasible entry feed rate can be defined:

$$V_f = \begin{cases} V, V < F \\ F, V > F \end{cases}$$

(Formula 2.3)

If the feed rate on the entrance to the current block - V_0, is larger than the programmed feed rate F, the feasible entry feed rate of the current block is F and the end feed rate is V_f. This start conditions are represented in the figure 2.51, [3].

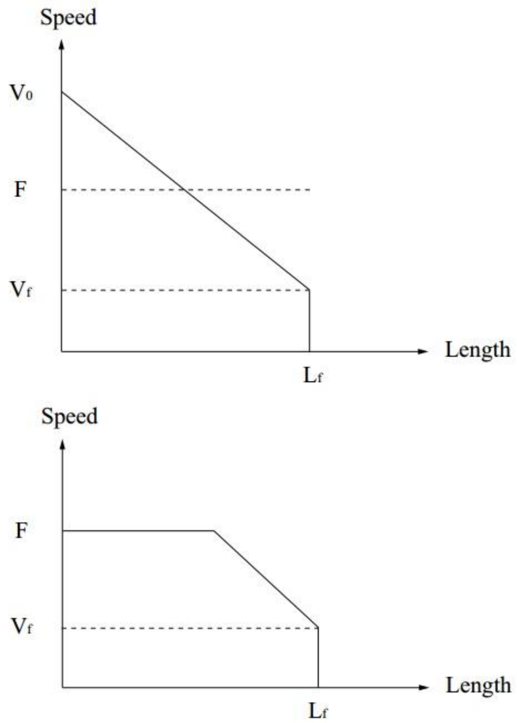


Figure 2.51 Start conditions and speed profile with Look Ahead, [3]

The speed in the end of the current block is calculated taking under the consideration the length of the block and corner speed between two blocks. The algorithm end speed determination is given in the figure 2.52. The available end speed is calculated as following:

$$\frac{V_e(1)^2 - V_s(1)^2}{2A} < S_{tot},$$

where

V_e - end speed of *i*th block

V_s - start speed of *i*th block

A - maximum allowable acceleration

S_{tot} - length of the current block

(Formula 2.4)

End speed of the block is proper one if the distance for acceleration and deceleration from start speed to end speed is smaller than length of current block. If is not the current block should e calculated again from the following equation:

$$V_e(1) = \sqrt{V_s(1)^2 + sign \times 2AS_{tot}}$$

(Formula 2.5)

Sign is equal to 1 if start speed is smaller than end speed. If is the otherwise then the sign is equal to -1.

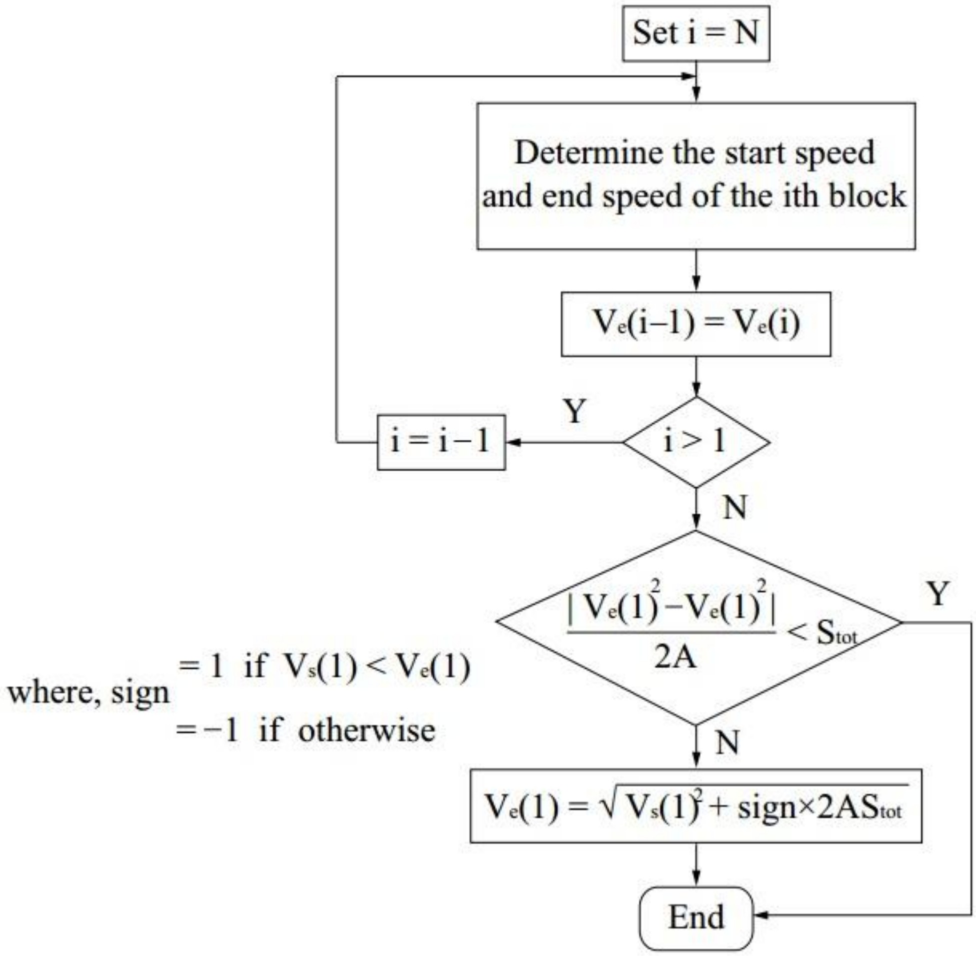


Figure 2.52 Algorithm for determining end speed with Look Ahead, [3]

2.7.2 Fagor 8055

The Look-Ahead function allows achieving high processing speeds in the implementation of these programs, by analyzing the paths to be machined even up to 75 blocks. This function calculates the maximum progress in each section. It allows for efficient and fast processing of programs with very little movement, even in micron range.

The best results when working with Look-Ahead are obtained by setting the machine axis with the least possible trace error, because the machining contour error will be at least equal to the tracking error.

CODE USE while working on Fagor 8055

A 0-255 - optional function; defines the percentage of acceleration applied

E 0-99999 – contour error; the smaller the error, the lower the feed rate.

B 0-180 - Indicates the angular value of the programmed corners below which they will be executed as sharp edges [5]

There are few working rules while working on Fagor 8055.

1. When calculating the feedrate, the CNC takes into account the programmed feedrate, the curvature, the corners, the maximum acceleration, the maximum feedrates of the axes.
2. The CNC slows down to "0" at the previous block and it recovers the machining conditions for Look-Ahead in the next motion block if there is:

- motionless block
- execution of auxiliary function,
- execution of single block mode,
- MDI mode,
- tool inspection mode.

3. Function is modal and incompatible with G05 (rounded corner), G07 (squere corner) and G50 (controlled rounded corner). If any of them is programmed, it will cancel the G51 function and start the work of a selected one.
4. To program function G51, it must be alone in a block.
5. If G51 is active and EMERGENCY or RESET mode will appear, the restared CNC will cancel look-ahead function, and it will assume G05 (rounded corner) or G07 (squere corner) according to the setting of general machine parameter "ICORNER".
6. IF G51 is active and any of G33 (electronic threading), G34 (variable-pitch threading), G52 (movement untill making contact), G95 (feedrate per revolution) is programmed, CNC will issue Error 7 (Incompatible G functions), [5]

3 OBJECTIVE AND THE SCOPE OF WORK

From the analysis of the current knowledge about use of the Look-ahead function during milling curvilinear contours, there is little data on the use of this function to optimize the production. G-code obtained by CAM systems for milling complex tool-paths are usually composed of many linear interpolations. This solution helps to get accurate results, but increases the cutting time. In order to overcome the previous problem, several controls such as "look-ahead" function, optimizes the feed rate of the milling tool looking forward blocks.

In view of the above, this work studies look-ahead function and its potential application to define a low-cost B-spline control through experimental tests.

The scope of work includes:

- to describe look ahead function and its main characteristics,
- to study its influence into cutting time,
- to analyze the similarities between actual tool-path and a B-Spline curve whose control points are the tool-path linear interpolation points.

4 METHODOLOGY OF RESEARCH

4.1 Materials

In order to fill the case of study, three main tests were performed. An Odisea CNC with a control Fagor 8055MC was used for their implementation and WinUnisoft was used to write, simulate and sent the code.

Odisea CNC Fagor 8055MC is a numerically controlled machine in four axes in the standard software, CPU up to 1 MB user RAM memory. The machine tool has an 6 tool magazine that offers automatic replacement. The most important technical parameters of this machine are showed in the Table 4.1, while the view of the milling machine is shown in Figure 4.1.

Working surface of table	200x200x200 mm
Max. revolution speed	4000 rpm
Max. feed rate	5 m/min
Power	1.5 kW

Tab. 4.1 Parameters of Odisea CNC Fagor milling machine

Figure 4.1 **Odisea CNC Fagor 8055MC**

The maximum look-ahead acceleration **A** on Odisea CNC Fagor 8055MC is 99999 and the maximum error **E** amounts 255.

4.2 Look ahead influence on the machining cycle time

Two types of paths were tested:

- two rectangular with different interpolation points,
- one spiral (pocket machining).

All of the codes were written in WinUnisoft program. First rectangular code includes 4 points with 3 changes the direction (Figure 4.2a). The second rectangular code has 3 changes of direction but is made from 16 points, which smaller the length between the points and therefore smaller the acceleration (Figure 4.2b). The third code has 13 changes of direction and much more instruction points (Figure 4.2c).

a)

```
%Test1,MX,
G90 G94 G97 F100 S1500 M3
T1 D1
M6
G43
GS1
G0 X0 Y0 Z10
G1 Z-1 F41
G1 Z-2. F100
 X40 Y0
 X40 Y-40
 X0 Y-40
 X0 Y0
 Z20
M5
M30
```

b)

```
%Test2,MX,
G90 G94 G97 F100 S1500 M3
T1 D1
M6
´G51
G43
G0 X0 Y0 Z10
G1 Z-1 F50
G1 Z-2. F100
 X10 Y0
 X20 Y0
 X30 Y0
 X40 Y0
 X40 Y-10
 X40 Y-20
 X40 Y-30
 X40 Y-40
 X30 Y-40
 X20 Y-40
 X10 Y-40
 X0  Y-40
 X0  Y-30
 X0  Y-20
 X0  Y-10
 X0  Y0
 Z20
M5
M30
```

c)

```
%Test3,MX,
G90 G94 G97 F100 S1500 M3
T1 D1
M6
G43
GS1
G0 X0 Y0 Z10
G1 Z-1 F50
G1 Z-2. F100
 X10 Y0
 X20 Y0
............
 X0  Y-6
 X1  Y-6
 X2  Y-6
 X3  Y-6
 X4  Y-6
 X5  Y-6
 X6  Y-6
 X7  Y-6
 X8  Y-6
 X9  Y-6
 X10 Y-6
 X11 Y-6
 X12 Y-6
 X13 Y-6
 X14 Y-6
 X15 Y-6
............
 X22 Y-21
 X22 Y-22
 Z20
M5
M30
```

Figure 4.2 Codes written in WinUnisoft: a) 3 direction changes and 4 points b) 3 direction changes and 16 points c) 13 direction changes

After codes were written and simulated they were sent to the Odisea CNC Fagor 8055MC milling machine by the serial communication system. The schema of the research station is shown in the figure 4.3.

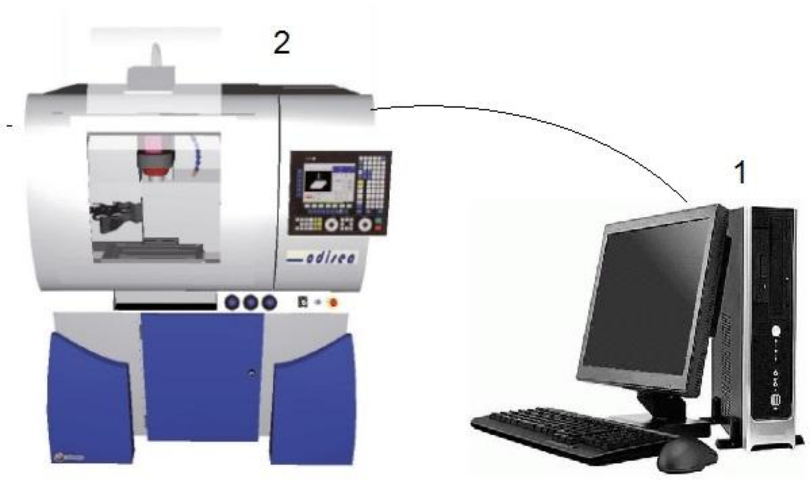


Figure 4.3 Schema of research station : 1 - WinUnisoft program , 2 - milling machine

The WinUnisoft does not have the Look-ahead function, which is the reason the codes needed to be edited on the milling machine in order to add look-ahead.

The parameters of Look Ahead for the two written codes were as following:

- A0 E0
- A99999 E0
- A0 E255
- A99999 E255

On the milling bed of the CNC machine was fasten in a vise of a wooden block with a sheet of paper (Figure 4.4a). The metal pen (Figure 4.4b) was fixed in the spindle. For all of the tests use the 100 mm/min feed rate. A paper with a higher value of feed rate could be torn. After the parameters of Look ahead had been edited, the program was run and the cycle times for each program were registered. The results of times and geometries were compared.

a) b)

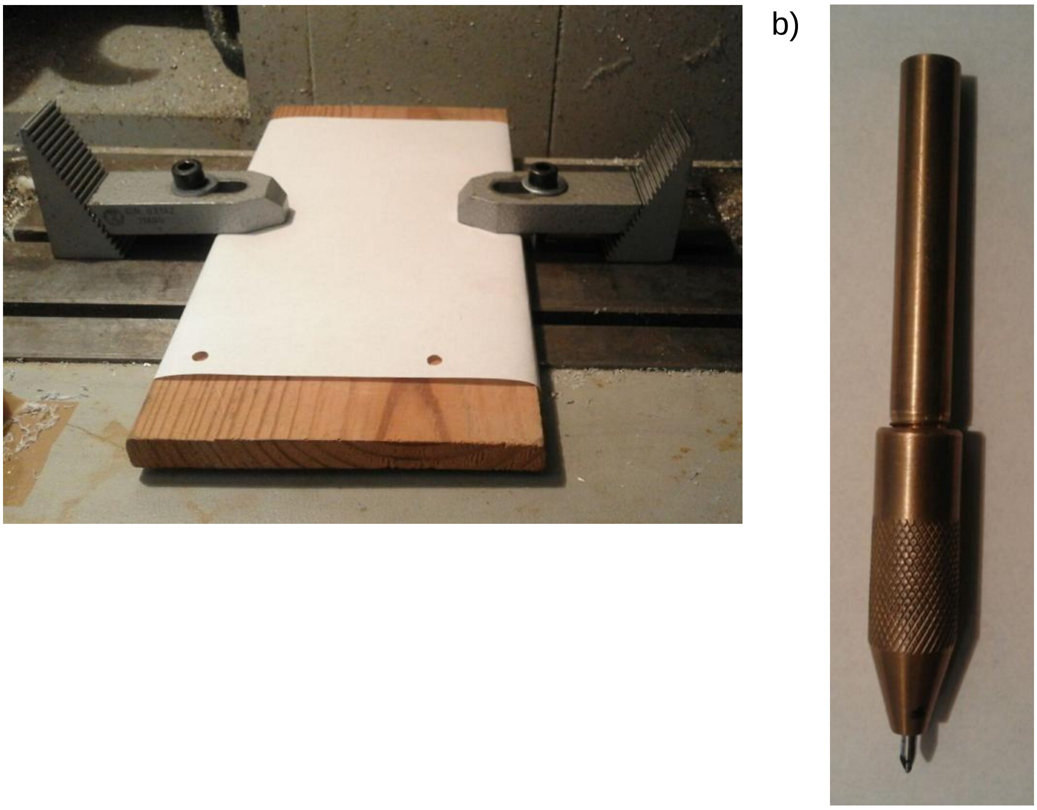


Figure 4.4 a) sheet of paper, b)the metal pen

4.3 Look ahead influence on the geometry

For registered the difference in geometry the camera SAMSUNG was used. Camera was putted inside of the milling machine, horizontally to the milling machine bed. It was fixed on the tripod to stabilize camera and register good quality videos. The schema of the research station is shown in the figure 4.5 and figure 4.6.

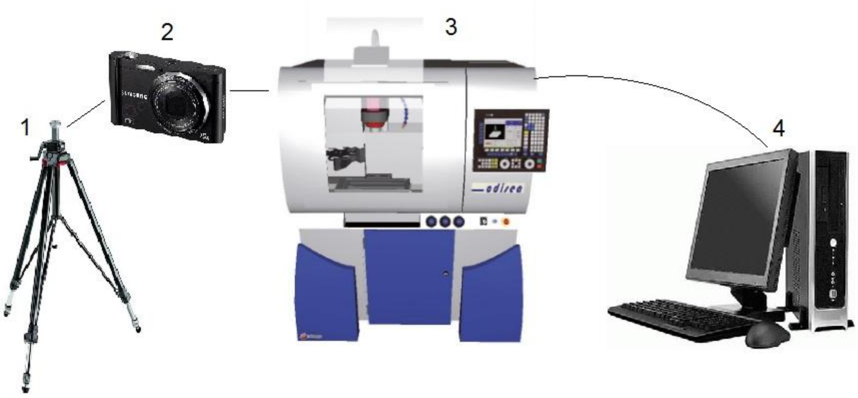


Figure 4.5 Schema of research station : 1 - tripod, 2 - camera, 3 -milling machine, 4 - Winunisoft program on the PC

The camera recorded the movements of the milling bed to, which was stacked ⦰20 mm red dot, when the programs were run. In the second research the same code of 13 direction changes (Figure 15c) were used.

The parameters of Look Ahead for the two written codes were as following:

- A0 E0,
- A99999 E0,
- A0 E255,
- A99999 E255.

In order to register bigger differences in geometry of couture and the times of cycle the code was run with difference feed rates parameters, which were:

- F100 mm/min
- F1000 mm/min
- F5000 mm/min, which is the highest allowed value when machining without load.

The recorded videos were sent to the PC, where each one was change for the frames per seconds. In the F1000 and F5000 were taken all 24frames/s. In F100 were taken every 10 frames/24s, because of large video and to big memory for process needed.

