

Translation



# Kinetic-NC Reference Manual

Software: Kinetic-NC with CncPOD und 2 LPT-Schnittstellen

Version: 1.74

Parameter settings, G-Code and macro programming, post processors

<https://www.cnc-step.com/>



powered by CNC-STEP.de

From 10.11.2018



## Reference Manual



### Short description

The Kinetic-NC software is a CNC control software for milling, engraving and cutting machines, which runs on a PC under Windows operating system. All signals are generated with an external hardware unit (so-called "CncPod"), so that no special connections (LPT / parallel port, plug-in cards) are required except for a network interface.

This manual is intended for system integrators who want to set up control on different types of machines, or for advanced users who write CNC programs by hand, or who want to create or customize macros for, for example, tool changes or post processors for CAM programs. Also the subject of parameter setting will be discussed in detail. If you have purchased a fully assembled machine and are producing the CNC programs using a CAM program, you do not need that specialized knowledge. For normal operation please refer to the *User manual*.

## Content

1	General information .....	10
1.1	Icon description .....	11
1.2	Safety instructions .....	11
1.2.1	Safety symbols .....	11
2	Parameter setting .....	12
2.1	Create backup .....	12
2.2	Connect hardware modules .....	13
2.3	Switch off the demo mode .....	13
2.4	Identify module .....	13
2.5	Possibly Update necessary .....	14
2.6	Select machine type .....	14
3	Axis definition .....	16
3.1	Rotational axes .....	16
3.2	Double axes (gantry) .....	17
3.3	Homing sequence .....	17
3.4	Backlash compensation .....	17
4	Axis parameters .....	18
4.1	The resolution .....	18
4.1.1	Belt or rack drive .....	19
4.1.2	Rotational axes .....	19
4.1.3	Servo drives .....	19
4.1.4	Calculator .....	19
4.2	Max. Speed .....	19
4.3	Working acceleration .....	20
4.4	Max. jerk .....	20
4.5	Activate display .....	20
4.6	Activate handwheel and hotkeys .....	20
4.7	Direction inverted .....	20
4.8	Backlash .....	20

---

4.9	Min. and max. Position.....	21
4.10	Homing .....	21
4.10.1	The switch .....	22
4.10.2	With the setting "in the middle" .....	22
4.10.3	Reference switch tolerance.....	23
4.10.4	Reference travel on gantry axes .....	23
4.11	Backlash compensation.....	24
4.11.1	Determining the play:.....	24
4.11.2	Enable backlash compensation: .....	25
5	Calculator for Axis parameters .....	26
5.1	Select motor type:.....	26
5.2	Select gear type:.....	26
5.3	In the field "Motor select" .....	27
5.4	In the field micro steps .....	27
5.5	In the middle field select the drive type. ....	27
5.6	Now in the field "spindle" .....	27
5.7	Explanation of the results .....	28
6	Other parameters .....	29
6.1	CV-Mode .....	29
6.2	Circle center.....	30
6.3	PWM frequency.....	30
6.4	Display of options.....	30
6.5	Spindle speed.....	30
6.6	D-Parameter.....	31
7	In- and outputs.....	32
7.1	The assignment .....	33
7.2	Inversion of inputs .....	33
7.3	Inversion of outputs .....	33
7.4	Emergency stop .....	34
7.5	Alarm signals.....	34

---

7.6	Step and direction signals .....	34
8	Manual travel speeds .....	35
8.1	Hotkeys .....	35
8.2	Save settings .....	36
8.3	Handwheel .....	36
9	Special processing types .....	37
9.1	Rotational axis .....	37
9.2	Laser cutting / engraving .....	38
9.3	Plasma cutting .....	38
9.4	Tangential knife .....	38
9.5	Wire cutting .....	38
10	Safety .....	39
10.1	Homing .....	40
10.2	Behavior after emergency stop .....	40
10.3	Security doors .....	40
10.4	Safety signals .....	41
11	Macro-Parameter .....	42
12	Hotkeys .....	43
12.1	What are they? .....	43
12.2	Definition .....	43
12.3	Key or input? .....	44
12.4	Tip: .....	44
13	Tools .....	45
13.1	Do I need it? .....	45
13.2	Radius compensation and length compensation .....	45
13.3	Add new tools .....	46
13.4	Assign names .....	46
14	Macros .....	47
14.1	User macros .....	48
14.2	Program yourself? .....	48

---

15	GUI-Settings.....	49
15.1	Change G-Code presentation .....	49
15.2	Change power settings .....	49
15.3	Accuracy vs. resolution.....	50
15.4	Import and Export .....	50
15.5	Machine parameters .....	50
15.6	Tool list.....	51
15.7	Modular.....	51
15.8	Create desktop icons .....	51
16	Path preview.....	52
16.1	What is it? .....	52
16.2	A comparison .....	52
16.3	Exact-Stop-Mode .....	53
16.4	Constant-Velocity-Mode.....	54
17	Post processors .....	55
17.1	What is it? .....	55
17.2	Program start.....	55
17.3	initialization .....	55
17.4	Typical commands .....	56
17.5	Formatting.....	56
17.6	Homing .....	57
17.7	Tool change.....	57
17.8	Arcs .....	57
17.9	End of program .....	57
18	G-Code Programming .....	58
18.1	Address letters.....	58
18.2	DIN/ISO Commands.....	59
18.3	special commands .....	61
18.4	G-commands .....	62
18.5	Other G-commands .....	70

19	M-Commands.....	71
19.1	Special commands .....	74
19.2	Compact code.....	74
20	Macro-Programming .....	75
20.1	What is it? .....	75
20.2	Where can I find the macros? .....	75
20.3	Re-definable commands and drilling cycles .....	75
20.4	Event Macros .....	77
20.5	User-Macros.....	77
20.6	Macro-Language .....	78
20.7	Calculation formulas (assignments "=") .....	78
20.8	#0..999 variables .....	79
20.9	#I0..999 Inputs .....	80
20.10	#O0..999 Outputs .....	81
20.11	Read tool parameters.....	81
20.12	IF THEN, ENDIF: Conditional execution .....	82
20.13	REPEAT: Loops .....	84
20.14	UNTIL: stop condition.....	84
20.15	SKIP: Jump to label .....	84
20.16	ASK: Dialogs for parameter input .....	85
20.17	ASKBOOL „message text“ I=1 .....	85
20.18	ASKINT „message text“ I=min J=max .....	85
20.19	ASKFLT „message text“ I=min J=max .....	86
20.20	CALL: subprogram call .....	86
21	Re-definable commands .....	87
22	Drilling cycles .....	91
23	Event-Macros .....	93
23.1	Startup .....	93
23.2	Shutdown .....	93
23.3	PrgStart .....	94

23.4	PrgEnd.....	94
23.5	PrgStop.....	94
23.6	PrgResume .....	94
23.7	Examples.....	95
23.7.1	Automatic tool change with linear magazine.....	95
23.7.2	Tool change with other magazine types .....	98
23.7.3	Tool length measurement with stationary probe .....	98
23.7.4	height measurement with mobile probe .....	100
24	Special processing types.....	103
24.1	Cylindrical engraving.....	103
24.1.1	What is it?.....	103
24.1.2	Works without CAM.....	103
24.1.3	Other work with rotary axis .....	104
24.2	Plasma cutting.....	105
24.2.1	What is it?.....	105
24.2.2	Particularities.....	105
24.3	Tangential knife cutting .....	106
24.3.1	What is it?.....	106
24.3.2	Easy Installation .....	106
24.3.3	System requirements .....	106
24.3.4	Transmission choice.....	106
24.3.5	What is supported?.....	107
24.3.6	Parameter settings .....	107
24.3.7	Track preview and excavation automatic.....	108
24.3.8	Macros and programming .....	108
24.4	Oscillating knife.....	111
25	User management.....	112
25.1	The users .....	112
25.2	Support-Admin: .....	112
25.3	Customer-Admin: .....	112

25.4	Default users: .....	112
25.5	Logged out user: .....	113
25.6	The administrative dialogue.....	113
25.6.1	Copy user.....	113
25.6.2	Rename user .....	114
25.6.3	Set / reset password .....	114
25.6.4	delete user.....	114
25.6.5	give out rights.....	114
26	Customer service .....	115

## 1 General information

The present control software was designed on the basis of all common at the time of development and known guidelines and was very elaborate and carefully tested. Nevertheless, a guarantee for faultless function cannot be given. The developers guarantee that the KinetiC-NC control, in conjunction with suitable electrical and mechanical components in the sense of the description and user instructions, is basically suitable for the intended purpose.

### **Liability**

Any liability for consequential damage or loss of profit, interruption of business, loss of information etc. is excluded. The KinetiC-NC control software is a component that can only be used in conjunction with a suitable personal computer and suitable hardware. It is by no means an independent controller.

Since mistakes, despite all efforts, can never be completely avoided, we are grateful for any note.

### **The user guides**

Since not all users need all the information about the control hardware and software, the documentation has been divided into several manuals. For hardware and wiring information, refer to the manuals for each hardware module. The software installation guide and general operating instructions are included in the *KinetiC-NC User Guide*.

The *Reference Manual* is intended for the advanced user who programs NC programs or macros "by hand" or for machine builders who need to adapt the controller to a new machine.

### **Manufacturer:**

***Benezan Electronics***

Stauffenbergstraße 26

72108 Rottenburg

Germany

### **Distribution:**

***CNC-STEP GmbH & Co. KG***

Siemensstraße 13-15

47608 Geldern

Germany

## 1.1 Icon description



**NOTE:** Special information regarding the economic use of the system



**ATTENTION:** Special information or rules and prohibitions for damage prevention.



**Statements or prohibitions on the prevention of personal injury or extensive property damage.**

## 1.2 Safety instructions



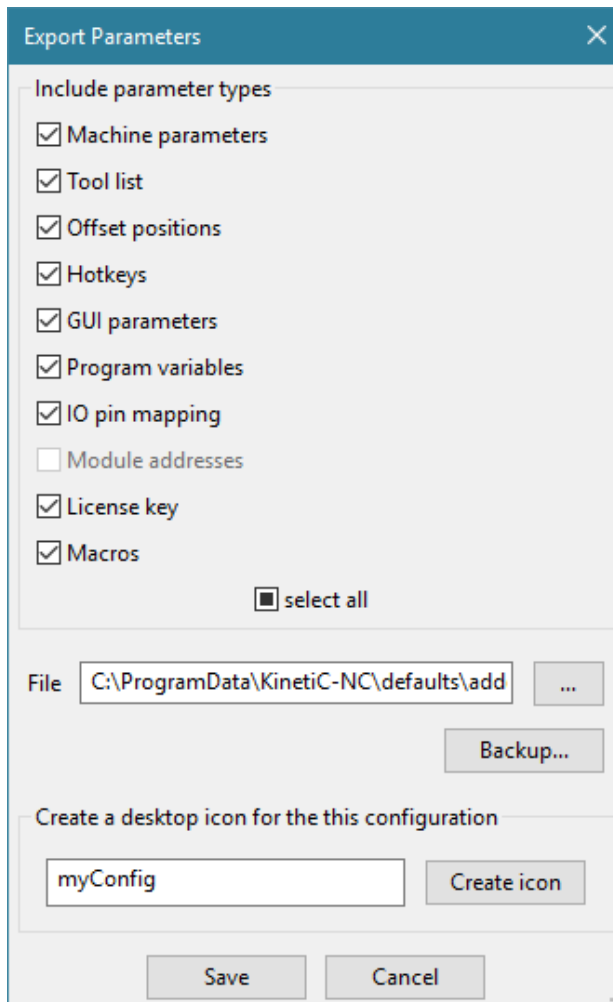
CNC machining always carries a certain risk of injury, because it works with fast rotating tools with sharp edges. Due to the computer-controlled movement of the drives, machine parts can unexpectedly start to move for the operator. Although the control software has been carefully tested, malfunctions can occur at any time. Therefore, follow the instructions of the machine manufacturer exactly. Never operate the machine without the prescribed safety devices. The plant operator is responsible for compliance with the statutory accident prevention regulations.

### 1.2.1 Safety symbols



## 2 Parameter setting

This chapter describes the procedure for adjusting the parameters of the control, such as resolutions, travel distances and speeds of the drives to a specific machine. If you bought the controller together with a finished machine, you do not need to do it yourself, but you can use a pre-made standard parameter set.



**Attention!** It may be that the software has already been developed and some of the input fields are not described in this manual. The presence of an adjustment option does not guarantee that this feature will work with all machine types. If in doubt, only change parameters whose meaning you know. If the machine is already running, be sure to make a **Backup of the parameters**, before making any changes so that you can restore it to its original state if something does not work as expected, or you accidentally change too much. It is also strongly recommended that after changing the parameters, carry out a test program "in the air" without tools and workpieces to avoid nasty surprises. A typo or an incorrect comma can have fatal consequences because the software has no way to check the entered values for plausibility.

