Manual of CNC programming (Vitralab_Title)

Doc. Type: Internal Working Document (Vitralab_Paragraph_1)
Authors: Name and surname of authors without titles (Vitralab_Paragraph_1)
Version: version number – 1.0 (Vitralab_Paragraph_1)

Place, Month, Year (Vitralab_Paragraph_centered)

Distribution:
Vitralab
Consortium
partners
1. Definition

**Computer numeric control production machines (CNC)** are characterized by central control system, driving main and auxiliary machine functions by program. Information is in a program written by alphanumeric characters. The program itself is given by sequence of separate group of characters called blocks or sentences. The program is designed to manage power elements of the machine and guarantees the manufacturing of the workpiece.

**Term CNC** (Computer Numerical Control) means: Computer (numerically) controlled machine.

Machines are flexible, can quickly adapt for different production and work in automatic cycle which is managed by numerical control. CNC machines applied in all sectors of industrial production (machining, forming, assembling, measuring). Their typical representatives are lathes and milling machines used for training programmers and operators.

The information in the program can be divided into:

**Geometric** – describe the tool path, which are given by workpiece dimensions, machining methods and describe feeding of the tool to the workpiece and from it. It is a description of the tool tracks in Cartesian coordinates, when the creation of the program needs the dimensions of workpiece blueprint. Description in program is in X,Z axis for the lathe, in axis X, Y, Z for milling machine (and also by other axes depending on construction of machine and workpiece complexity), by given functions which provides ISO and the individual producers of control systems.

**Technological** – provide machining technology in terms of cutting conditions (especially the speed or cutting speed, feed or depth).

**Auxiliary** - it's information, commands for the machine for some auxiliary functions (such as switching on coolant pump, direction of spindle rotation etc.)
1.1. Scheme of CNC machine tool

![Diagram of CNC machine tool]

**Fig. 1 - Block diagram of the CNC machine tool – simplified**

- **Computer** – it is an industrial computer with pre-recorded control system, which is part of the machine. Is given by a screen and control panel. Via control panel is possible to perform requested steps during manual servicing, for adjusting CNC machining and work in other modes of machine. The computer enables, by using the software control system, to create the required CNC program.

- **Control circuits** - In these circuits the logic signals are converted to high-voltage electrical signals, which directly control each part of the machine - the spindle and feed motor, valves, etc.

- **Interpolator** - Managing the tool path, which is given by geometry, calculations of the length and radius corrections of tool. So it calculated equidistant movement, which is shifted from the calculated correction of the geometric contour. Guarantees the geometric accuracy of the product.

- **Comparative circuit** - The machine must be equipped with the feedback (with exceptions for simple CNC machines for staff training), which transmits information about the geometric values in coordinate axes at various points within the range of motion. These coordinates are compared with the values that are given by the program (and modified in interpolator). If a difference appears, feed drives receive the command to achieve the desired coordinate values. The machine must be equipped with a transducer, which is used to obtain the coordinates.

- **Control panel** – (can be solved as a figure 2) divided into several parts, which differ with their meanings:
- data input – alpha numeric section, which is used to manually write for example a program, data instruments, adjusted for machinery, etc.;
- machine control – special section used to tool or workpiece motion, triggers the spindle speed, affects the size of the hand-feeds, speeds, etc.;
- choice of operation mode - you can choose the manual mode, automatic mode, workshop programming, etc.;
- memory activation - induce different types of memory;
- tests activating – calling tests of programs and test of machine, simulation of programs;
- screen - used to control process;
- portable panel - for controlling the basic physical functions of the machine as a basic part of the keypad. Allows the operator to go to places that offer the possibility of more accurate and more complete visual inspection.

*Fig. 2 - CNC machine control panel – example of the on of many design*
1.2. Schemes of work CNC machining machine

During operations, we may encounter several types of operations of the machine or only types of machine control system. They can be set on the control panel by buttons. Typically, control systems have these schemes:

- **Manual mode** - is used for resetting tool or measuring equipment in the desired position, tool change, approaching to the workpiece, start-up speeds, etc.