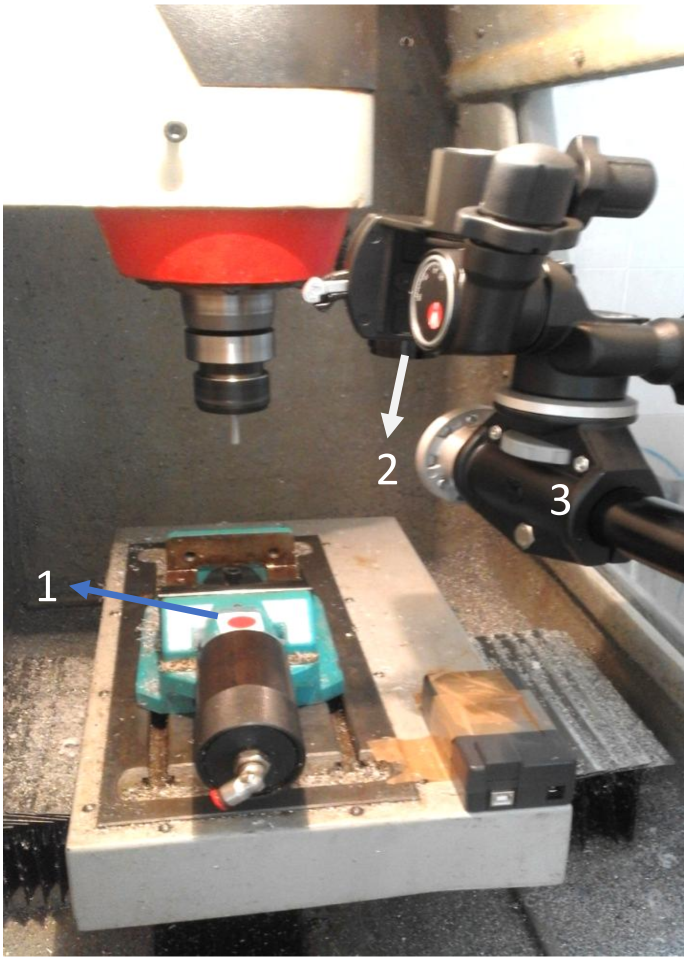


Figure 4.6 The research station inside of the milling machine in reality: 1 - red dot, 2 - camera, 3 - tripod

For further geometry calculation, frames were put to the mathematical program Mathematica Wolfram 2011, in which were generated to the trajectories. The code used for the change of video for trajectories was modified on the basis of another diploma thesis. The points which construct the geometry where exported to the Microsoft Excel.

In order to calculate the differences between generated trajectories the Hausdorff distance were used. The Hausdorff distance is measures how far two subsets of a metric space are from each other and is represented as following:

$$d_H(X,Y) = \max\left\{\sup_{x\in X}\inf_{y\in Y} d(x,y), \sup_{y\in Y}\inf_{x\in X} d(x,y)\right\},$$

where:

sup - supremum,

inf - infimum.

Program Mathematica Wolfram was also used to calculated the real feed rate as the derivative of the distance over time.

$$\lim_{t \to 0} \frac{f(s+t) - f(s)}{\Delta t}$$

4.4 Similarities between actual tool-path and a B-Spline curve

The idea of this research is to know if is possible to represent the actual tool-path with a NURBS. If it is possible with the further researches about the relation between acceleration and weights it could be get a spline interpolator with the look-ahead function.

In order to check the similarities between the actual tool-path and the B-Spline curve the theoretical points of the corners, where direction is changed, were implement to the Mathematica Wolfram program. Points were the data for the B-Spline Function calculated in the program.

The trajectory points which had been registered from the camera during the program run were imported from Microsoft Excel.

To gets the results, manipulation of the weights and control points of B-Spline curves needed to be made. Weights were determinate when the theoretical and recorded trajectories had similar geometries.

In the end the Hausdorff Distance between the registered geometry and theoretical B-Spline curve were calculated.

5 RESEARCH RESULTS

The results of three main researches are described below in this chapter.

5.1 Look ahead influence on the machining cycle time

As a result of the tests, 15 machining times were determined and 15 different geometries were made, 5 for each code.

Times for rectangular code with 4 interpolation point with Look-Ahead at different values A and E are shown in Table 5.1 - Points 1 - 4. Time for code without Look-Ahead is shown in Table 5.1 - Point 5. Table shows that the times of machining, with and without using the Look Ahead do not differ significantly and range from 5-37 millisecond. The smallest time is registered for the code with Look-ahead A255 and E0. Difference in time between the first and the second cycle time with A255 E99999 is 18 milliseconds. Time between the second and the third machining time with A0 E0 is almost the same, while the difference between the fourth codes with G51 A0 E99999 is 5 milliseconds.

nr	A	E	CZAS
1	255	0	02:06:51
2	255	99999	02:06:79
3	0	0	02:06:78
4	0	99999	02:06:83
5	-	-	02:06:88

Tab. 5.1 Times for rectangular code with 4 interpolation point with Look-Ahead and without Look ahead at different values A and E

The machining contour with Look-ahead is shown in Figure 5.1a - 5.1d; and without Look-ahead is represented on the Figure 5.1e. The differences in the geometry are imperceptible. The geometry with look-ahead A0 E99999 has intermitted line.

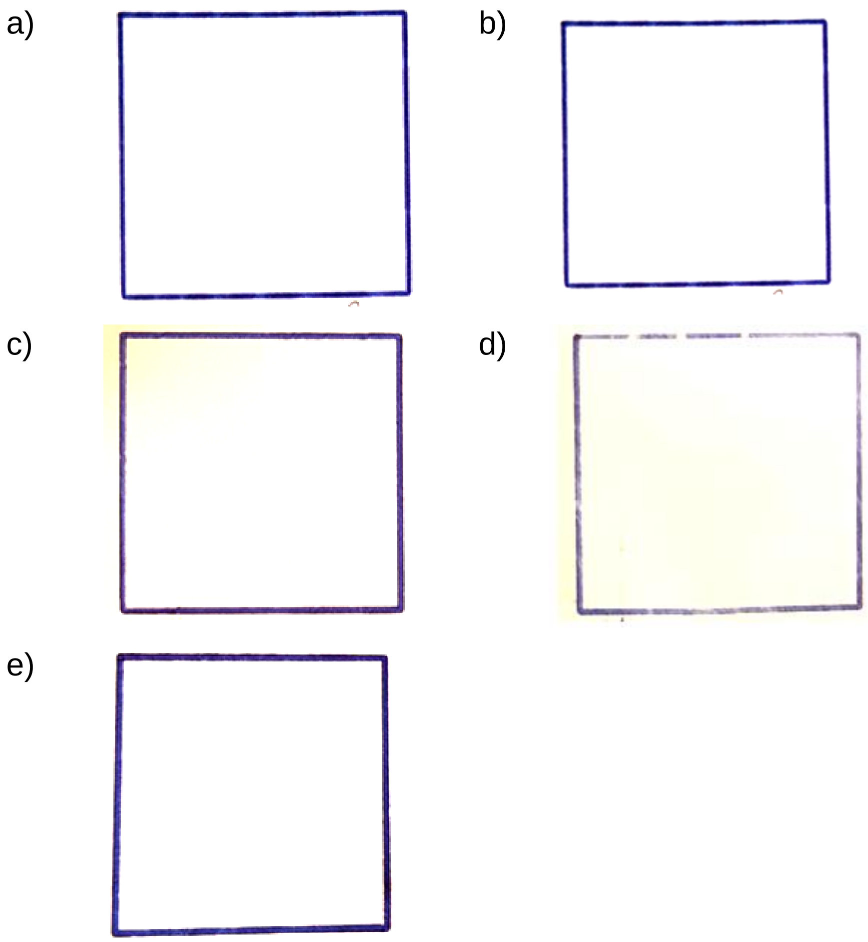


Figure 5.1 **The machining contour for G-code nr 1: a) G51 A255 E0, b) G51 A255 E99999, c) G51 A0 E0, d) G51 A0 E99999, e) without G51**

Times for rectangular code with 16 interpolation points with Look-Ahead at different values A and E are shown in Table 5.2 - Points 1 - 4. Time for code without Look-Ahead is shown in Table 5.2 - Point 5. Results from the table shows that the time for the code without Look-ahead increase to 2:08:15, which is 1 second and 27 milliseconds. The times with Look ahead do not increase significantly. The shortest time is registered for the code with Look-ahead and the parameters A255, E99999. Difference in time between the first and the second cycle time with A255 E0 is almost the same and values 3 milliseconds. Time between the second and the third machining time with A0 E0 differs 5 milliseconds, while the difference between the fourth code with G51 A0 E99999 is 1 milliseconds.

Tabela 1

nr	A	E	CZAS
1	255	0	02:06:82
2	255	99999	02:06:79
3	0	0	02:06:87
4	0	99999	02:06:88
5	-	-	02:08:15

Tab. 5.2 **Times for rectangular code with 16 interpolation point with Look-Ahead and without Look ahead at different values A and E**

The machining contour with Look-ahead is shown in Figure 5.2a - 5.2d; and without Look-ahead is represented on the Figure 5.2e. The differences, similar like in the first test, in the geometry are imperceptible.

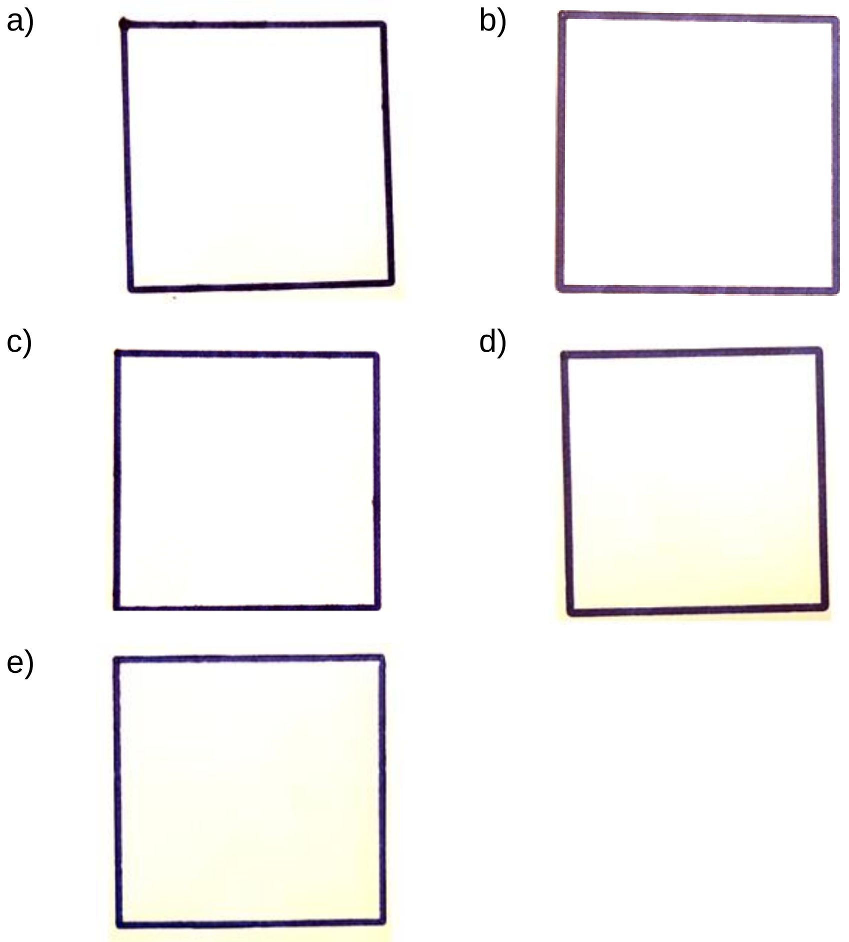


Figure 5.2 **The machining contour for G-code nr 2: a) G51 A255 E0, b) G51 A255 E99999, c) G51 A0 E0, d) G51 A0 E99999, e) without G51**

Times for spiral code with Look-Ahead at different values A and E are shown in Table 53 - Points 1 - 4. The code for time without Look-Ahead is shown in Table 5.3 - Point 5. Results from the table show that the time for the code without Look-ahead increase to 3:42:13. Times of machining, with and without using the Look Ahead differ significantly and range from 24 seconds 84 milliseconds to 24 seconds 75 millisecond. The times between code with different parameters of Look-ahead do not change significantly. The difference is between 4-9 milliseconds. Similar like in the code with 16 points the fastest cycle time is the one with Look-ahead with parameters A255 E99999. The code with Look-ahead A255 E0 and A0 E9999 have exactly the same time, which are 5 milliseconds bigger than A255 E99999 and 4 milliseconds smaller than A0 E0.

nr	A	E	CZAS
1	255	0	03:17:34
2	255	99999	03:17:29
3	0	0	03:17:38
4	0	99999	03:17:34
5	-	-	03:42:13

Tab. 5.3 Times for spiral code with Look-Ahead at different values A and E and without look ahead

The machining contour with Look-ahead is shown in Figure 5.3a - 5.3d; and without Look-ahead is represented on the Figure 5.3e. The differences, similar like in the first test, in the geometry are imperceptible.

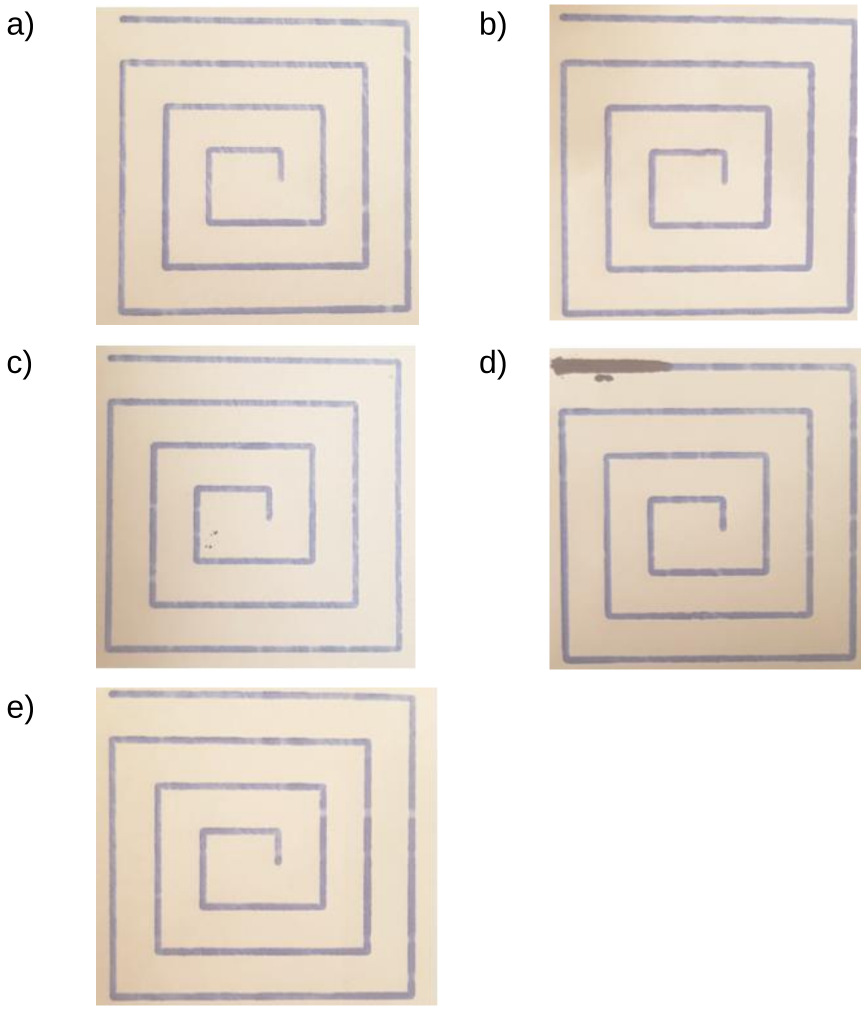


Figure 5.3 **The machining contour for G-code nr 3: a) G51 A255 E0, b) G51 A255 E99999, c) G51 A0 E0, d) G51 A0 E99999, e) without G51**

5.2 Look ahead influence on the geometry

As a result of the tests, where the feed rate were F100 mm/min ,F1000 mm/min, F5000 mm/min, 15 machining times were determined and 15 geometries were calculated, 5 for each feed rate. Also the 15 real lengths of tool path were computed as well as Hausdorff distance comparing the Look-ahead codes with code without this function. In the end the 15 graph of time in the function of length and 15 real feed rate graphics were determined.