### 2.1 Create backup

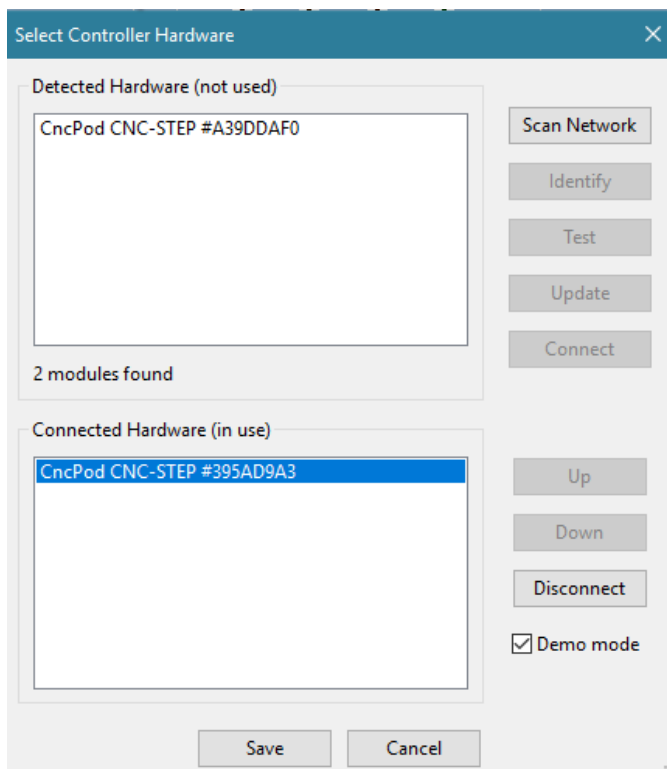
To save a backup, i.e. a backup of the parameters in a file, select "File -> Export settings" in the menu. The parameter settings of the software are divided into several groups, which can also be saved and restored individually. For a complete backup, it's best to select all the groups as shown in the picture above.

The button with the dots to the right of the file name opens a file dialog for selecting the directory and file name. However, the backup is not saved until you click on "Save" in the dialog on the right.

## 2.2 Connect hardware modules

Because some settings, such as If the assignment of the inputs / outputs depends on the connected hardware, it is recommended to connect the hardware modules first. Open the hardware dialog in the menu „*Configuration -> Hardware*“. If all modules of the machine are already listed in the lower left corner "Connected hardware (used)" (see picture below), the modules are already connected and you do not need to do anything anymore. If not, at least one module should be listed in the upper left corner of "detected hardware (unused)". If nothing is displayed here, make sure the controller is plugged in and the power is turned on, and then click "Scan Network" if necessary.

## 2.3 Switch off the demo mode

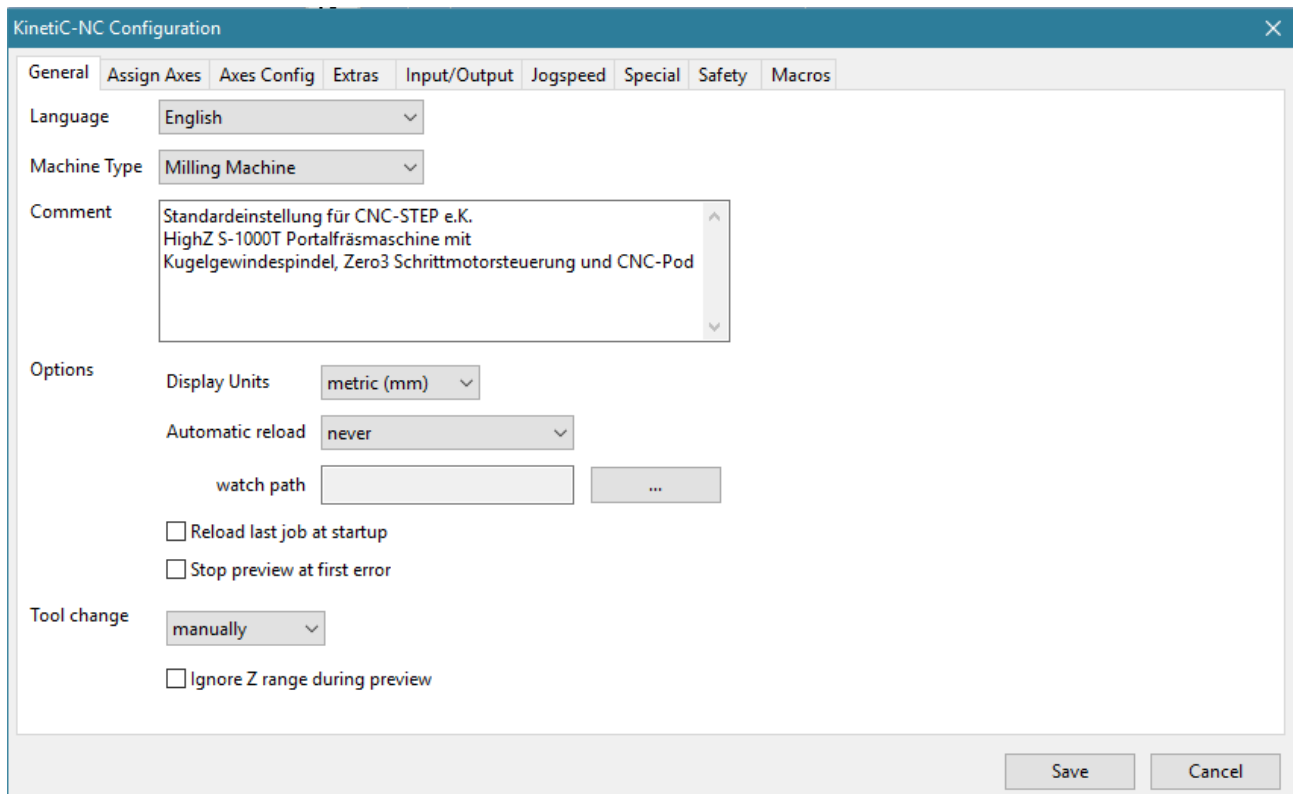


Most controllers consist of only one module. In this case you only need to click on the module and connect it to the button on the right. The demo mode is then automatically turned off, and after "saving" you are done.

## 2.4 Identify module

If the machine control includes several modules, or several machines are connected in the network, then several modules may appear in the list. To find the right module, you can click on it in the list and then activate "Identification". The selected module responds by quickly flashing the Traffic LED (yellow) on the network jack. For all other modules, only the green connection LED lights up permanently. Identify all modules of the controller in this way. Then connect the main module (NetBob or CncPOD) first, then the IO module (s) (NetIoExt). If necessary, take care not to "hijack" modules from other machines.

## 2.5 Possibly Update necessary



As the software evolves, you may be prompted to update the firmware when you connect the hardware. This means that the software, which is stored in the processor of the hardware module, must be adapted to the state of the PC software. This will happen automatically, just follow the instructions on the screen after selecting the module and pressing the Update button (top right picture). The firmware update dialog appears and you can select a file with the "..." button on the right. Choose the one with the highest version number that matches the hardware module. After transferring the firmware file, it is important to wait until the alternately blinking of the LEDs on the network connector of the hardware module stops. Only then restart the module by briefly switching the power supply off and on again.

## 2.6 Select machine type

The menu item "Configuration -> Machine" opens the dialog for the machine parameters. This is divided into several pages. The first page is used to select the type of machine and some general information about the machine. The **comment text** is unimportant for the function, but facilitates the later assignment of the files to the machine, and helps to avoid confusion, for example, if you operate multiple machines, or contact the support.

The **language** for the user interface of the software is normally selected after installation at first startup. If you want to change these later, please note that the change only becomes effective for texts of newly opened windows. In order for all messages and labels to appear in the new language, the software must be closed and restarted.

Use the **Display Unit** option to specify whether the coordinate display should be in millimeters (metric) or inches. This only affects the screen display. The unit in the NC programs is selected in the program text with the G20 or G21 command and may differ from the display unit. The machine parameters (resolution, speed and acceleration of the final drives, etc.) are always specified metric, regardless of the unit chosen for the display.

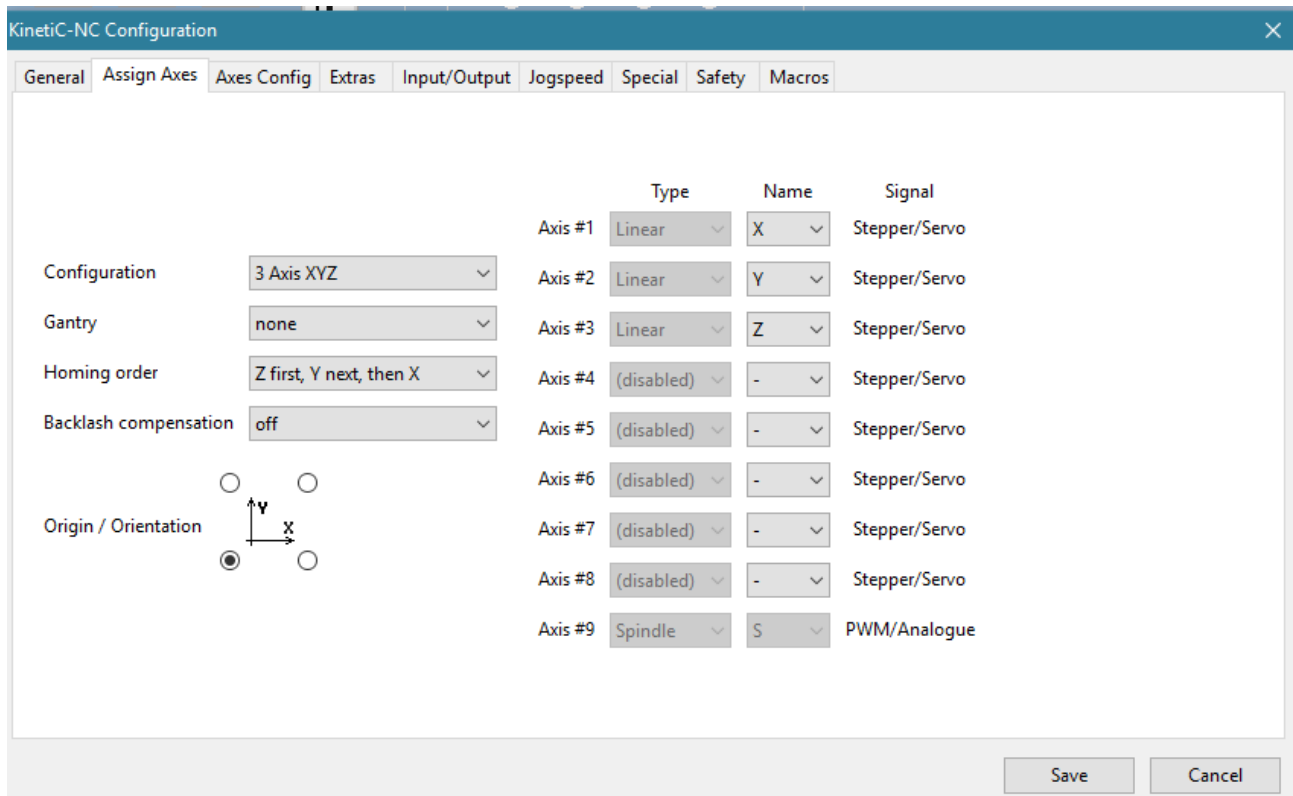
If you have activated "**automatic loading**", the system constantly checks whether the last loaded NC program file has changed or whether a new file has been saved in the **monitored directory**. The file is then automatically loaded either immediately or on demand. This can be useful when making changes in the CAM software. Immediately after the NC program has been saved, you will switch to the KinetiC NC software and save manual switching and loading.

If the option "**Load last program at startup**" is activated, the program loaded before the last exit is automatically reloaded the next time the software is started (if the file still exists). This is useful if the same series parts are often produced for several days, or if the work is to be continued the next day.

The option "**Pause preview on first error**" determines whether the remainder of the path preview should nevertheless be displayed in the event of a non-serious error. If the option is not set, the preview is not interrupted, but instead, for example, lines that are outside the permitted range are only marked red as a warning.