- **Automatic mode** - a smooth implementation of the program. After the block processing machine reads and processes the next block automatically - smooth machining process.

- **Mode B - B (block by block)** - the machine stops after processing the block and after re-start reads and processes next block. BB scheme is one of the options for checking correctness of CNC program.

- **Setting (impact speed, work shift, fast feed)** - The amount of movement can affect by hand control, by potentiometer, where you can adjust range usually between 5-150% of the value set in manual or automatic mode.

- **Tool memory mode (the tool data memory)** - you can save and recall tools data, including corrections.

- **Teach in mode** ("learning" or "lead-in and storage") - the machine has the "ability" to learn. Operator manually (via keyboard) perform required movements to manufacture workpiece.

- **EDIT mode program** - a program for processing is entered directly into the editor of the machine or is "loaded" into the machine control system externally. In the editor of the machine programs can be repaired as required.

- **Diagnostic mode** – reports, locates, diagnoses defects for quick removal. It also allows remote service.
1.3. Coordinate system of the machine

Production machines use a Cartesian coordinate system. The system is right-handed, rectangular with axes X, Y, Z, rotary motion, whose axes are parallel to the axes X, Y, Z, marked A, B, C - Figure 3. True that the Z axis is parallel to the axis of the work spindle, and a positive sense takes place from the workpiece to the tool. Values are present also in the negative field of coordinates.

![Cartesian coordinates](image)

*Fig.3 - Definition of Cartesian coordinates – right-handed system*

Cartesian coordinate system is necessary for the machine control and for the measuring of the tools. Tool, in the machine, moves according to orders from the CNC machine control panel or under the CNC program commands. If necessary, coordinate system can move and rotate. In the case of measurement instruments (surveys corrections) is a Cartesian system placed in a point of exchange tools or tool tip.

**The coordinate system is placed in the machine according to the following rules:**

1) Start from a stationary workpiece.
2) Always must be defined X-axis.
3) The X axis lies in the plane of the fixture or the workpiece or is parallel to its plane.
4) Z-axis is identical or parallel to the axis of work spindle, which grants main cutting movement.
5) A positive axis sense goes from the workpiece to a tool in the direction of workpiece growing.
6) If there are other additional movements in the axes X, Y, Z on the machine, these are called U, V, W.
7) If the workpiece is moved against the tool, coordinates are called X', Y' and Z'.
Fig. 4 – Lathe coordinate system (spindle without driven tools)

Fig. 5 – Multi-axes lathe
Fig. 6 – Coordinate system for CNC milling and drilling

Fig. 7 – Coordinate system of 5-axes centre
On top of the basic coordinate system is necessary to reference points define in the workspace of the machine, which help determine the relative position of machine, tool and workpiece.

---

Fig.8 – Coordinate system of machining center
1.4. Zero and reference points on CNC machines

After switching CNC control system on the machine, coordinate system in its own machine is activated. The coordinate system has its origin - the zero point, which must be specified. According to the application the zero points have their names. The CNC machines have other important points:

- **M – zero point of the machine**

  This is the beginning point of the coordinate system of the machine work area. It is a fixed structure (usually the intersection of the main spindle axis and workpiece mounting plane) and can’t be changed. It’s an absolute beginning of the coordinates.

- **W – zero point of the workpiece**

  It is the beginning of the workpiece coordinate system. The position chosen freely by the programmer and can be changed during the program. Zero point is usually chosen in the axis of symmetry and at the upper surface of the workpiece (semifinished).

- **R – reference point of the machine**

  It is a place on the machine (usually in the working space of the machine), in the maximum possible distance from the zero point of the machine, due to end switches in each axis. Only after moving the reference point relative to the point M, machine “knows where it is”. Distance from reference point and the zero point is stored in the machine table of the machine constants. Without moving the reference point R, machine can’t work in absolute coordinates input.

- **P – point of the tool tip**

  It is necessary for determining the length correction and subsequent radius correction (nose compensation). It is a point which movement is theoretical programmed (if using radius correction).