The results of the test with a feed rate F = 100 mm/min are shown in Table 5.4. Times for code with Look-Ahead at different values A and E are shown in Table 5.4 - Points 1 - 4. The code without Look-Ahead is shown in Table 5.4 - Point 5. Results from the table shows that the cycle time for the code without Look-ahead is bigger than the cycles times and the range from 22 seconds 21 milliseconds to 22 seconds 8 milliseconds. The times between codes with different parameters of Look-ahead do not change significantly. The difference is between 3-13 milliseconds. The fastest cycle times were registered for both codes with look-ahead A255, E0 and E99999. The code with Look-ahead A0 E0 and A0 E9999 have almost the same time. Difference is equal to 3 milliseconds.

nr	A	E	czas
1	255	99999	03:49:69
2	255	0	03:49:69
3	0	99999	03:49:79
4	0	0	03:49:82
5	-	-	04:11:90

Tab. 5.4 **Machining times for F = 100 mm/min**

The results of the test with a feed rate F = 1000 mm/min are shown in Table 5.5 . Times for code with Look-Ahead at different values A and E are shown in Table 5.5 - Points 1 - 4. The code without Look-Ahead is shown in Table 5.5 - Point 5. Results from the table shows that the cycle time for the code without Look-ahead is bigger than the cycles times and the range from 45 seconds 28 milliseconds to 45 seconds 42 milliseconds. The times between codes with different parameters of Look-ahead do not change significantly. The difference is between 4-14 milliseconds. The fastest cycle times were registered for codes with look-ahead A255, E99999. The slowest from codes with look-ahead is the one with parameters A0 E0.

nr	A	E	czas
1	255	99999	00:31:52
2	255	0	00:31:56
3	0	99999	00:31:61
4	0	0	00:31:66
5	-	-	01:16:94

Tab. 5.5 **Machining times at F = 1000 mm/min**

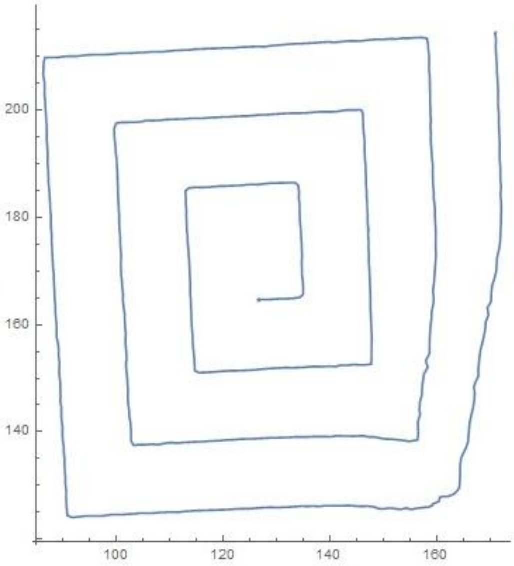
The results of the test with a feed rate F = 5000 mm/min are shown in Table 5.6 Times for code with Look-Ahead at different values A and E are shown in Table 5.6 - Points 1 - 4. The code without Look-Ahead is shown in Table 5.6 - Point 5. Results from the table shows that the cycle time for the code without Look-ahead is bigger than the cycles times and the range from 50 seconds 66 milliseconds to 51 seconds 19 milliseconds. The times between codes with Look-ahead parameters A255 E99999, A255 E0, A0 E99999 do not change significantly. The difference is between 2-15 milliseconds. The fastest cycle times were registered for codes with look-ahead A255, E99999. The slowest from codes with look-ahead is the one with parameters A0 E0. The difference in time to the rest of the codes with look-ahead is range 38 - 53 milliseconds.

nr	A	E	czas
1	255	99999	00:13:82
2	255	0	00:13:95
3	0	99999	00:13:97
4	0	0	00:14:35
5	-	-	01:05:01

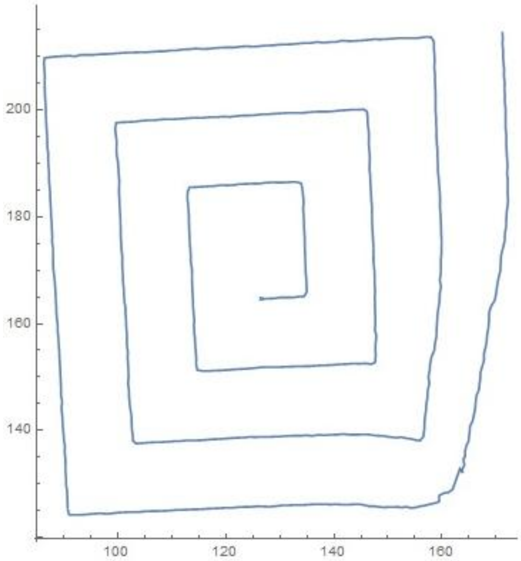
Tab. 5.6 **Machining times at F = 5000 mm/min**

The results of trajectory for feed rate equal to F = 100 mm/min, without the look-ahead and with, which were processed in the Mathematica program are shown in Figure 5.4. The geometry according to the theoretical assumptions is changed in the all the trajectories, which is mostly cause of the camera. It was not aligned properly to get high quality video. However, between the trajectories there are slightly differences.

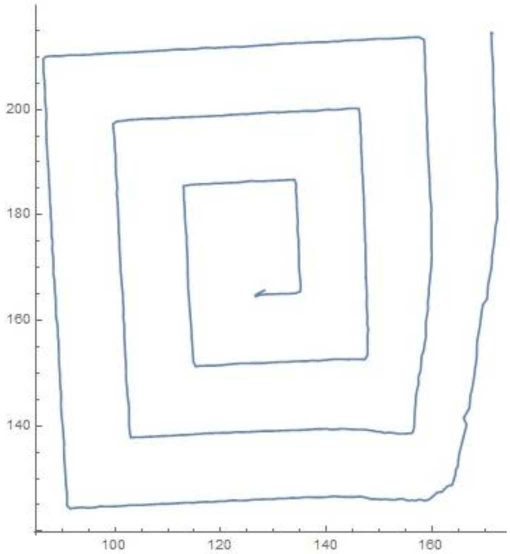
a)



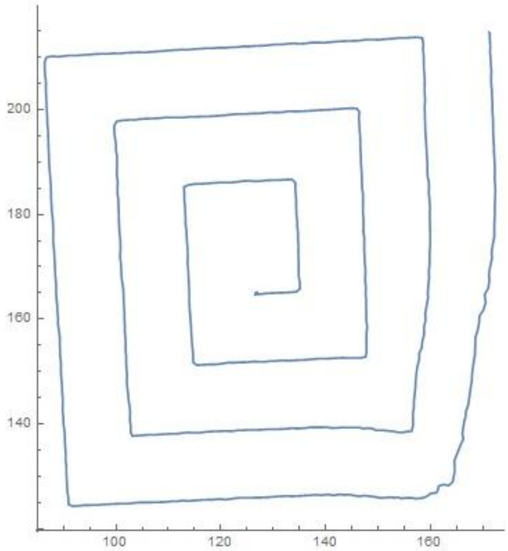
b)



c)



d)



e)

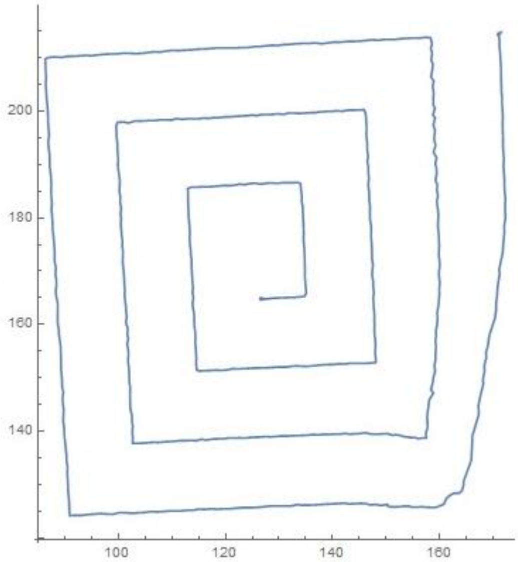


Figure 5.4 Trajectory machining for F=100 mm/min processed by program Mathematica: a) A0 E0, b) A0 E99999, c) A255 E0, d) A255 E99999, e) without Look-ahead

In the Table 5.7 were presented the results of the test with feed rate equal to F = 100 mm/min:

– Hausdorff Distance calculation for codes with Look-ahead in comparison with code without Look-ahead,

– calculation of trajectories length (mm)

– differences of real length to the theoretical.

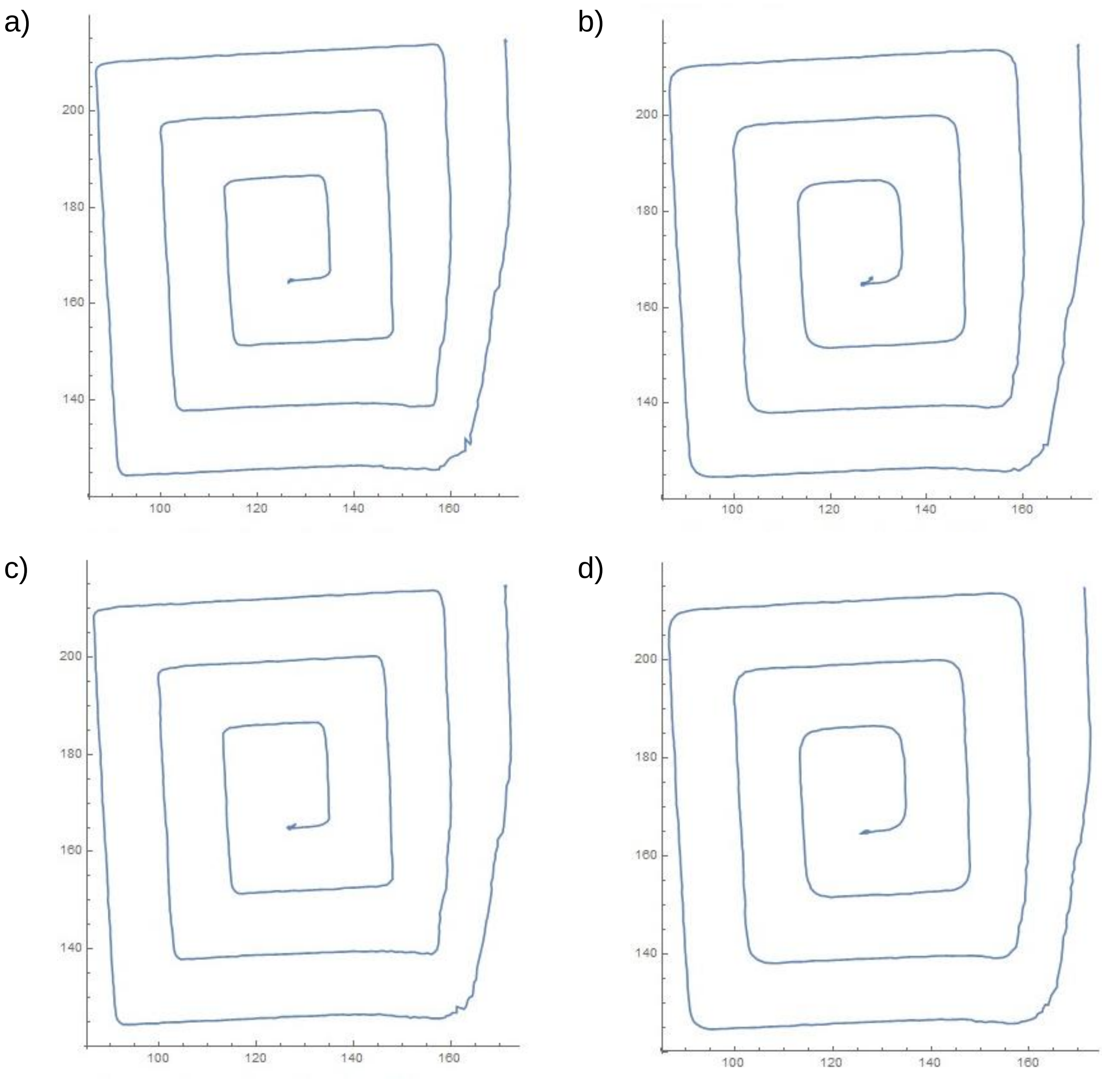
All the results had been processed in the Mathematica Wolfram. Theoretical length path is 344 mm. The code without Look-ahead has the smallest difference in length according to theoretical path and amounts 4,265. The lengths of the tool paths with Look-ahead are around 9,097 - 9,398 mm smaller than the theoretical path. The biggest length difference is to the theoretical path has the code with G51 A0 E0. The tool paths lengths between the codes with Look-ahead do not differ significantly from each other.

When it comes to Hausdorff Distance (Hd), the results hesitate from 0,68443 - 1,15859 mm. The differences between Hd are not notably. The smallest Hausdorff Distance has been calculated for the code Look-ahead parameters A0 E0. The biggest Hd were computed for the code G51 A0 E99999.

Number	Feed rate, mm/min	A	E	Length, mm	Length difference to theoretical path, mm	Hausdorff Distance, mm
1	100	0	0	334,602	9,398	0,68443
2	100	0	99999	334,903	9,097	1,15859
3	100	255	0	334,851	9,149	0,78299
4	100	255	99999	334,671	9,329	0,82626
5	100	-	-	339,735	4,265	-
6	Theoretical path			344	-	

Tab. 5.7 Feed rate, Length and Hd for feed rate 100 mm/min

The results of trajectory for feed rate equal to F = 1000 mm/min, without the look-ahead and with, which were processed in the Mathematics program are shown in Figure 5.5. Similarly like in the F=100 mm/min the geometry according to the theoretical assumptions is changed in the all the trajectories. The geometry of codes with the Look-ahead are different to the one where function were not added. The corners, when direction of tool path change, for the ones with look-ahead are rounded. What is more the trajectory is much smoother for the ones with G51 code. The difference especially is visible for the code which consists E99999, figure 5.5b and 5.5d.



e)

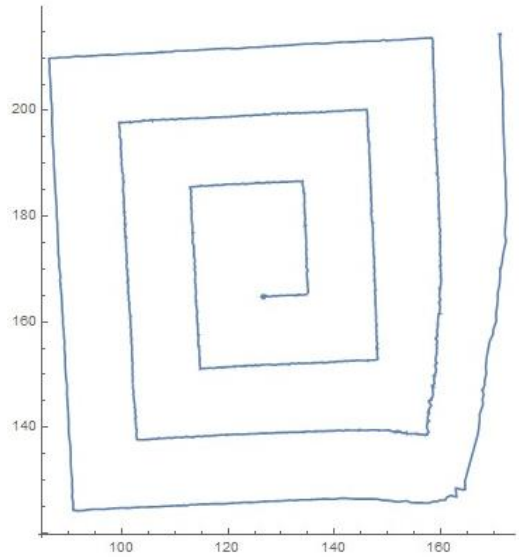


Figure 5.5 Trajectory machining for F=1000 mm/min processed by program Mathematica: a) A0 E0, b) A0 E99999, c) A255 E0, d) A255 E99999, e) without Look-ahead

In the Table 5.8 were presented the results of the test with feed rate equal to F = 1000 mm/min:

- Hausdorff Distance calculation for codes with Look-ahead in comparison with code without Look-ahead,
- calculation of trajectories length (mm)
- differences of real length to the theoretical.

All the results had been processed in the Mathematica Wolfram. Theoretical length path is 344 mm. The code without Look-ahead has the biggest difference in length according to theoretical path and amounts 15,701. The lengths of the tool paths with Look-ahead are around 7,711 - 13,417 mm smaller than the theoretical path. The smallest length difference to the theoretical path has the code with G51 A0 E0. The second smallest length difference represents code with look ahead and parameters A255 E0 and is 1,172 mm bigger than the one the error 0. The length tool paths difference between the codes with look ahead and E9999 is 0,222mm.

When it comes to Hausdorff Distance (Hd), the results hesitate from 0,955261 - 1,22371 mm. The differences between Hd are not notably. The smallest Hausdorff Distance has been calculated, dissimilarly like in F100 mm/min, for the code with the

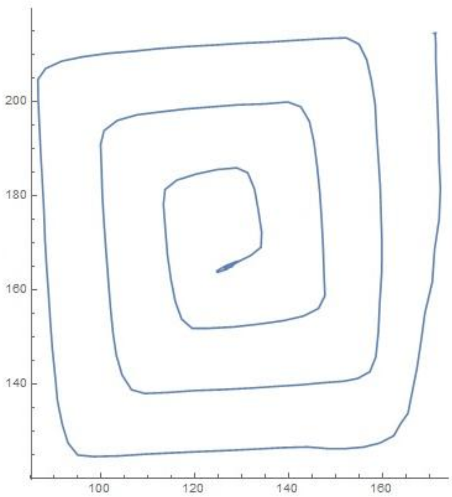
Look-ahead parameters A0 E99999. The biggest Hd were computed for the code G51 A255 E0.

Number	Feed rate	A	E	Length in mm	Length difference to theoretical path	Hausdorff Distance mm
1	1000	0	0	336,289	7,711	1,17692
2	1000	0	99999	330,805	13,195	0,955261
3	1000	255	0	335,117	8,883	1,22371
4	1000	255	99999	330,583	13,417	0,971998
5	1000	-	-	359,701	-15,701	-
6	Theoretical path			344	-	

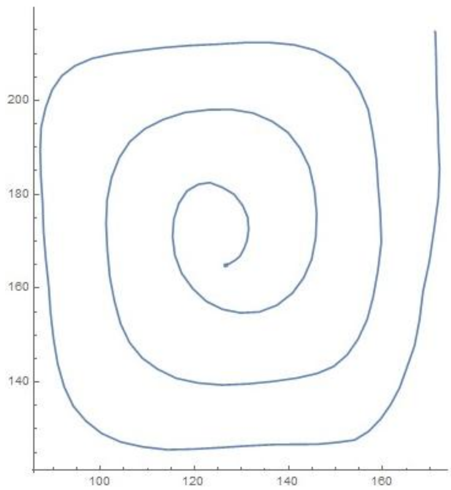
Tab. 5.8 Feed rate, Length and Hd for feed rate 1000 mm/min

The results of trajectory for feed rate equal to F = 5000 mm/min, without the look-ahead and with, which were processed in the Mathematics program are shown in Figure 5.6. Similarly like in the F=100 mm/min and F=1000 mm/min, the geometry according to the theoretical assumptions is changed in the all the trajectories. The geometry of codes with the Look-ahead are different to the one where function were not added. The corners, when direction of tool path change, for the ones with look-ahead are much more rounded. What is more, the trajectory is much smoother for the ones with G51 code. The difference especially is visible for the code which consists E99999, figure 5.6b and 5.6d.

a)



b)



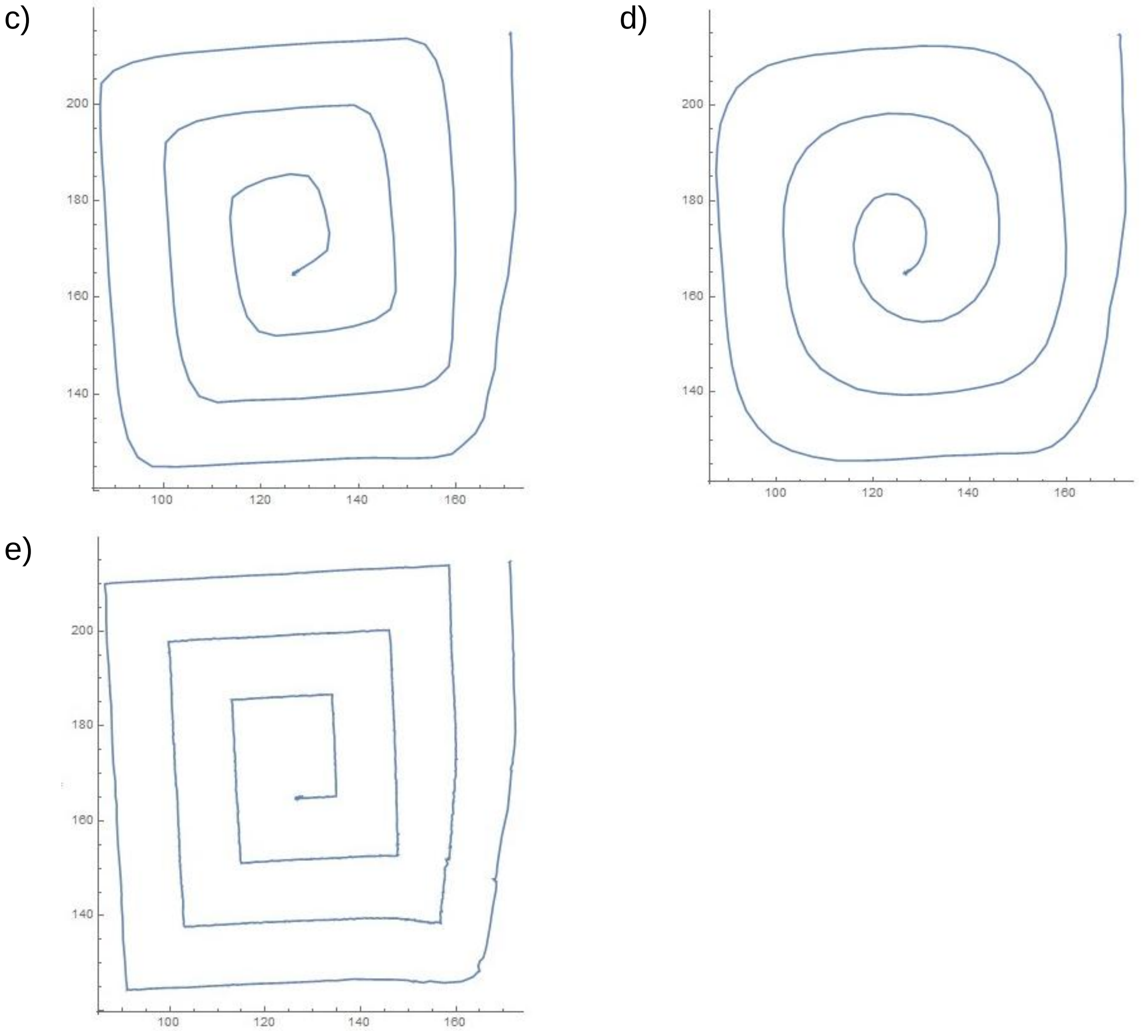


Figure 5.6 **Trajectory machining for F=00 mm/min processed by program Mathematica: a) A0 E0, b) A0 E99999, c) A255 E0, d) A255 E99999, e) without Look-ahead**

In the Table 5.9 were presented the results of the test with feed rate equal to F = 5000 mm/min:

- Hausdorff Distance calculation for codes with Look-ahead in comparison with code without Look-ahead,
- calculation of trajectories length (mm)
- differences of real length to the theoretical.