The setting of the **tool change** (manual or automatic) determines which macro is executed when changing (see chapter "Macro programming"). In the case of direct change systems, if the tool is inserted manually or automatically directly into a collet chuck in the spindle shaft and without tool draw (taper), the actual tool length is known only after the length measurement. Therefore, it can occur with tools of very different lengths that error messages appear during the preview (simulation) that turn out to be unfounded in the later execution of the "real" program with measurement. The "**Ignore Z Height During Preview**" option suppresses such error messages.

### 3 Axis definition



The second page "Assign axes" defines the number of axis drives and their function. It may be. not all possible combinations supported. It is best to select one of the predefined configurations in the upper left corner. The "Configuration - Special" option should only be used after consultation with *Benezan-Electronics*.

The type of an axis (linear or rotary) also defines its unit (mm or °) at the same time. The axes are assigned in order to the hardware connectors as they appear in the list on the right. By selecting the location of the coordinate origin at the bottom left you indicate how the machine is set up, i. how the location of the X and Y axes is aligned relative to the operator's point of view. This only affects the assignment of the jog buttons (manual driving) on the "Setup" page of the main window.

#### 3.1 Rotational axes

The Beamicon2 software supports the display of rotations in the graphics window if the axes of rotation (A or B axis) rotate the workpiece. For this to work properly, the position of the rotary axis in the machine room must be entered on the „*Special*“ . If the rotation axis does not rotate the workpiece but is used, for example, only as an auxiliary axis for a tool changer (revolver magazine), or if the machine has a swivelling spindle head, this function must be deactivated by selecting the option "No workpiece rotation axis" for "Special drum engraving" is selected.

### **3.2 Double axes (gantry)**

A double axis refers to a system in which two separate motors drive a common bridge gantry (gantry). The motors perform exactly the same movements during normal work. Homing is compensated for by using two separate reference switches, if any, minimum offset. The output stages or controllers for the motors are therefore connected to two separate signal outputs of the controller. The sequence of homing for gantry drives is described in the section „Gantry reference travel“ below.

If this compensation is not required because the motors are mechanically rigidly coupled anyway, then the gantry option does not have to be activated either. In this case, a single signal output (step / direction) is sufficient for the output stages of both motors and a single reference switch.

Although dual axes have two signal outputs for the power amplifiers or servo drives and two signal inputs for the reference switches, all parameters such as resolution, travel distances and speeds must be the same.

### **3.3 Homing sequence**

Here the order of the axes can be selected during homing. In the case of milling machines, Z must always be referenced first so that no collision with the workpiece can occur during the subsequent reference travel of X and Y. For a large machine, taking advantage of the homing of X and Y at the same time can be a time saver.

### **3.4 Backlash compensation**

The backlash compensation can be switched on or off simultaneously for all axes. With the selection "on, Z always up", the Z-axis is briefly reversed after a ride down, so that the movable part always sits on the lower "stop" of the play area. Thus, a subsequent "sagging" is prevented. The detailed procedure for setting the backlash will be described in a later Chapter .

## 4 Axis parameters

The actual parameters of the final drives can be entered on the next page. For better clarity, not all parameters of all axes are visible at once.

The axis must first be selected in the top left corner (see picture), then the associated parameters are displayed

### 4.1 The resolution

The resolution is the number of motor steps per unit travelled path of the axis, ie steps / mm for linear or steps / ° for axes of rotation. The resolution of linear axes can be calculated from the formula below:

$$\text{Resolution} = \text{full steps / revolution} * \text{Micro step factor} * \text{Gear factor} / \text{spindle pitch}$$

**Example:**

A stepper motor has 200 full steps / revolution. The stepper motor output stage is set to 8x microstep. A toothed belt transmission is used 1: 2, and the spindle pitch is 5mm. Then the resulting  $\text{Resolution} = 200 \text{ steps} * 8 * 2 / 5\text{mm} = 640 \text{ steps} / \text{mm}$ .

If the result is a periodic decimal fraction, it is recommended to enter at least six and no more than fifteen decimal places. The remaining rounding error is negligible and does not add up.

### 4.1.1 Belt or rack drive

For belt or rack drives, replace the spindle pitch with the effective circumference of the drive gear. Example belt drive with  $Z = 20$  and  $T = 5$ : circumference =  $Z * T = 100\text{mm}$ .

Example:

Rack with  $Z = 20$ ,  $M = 2$ : effective pitch circle diameter  $D = Z * M$ , effective circumference =  $D * \pi = 20 * 2\text{mm} * 3.14159 = 125.6637\text{mm}$ .

### 4.1.2 Rotational axes

For rotary axes, a similar formula applies:

*Resolution = full steps / revolution \* Micro step factor \* Gear factor / 360°*

### 4.1.3 Servo drives

For servo drives, the full-step resolution is replaced by the number of pulses of the encoder and the micro-step factor is replaced by the encoder signal evaluation factor of the controller.

**Example:**

The encoder has a pulse number of 1000 / revolution. With 4-fold evaluation, this results in 4000 steps / revolution. If a harmonic drive with ratio 1:50 is used on the rotary axis, the result is a resolution of  $4000 \text{ steps} * 50/360^\circ = 555.555555 \text{ steps}/^\circ$ .

### 4.1.4 Calculator

To simplify the calculation of the drive data, you can open an extra calculator window with the button on the top right (*see chapter Calculator for Axis Parameter*).

## 4.2 Max. Speed

The max speed is the highest permissible speed of the axis in rapid traverse. Please note that the unit is expressed either in units / second or units / minute. This can be selected with the selection box in the top center. In physical calculations such as ramp times and braking distances, for example, mm / s is simpler because no conversion factors are required when calculating with SI basic units. In NC programs, however, mm / min are common.

### 4.3 Working acceleration

Work acceleration is the maximum allowable acceleration and deceleration of the axis in normal operation, manual or automatic. The duration for the acceleration and deceleration ramp is calculated as speed / acceleration.

Example:

To accelerate to a rapid traverse speed of 100mm / s = 6m / min, a ramp time of 100mm / s / 500mm / s<sup>2</sup> = 0.2s is needed. The braking distance is 1/2 \* speed<sup>2</sup> / acceleration. At the above values this would be 1/2 \* (100mm / s)<sup>2</sup> / (500mm / s<sup>2</sup>) = 10mm.

### 4.4 Max. jerk

The max. Jerk is important for the web preview in Exact-Stop mode. This is the speed that the drive can accelerate or decelerate without ramping without excessive impact or step losses. This is explained in more detail in the chapter. "*Exact-Stop-Modus*".

### 4.5 Activate display

Determines whether the axis is included in the display of coordinates in the main window.

### 4.6 Activate handwheel and hotkeys

Defines whether the axis may be moved manually (with hotkeys or remote control). Disabling this option prevents inadvertent misalignment of axes that are not relevant to primary tool movement (e.g., tool changer magazine).

### 4.7 Direction inverted

With the option "Direction inverted", the direction of movement of the axis (the sign) is reversed. If the axis is moving in the wrong direction during the test drive, change this setting. For gantry axes, the direction of both drives (master and slave) can be set separately, which is required in a mirror-image arrangement (for example, in rack-and-pinion drive with right / left gear).

### 4.8 Backlash

The topic of backlash compensation will be explained in an extra Chapter below.

## 4.9 Min. and max. Position

Specify the permissible travel range of the axis. This is the path (or angle) that can be safely driven without reaching a stop or limit switch. In contrast to other control software, the travel limits are regarded as an elementary property of the axes. If the so-called "soft limits" are not explicitly deactivated, and a reference run is always triggered before the actual operation, then there is actually no danger of driving on a stop.

Homing and all related parameters are described in the following chapter.

## 4.10 Homing

Reference travel is called the first position determination. Since only relative position information is used for the positioning with step and direction signals, the absolute position of the axes after switching on is initially unknown. The controller therefore moves all axes one after the other to a reference point that is defined by the position of a reference switch.

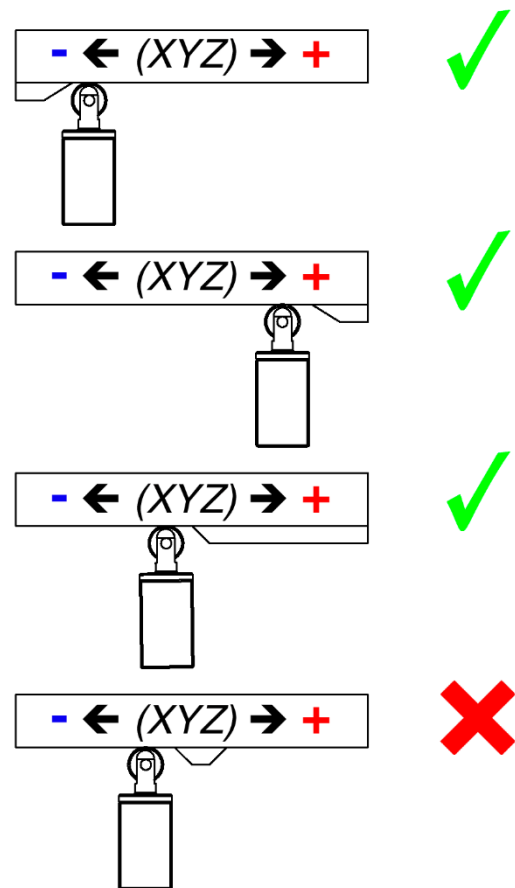
The homing procedure is as follows:

If the reference switch is already actuated in the initial state, the vehicle is first moved away from the switch until it is no longer actuated.

It is driven with the set homing speed in the direction of the switch until it is actuated.

It is driven away from the switch at an even slower speed (crawl speed) until it is no longer actuated.

The switch-off point marks the actual reference position.



### 4.10.1 The switch

The switch (or the switch's actuator) may optionally be placed at the positive or negative end of the axis. This should be chosen so that the machine is in a comfortable position for the operator after the reference run, and no collision is to be feared. In particular, the reference position of the Z axis should always be in the upper position. If "Reference switch at max. Position "is selected," towards the switch "in the direction of positive coordinates and" off the switch "in the direction of negative coordinates. This corresponds to the highest case in the picture (carriage moves, switch is fixed). With "Reference switch at min. Position "is the other way around (second case in the picture). With "Reference switch in the middle" (third case) it behaves like "at min position". In contrast to the previous two settings, the switching point can be at any position within the travel limits. This position can be specified with "Reference switch position". In the upper two cases (min / max), the switching point is at the ends or outside the travel range.

### 4.10.2 With the setting "in the middle"

It is important that the switch remains depressed in the negative range beyond the switching point over the entire distance (long actuation ramp, case 3). If this is not the case (bottom case in the picture), then there are problems if the axis is behind the switch before the reference travel, ie the switching cam in the picture is located to the left of the switch. In this case, the controller would assume that the position is in the positive range of the switching point and would erroneously move further in the negative direction, ie from the switching point. The switching point would never be reached and the axis would end position point.

An exception to this rule are rotary axes with an unlimited travel range  $> 360^\circ$ . Here is the direction, with which the switch is driven, initially does not matter, because after one revolution at the latest this is achieved anyway. If there is no reference travel and no reference switch for the axis, select "Reference switch: none". This may be useful, for example, for rotary axes for pipe machining where the starting position does not matter.

For proper operation it is important that the homing speed is not set too fast. The braking distance depends on the selected speed and acceleration and may not be greater than the distance of the switch, i.e. less than the distance from the switching point to the stop. It is therefore advantageous if it is not driven perpendicular to the switch, but the switch is actuated laterally with a bevelled cam, as shown in the picture above. For inductive or optical sensors (photocell), the actuator can of course be rectangular.

The maximum possible speed can be calculated from the available braking distance as follows:  
Speed =  $\sqrt{2 * \text{braking distance} * \text{acceleration}}$ . Example: With an acceleration of 500mm / s<sup>2</sup>

and a max. Braking distance of 1mm is the max. Homing speed equal to  $\sqrt{2 * 1\text{mm} * 500\text{mm} / \text{s}^2} = 31,6\text{mm} / \text{s}$ .

In order to achieve a high accuracy in the homing, not the switching point is used when driving on the switch for the actual position determination, but the switching point in the subsequent driving away from the switch. The speed should once again be significantly less than the reference travel speed - a maximum of 1000 steps / second, or if the switch is sensitive to shocks even smaller. If the check mark "crawl speed" is activated, this reduced speed can be entered here. If not, one tenth of the reference travel speed is taken.

### 4.10.3 Reference switch tolerance

indicates the path from the stop to the switching point of the switch. With reference switch offset, the path is specified, which is cleared after referencing. This is a kind of "safety margin". With  $\text{offset} > 0$ , the switching point is then completely outside the travel range.