- **F – reference point of the slide or spindle**

  It is a point on the fixture (seating) area of the media tools (for example end of the spindle in the spindle axis). This point manages the control system according to the program. In the point F has the tool zero value, so it is therefore necessary to correct the actual tool path. Corrections of the tool are applied to this point.

- **E – set-point machine**

  Point of the tool mounting, which is identical with F pint after workpiece fixation (it is necessary to ensure the correction of the tool on the device outside the machine).
At the beginning of processing (programming) it is necessary to move the coordinate system from the machine zero point into the zero point of the workpiece.

The current system allows two types of beginning displacement:

- absolute (set) offset – calls by preparatory function in the program (G54 - G57) - individual displacements are absolute - they indicate the distance from point W to the point M - each new shift cancels the previous
- programmable (additive) offset (G58 - G59) - is relative - indicates the distance from the W hotspot - added to the absolute displacement - applied only in the sentence, which was called
1.5. Determination of the zero point of the workpiece W

- scratch by tool - it is not accurate (ovality, throwing blank, operator skills), but don’t requires the equipment costs
- by measuring eccentric touch
- using a probe
- using optical devices

Fig.11 - Offset of workpiece zero point - W

Fig.12 - Scratch by tool
Eccentric contact has two parts - clamping and touch. Touch part move to the semiproduct. Eccentricity is reduced to zero - at this point will be position deducted. A little touch of overrunning evoke the vibrated eccentrically again.
1.6. Tools correction

Location of media tools in machine coordinate system is relative to point F (zero-point on the machine carriers). The surface of the workpiece is created by the tool tip, point F shall therefore describe ecvidiants so therefore must be activated corrections, which automatically handle the interpolator.

One other reason for the use of corrections is that different instruments have different dimensions. If it was not treated by corrections, the various instruments in the same sentence of the program held a variety of tracks to the workpiece. On the following picture the shape of the workpiece is made by the black tool with black color. Red tool, which has a tip after the clamping into the fixture in another place under the same program without correction created the red shaded shape.

![Fig.15 - Workpiece shape changing in machining with various uncorrected tools](image)

It is true that in many cases it is possible to set the instrument into the module clamps so that the edge of each instrument was set to the same point, but it is tedious and difficult.

All corrections are stored in memory of corrections.

Corrections are divided into:

- length
- diametric (radius)

Tools corrections are usually measured on a special machine outside the workplace in order to maximize machine time. To that are used special optical instruments.
1.6.1. Length corrections

They are used both in turning and in milling.

When turning the surface it does not follow the required theoretical tool tip (with zero radius), but the actual tip with radius of a certain size. This results in the bevels and rounds the variations of the theoretical and actual shape. Therefore it is necessary in memory of corrections specify a tool nose radius and tool position due to the machined surface, in order interpolator can calculate the ecvidistant of the track.

Fig.16 - Contour defect without correction of tip radius

1.7. Diametric (radius) corrections

For the activation radius compensation are used preparatory functions G41 or G42. These functions are so-called modal (it means valid until further notice). The validity of the functions is terminated by a preparatory function G40.

G 41 - CORRECTION OF THE TOOL RADIUS IN THE LEFT OF OUTLINE

G 42 – CORRECTION OF THE TOOL RADIUS IN THE RIGHT OF OUTLINE

Evaluation if the tool is on the left or right is done from the perspective of the tool feed direction.

When we are not using averaging correction (G41, G42), then the system operates a zero tool holder (F) as the tool axis. This means that if we use tools with different diameters would be during the execution of the program made components of different sizes. Sizes of ground-plans machined using various tool diameters are shown in the next picture.

Fig.17 - Workpiece dimensions without diameter corrections
2. Numerical control systems

Numerical control systems can be divided into two groups:

- NC
- CNC

Computer Control Systems (CNC), for all its advantages prevail, whereas the NC systems are both technically and morally obsolete and are used only in the depletion of their life.