All the results had been processed in the Mathematica Wolfram. Theoretical length path is 344 mm. The code without Look-ahead has the smallest difference in length according to theoretical path and amounts 7,573. The lengths of the tool paths are much shorter for feed rate F5000 than for the both feed rate F100 and F1000. The differences in length according to theoretical path with Look-ahead are around 20,12 - 64,092 mm smaller. The biggest length difference to the theoretical path has the code with G51 A255 E99999. The second biggest length difference represents code with look ahead and parameters A0 E99999 and is 0,300 mm smaller than the one the acceleration 255.

When it comes to Hausdorff Distance (Hd), the results hesitate from 1,98862 - 4,59859 mm. The differences between Hd are notably. The smallest Hausdorff Distance has been calculated, similarly like in F100 mm/min for a code with Look-ahead parameters A0 E0. However the Hd for feed rate 5000 mm/min is almost 3 times bigger. The biggest Hd were computed for the code G51 A0 E99999. It needs to be notice that for the codes, where parameters of error where 0 the Hd is 2 smaller than for the programmed error 99999.

Number	Feed rate, mm/min	A	E	Length in mm	Length difference to theoretical path, mm	Hausdorff Distance, mm
1	5000	0	0	323,88	20,12	1,98862
2	5000	0	99999	280,208	63,792	4,59859
3	5000	255	0	311,582	32,418	2,35008
4	5000	255	99999	279,908	64,092	4,50238
5	5000	-	-	351,573	7,573	-
6	Theoretical path			344	-	

Tab. 5.9 Feed rate, Length and Hd for feed rate 5000 mm/min

The results of length in function of time (and its derivatives) help to understand the dynamic of the machine through the tool-path. Length and feed rate as a function of time for programmed feed rate equal to F = 100 mm/min, without the look-ahead and with, which were processed in the Mathematica program, are shown respectively from Figure 5.7 to 5.12. There is no notable difference between the length profiles. The process times of the function with look ahead are although shorter than for the one without Look-ahead and the machining starts from the beginning. The differences in feed rates between the codes where were used the Look ahead function are not significant. The feed rate for a code without look ahead has more deceleration and acceleration changes. The feed rate is not stable like in the codes with look ahead.

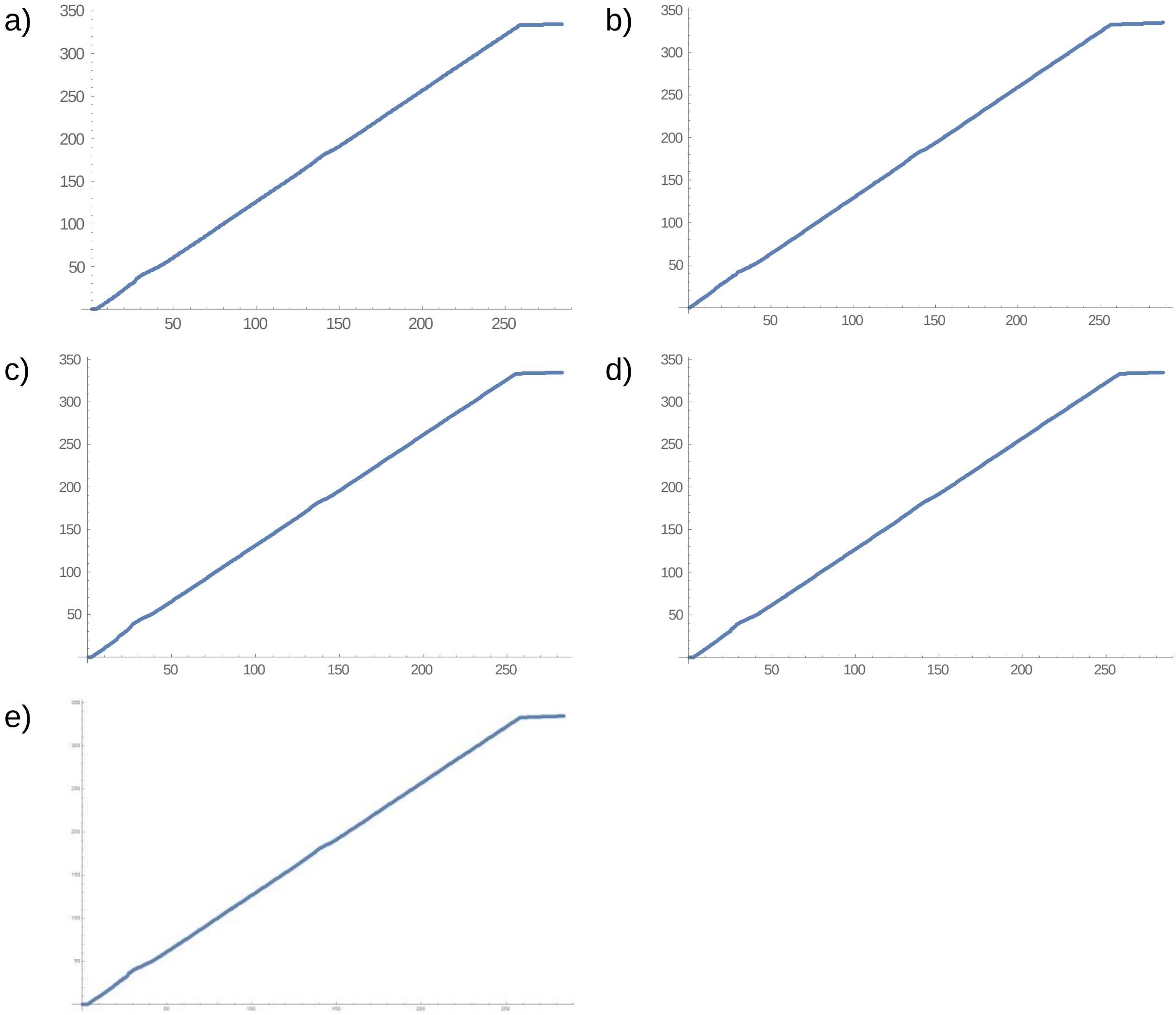


Figure 5.7 Length in the function of time for F=1000 mm/min processed by program Mathematica: a) A0 E0, b) A0 *E99999, c) A255 E0, d) A255 E99999, e) without Look-ahead*

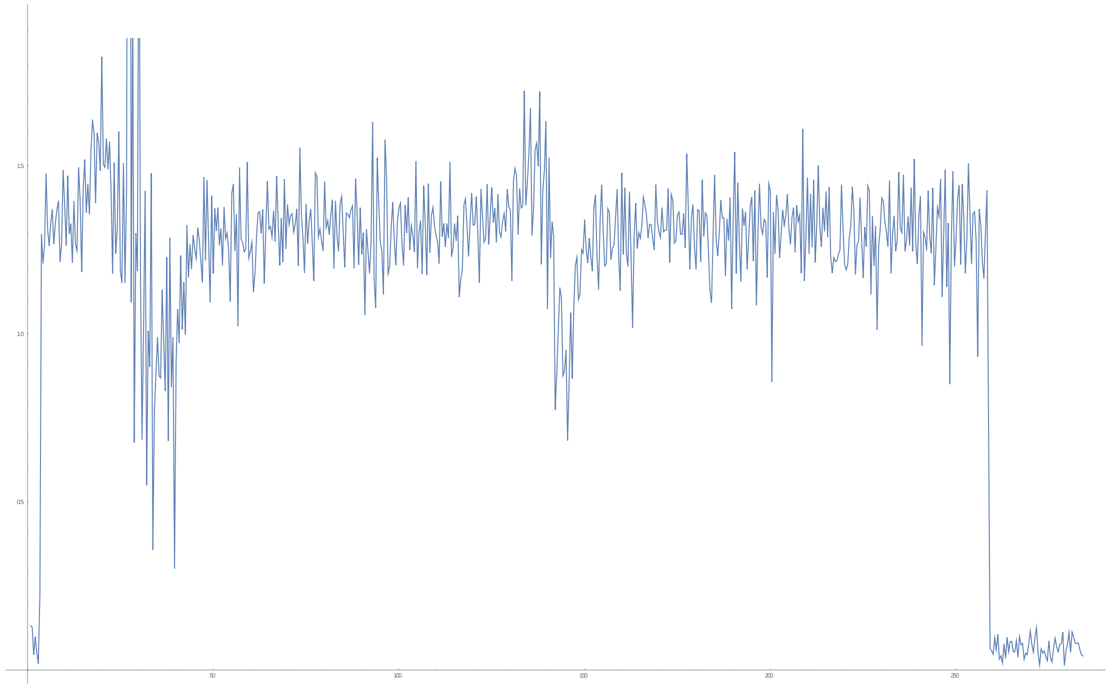


Figure 5.8 Real feed rate for code with Look ahead and parameters A0 E0 programmed for programmed F=100 mm/min

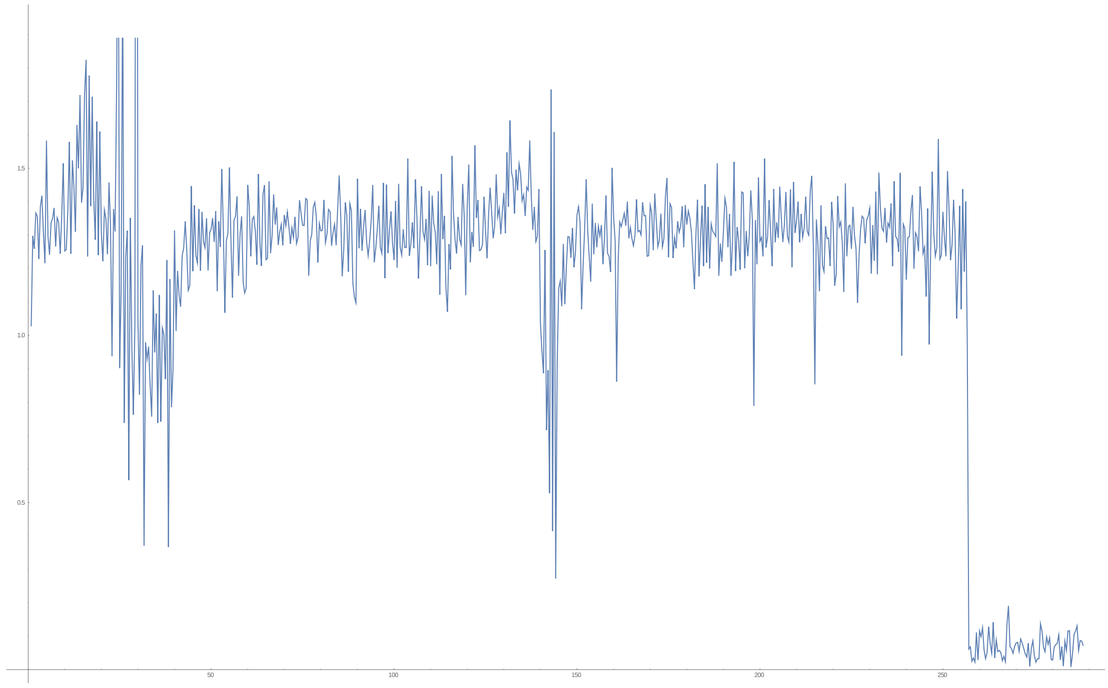


Figure 5.9 Feed rate for code with Look ahead and parameters A0 E99999 programmed for programmed F=100 mm/min

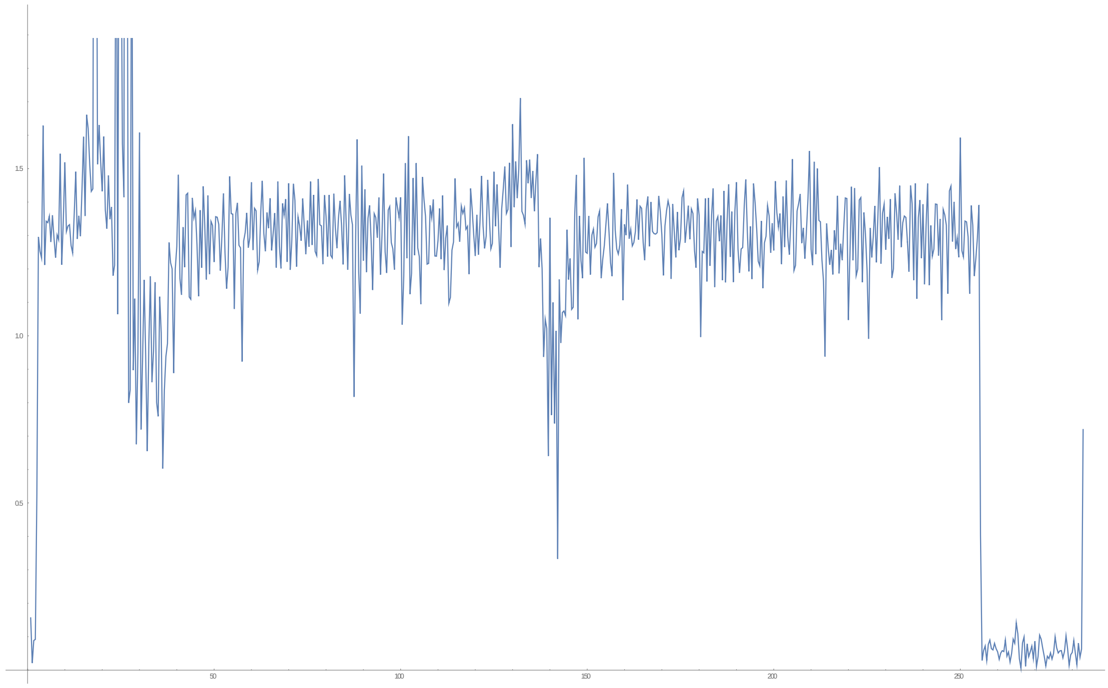


Figure 5.10 Feed rate for code with Look ahead and parameters A255 E0 programmed for programmed F=100 mm/min

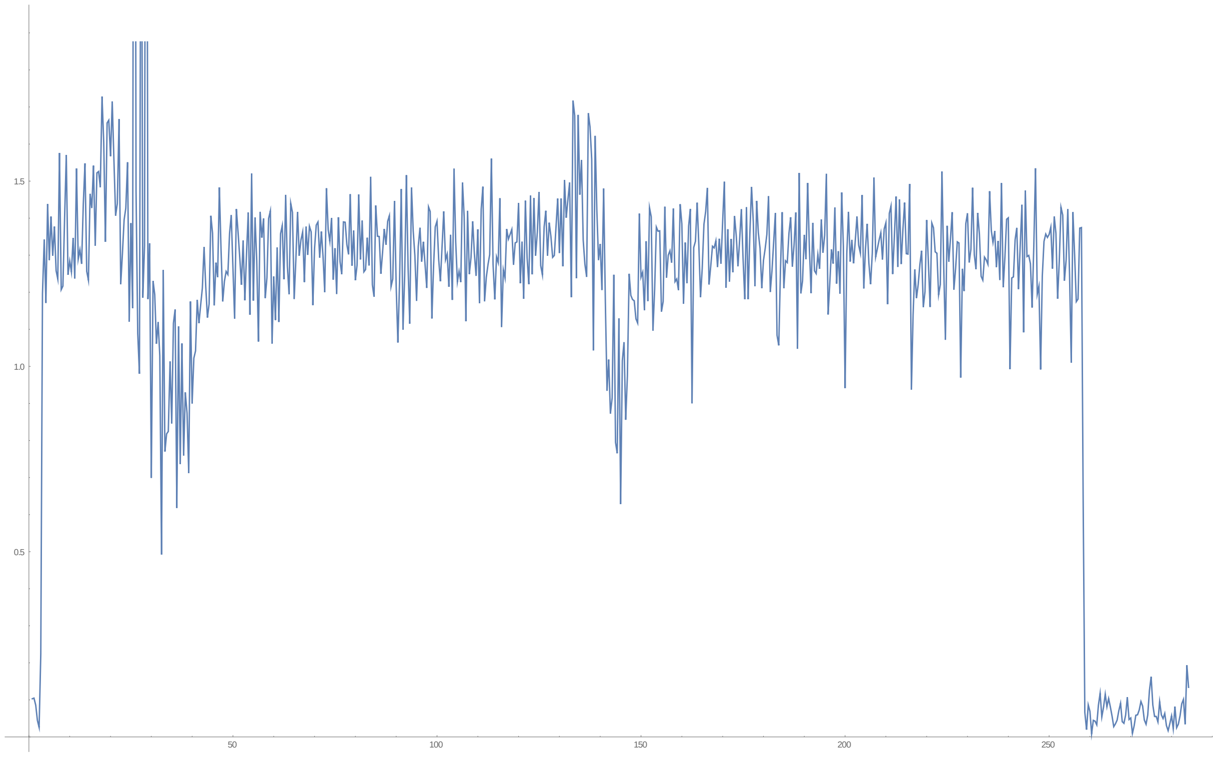


Figure 5.11 Feed rate for code with Look ahead and parameters A255 E99999 programmed for programmed F=100 mm/min