### 4.10.4 Reference travel on gantry axes

Axes with double drive are a special case of homing. Such axes have two motors and two reference switches, one on each side. The two separate switches are used to detect and compensate for a possible offset (angle not exactly right-angled). During homing, both drives always run simultaneously and in parallel. This is the only way to ensure that the two switching points of the switches can be determined independently and without undue delay of the portal.

As in the case of simple homing, the switches are first moved in the direction of the switches until both switches are actuated. Then it is driven creeping away from the switches until both are free. Only then is the determined path difference between the two switching points offset by only one of the two drives is moved.

The additional travel path, until both switches are pressed, must be considered, i.e. there must be sufficient distance reserve behind the switch, for the braking distance plus the maximum possible offset.

With **reference offset**, the path is set which drives the axis additionally away from the switch after the actual homing run. This can either be used to maintain a certain safety margin, or specifically for gantry axes, to deliberately set the offset differently on both drives (master and slave) to compensate for mechanical misalignment.

## 4.11 Backlash compensation

It is expressly pointed out at the outset that the backlash compensation is only a compromise solution, and you should not set the expectations too high. You can not turn a "wobbly mechanic" into a precision machine. The function of software backlash compensation presupposes that the actual position of the mechanism is always behind the target position specified by the drive. This works normally well for slow driving and heavy-duty mechanics (plain bearings, dovetail guides). In light and smooth-running machines, and especially in synchronous milling, however, it can happen that the tool "pulls" into the workpiece in the direction of movement, thus leading the way in the direction of travel. In addition, since the reaction rate of lash equalization is finite (and must be limited to avoid step losses), short-term path deviations and additional chatter marks may result from transients. A small path deviation, especially in the reversal points in arcs, is therefore inevitable.

If the existing play is only a few micrometers to a few hundredths of a millimeter and can be tolerated for the required accuracy, it is recommended that you work without software clearance compensation because the dynamics and surface quality are better. Improvements in mechanics, meaning, a change from trapezoidal (possibly biased) ball screws or the replacement of gears by lower-back models is indeed more expensive, but brings much better results than the backlash compensation by software.

### 4.11.1 Determining the play:

This procedure must be performed at least once for each axis. First turn off the backlash compensation and set the jog step size to a level that is definitely above the largest expected play, for example 1mm. Mount a dial gauge on the axis. Move at least one step (1mm) to the right or in the direction of larger coordinates. Now reset both the coordinate display and the dial gauge to zero. Now take another step to the right, then one to the left. The coordinate display must now show zero, the dial gauge will display a non-zero value if there is clearance. Write down the value and carry out the measurement at several positions of the travel path, for example once at the beginning, once in the middle and once at the end. Strongly deviating values indicate wear or misalignment of the mechanics, which must be remedied.

#### **4.11.2 Enable backlash compensation:**

enter the average value of the calculated play values for all axes (page „*Axis parameters*“ of the machine configuration). This is only possible if the backlash compensation is switched on on the "General" page, otherwise the fields on the "axis parameters" page are greyed out. After saving, it is best to run a test program that cycles around. If strong blows occur in the reversal points of the circle, reduce the working acceleration of the axis where the shocks occur.

## 5 Calculator for Axis parameters

The calculator is called up via Menu-> Configuration-> Machine Axis Parameters tab. There is a button on the top right "Open calculator"

The calculator for the axis parameters supports the user in the setting and also provides information about the constructive properties of the machine, for example, to select the ball screws or the size of the motors. One can thus determine in advance whether the envisaged motors are suitable for the desired speeds, or whether the bearing of the ball screw is sufficient.

### 5.1 Select motor type:

Here it is selected whether it is a servo motor or a stepper motor. The selection of the motor type changes the further options.

### 5.2 Select gear type:

If a gearbox is used between the motor and the drive, the tick must be set. Then the gearbox type can be selected from the following 4 possibilities: toothed belt, helical gearbox, gearbox in oil bath or worm gear. In the corresponding fields "number of teeth pinion" comes the number of teeth of the gear on the motor shaft, in the field "teeth gear" the number of teeth of the driven gear. The transmission will be calculated later by the program.

### 5.3 In the field "Motor select"

a motor can be selected from a list. If you are not using a predefined motor, you can define your own motor under "Own values".

### 5.4 In the field micro steps

Enter the number of microsteps per full step. Usually this value is also called "multiplier", or the number is directly on the power amplifier. These are usually values between 4 and 64.

### 5.5 In the middle field select the drive type.

Currently supported are ball screw, trapezoidal thread spindle and toothed belt (or toothed rack). Depending on the choice of drive type, the input fields change. If a spindle drive has been selected, the bearing can now be defined.

The storage type means:

- fixed: a preloaded bearing, for example, tapered roller bearings, shoulder bearings, preloaded thrust bearing,
- loose: a deep groove ball bearing or needle roller bearing,
- none: the spindle end is free or insufficiently supported

The first value is the bearing on the drive side, the second value is the bearing at the non-driven end  
**In case of a driven nut this calculation does not apply!**

### 5.6 Now in the field "spindle"

a predefined spindle can be selected or, if the data is known, one can define a separate spindle under "own values" (top entry in the list). The length in mm is specified in the "Length" field.

In the case of the "toothed belt" drive type, only the number of teeth of the toothed wheel and the pitch of the toothed belt are possible. The pitch of the toothed belt is the distance between two teeth.

For the rough calculation of accelerations, the mass of all parts to be moved on the axis is added in the field "Weight of all moving parts". In the case of the Z-axis, this would be in one example: ball nut, linear carriage, carrier plate, milling spindle with tool, and other parts attached thereto. The supply voltage of the motors is important for the maximum achievable speed.

The actual calculation is triggered by pressing the "Calculate" button. Then the results appear under "Results" on the right. If a parameter necessary for the calculation is wrong or not specified, an error message appears. The value is highlighted in red and must first be corrected before the calculation can be performed.

## 5.7 Explanation of the results

**Steps/mm** is the resolution of the axis that should be entered in the axis parameters.

**The critical speed** is the speed at which a lead screw begins to vibrate and buckle. By changing the diameter and the storage this value can be increased.

**Corner frequency stepper motor:** This is the frequency above which the motor begins to lose torque. This frequency should not be exceeded under load. In rapid traverse without tools engaged the speed can be quite higher.

It only needs to be considered that the reduced torque for friction and acceleration forces (plus safety margin) is sufficient. You can increase the corner frequency by selecting a different motor or increasing the supply voltage.

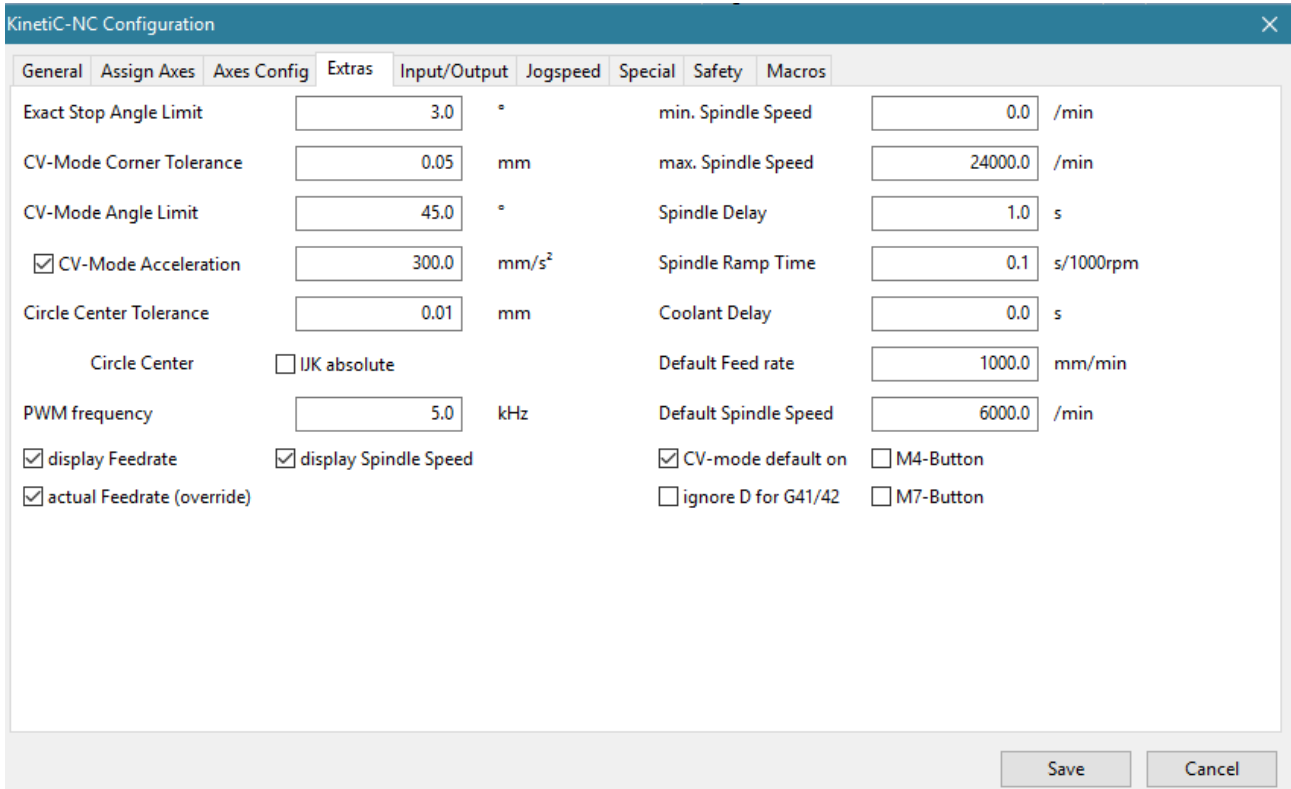
**The pulse rate** is the step rate at which the motor is driven. Make sure that the hardware used (motor power amplifiers) supports this frequency.

The fields underneath show some **values or assessing the drive**.

The calculated values are NOT a guarantee that they will actually be achieved in practice but are merely indicators of the design of the machine. The actual achievable acceleration / speed values are highly dependent on the stability of the machine and must be checked by the machine designer. The displayed values are only the purely mathematically maximum possible limit values on the basis of the specified parameters and not an assurance of properties.

The **Notes field** evaluates the parameters for their results and provides advice on how to improve the dynamics of the machine.

## 6 Other parameters



The screenshot shows the 'Extras' tab of the 'KinetiC-NC Configuration' dialog. The settings are as follows:

Parameter	Value	Unit	Parameter	Value	Unit
Exact Stop Angle Limit	3.0	°	min. Spindle Speed	0.0	/min
CV-Mode Corner Tolerance	0.05	mm	max. Spindle Speed	24000.0	/min
CV-Mode Angle Limit	45.0	°	Spindle Delay	1.0	s
<input checked="" type="checkbox"/> CV-Mode Acceleration	300.0	mm/s <sup>2</sup>	Spindle Ramp Time	0.1	s/1000rpm
Circle Center Tolerance	0.01	mm	Coolant Delay	0.0	s
Circle Center	<input type="checkbox"/> IJK absolute		Default Feed rate	1000.0	mm/min
PWM frequency	5.0	kHz	Default Spindle Speed	6000.0	/min
<input checked="" type="checkbox"/> display Feedrate			<input checked="" type="checkbox"/> CV-mode default on		
<input checked="" type="checkbox"/> actual Feedrate (override)			<input type="checkbox"/> ignore D for G41/42		
			<input type="checkbox"/> M4-Button		
			<input type="checkbox"/> M7-Button		

Buttons: Save, Cancel

The “Extras” page contains settings that cannot be assigned to a particular axis. On the top left are the parameters for the *Exact-Stop*- or *Constant-Velocity-Mode* (CV-mode). In the Exact-Stop mode, the brakes are applied when the angle between two consecutive line segments is greater than the limit angle specified above or the maximum jerk (see above) of at least one axis is exceeded, which is speed-dependent.

### 6.1 CV-Mode

For the CV mode, a maximum chord error (path deviation perpendicular to the direction of movement) and an extra critical angle can be specified. The maximum acceleration transverse to the direction of movement can be limited if the corresponding tick is set. All these parameters have an effect on the path preview, which will be dealt with separately in the later section „*Path preview*“ and „Constant-Velocity-Mode“.

## 6.2 Circle center

The circle center point can be given either relatively or absolutely with the I, J, and K words for G2 and G3 commands. It may happen that due to rounding errors, the distance of the center to the sheet start and end point is not exactly the same. This is automatically corrected by the software if the maximum tolerance for the circle center is not exceeded.

## 6.3 PWM frequency

The PWM frequency can be set between 0.1 and 5kHz. This only makes sense in laser or GranitoGrav engraving. For the control of frequency converters with 0 ... 10V analog signal during milling, the default setting of 5 kHz is best used.