2.1. NC control systems

- Into the memory system is read only one sentence which will be executed.
- After the sentence is read, loads a new one.
- When the new sentence loads, the current content of memory is overwritten.
- Information is entered into the program on paper tape, or manually from the keyboard.
- Program on paper tape is read again and again during the production of other items.
- For making the next piece the tape has to be rewind to its beginning.
- Any treatment of the program is possible only by adjusting the paper tape.
- In the program can not be used parameters and user subroutines, program can not branch out.

Fig. 18 – Scheme of NC control system
2.2. CNC control systems

System is reading whole program from diskette or other medium for storing data, or from LAN net (cable or wireless). Difference from NC systems is, that interpolator is not hardware but software. To generate profile of trajectory is possible to use mathematical description. So it is also possible to generate parabolas and high-order curves. Systems with better processor performance can realize dimensional circular interpolation, but in practical use linear and circular interpolation is sufficient.

Programmable logic controller (PLC) is used to processing technological information for CNC systems.

**CNC systems benefits:**

- easy to edit program
- expand program
- to use parameters
- work with sub-programs
- to use graphic simulation of machining
- to use diagnostic programs
- to offset inaccuracies of system and machines parts

![Fig.19 – Scheme of CNC control system](image)
2.3. Digital systems according tool’s trajectory control to the workpiece

2.3.1. Continuous control systems

Coordinates system input
- Interpolation is missing.
- Tool is moved with rapid feed to the programed point. Carried trajectory is not issue (tools is moved in a plane of one axis an then in a plane of second axis to the designated point).
- After reaching designated point, movement in other axis is performed
- Useful for drillers and forming machines.

Rectangular control
- Tool movement is parallel with coordinates systems.
- There is movement in one axis, after finishing movement in other one.
- Useful for drillers and forming machines and lathes.

2.3.2. Coherental control systems

Systems enabling computing of corrections and geometry.
- Tools is moved in plane X-Z (2D) in lathes
- For millers, linear interpolations are possible in one plane, X-Y, X-Z, Y-Z (2,5D). By using powerful processor, it is possible to machine various shapes and 3-D surfaces. If other movements, out of movements in axis, are possible (rotations), we are talking about 4D and 5D control.

Fig.20 - 2D control  
Fig.21 - 2,5 D control
2.4. According to the programming method of tool location against the workpiece

2.4.1. Absolute programming (G 90)

- all the programmed tool path points are related to a pre-selected point – zero point of the program (W), the location chosen arbitrarily by programmer
- for needs of absolute programming is better to use quotations from the base (grid dimensions)

!! In program the end-point position is adjusted !!

During setting of dimensional words (X,Y,Z, ...) in absolute programming is the basic question:

What distance from the zero point of the program should the tool reaches (in each axis)?
2.4.2. Incremental programming (G 91)

- coordinates of all programmed points are given due to the previous point, which is regarded as an initial
- for needs of incremental programming are so used string dimensions

During setting of dimensional words (X,Y,Z, ...) in absolute programming is the basic question:

What kind of distance should tool draw forward from endpoint of preceding movement (in each axis)?

Fig.25 - Incremental programming - dimensions
2.5. Information processing in control system

Information which management system needs to correct action can be divided into:

- geometrical
- technology and support
- necessary for the organization of the program

2.5.1. Geometric information

Information about track media of tool are processed in the interpolator. Interpolator is an arithmetic unit which calculates the path elements in each coordinate axis so that the resulting movement between two given points is:

- linear – linear interpolation
- around circular arc – circular interpolation
- parabola or a general curve

Interpolator generates signal of the desired path. Measuring device generates a signal on the actual track. Both signals are compared in a differential element - their difference is the regulation divergence, which after amplification and transformation creates a action quantity. In other words - differential member sends impulses to the motor until rest reached the desired position. Measuring devices work after some non-zero "jumping" – increments. Increment is the smallest measurable and therefore programmable path. At present is widely used increment 0.001 mm.