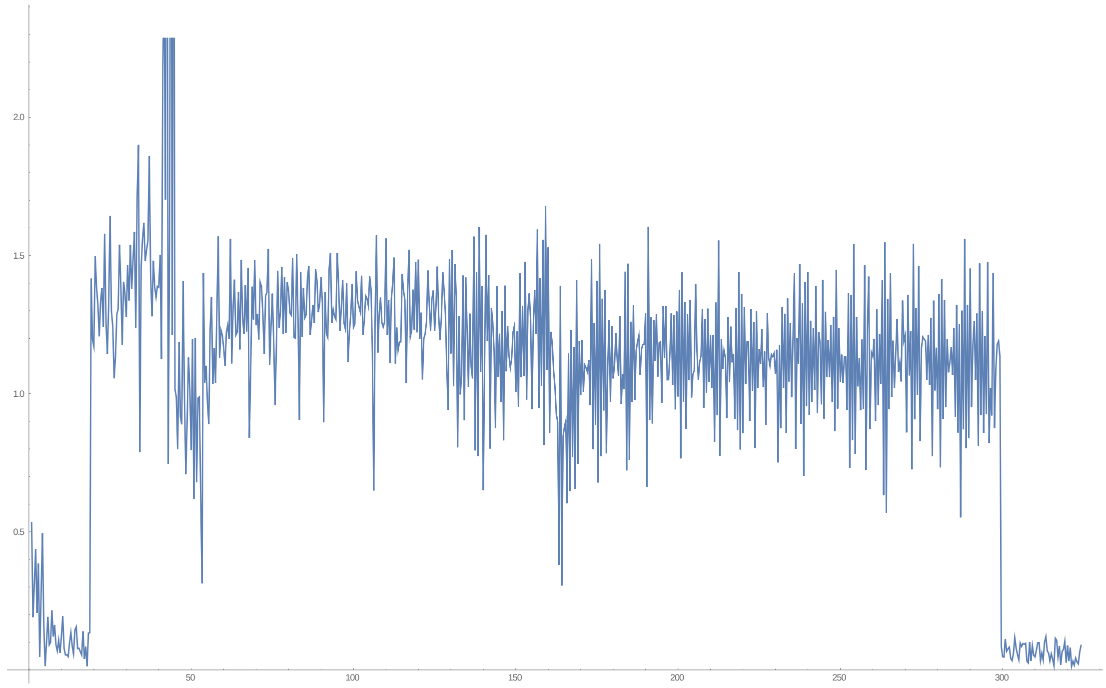


Figure 5.12 Feed rate for code without Look ahead for programmed F=100 mm/min

The results of length in function of time (and its derivatives) for programmed feed rate equal to F = 1000 mm/min, without the look-ahead and with, which were processed in the Mathematica program, are shown respectively from Figure 5.13 to 5.18. There is no significant difference between the length profiles with Look ahead function. The process times of the function with look ahead are although shorter than for the one without Look-ahead and the machining starts from the beginning and increase proportionally to the time. In the code without Look-ahead machining also starts from the beginning and grow proportionally until the 18 seconds and after lowers the value of increment of machining. Time of the machining for code without look-ahead is about 3 times longer than for the codes with Look-ahead.

The differences in feed rates between the codes where were used the Look ahead function are not significant. The feed rate for a code without look ahead has a numerous deceleration and acceleration. The feed rate is not stable.

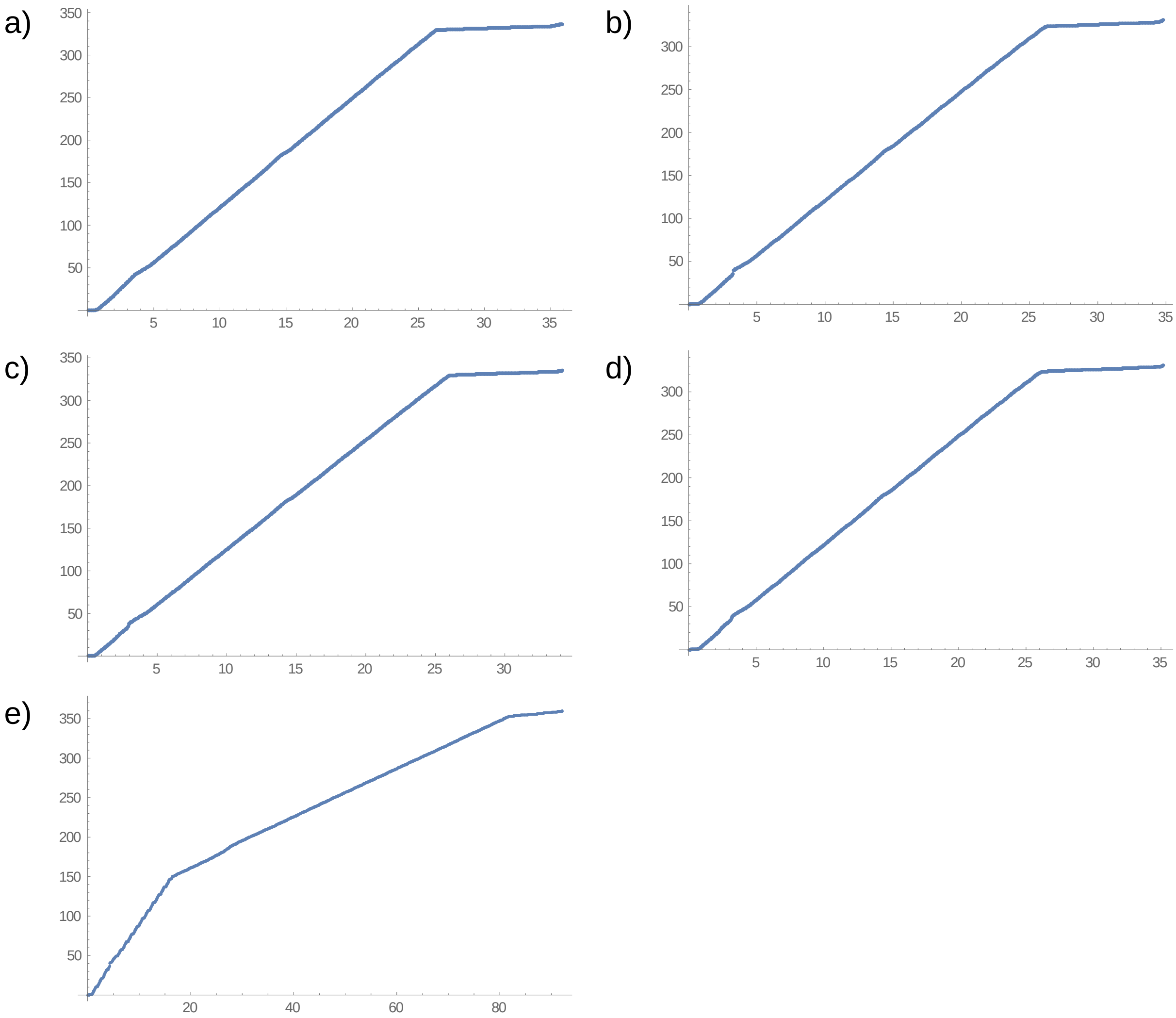


Figure 5.13 Length in the function of time for F=1000 mm/min processed by program Mathematica: a) A0 E0, b) A0 E99999, c) A255 E0, d) A255 E99999, e) without Look-ahead

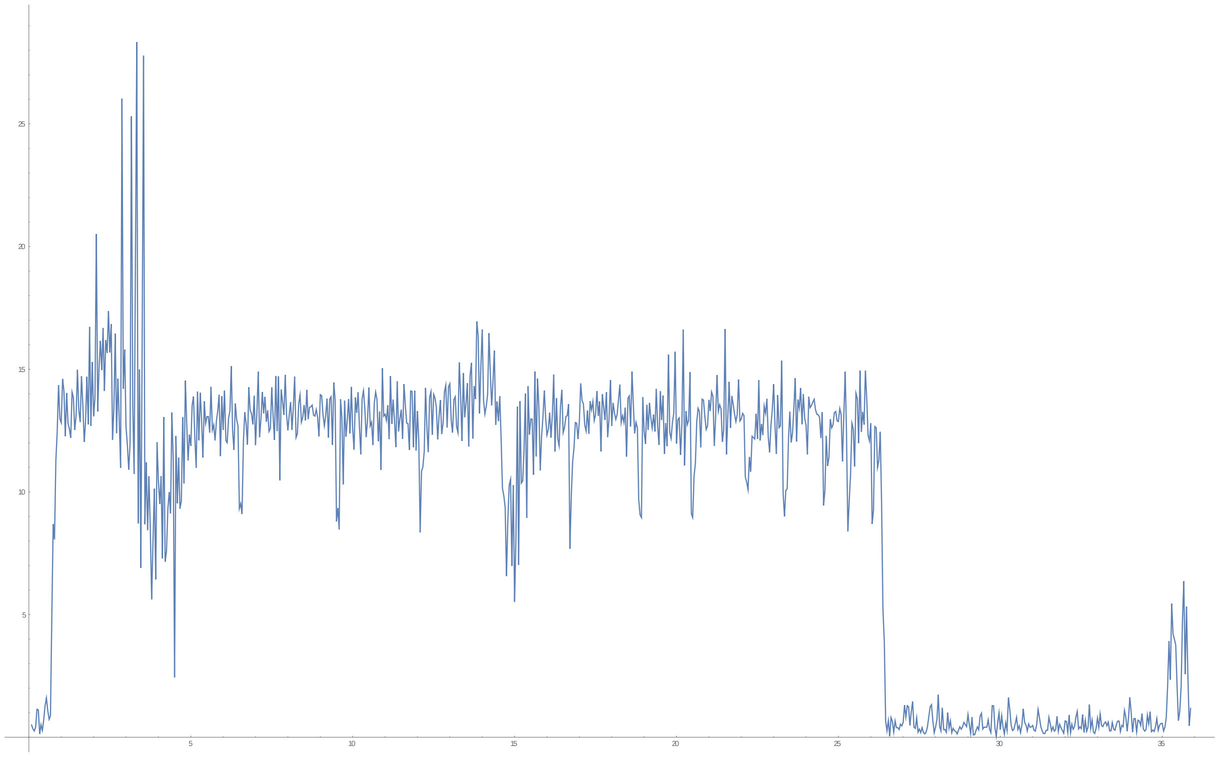


Figure 5.14 feed rate for code with Look ahead and parameters A0 E0 programmed for programmed F=1000 mm/min

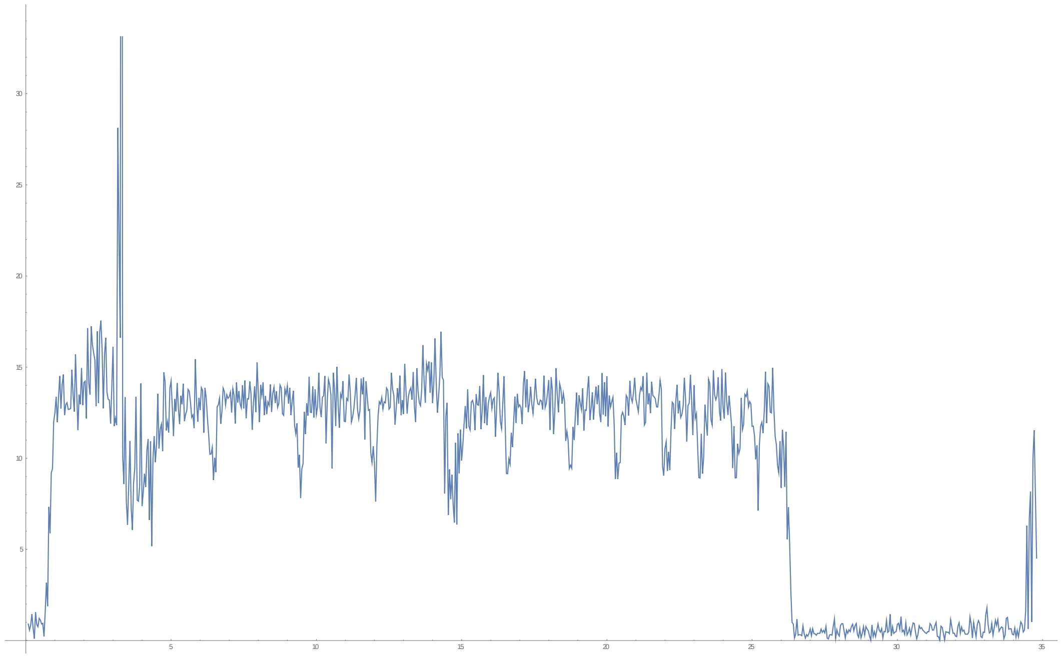


Figure 5.15 feed rate for code with Look ahead and parameters A0 E99999 programmed for programmed F=1000 mm/min

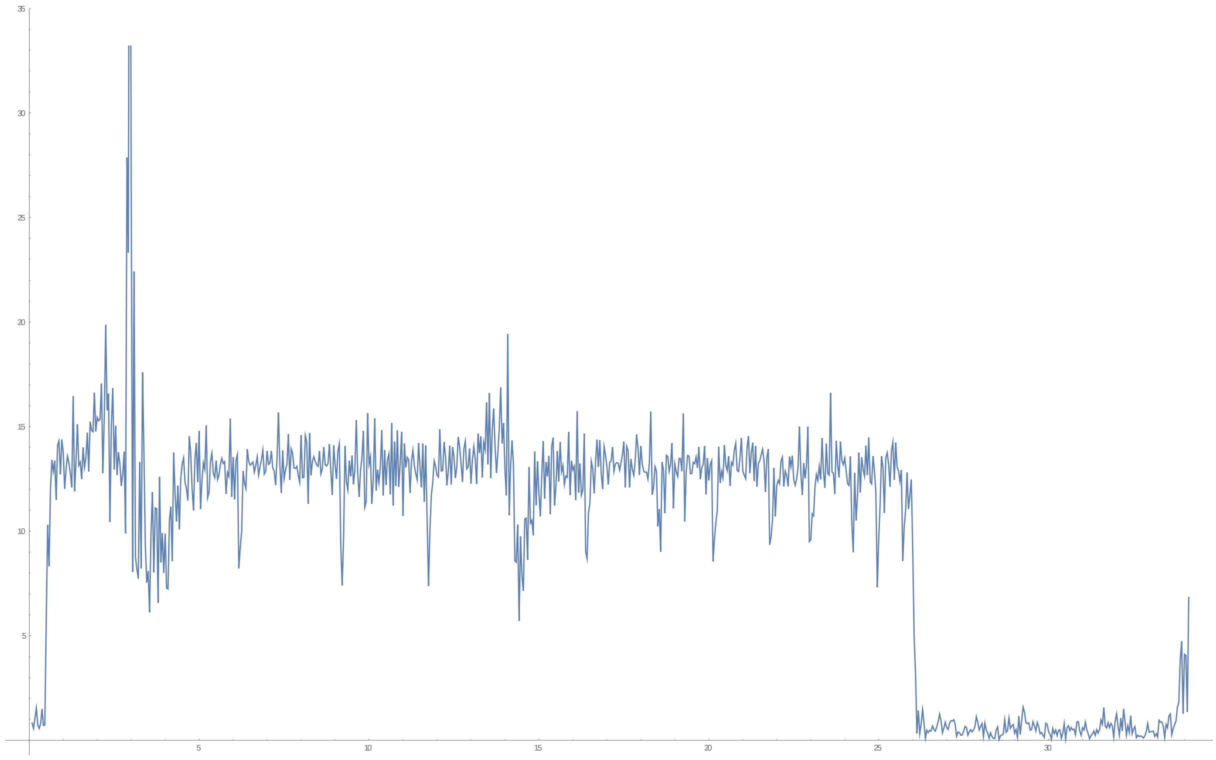


Figure 5.16 feed rate for code with Look ahead and parameters A255 E0 programmed for programmed F=1000 mm/min

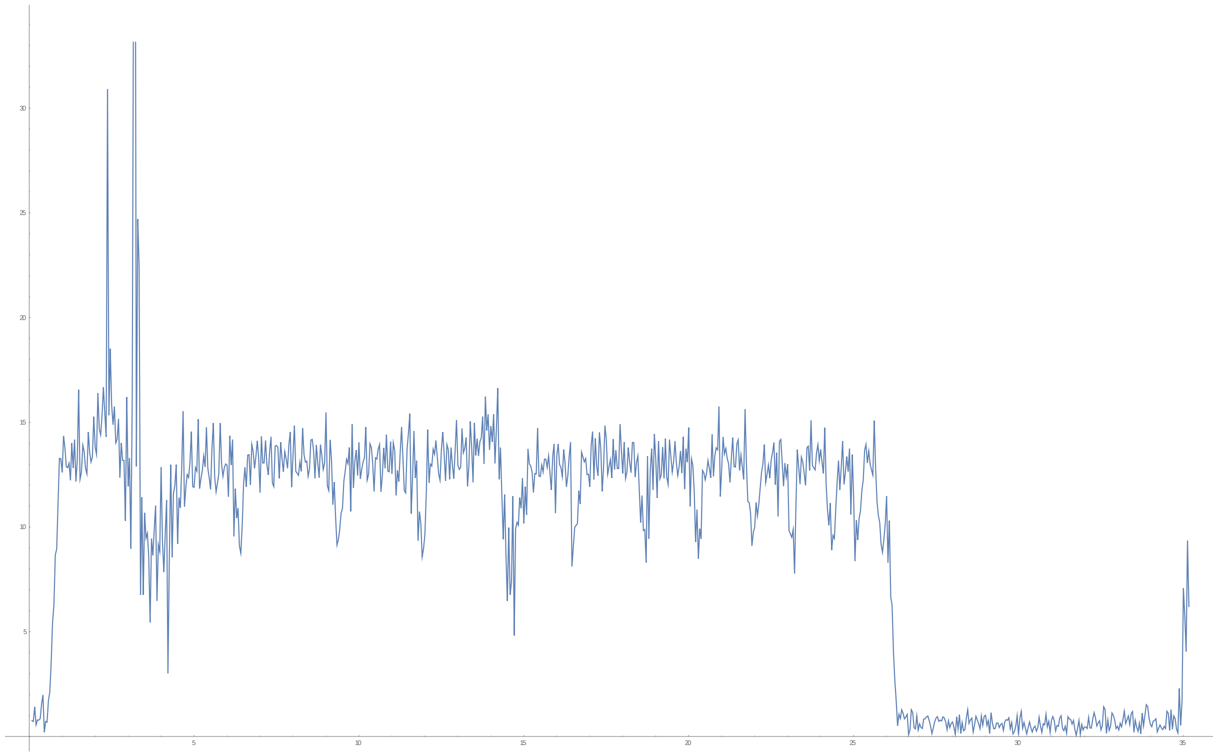


Figure 5.17 feed rate for code with Look ahead and parameters A255 E99999 programmed for programmed F=1000 mm/min