## 6.4 Display of options

Depending on whether the checkmarks are set on the lower left, the feedrate and spindle speed are displayed in the main window or not. For laser cutting or engraving machines, the S value corresponds to the laser power instead of the spindle speed. "Feed with override" determines whether the current override factor is taken into account when displaying the F value (actual display) or not (setpoint display).

## 6.5 Spindle speed

Min. And max. Spindle speed specifies the permissible limits for the S value in CNC programs. In addition, the range for the PWM output or the 0..10V analog output for the spindle speed is specified. A more precise assignment of the PWM output values to the S values can be made in the menu item „*Configuration -> Calibration -> PWM output*“ .

The spindle delay and spindle ramp time determine the waiting time which is waited after the spindle is switched on (M3 or M4) before the next travel command is executed. The waiting time is the sum of the delay and the ramp time multiplied by the speed divided by 1000 / min.

### **Example:**

When accelerating to a speed of 10000 / min, and the above settings (1.0s, 0.1s) hold, the waiting time is  $1.0s + 10 * 0.1s = 2s$ .

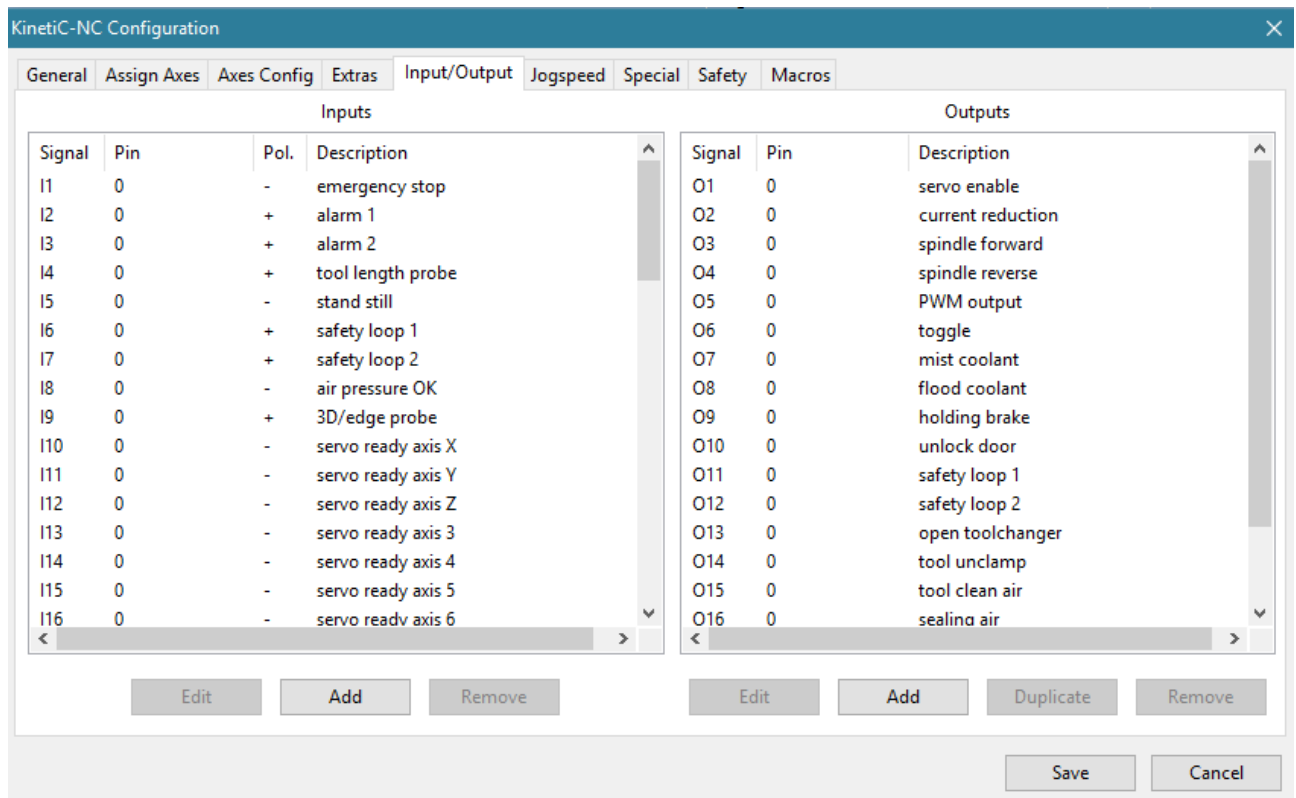
The coolant delay determines the waiting time after switching on the cooling (M7 or M8). The settings for the default values for the preset spindle speed and traversing speed follow. These are used if no values are specified in the program. If no explicit mode with G61 or G64 is selected in the program, then what is selected in "CV-Mode preset" applies.

With M4-Button and M7-Button it can be determined, whether an extra button for spindle forward and backward rotation (M3, M4) as well as an extra button for spray cooling and flood cooling (M7, M8) should be available, or only one button at a time.

## **6.6 D-Parameter**

Some post processors issue a D parameter for G41 / G42 commands indicating the tool memory number. The KinetiC-NC software interprets the D value as a diameter specification. This can lead to incorrect correction values for the cutter radius. Therefore you can optionally ignore D in G41 / 42.

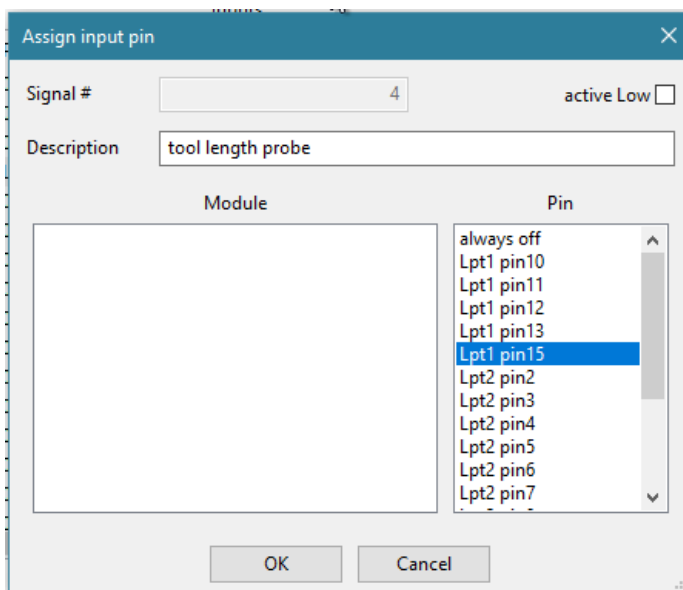
## 7 In- and outputs



The KinetiC-NC-CNC control allows the use of numerous inputs and outputs, which can be freely assigned to the connections of the hardware with a few exceptions. If required, the number of available inputs and outputs can be extended almost indefinitely by adding additional IO modules. There are predefined inputs and outputs for frequently required functions such as reference switch and the tool spindle motor. In addition, user-defined inputs and outputs can be created.

In the left column of the list, the (logical) signal number is displayed, for e.g. "I4" for the tool probe input or "O8" for the coolant output. Predefined signals have numbers less than 100, user defined values greater than or equal to 100. The signal number remains the same regardless of the pin assignment and is relevant for use within the G code or macros.

## 7.1 The assignment



The second column shows the assigned hardware pin or a zero, if no pin is assigned, followed by the polarity in the third column (+ = high-active, - = low-active or inverted). After that is the name of the signal is ready. If there are several hardware modules, there is another column with the module number. With "Change" the assignment to a hardware pin can be changed. "Add" creates a new, user-defined signal. "Remove" completely clears a user-defined signal or removes the pin assignment of a predefined signal. Predefined signals can not be deleted completely.

## 7.2 Inversion of inputs

Inputs can be inverted by clicking on the check mark "low-active". Inputs to which no pin is assigned are treated as if the input were always low. A high-active input is thus always switched off (0), whereas an inverted (low-active) input is always switched on. Unused error inputs (alarm) should always be switched off (high active), unused ready inputs always switched on (low active) so that no malfunction is signalled during normal operation. It is allowed to assign several inputs to the same hardware pin. This is useful if, for example, a reference switch is a limit switch at the same time or only one single signal input for "Servo Ready" of all axes is available.

## 7.3 Inversion of outputs

Outputs can not be inverted, but are always high-active. This must be the case because all outputs are switched off (low, de-energized) when the controller is switched on or in the event of communication failure or power failure. Inverted outputs could cause undefined levels, and devices could be turned on inadvertently, which could cause dangerous situations. It is possible to assign a plurality of output pins to the hardware to a logic signal. The pins are then all connected in parallel. For this purpose, the output signal to which several pins are to be assigned must first be selected, then "Duplicate" must be selected instead of "Change". "Add", on the other hand, generates a new signal and no parallel connection.

## 7.4 Emergency stop

The emergency stop input (I1) and the alarm inputs (I2...I3) play a special role. The emergency stop input is permanently connected to a hardware pin (e.g., LPT1 pin 11 on the CncPod). The assignment and also the polarity cannot be changed for security reasons. If you require a different assignment, you must request a special version from the manufacturer of the software. The alarm inputs, on the other hand, can be assigned to any free input pins. This can be used, for example, to ensure that the machine stops when a frequency converter reports a fault.

## 7.5 Alarm signals

If the signal name of the alarm input is changed, for example to "Frequency inverter fault", an understandable text and not just "Alarm 1" is displayed as a message in case of an error.

## 7.6 Step and direction signals

Also, not (any) changeable are the assignments of the step and direction signals for the positioning drives. Therefore, these outputs also do not appear in the list of pin assignments on the "Inputs / Outputs" page. The order of the axes is set on the „Assign Axes page“ The signals cannot be applied to any pins.

## 8 Manual travel speeds

KinetiC-NC Configuration

General Assign Axes Axes Config Extras Input/Output **Jogspeed** Special Safety Macros

**Speed**

1.0 mm/min  
10.0 mm/min  
100.0 mm/min  
1000.0 mm/min  
rapid

include rapid

Add Delete

**Step size**

0.01 mm  
0.1 mm  
1.0 mm  
10.0 mm  
continuous

Add Delete

**Handwheel**

Linear fast speed 1.0 mm/min

Step size 0.01 mm

Linear slow speed 1.0 mm/min

Step size 0.01 mm

Rotary fast speed 1.0 °/min

Step size 0.01 °

Rotary slow speed 1.0 °/min

Step size 0.01 °

Position mode, allow delayed following  
 Velocity mode, ignore overturning

Reduce Acceleration 50 %

Smoothing filter 0 ms

Save Cancel

This page of the Machine Parameters dialog is used to set the speeds and increments for the manual run (Jog). The values entered are used as default on the "Setup" page of the main window. This allows the speeds to be adapted to the machine. For small machines with high-speed drives, it may be useful, for example, to exclude rapid traverse for manual driving because the risk of collision is too high. If the given values are not sufficient, additional speeds can be included in the lists with "Add".

### 8.1 Hotkeys

From the list of the total available speeds, you can also select at the bottom left which speeds should be selected for the specially designated hotkey speeds "fast" and "slow". These are only needed if hotkeys are used with additional selection keys, e.g. CTRL for fast and ALT for slow. For hotkeys without selection button, the last activated speed remains.

## **8.2 Save settings**

The option "Save jog settings on exit" indicates whether the software should remember the values for speed and increment currently set on the setup page, or whether the default value should be set again the next time the software is started.

## **8.3 Handwheel**

In the right field you can set extra speed values for the optional handwheel (not yet supported).

## 9 Special processing types

On this page are settings that are only relevant for special processing types. The individual parameters are explained in detail in the separate chapters and application notes for the processing types. Only those parameters that are relevant to the machine type selected on the "General"- page are accessible.

### 9.1 Rotational axis

If a rotation axis is present, the orientation and position of the axis can be specified. If the option "Position is determined from current zero point" is selected, the coordinates do not have to be entered as numerical values but are calculated automatically. This is easier for the user during cylindrical engraving but requires that the zero point is always placed vertically above the axis on the material surface. This is not always the case when working on dental equipment, for example, so in this case the option should be switched off and the exact position of the axis of rotation should be specified. The workpiece diameter is not only relevant to the distance of the axis to the zero point and for the representation of the graphic, but also determines the ratio of rotation angle to the travel during movements of rotary axes and for the calculation of the traversing speed during drum engraving or milling on the surface of turned parts. The workpiece diameter can also be accessed via the tool path dialog from the main page and can thus be changed later without having to call up the machine parameters.