![Circular interpolation – interpolar function](image)

*Fig.26 - Circular interpolation – interpolar function*
Operation principle of interpolator of circular interpolation clockwise (G02) is on the previous figure:

1. Creates an equation of a circle in the XY plane \((X-0,026)^2 + (Y-0,001)^2 = 0,022\)

2. Send a unit impulse in the direction+X

3. Substituting the coordinates of point 1 in the equation of a circle and find that the left side of the equation is smaller than the right. This means that the point 1 lies within the arc.

4. Change the direction of movement and sends unit pulses until it finds that the point lies outside the arc, see section 2)

5. Repeats the previous paragraphs, until it gets to the end point.

### 2.5.2. Technology and support information

The control system must handle not only information about the geometry of movement, but also its speed, then feed per revolution or per unit of time, cooling type, blowing etc. Matching Logic deals with other helpful information - the logical relationships between the control commands and signals from the machine, which report the status of the various mechanisms – for example:

- spindle starts only when the chuck is clamped shut and the housing is closed
- during spin the spindle working shifts starting to work
- shifts and speeds of spindle stops working when open the door of the working place
- translation program doesn’t run in a loss of information on the reference.

### 3. CNC machines programming

#### 3.1. Structure of program

Program is sequence of sentences. Every sentence is sequence of words.

Program is bounded:

- At the begining by sentence.
- At the end of the program has to be one of the assistance functions M02 or M30.

Every sentence is bounded by agreed characters:

- Beginning of the sentence.
- End of the sentence.
Starting of the sentence:

- Character N.
- : (colon) - for some systems for main sentence – sentence, which has all necessary dates for continuing program. So called subordinate sentence starting with character N has only functions, which has been changed from the last sentence.

End of sentence

- With character LF or EOB.
- There could be character / (slash) in front of the first character of the sentence, which is marking sentence not used in the program.

Example:

```
%MPF<file>
:
N<file>
N<file>
```

LF

```
: slovo slovo slovo ...
N slovo ...
N slovo ... M02
```

Numbering of sentences is arbitrary. Same numbering can’t be used for more than one sentence. Some systems are ignoring sequence of numbering and using sequence of sentences. Which means, as in following example, sentence 1000 will be executed before sentence 5.

Sample:

```
.
.
N 200 G0 X0 LF
N 1000 G1 X100 LF
.
.
N 5 T1 D5 L96 LF
.
.
```
3.2. Sub-programs

- Is a closed part of the program, which could be repeated several time in the main program, or could be used in other program,
- Is made by programmer,
- is mentioned behind main program in some systems, but usually it is independent part of code, which is called by other program (main or by sub-program),
- has similar structure as main program,
- is re-called by word (for SINUMERIK) with address L <number of program>,
- could be re-called numerous times by using word (for SINUMERIK) with address P <number>,
- podprogramy je možné vnořovat,
- ends with word M17 and sends control back to main program

3.3. Cycles

Is a sub-program added and fixed by manufacturer of control system. Cycles are used, for example in turning, for:

- roughing lengthwise and crosswise
- grooving
- drilling holes
- threading

for milling:

- drilling
- pocketing
- grooving
- etc.

3.4. Sentence formats „Blocks“

Form of the sentence can be divided by the length:

- With fixed (constant) length
- With variable length
In formats with fixed length is necessary to use, according to the type of used word, syntactically complete words.

In the format of variable length sentence it is not necessary, system retrieves the specified word from memory of words, which pulls the contents of memory and the sentence is carried out by substituting. So until there is no change in the words, it is not necessary to write the word in the sentence.

Each phrase (block) includes characters in addition to the beginning and end of several groups of characters, known words.

Each word consist of two parts:

- adress
- semantic

Example:
4. Computer aided tool paths design – CAM

Computer aided tool paths design, part of an integrated production system, not only generates NC code for the machine (this is a fundamental output), but output may be (depending on application) as well as drawings, data on the use of materials, tools and machines, and more.