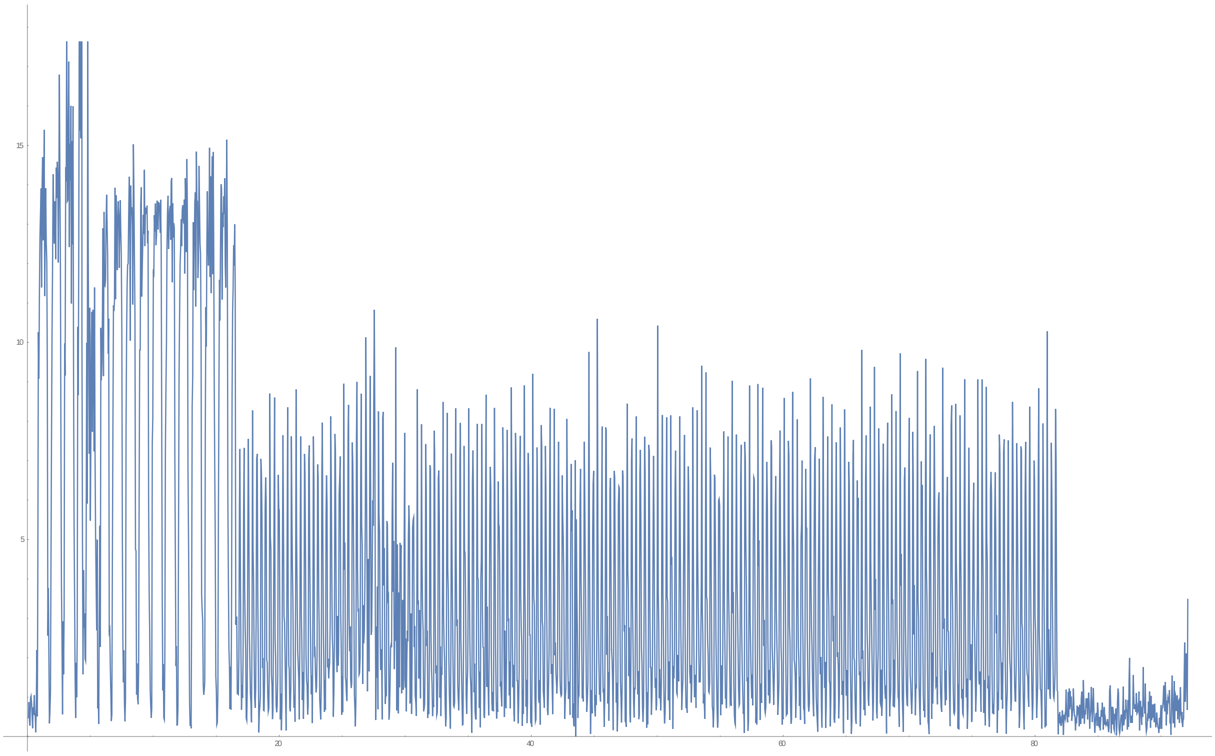


Figure 5.18 feed rate for code without Look ahead for programmed F=1000 mm/min

The results of length in function of time (and its derivatives) for programmed feed rate equal to F = 5000 mm/min, without the look-ahead and with, which were processed in the Mathematica program, are shown respectively from Figure 5.19 to 5.24. There is no significant difference between the length profiles with Look ahead function. The length of the tool path for the codes with look ahead G51 A0 E0 and A255 E0 is about 30 mm longer than for the code with look-ahead G51 A0 E99999 and G51 A255 E99999. The process times of the function with look ahead are although shorter than for the one without Look-ahead and the machining starts from the beginning and increase proportionally to the time. In the code without Look-ahead machining also starts from the beginning and grow proportionally until the 10 seconds and after lowers the value of machining increment. Time of the machining for code without look-ahead is about 12 times longer than for the codes with Look-ahead.

The difference in feed rates between the codes where were used the Look ahead function and the code without look ahead are significant. The feed rate for a code without look ahead has a numerous deceleration and acceleration, it is not stable. The feed rates of the codes with look ahead A0 E0 and A255 E0 have 4 more feed rates changes than the codes with look ahead A0 E9999 and A255 E99999. The feed rate in all the codes with look ahead is around 80 revolutions per mm. The feed rate in the code without look ahead is only 17 revolutions per mm.

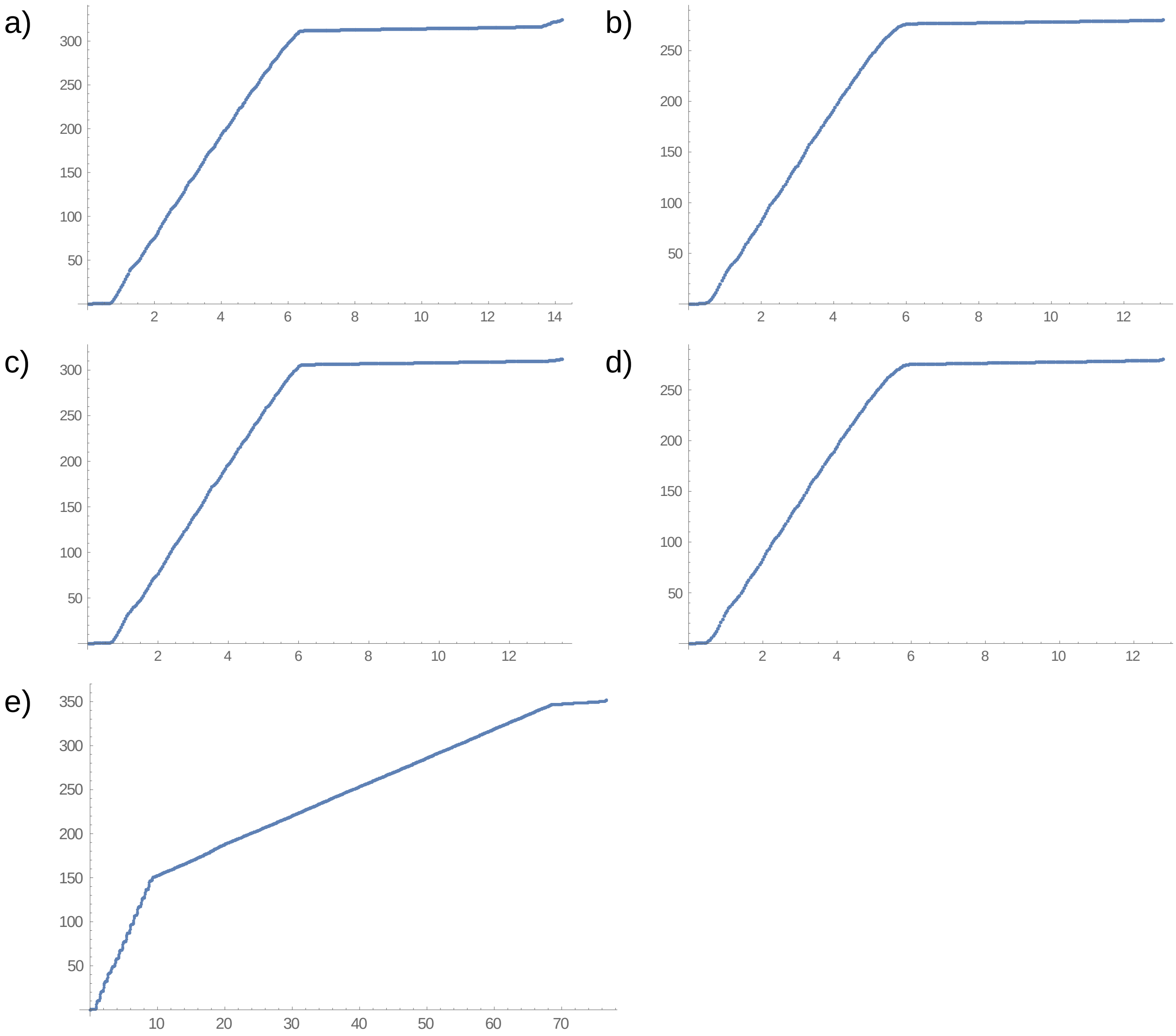


Figure 5.19 Length in the function of time for F=5000 mm/min processed by program Mathematica: a) A0 E0, b) A0 E99999, c) A255 E0, d) A255 E99999, e) without Look-ahead

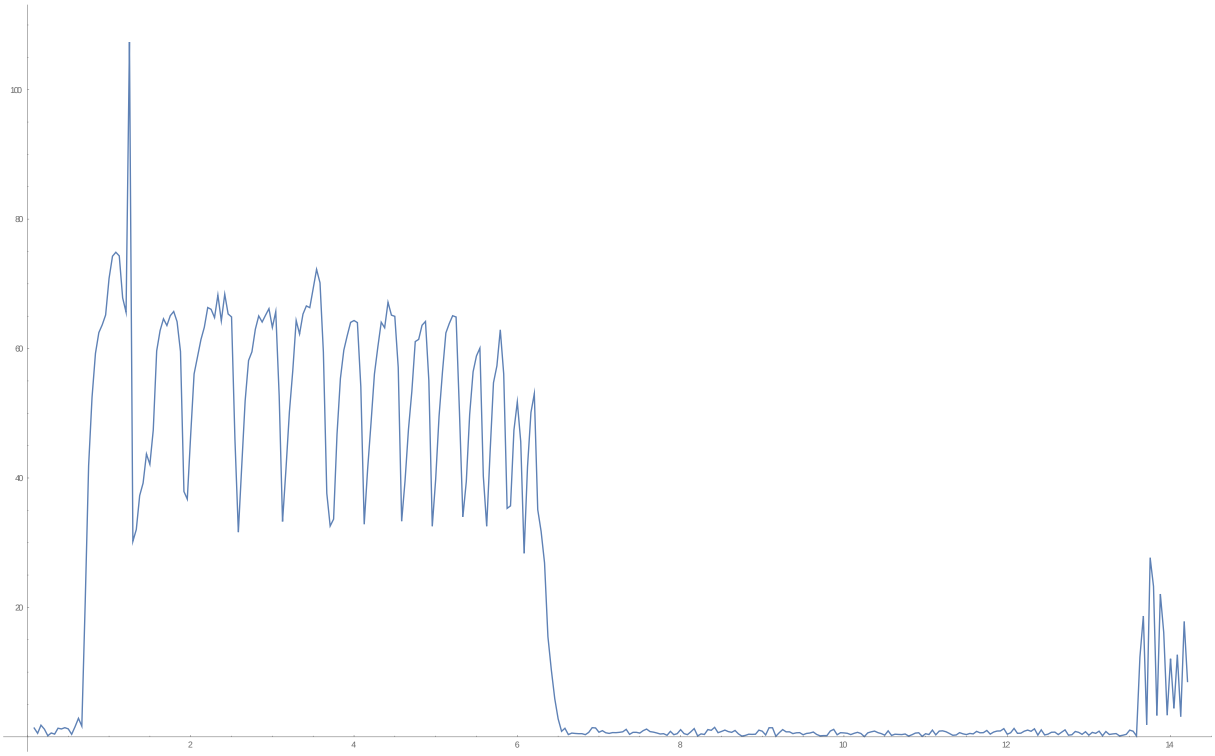


Figure 5.20 feed rate for code with Look ahead and parameters A0 E0 for programmed F=5000 mm/min

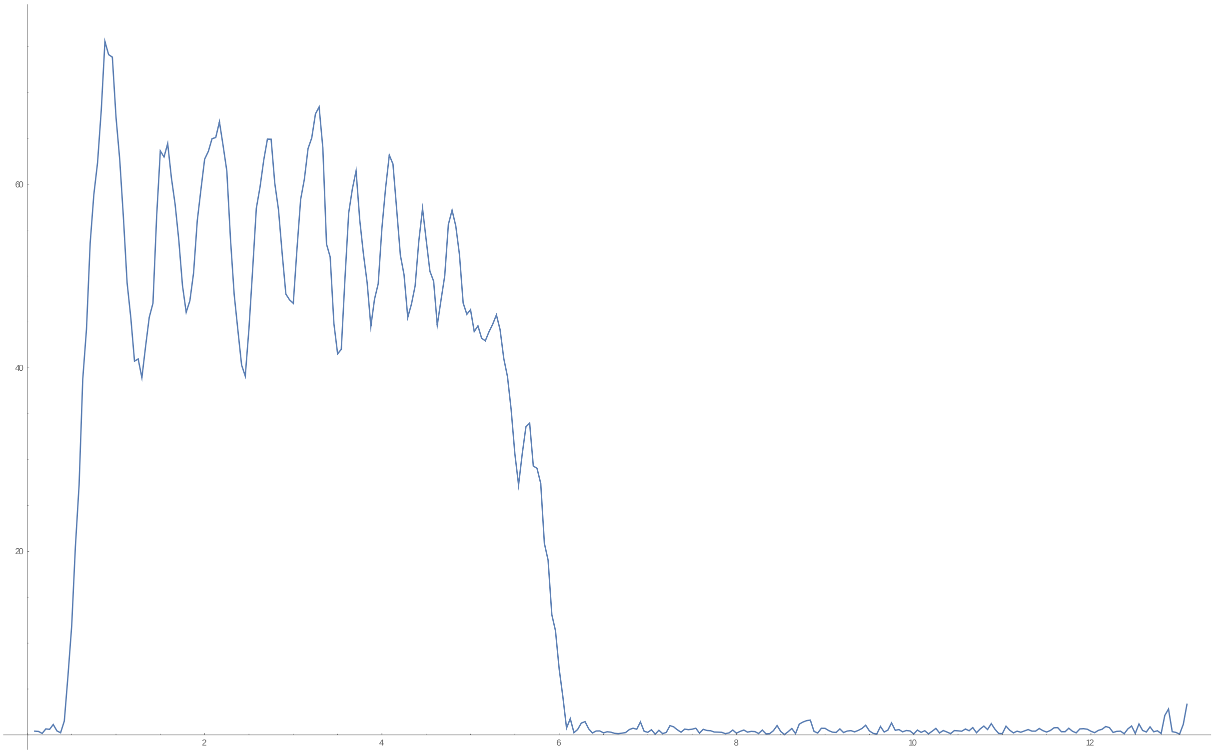


Figure 5.21 feed rate for code with Look ahead and parameters A0 E99999 for programmed F=5000 mm/min

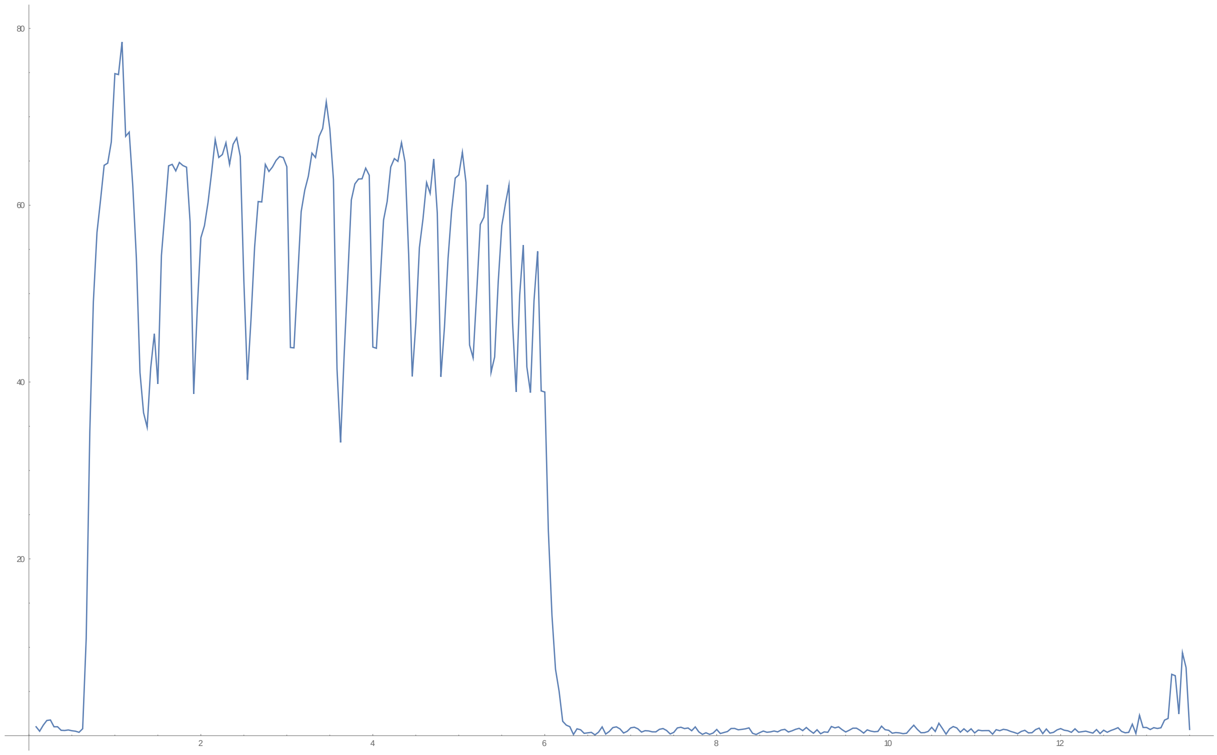


Figure 5.22 Feed rate for code with Look ahead and parameters A255 E0 for programmed F=5000 mm/min