## **9.2 Laser cutting / engraving**

“Laser Power Threshold” specifies the minimum laser power (in percent) at which power in sharp corners of the web is reduced. The power reduction can prevent burns, which can otherwise be caused by the reduced speed through the braking ramp in the corners. If no speed-dependent power reduction takes place, set the value to 100%.

For some laser units, the PWM signal is low active. This can be changed with "PWM inverted".

## **9.3 Plasma cutting**

In plasma cutting with height control (THC), the voltage divider ratio and the gain for the height control must be specified.

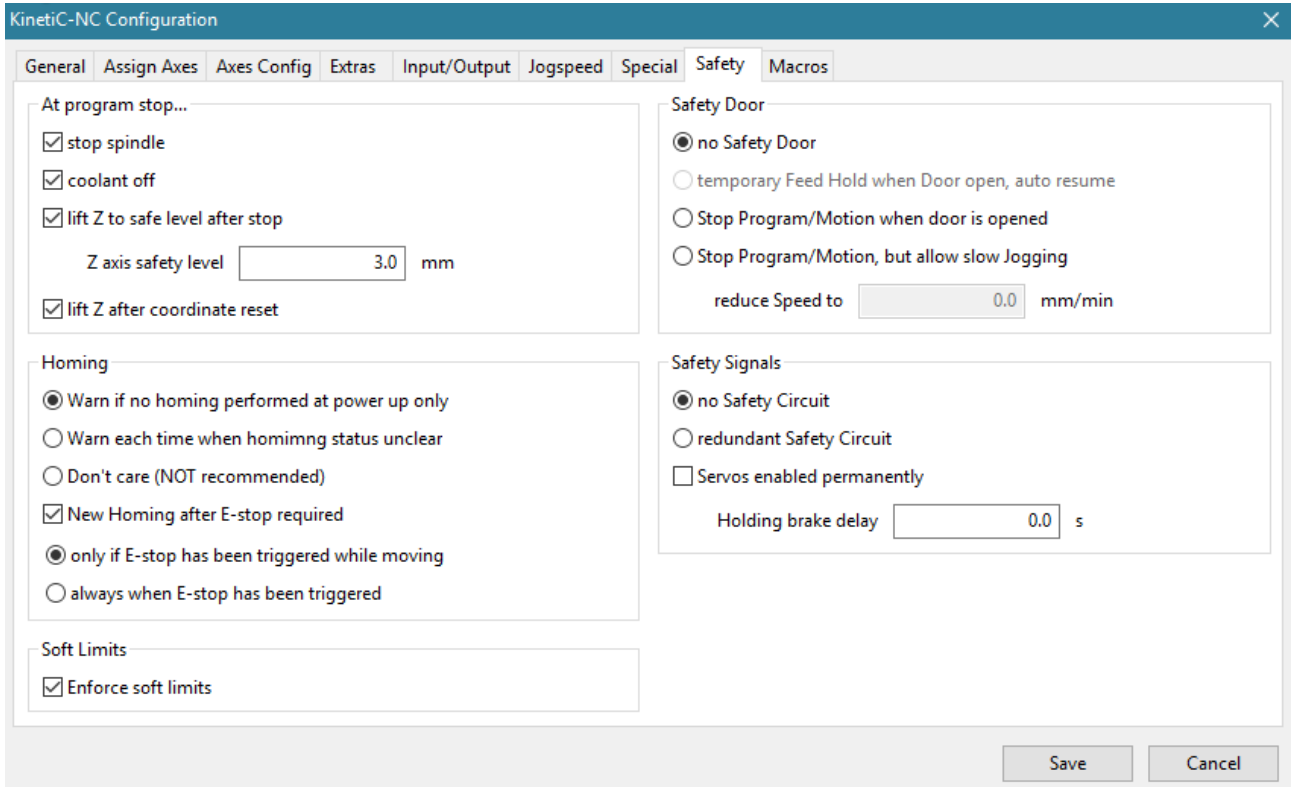
## **9.4 Tangential knife**

With Tangential knife cutting two limits for angles between successive line segments can be specified. If the angle of the travelled lane is greater than the first limit (maximum angle without stopping), then it is not driven through continuously, but first braked, the knife is turned, and then continued. If the second limit value (lifting if angle >) is exceeded, the knife is additionally lifted out before turning and then inserted again after turning.

## **9.5 Wire cutting**

For Styrofoam wire cutting, the length of the cutting wire can be entered. This is only relevant for the scaling of the graphical display.

## 10 Safety



The screenshot shows the 'KinetiC-NC Configuration' dialog box with the 'Safety' tab selected. The dialog has several sections:

- At program stop...:**
  - stop spindle
  - coolant off
  - lift Z to safe level after stop
    - Z axis safety level:  mm
  - lift Z after coordinate reset
- Homing:**
  - Warn if no homing performed at power up only
  - Warn each time when homing status unclear
  - Don't care (NOT recommended)
  - New Homing after E-stop required
    - only if E-stop has been triggered while moving
    - always when E-stop has been triggered
- Soft Limits:**
  - Enforce soft limits
- Safety Door:**
  - no Safety Door
  - temporary Feed Hold when Door open, auto resume
  - Stop Program/Motion when door is opened
  - Stop Program/Motion, but allow slow Jogging
    - reduce Speed to:  mm/min
- Safety Signals:**
  - no Safety Circuit
  - redundant Safety Circuit
  - Servos enabled permanently
    - Holding brake delay:  s

Buttons: Save, Cancel

On the page "Safety", the behavior of the controller during interruptions, emergency stop and reference travel is influenced. The top left determines how the machine should behave in the event of an interruption ("Stop program" button). "Spindle off" and "Cooling off" cause the corresponding outputs to be switched off automatically. If "Z at retraction height" is activated, then the Z-axis is additionally raised to the specified Z-height. Depending on the processing, this may be advisable, for example to avoid burns. But there are also cases where this does not make sense, for example with 5-axis machines or with T-slot milling. Z raise after coordinate reset is intended for users who are used to the behavior of WinPCNC. There, after touching ("zeroing") the workpiece surface, the Z-axis is raised automatically.

## 10.1 Homing

Bottom left, it is determined when a reference run should be performed or requested. The default setting is "Warn if no homing is performed after switching on". In order not to jeopardize the safety of the operator, it is not permissible to perform homing immediately after switching on or automatically when the software is started. The user is only warned if no homing has taken place at the time a program or manual drive is started. With "Never warn" this request can be suppressed. This is specifically discouraged because it can cause collisions with stops, tool changers and other objects if the reference travel is forgotten.

## 10.2 Behavior after emergency stop

Depending on the type of drive and the control electronics, there may be a loss of position during an emergency stop so that a new reference run is required. If "Reference travel required after emergency stop" is selected, the software warns after an emergency stop and prompts for homing before the next program or jog movement is started. Stepper motor drives with self-locking threaded spindles normally hold their position if the emergency stop switch has only been triggered at standstill. For non-self-locking drives (racks or heavy Z axes with holding brake), however, position loss may occur even if the motor current is switched off.

## 10.3 Security doors

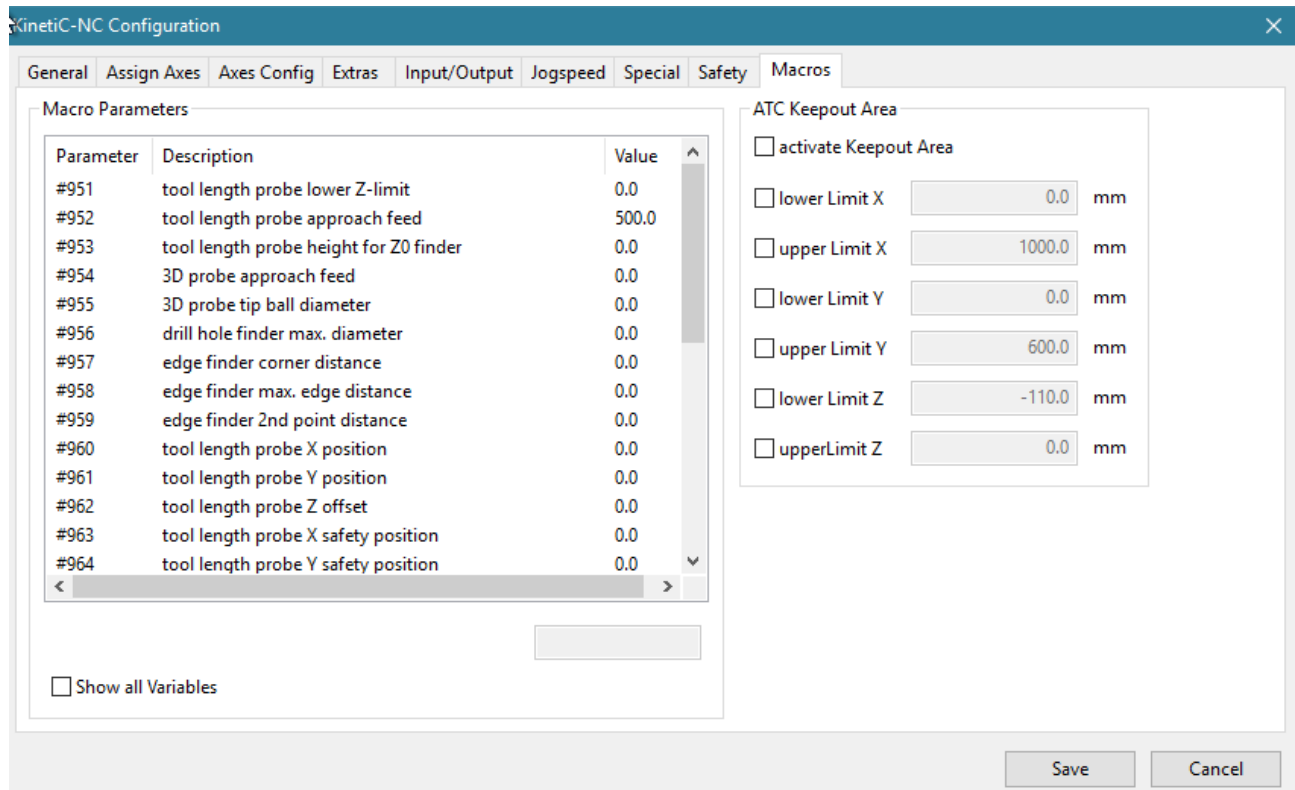
In the "Safety door" field, the behavior of the controller can be defined when one of the door contact inputs becomes active. When "stop when door is opened" is stopped immediately, in contrast to the emergency stop but with normal braking ramp, as if the "stop" button was pressed on the program side. Any further execution of programs or manual jog movements are then prevented. With "stop but allow slow jogging", however, jog movements are allowed with the door open, but are limited to the maximum speed specified below. See also: Maintenance mode.

## 10.4 Safety signals

"Redundant safety circuit" and "Servos permanently active" are special options for machines with servo drives and safety doors with redundant safety circuit. They should only be used by experts and in consultation with Benezan-Electronics.

The delay time for the holding brake indicates how long to wait after the brake is opened (switching on the output signal) until the first travel movement. This prevents unintentional blocking of the drive when the brake reacts (mechanically) with a delay. Values of approx. 0.5s are usually sufficient.

## 11 Macro-Parameter



The macros page allows you to enter parameters for the tool change and length and edge survey macros. This makes the input more user-friendly and a change of the program text is avoided in most cases. This also reduces the risk that inexperienced users will disable the macros due to typos. For details on the macro parameters, refer to the chapters on the Tool length probe and Tool change.