When working with the CAM system is need to select, as input information, information about:

- workpiece (geometry, material)
- technology (material, cutting conditions)
- machine on which runs the final control program

GEOMETRY OF THE WORKPIECE

CAM system uses geometry in digital form. Geometric elements that characterize the shape of the workpiece or semiproduct, may constitute such contours, surfaces, and 3D models.

Applicable digital data can arise:

- In the CAD part of CAM application.
- Retrieved from another application.
- By model digitization.

Getting geometry is shown in more detail in next image.

![Diagram](image)

*Fig.27 - Capture of geometry of workpiece for CAM applications*
4.1. TECHNOLOGY

The most popular technology used in CAD / CAM systems are shown in the following diagram.

![Technology Diagram]

*Fig.28 – Technology supported by CAM*

4.2. Procedures for making technology

Before the start of production technology is needed to know some basic information regarding the machining process. These values include the default information for creating the NC program, such as:

Definition of the workpiece

- material of the workpiece
- default semiproduct

Definition of the tool

- selection of library tools
- creation of new tools

Data of the machine and control system

- postprocesor
5. CNC machinery and the technological development

5.1. CNC machines, current status and trends of development

Status of development in the modernization and automation, the use of CNC technology in the manufacturing sector shows the following picture. Intensity of production and number of units on the axes of the graph illustrates how a given state corresponds to the use of production techniques and the type of programming. The conventional technique is likely to have future success only in single piece production and repair.

![Diagram: Deployment of production machines – equipment in factories in dependence of type and labor intensity](image)

Continuous development and upgrading of machine tools is rapidly applied in practice. This is because reduction of prices of machinery and control equipment due to their increasing utility value. Machines provide more convenience in programming, include more features, and reduce production downtimes. This leads to a reduction in traditional conventional machines deployed in production.

5.2. CNC machining centers

Diagram (Figure 30) shows a sorting machining machines, originally dedicated, sorted by technology of machining (figure doesn’t capture all the technologies and their combinations). There are very few parts which are made of only one technology, such as for turning the shaft is needed to mill the groove. Economy of operation leads to the integration of several methods of machining in one machine (center). The reasons are in the reduction (removal) in the operation time, it also increases the accuracy of operation production. Further integration of technology into the machine leads to the universal machining centers. That means for the economy:

- Reducing lead times and increased accuracy of work.
- Reducing the costs of production (instead of several machines be drawn one - saving production areas, saving depreciation costs).
- Possibility to easily automate the production (construction of flexible production lines - CIM).
For machines with HSC technology is provided as a fivefold increase in productivity, this ratio can be expected in economic savings.

Fig. 30 - Evolution from simple machines to machining centers

5.2.1. Requirements on modern CNC machines, production centers

Diagram (Fig. 30) demonstrates the current requirements for developed and sold by CNC machines (center), which are economically successful. Already in the development of the machine are taken into account the economic demands, giving rise to technological requirements and lead to the construction, design machines with advanced manufacturing technology HSC. That the machine complies with those requirements, the design shall also include other advanced features and accessories.
5.3. Examples of modern CNC technology
6. Sample of example

Example 1 (programming in control program FANUC)

Contouring, pocket

O0002 (CVIC1)

N5  T1  M6
N10 G43 H1  N91 G91
N15 M3  S1800  N100 G1 Z-4 F200
N20 G0 X50 Y-22 Z2  N105 G1 X5
N25 G1 Z-2 F200  N106 G17
N30 G41 H11  N110 G3 X-10 R5
N35 G1 Y5 F400  N115 G3 X10 R5
N40 G1 X35  N120 G1 X5
N45 G2 Y55 R25  N125 G3 X-20 R10
N50 G1 X95 C10  N130 G3 X20 R10
N55 G1 Y5 C10  N135 G90
N60 G1 X48  N140 G0 Z0
N65 G1 Y-22 G40  N145 G0 Z50
N70 G0 Z50  N150 G0 X-50 Y100
N75 T2  M6
N80 G43 H2
N85 M3  S2400
N90 G0 X35 Y30 Z2