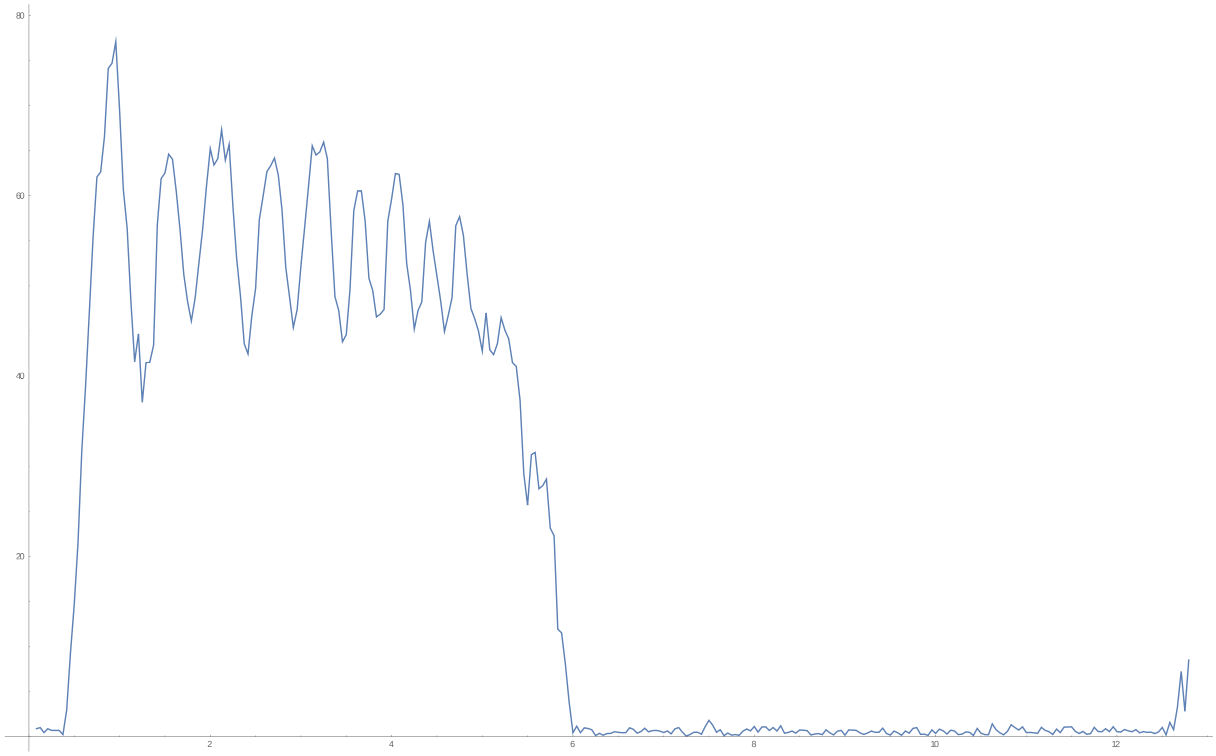


Figure 5.23 Feed rate for code with Look ahead and parameters A255 E99999 for programmed F=5000 mm/min

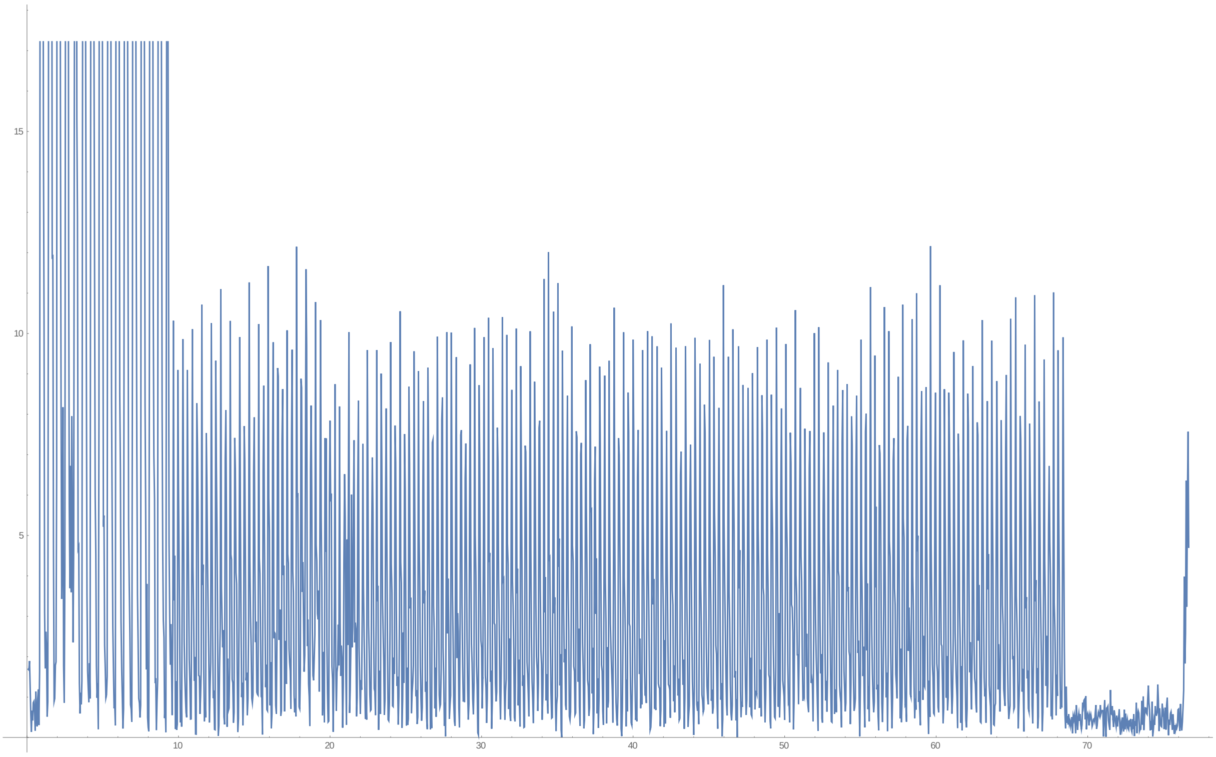


Figure 5.24 Feed rate for code without Look ahead for programmed F=5000 mm/min

5.3 Similarities between actual tool-path and a B-Spline curve

The idea of this research is to know if is possible to represent the actual tool-path with a NURBS. As a result of the tests, weights for each code were calculated to get the most similar geometries between the actual path and B-Spline curve. The weights for the codes with Look-ahead and with feed rate 100 mm/min are presented in the Tables 5.11 - 5.14, also with the geometries of computed spine interpolation and the actual path. The Hausdorff distance between the spline curves and the actual tool-paths is shown in the Table 5.10.

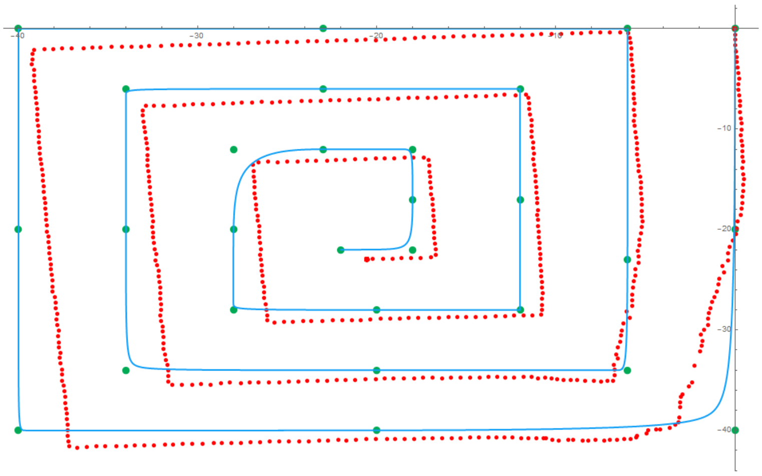
All the codes have the 26 interpolation points and 26 weights. The equal number is needed in order to make the spine interpolation. Interpolation points are the corners of the theoretical path and also in the middle of the length between them. The weights' values change in almost all the codes from 0 to 10 and are different for each tool path. Only in the code with look ahead parameters A255 E9999 weights' values hesitate from 0,00 to 8,55. With the bigger number of weight the spline is respectively further from its interpolation point. In all of the codes the actual tool path is not the same as the computed spline curves.

From the Hausdorff distance calculation the most similar tool path for codes with feed rate 100 mm/min is A255 E99999. The least similar to computed spline curve is tool path with Look ahead's parameters A0 E0. The Hausdorff distance hesitates from 2,0448 to 2,6378.

Number	Feed rate (mm/min)	A	E	Hausdorff distance (mm)
1	100	0	0	2,6378
2	100	0	99999	2,2690
3	100	255	0	2,0481
4	100	255	99999	2,0448

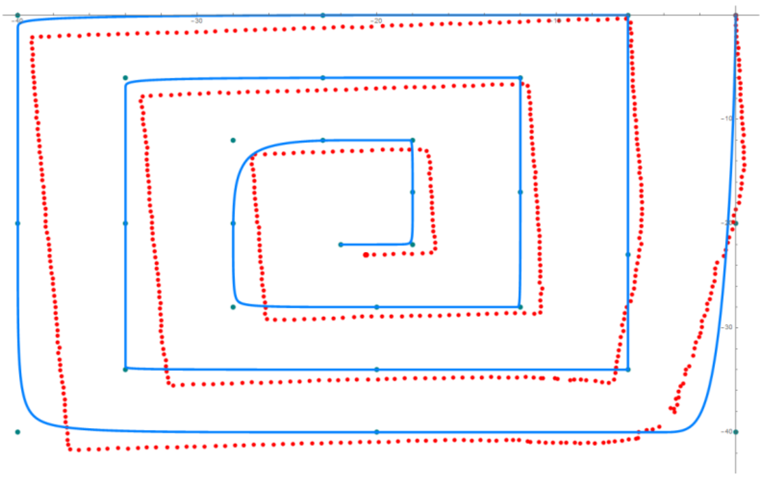
***Tab. 5.10* Feed rate, and Hd for feed rate 100 mm/min**

Number	Weights' name	Weights' value
1	na	0,80
2	nb	2,40
3	nc	0,50
4	nd	5,70
5	ne	0,00
6	nf	2,50
7	ng	0,00
8	nh	5,60
9	ni	0,00
10	nj	4,80
11	nk	0,90
12	nl	4,90
13	nm	0,70
14	nn	3,50
15	no	0,00
16	nq	2,00
17	np	0,00
18	nr	9,50
19	ns	0,00
20	nt	10,00
21	nu	3,80
22	nw	2,40
23	nx	2,40
24	ny	3,90
25	nz	0,00
26	nv	2,00



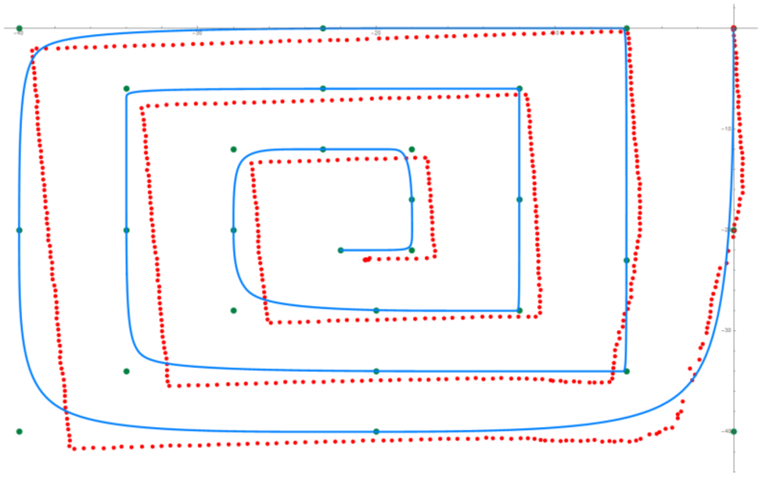
Tab. 5.11 Weights for F100* mm/min and Look ahead parameters *A0 E0

Number	Weights' name	Weights' value
1	na	0,00
2	nb	5,80
3	nc	0,00
4	nd	5,40
5	ne	0,00
6	nf	2,15
7	ng	0,00
8	nh	8,20
9	ni	0,00
10	nj	10,00
11	nk	0,00
12	nl	8,98
13	nm	0,97
14	nn	3,28
15	no	0,00
16	nq	10,00
17	np	0,00
18	nr	8,28
19	ns	0,86
20	nt	9,30
21	nu	4,62
22	nw	2,31
23	nx	1,77
24	ny	9,52
25	nz	0,00
26	nv	9,25



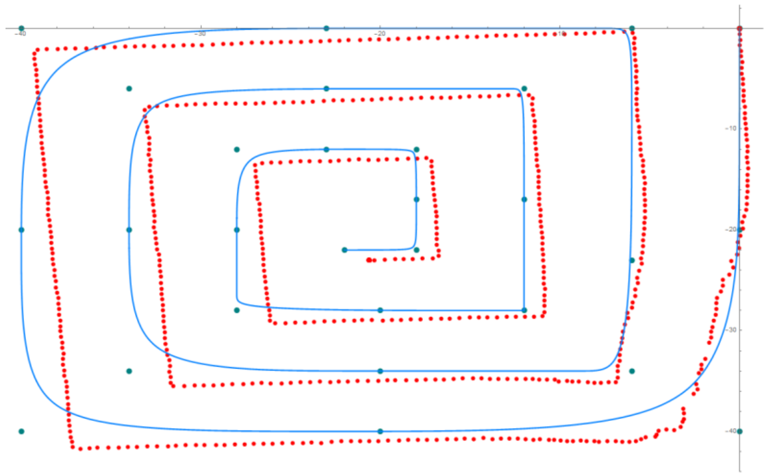
Tab. 5.12 Weights for F100 **mm/min and Look ahead parameters** ***A0 E99999***

Number	Weights' name	Weights' value
1	na	0,75
2	nb	1,51
3	nc	1,51
4	nd	2,53
5	ne	1,56
6	nf	2,00
7	ng	0,43
8	nh	10,00
9	ni	6,08
10	nj	0,00
11	nk	0,38
12	nl	1,83
13	nm	1,45
14	nn	4,89
15	no	0,00
16	nq	9,89
17	np	0,00
18	nr	10,00
19	ns	0,81
20	nt	1,67
21	nu	3,17
22	nw	3,23
23	nx	1,24
24	ny	0,81
25	nz	0,00
26	nv	1,99



Tab. 5.13 Weights for F100 mm/min and Look ahead parameters A255 E0

Number	Weights' name	Weights' value
1	na	3,23
2	nb	3,71
3	nc	2,63
4	nd	5,91
5	ne	4,57
6	nf	3,28
7	ng	2,00
8	nh	4,95
9	ni	0,00
10	nj	2,00
11	nk	2,00
12	nl	3,01
13	nm	2,63
14	nn	3,44
15	no	2,10
16	nq	7,74
17	np	0,00
18	nr	7,69
19	ns	0,00
20	nt	2,10
21	nu	2,00
22	nw	2,44
23	nx	1,99
24	ny	7,96
25	nz	1,34
26	nv	8,55



Tab. 5.14 Weights for F100 mm/min and Look ahead parameters A255 E99999

The weights for the codes with Look-ahead and with feed rate 1000 mm/min are presented in the Tables 5.16 - 5.19, also with the geometries of computed spine interpolation and the actual path. The Hausdorff distance between the spline curves and the actual tool-paths is shown in the Table 5.15.

Almost all the codes have the 23 interpolation points and 23 weights. The only code which has 26 weights is with look ahead parameters A0 E0. For this code interpolation points are the corners of the theoretical path and also in the middle of the length between them. For the rest of the tool paths the interpolation points are the corners and only 10 of them are in the middle between them.

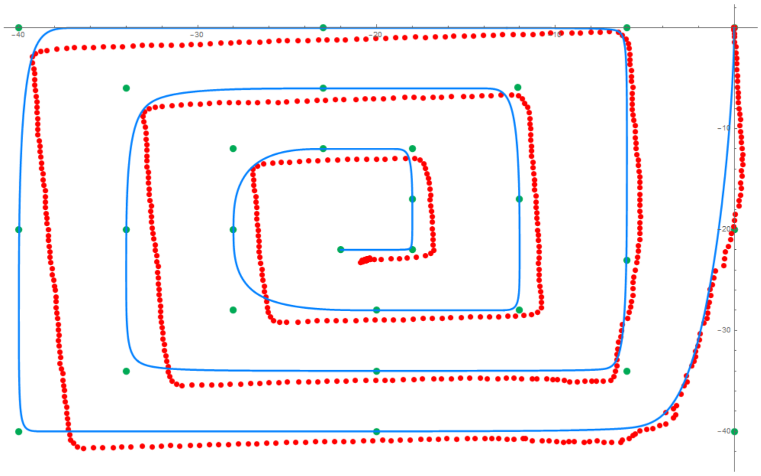
The weights' values change in almost all the codes from 0 to 10 and are different for each tool path. Only in the code with look ahead parameters A255 E9999 weights' values hesitate from 0.00 to 7.69. With the bigger number of weight the spline is respectively further from its interpolation point. In all of the codes the actual tool path is not the same as the computed spline curves.

From the Hausdorff distance calculation the most similar tool path for codes with feed rate 1000 mm/min is A255 E99999, which is similar like in the tool paths for feed rate 100 mm/min. The least similar to computed spline curve is tool path with Look ahead's parameters A255 E0. The results of Hausdorff distance hesitates from 2.1100 - 2.6883 mm, which is 0.0652 mm higher than for the tool-paths with feed rate 100 mm/min.