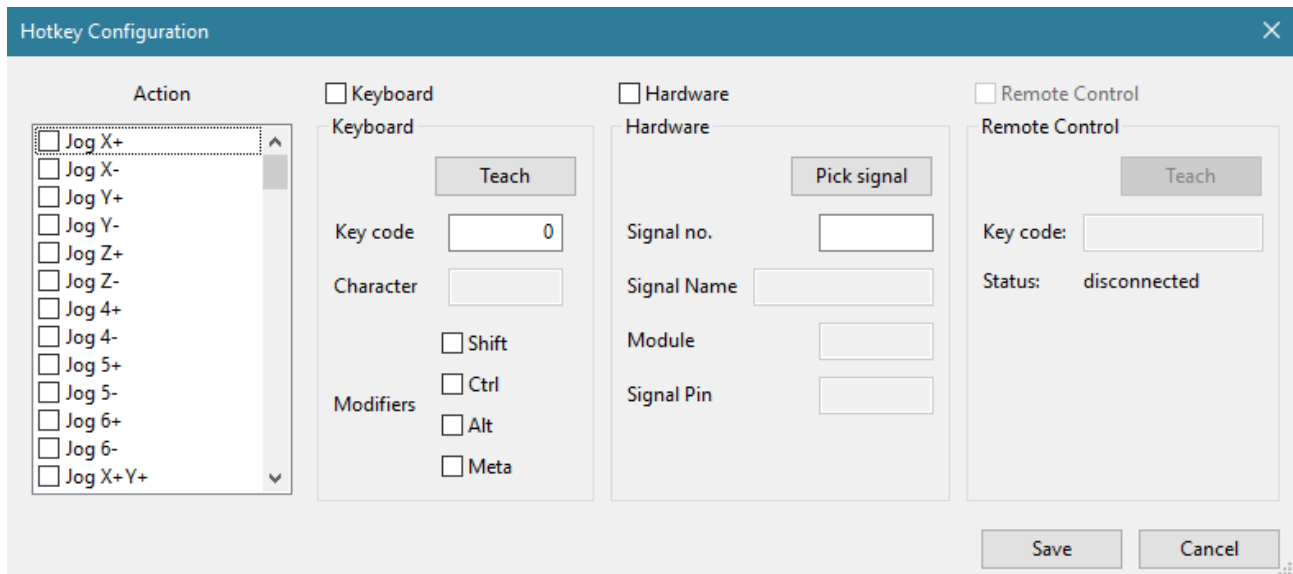
If an automatic tool changer is available, a "**prohibited area**" can be defined for it. This area then appears as a red box (wireframe) in the graphic display and can no longer be navigated with the manual movement (jog) or travel commands in the (main) program. The movement stops in this case from entering the forbidden area and a corresponding message appears. Only the tool change macro itself can travel through the protected area.

It should be understood that this cannot guarantee 100 percent protection for all conceivable cases. The monitoring only applies to the coordinates of the X, Y and Z axes, regardless of the tool length and diameter. Inclination of swivel axes (if available, in 5-axis machines) are also not taken into account.

If limits of individual axes are not selected, they are automatically set to the corresponding travel limit of the axis. This means that if, for example, "upper limit Y" is not selected, then the forbidden range extends up to the maximum value of the Y axis, in this example up to 300 mm.

## 12 Hotkeys

### 12.1 What are they?



So-called "hotkeys" are keys or key combinations with a special assignment, which trigger a specific action when pressed. The best-known application is the manual movement of the axes, which can be done with hotkeys without having to press the arrow buttons on the Setup-page with the mouse.

In the Hotkey dialog, an associated key combination can be defined for each action in the list on the left. Each jog movement direction occurs a total of three times. Once without speed, once "fast" and once "slow". At the first option without specification, the last set speed is retained. With "fast" or "slow", the speed is set to the value set on the „Speed page“.

### 12.2 Definition

To define a new hotkey, first select the action on the left, activate the "Keyboard" function at the top, then click "Learn" below it. Then press the desired key combination on the keyboard. The key code and its associated character, if it is a printable character, will be displayed underneath. You can also subsequently change the corresponding toggle keys that must be pressed simultaneously with the hotkey using the selection boxes on the bottom right. To clear the hotkey, you can turn off Keyboard, or turn off the check mark at the beginning of the line in the list.

### 12.3 Key or input?

Hotkeys must be unique, i.e. two different actions must use different key codes and toggle key combinations when they are occupied. If you receive an error message that a combination is already in use, you must either use another combination or delete the other hotkey that uses the same one first.

Hotkeys can also be set to hardware input signals instead of PC keyboard keys. This is for example useful when an action is to be triggered by a foot pedal, photocell or other sensors. For this, a corresponding input signal must first be defined in the machine configuration (Configuration → Machine → In-/Outputs) Then you can use it in hotkey configuration by switching from "keyboard" or "not busy" to "hardware input" in the top right corner and selecting the desired signal. (Of course, this does not work in demo mode)

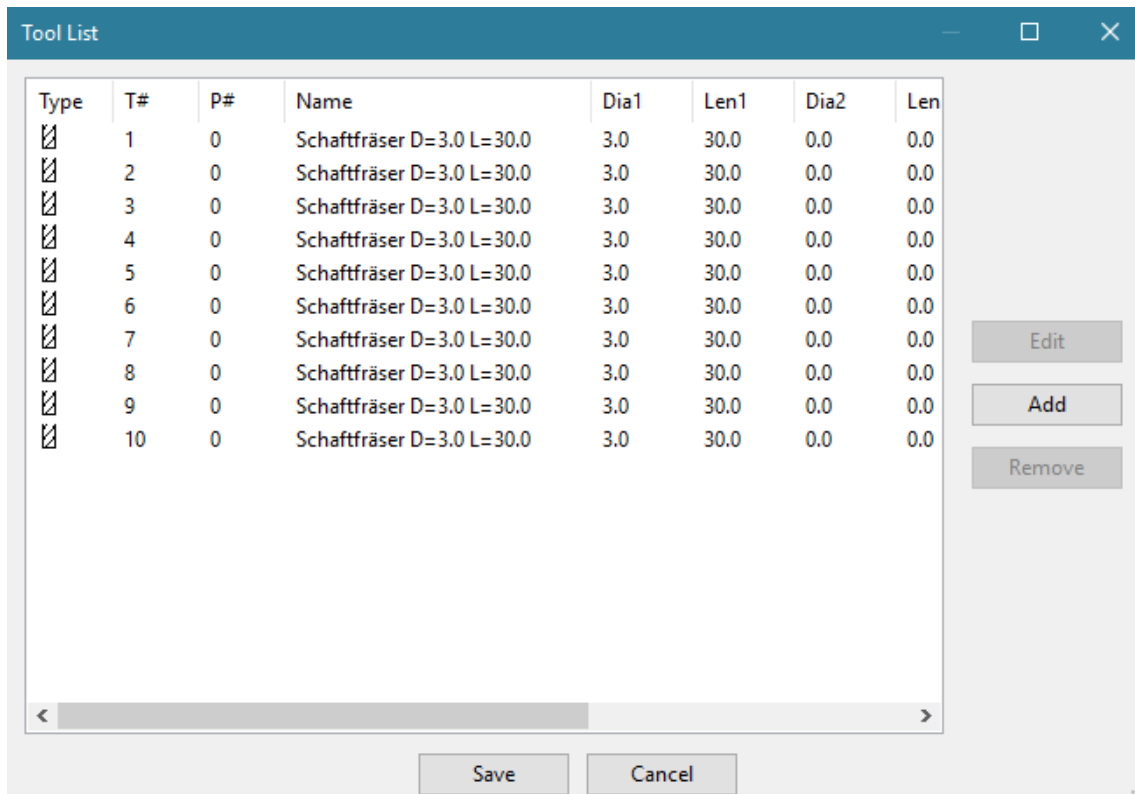
### 12.4 Tip:

You can also connect more than one keyboard to a PC. There are e.g. wireless numeric keypads ("ten-key") that work with a wireless USB dongle. If you use such a wireless keyboard in addition to the normal PC keyboard, you can use the jog function while you move freely on the machine.

The "Remote control" field on the right is only intended for special remote control devices that are not connected as a normal PC keyboard but require their own driver. This is currently not supported.

## 13 Tools

### 13.1 Do I need it?



Tool management is only required if you have an automatic tool changer or if you are working with tool radius compensation or length compensation (G41 / G42 or G43). If you load the tool manually, and only ever use one program per program, you can always work with only one tool number (T1), and you do not need to create a tool list.

### 13.2 Radius compensation and length compensation

The image above shows the tool list with the most important parameters. Type and name are for convenience only. The T number is the number used to identify the tool in the G code. The P number is the number of the place in the tool magazine. This is only needed if an automatic tool changer is used where the T number is not the same as the location number. Diameter 1 (DM 1) is the effective diameter of the main cutting edges and is also used for radius compensation (G41 / G42). Length 1 (L. 1) is the total length of the tool and is used for length compensation (G43). Depending on the tool type, a second diameter and a second length can be specified. This can be, for example, shaft diameter and cutting length. The secondary parameters are only used for the visualization and play no role in the program flow.

### 13.3 Add new tools

"Change" or "Add" opens the tool dialog (see picture below) with either the currently selected or a new tool. The type and parameters of the tool can be entered. Service life and max. Speed is currently not used. The track color is used for the 3D graphics. If different colors are chosen, the paths of the tools can be better distinguished.

**Tool Data**

**Tool Type**

- ? custom
- end mill
- drill
- ball nose
- V-tip
- T-slot
- torus
- lollipop
- countersink
- dove tail
- reamer

Name: Schafftfräser D=3.0 L=30

automatic

Number: 1

Place: 0

Diameter1: 3.0

Length1: 30.0

Diameter2: 0.0

Length2: 0.0

max. RPM: 0.0

default RPM: 0.0

default Feed: 0.0

max. Life:

Time used:

Path color:

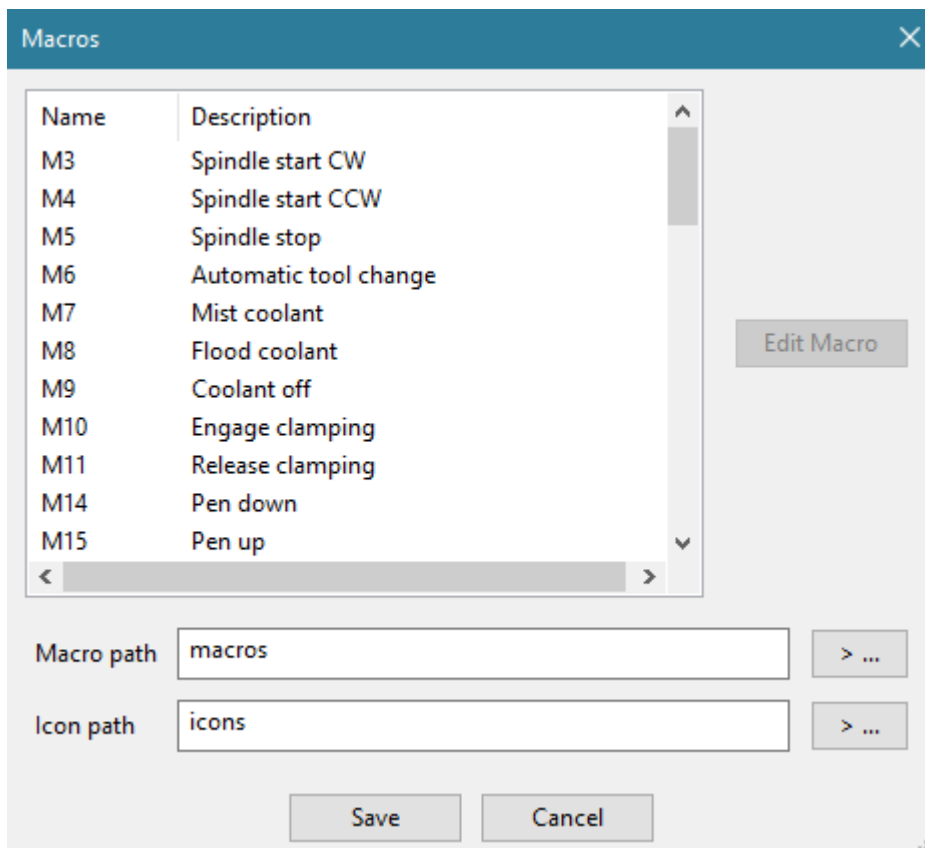
OK Cancel

### 13.4 Assign names

If "automatic" is activated, a name is automatically generated from the selected settings. You can also replace the tool name with your own input.

## 14 Macros

The controller can be flexibly adapted to different machines by programming so-called macros. Macros are small programs written in an extended G-code programming language and executed whenever special commands such as automatic tool change (M6) are executed. The macro defines the behavior of the controller for these commands. In addition, there are macros that are always executed at certain times or events, for example the startup macro when starting the control software.



If you want to change the machine or the type of processing, changing the macro directory makes it easy to toggle between two or more sets of macros without having to change all the macros one at a time. By pressing the "Change" button on the right, a text editor is started and you can edit the program text of a single macro. The macro programming language is discussed in more detail in the extra chapter „*Macro-Programming*“.

## 14.1 User macros

The so-called user macros "User01" to "User10" can be started on the "Special" page of the main window by clicking on a symbol (icon). The image that is displayed there as an icon can be replaced by an image file in the icon directory. To do this, you must copy a PNG file (for example "User01.png") into the icon directory. The path name of this directory depends on the operating system version and is displayed when you press the "> ..." button on the right. There you can also change the directory to change the complete set of symbols.

Changes to the macro program texts and the icon symbols only become effective the next time an NC program is loaded. If you want to test a change, it is best to select the current program from the "File - Recently used files" menu in the top line.