Number	Feed rate (mm/min)	A	E	Hausdorff distance (mm)
1	1000	0	0	2,5185
2	1000	0	99999	2,3421
3	1000	255	0	2,6883
4	1000	255	99999	2,1100

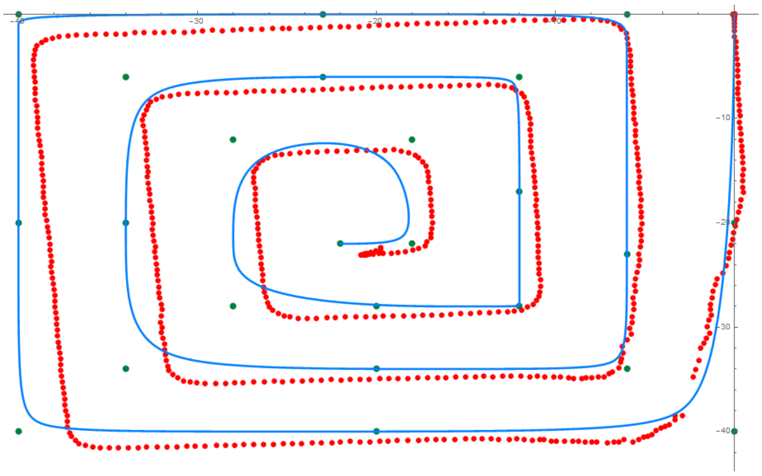
Tab. 5.15 **Feed rate, Length and Hd for feed rate 1000 mm/min**

Number	Weights' name	Weights' value
1	na	0,05
2	nb	1,13
3	nc	2,00
4	nd	6,30
5	ne	0,00
6	nf	2,53
7	ng	2,42
8	nh	10,00
9	ni	0,27
10	nj	5,05
11	nk	2,80
12	nl	5,43
13	nm	0,70
14	nn	2,26
15	no	2,37
16	nq	2,42
17	np	0,00
18	nr	7,10
19	ns	6,13
20	nt	6,02
21	nu	8,12
22	nw	5,86
23	nx	10,00
24	ny	8,23
25	nz	0,00
26	nv	10,00



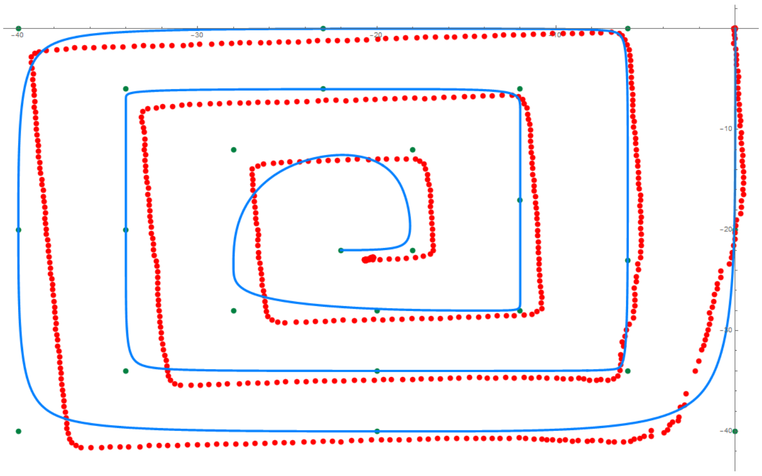
Tab. 5.16 Weights for F1000 mm/min and Look ahead parameters A0 E0

Number	Weights' name	Weights' value
1	na	0,32
2	nb	1,83
3	nc	1,88
4	nd	5,32
5	ne	0,75
6	nf	2,85
7	ng	0,00
8	nh	4,46
9	ni	2,85
10	nj	10,00
11	nk	2,74
12	nl	3,44
13	nm	2,42
14	nn	2,26
15	no	1,08
16	nq	5,16
17	np	0,81
18	nr	9,03
19	ns	0,00
20	nt	2,20
21	nu	6,24
22	nw	10,00
23	nx	10,00



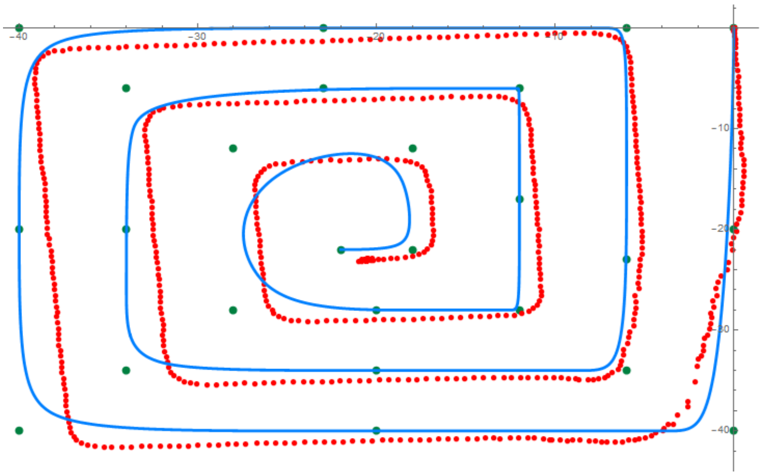
Tab. 5.17 Weights for F1000 mm/min and Look ahead parameters A0 E99999

Number	Weights' name	Weights' value
1	na	2,74
2	nb	4,46
3	nc	2,10
4	nd	2,90
5	ne	0,81
6	nf	3,12
7	ng	2,53
8	nh	10,00
9	ni	0,43
10	nj	6,67
11	nk	1,67
12	nl	4,52
13	nm	0,65
14	nn	2,20
15	no	0,00
16	nq	4,25
17	np	4,41
18	nr	10
19	ns	0,00
20	nt	1,50
21	nu	2,37
22	nw	5,91
23	nx	8,66



Tab. 5.18 Weights for F1000 mm/min and Look ahead parameters A255 E0

Number	Weights' name	Weights' value
1	na	0,00
2	nb	2,42
3	nc	2,04
4	nd	4,25
5	ne	1,72
6	nf	4,52
7	ng	2,74
8	nh	7,69
9	ni	0,00
10	nj	5,16
11	nk	5,00
12	nl	5,75
13	nm	1,34
14	nn	0,65
15	no	0,11
16	nq	3,39
17	np	0,00
18	nr	4,62
19	ns	1,45
20	nt	1,45
21	nu	2,26
22	nw	6,77
23	nx	7,63



Tab. 5.19 Weights for F1000 mm/min and Look ahead parameters A255 E99999

The weights for the codes with Look-ahead and with feed rate 5000 mm/min are presented in the Tables 5.22 - 5.25, also with the geometries of computed spine interpolation and the actual path. The Hausdorff distance between the spline curves and the actual tool-paths is shown in the Table 5.21.

All the codes have the 18 interpolation points and 18 weights. The weights' values change is different for each tool path. For the code with look ahead parameters A0 E0 weights' values hesitate from 0.27 to 9.84. For the code with look ahead parameters A0 E99999 weights' values hesitate from 0.27 to 2.58. For the code with look ahead parameters A255 E0 weights' values hesitate from 0.38 to 10. For the code with look ahead parameters A255 E99999 weights' values hesitate from

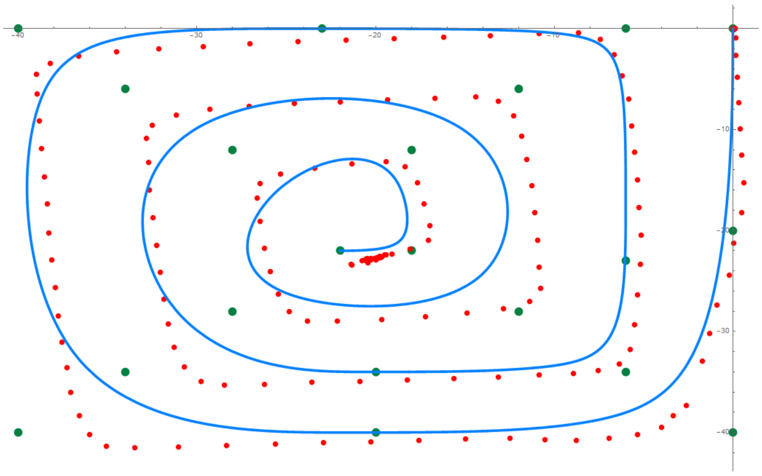
0.27 to 3.76. With the bigger number of weight the spline is respectively further from its interpolation point. In all of the codes the actual tool path is not exactly the same as the computed spline curves, but it is more similar than in the codes with feed rate 100 mm/min or 1000 mm/min.

From the Hausdorff distance calculation the most similar tool path for codes with feed rate 1000 mm/min is A0 E99999, which has the smallest values of weights. The least similar to computed spline curve is tool path with Look ahead's parameters A0 E0 similar like in the tool path with feed rate 100 mm/min. The results of Hausdorff distance hesitates from 2.0063 - 3.4978 mm, which is 0.0385 mm smaller than for the tool-paths with feed rate 100 mm/min. However, the code with look ahead parameters A0 E0 has the biggest value of the Hausdorff distance from all the tool-paths.

Number	Feed rate (mm/min)	A	E	Hausdorff distance (mm)
1	5000	0	0	3,4978
2	5000	0	99999	2,0063
3	5000	255	0	3,3636
4	5000	255	99999	2,0600

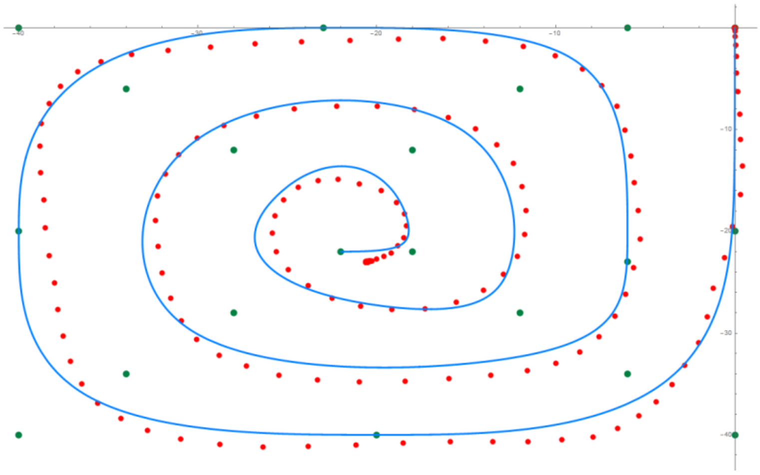
Tab. 5.20

Number	Weights' name	Weights' value
1	na	0,48
2	nb	1,77
3	nc	1,67
4	nd	3,44
5	ne	2,69
6	nf	0,27
7	ng	2,80
8	nh	4,57
9	ni	9,84
10	nj	2,26
11	nk	0,97
12	nl	2,15
13	nm	5,48
14	nn	4,68
15	no	3,82
16	nq	4,14
17	np	9,57
18	nr	9,19



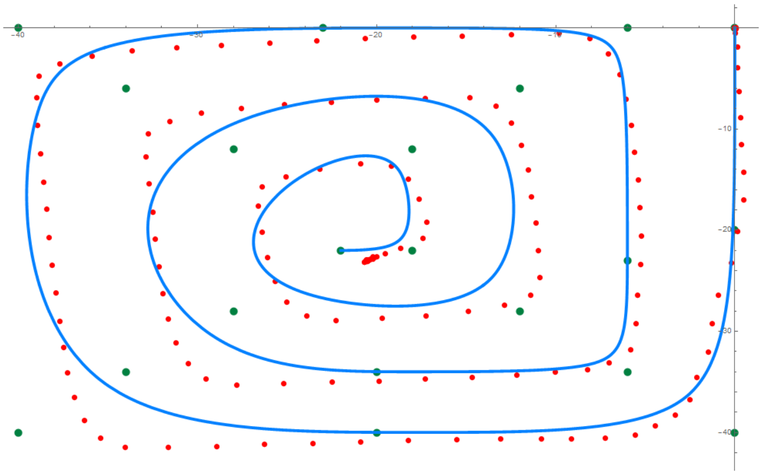
Tab. 5.21 Weights for F5000 mm/min and Look ahead parameters A0 E0

Number	Weights' name	Weights' value
1	na	2,15
2	nb	1,67
3	nc	1,88
4	nd	1,56
5	ne	2,00
6	nf	1,40
7	ng	2,58
8	nh	2,26
9	ni	1,61
10	nj	1,77
11	nk	2,58
12	nl	2,31
13	nm	2,58
14	nn	2,80
15	no	0,65
16	nq	0,27
17	np	0,54
18	nr	1,88



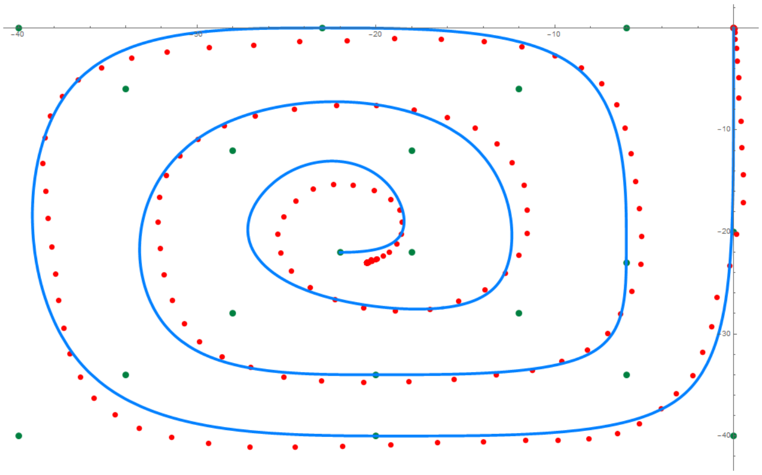
Tab. 5.22 Weights for F5000 mm/min and Look ahead parameters A0 E99999

Number	Weights' name	Weights' value
1	na	1,72
2	nb	2,74
3	nc	1,83
4	nd	1,34
5	ne	1,13
6	nf	0,38
7	ng	2,47
8	nh	1,51
9	ni	3,76
10	nj	2,74
11	nk	1,72
12	nl	3,06
13	nm	8,17
14	nn	10,00
15	no	5,11
16	nq	2,85
17	np	8,01
18	nr	5,75



Tab. 5.23 Weights for F5000 mm/min and Look ahead parameters A255 E0

Number	Weights' name	Weights' value
1	na	1,51
2	nb	2,10
3	nc	1,83
4	nd	1,51
5	ne	1,67
6	nf	2,15
7	ng	2,37
8	nh	2,00
9	ni	1,61
10	nj	3,06
11	nk	1,61
12	nl	2,00
13	nm	2,00
14	nn	1,40
15	no	0,27
16	nq	0,11
17	np	0,32
18	nr	3,76



Tab. 5.24 Weights for F5000 mm/min and Look ahead parameters A255 E99999

6 DISCUSSION OF CONCLUSION

In the presented researches were observed influence of Look-ahead function on the cycle time. When the length of the block is not big, the look ahead festinates the machining. This acceleration results in the changes in the geometry of the work piece. Although, in the tool paths made with feed rate 100 mm/min do not observed significant change in the geometries, the time even in the simple contour changed. The biggest influence on the geometry is presented in the code made with the feed rate 5000 mm/min.

The differences in length according to theoretical path with Look-ahead are around 20,12 - 64,092 mm in the codes with feed rate 5000 mm/min, which is 2-5 times bigger than for the codes with 100 mm/min. The error of the camera needs to be taken under the consideration. In the feed rate 100 mm/min, we distinguish 0,069 mm. While in the codes with a feed rate 5000 mm/min the resolution is 3,5 mm, which is one of the reason of such a big change in the geometry.

If it comes to Hausdorff distance, it could not be calculated from the theoretical tool path because of the displacement of the trajectories computed in Mathematica program. This displacement is caused by bad fix camera. The use of Look-ahead function provoked in code with the feed rate 100 mm/min shape changes from 0,684 to 1,156 mm and it is similar like for the codes with feed rate 1000 mm/min, where the difference from actual shape from 0,955 to 1,224 mm. The Hausdorff distance increase for the codes feed rate 5000 mm/min and it varies from 1, 989 to 4,599 mm. The smallest Hausdorff distances in code with feed rates 100 m/min and 5000 mm/min were calculated for the codes where programmed errors were 0.

The use of Look-ahead function has a big impact on the feed rates in the code made with F=1000 mm/min and F = 5000 mm/min. It causes the stability of the feed rates and the smooth transition from one block to another.

In all of the codes were impossible to get exactly the same like theoretical tool path. However, it has been found the similarities between actual tool path and B-Spline. To better match the Spline curves to the shape of the Look-ahead tool paths, the greater number of weights needs to be added. In the tool paths with feed rate 5000 mm/min it was possible to get the smaller number of weights because the actual tool path is more similar to the spline interpolation.

The further researches about the look ahead parameters like error, acceleration and its connection with the weights and control points can provoke in the future possibility of getting the spline interpolator with the look-ahead function. This could results the optimizing the production and make the work easier even on the conventional machines.

BIBLIOGRAPHY

[1] Al-Habahbeh O., Numerical Control, Presentation, Jordan

[2] Collins D., What's the difference between contouring and point-to-point motion, May 2016, Available online: http://www.motioncontroltips.com/faq-whats-difference-contouring-point-point-motion/

[3] Dae-Hyuk Ch. et al., Theory and Design of CNC Systems. Springer, Cardiff 2008

[4] Denford, G and M Programming for CNC Milling Machines, England

[5] Fagor Automation, Operating manual CNC 8055,

[6] Farin G.E., NURBS From Projective Geometry to Practical Use, Tempe

[7] Groover M.P, Fundamentals of Modern Manufacturing, John Wiley&Sons 2010

[8] Groover M. P., Automation, Production Systems, and Computer-Integrated Manufacturing. Third Edition, Pearson Education, New Jersey 2008

[9] Grzesik W. et al., Programowanie obrabiarek NC/CNC. Warszawa, WNT 2010

[10] Heinrich A., Tech recent history of the machine tool industry and effects of the technological change, University of Munich, 2001

[11] Hollister S.M., The Dirty Little Secrets of NURBS, New Wave System, 2001

[12] Industrial Center, Computer Numerical Control (CNC), Hong Kong 2009

[13] Klocke F., Manufacturing Processes 1.Spinger, Aachen 2010

[14] Krar S., Gill A., Computer Numerical Control Programming Basics, Machinery's Handbook, New York

[15] Kształcenie na odległość: Moduł 7 Układy sterowania numerycznego, Kapitał ludzki, Warszawa 2008

[16] Madison J., CNC Machining Handbook: Basic Theory Production Data, and Machining Procedures, Industrial Press, New York 1996

[17] Makhanov S. S. et al., Advanced Numerical Methods to Optimize Cutting Operations of Five-Axis Milling Machines, Springer, Berlin 2007

[18] Mechatronics and Manufacturing Automation, Module 4 Drives and Mechanism

[19] Nikiel G.: Programowanie obrabiarek CNC na przykładzie układu sterowania Sinumerik 810D/840D. Bielsko-Biała, ATH 2004.

[20] Pabla B.S., Adithan M., CNC Machines, New Age International, New Delhi 2005

[21] Patrikalakis N.m., Maekawa T., Shape Interrogation for Computer Aided Design and Manufacturing, Springer 2001

[22] Piegl L., Tiller W., The NURBS Book, Springer 1997

[23] Ptaszyński W., Programowanie obrabiarek CNC, Nr 1 Obróbka prostych kształtów, Politechnika Poznańska, Poznań 2015

[24] Radhakrishnan P., et al.: CAD/CAM/CIM, New Age International, New Delhi 2004

[25] Sarcar M.M.M, Mallikarjuna RAO K., Lalit Narayan K., Computer Aided Design and Manufacturing, Prentice Hall of India, New Delhi 2008

[26] Tokarska M., Rabiasz S., Operator Obrabiarek Sterowanych Numerycznie, Państwowa Wyższa szkoła zawodowa, Krosno 2009

[27] Vasandia N., Introduction to NC, CNC, DNC machine tools, Shree S'ad Vidya Mandal Institute of Technology, 2016

[28] Wang L., A look-ahead and adaptive speed control algorithm for high-speed CNC equipment, Springer, London, January 2012

[29] Younkin G., Orgin of Numerical Control, Available online: http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=4609884

[30] Zielińska A., Herma S., Historie i tendencje rozwoju obrabiarek

Webpages:

[31] cnc.pl

[32] cncwikipedia.com

[33] http://nptel.ac.in

[34] http://www.rhino3d.com/nurbs/