## 14.2 Program yourself?

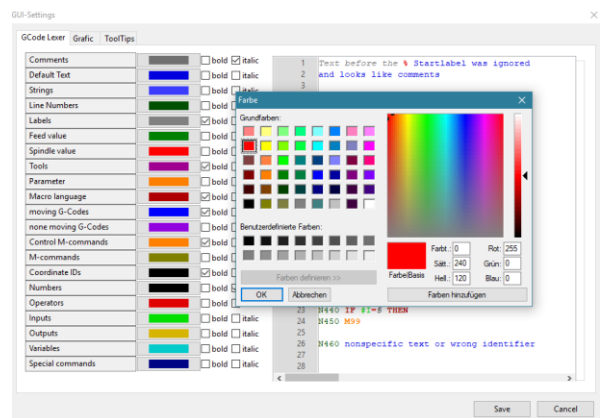
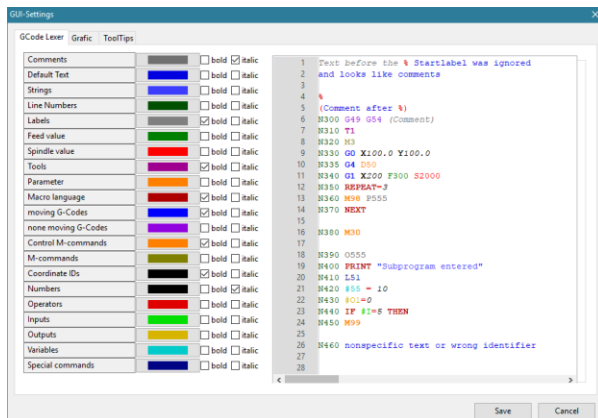
Not in all cases you have to program macros completely yourself. There are predefined macros for the most frequently needed cases, such as tool length measurement and tool change with linear magazine. You can install the macros for height measurement and tool length measurement by importing the file "HighZ\_LengthProbe.ini", which is located in the subfolder "Addons" when you click on "Standard Parameter" in the *Import-Dialog*. You can parameterize these macros via the macro parameters in the machine configuration dialog (*Configuration* → *Machine* → *Macros*) or adapt them to your own requirements by making minor changes.

## 15 GUI-Settings

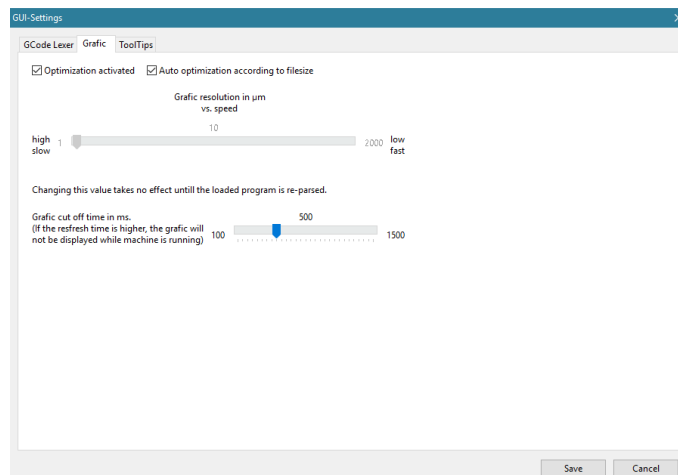
These settings affect the graphical user interface (GUI), ie the display of the text in the program window and the tool path in the graphics window is influenced. This opens a dialog box with two pages (tab).

### 15.1 Change G-Code presentation

The G-Code Lexer page allows you to change the color and font attributes for all the text elements of the syntax highlighting.



### 15.2 Change power settings



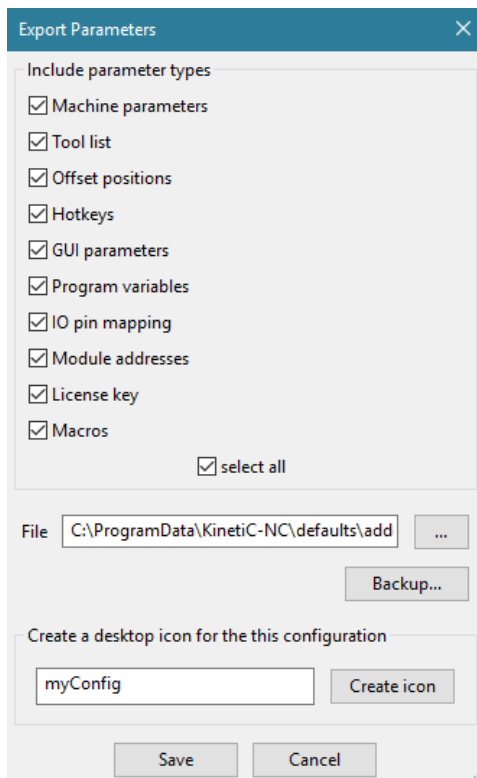
On the Graphics page you can change the performance of the graphic display. For very large programs with several million line segments (depending on the performance of the PC and the graphics card from about 30MB file length) can lead to performance degradation. This is noticeable by the fact that loading takes a long time, and moving or rotating the 3D display is jerky and delayed.

The response time of the other screen elements such as the coordinate display can be delayed. By reducing the pixels and the data volume, this can be counteracted. When the optimization is active, very short lines of the tool path are summarized in the graphic display.

### 15.3 Accuracy vs. resolution

This has no effect on the accuracy of the actual tool path of the machine but only causes a barely noticeable, coarser representation on the screen. The limit for summarizing lines can be either via the upper slider or automatically. The lower slider determines the maximum screen build time. If the update time for the graphic is longer, it will be switched off completely while the machine is running to avoid long reaction times.

### 15.4 Import and Export



The Kinetic-NC software allows the import and export of parameter sets. This makes it easy to transfer parameters from one controller to another, to back up the parameters, or to change the machine if one machine is to operate multiple machines alternately, or different people with different personal preferences work on the same machine. The special feature is that not only the complete parameter set can be exchanged, but also a subset of it. Thus it is possible, for example, to change only the tool list or just the hotkey settings, leaving all other parameters unaffected.

The parameters are divided into groups. The dialog shown on the right can be accessed via the menu "File -> Export settings". Parameter export saves only the selected groups to the file. If this file is later imported to the same or another PC, the remaining groups remain unchanged.

### 15.5 Machine parameters

Machine parameter means all parameters that affect the positioning drives of the axes and the machine in general, i.e. machine type, resolutions, speeds, accelerations and reference switch functions (i.e. all sides of the machine dialog except the input / output configuration). The assignment of the input / output signals to the hardware pins is listed as a separate group. Since each machine has modules with different (MAC) addresses, it does not make sense to transfer them to another PC when copying the parameters, unless the same machine is to be controlled (of course not simultaneously). If, on the other hand, several different machines are to be controlled with one PC (not simultaneously), then the module addresses must also be saved. Therefore, the module addresses also form an extra parameter group.

## 15.6 Tool list

The tool list contains the definition of all tools with all properties such as lengths and diameters. The parameters of the zero points store the coordinates of the workpiece zero points (G54 to G59) as well as the parking position. Hotkeys are the settings for the manual Jog (Jog) and other special function shortcuts. The GUI parameters store the settings for the 3D display, the syntax highlighting colors of the text editor and the last used paths and window positions. Program variables are the values of the nonvolatile variables in the G code (# 100 to # 999). This also includes the temporary zero point and the number of the last substitute tool as well as the macro parameters.

During the import, the parameter groups are shown in the file after the file name has been entered (in the text field or with the file dialog with the "..." button). Non-included groups are disabled (tick box is grey) and cannot be selected.

## 15.7 Modular

Advanced users or manufacturers of control accessories can further limit the scope of parameter groups by editing the exported files with a text editor. Parameters that should not be overwritten during import can simply be deleted from the file. Each line corresponds to a parameter. The parameter name precedes, the value after the equal sign. Groups are separated by identifiers in square brackets. So, it is possible to offer Modular Extensions to certain machines, and to deliver a parameter file, which the customer only has to import during the installation. All his remaining settings on the machine will be retained.

For example, the following extensions are conceivable:

Type of extension	supplied parameters
additional rotary axis for 3-axis machine	Resolution, speed and acceleration values only for the 4th axis
Laser module	Machine type, pin assignment for PWM output, limits for S-parameters
Tool changer	Pin assignment for changer pneumatics and sensors

## 15.8 Create desktop icons

With the button "**Create icon**" you can create a link of the current machine parameters on the desktop. To do this, enter the desired name of your configuration on the left next to it.

Double-clicking the icon on the desktop starts the software with this configuration. This is useful if you have multiple machine configurations and you want to select one at the start of the software.

The software then starts directly with these settings without you having to select "import settings".

## **16 Path preview**

### **16.1 What is it?**

In a CNC program, the path of the tool is given as a sequence of line and circular arc segments. For a "liquid" processing without unnecessarily many acceleration and braking operations, it is important that not every line segment is considered individually, but contiguous curves are driven as possible in one pass. For this purpose, it is necessary that the control software analyses the lane in advance. The achievable processing time and surface quality are decisive for the strategy used.

### **16.2 A comparison**

The problem is similar to driving a car. On some routes there is no speed limit. Here only chassis design and engine power is crucial (highway or rapid traverse). On most sections, however, a predetermined maximum speed applies. The F-values in the program correspond to the signs with speed limits in traffic. In addition, it may happen that the speed must be further reduced due to tight bends, poor road conditions or limited visibility. The driver must always adjust the speed so that he can always stop within sight to avoid accidents. Before bends, he has to estimate how fast he can drive through them without leaving the road because of lack of grip.

At high speeds and with little curve curvature, it can happen that the braking distance is longer than a single road section. Since everything that is not an exact arc of a circle must be approximated in the G-code as a result of small line segments, a single acceleration or deceleration may even drag on over several hundred line segments.

### 16.3 Exact-Stop-Mode

If the specified path in the program is to be maintained exactly, this means that the road is just as wide as the car. It is immediately obvious that cornering is only possible if the curves are circular arcs that merge tangentially into the adjacent line segments. If the track makes a sharp-edged kink, there is no other option than to slow to a stop, turn in place, and continue in the other direction.

When curves such as ellipses or splines are approximated by many short, straight line segments, it means that it has to be braked and accelerated countless times, generating high levels of vibration, and significantly increasing processing time. The Exact-Stop mode is therefore only suitable for prismatic, "square" workpieces. He is activated with G61.

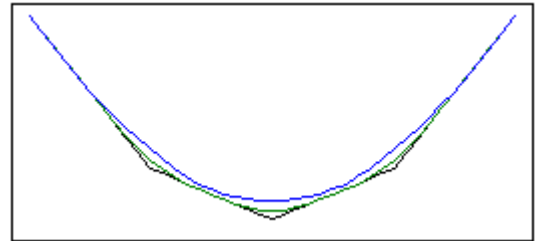
In order to alleviate the problem somewhat, a critical angle and a permissible "jerk", i.e. a discontinuous change in speed is defined. If these limits are not exceeded at a kink, the machine may continue without stopping. Although the direction of movement at the kink changes abruptly, and thus the acceleration is theoretically infinitely high, the real mechanics is always somewhat elastic and tolerates these shocks to some extent. In order to avoid significant chatter marks or even step losses, the limit values must be chosen to be relatively small, usually numbers in the small, single-digit range for angle (degree) and jerk or speed change (mm / s).

If, for example, a limit angle of 2 ° and a permissible jerk of 1mm / s are allowed, then a circle must be broken down into more than 180 line segments, and the feed may not exceed 1719mm / min, so that the "circle" (actually the 180-corner ) can be driven in one piece.

## 16.4 Constant-Velocity-Mode

For workpieces with curved contours or 3D free form surfaces, another strategy is an advantage. It is allowed a certain track tolerance, which may deviate the real tool path from the theoretically exact line. Figuratively speaking, the road is made wider than the car. The driver can then easily cut the turns so that higher speeds are possible without too much impact from abrupt changes of direction. Since the priority here is more on a uniform speed than on the most accurate compliance with the trajectory in the corners, the mode is called "constant velocity". It is activated with the command G64 (Mach3 compatible) or G62.

In Constant Velocity (CV) mode, the controller decelerates to a stop only when the CV limit angle is exceeded. This can be chosen much higher than in the Exact-Stop mode, usually 45 to 90 °. In the case of kinks with a smaller angle, an attempt is first made to reduce the lateral acceleration to such an extent that the permissible values for the drives are not exceeded by rounding off the corner ("cutting", means shortening the curve) within the permissible tolerance.



Only if the acceleration values cannot be maintained despite the maximum path deviation, the speed must be reduced. The controller then does not brake to a standstill, but only so far that acceleration and track tolerance are being maintained. It is then braked before the critical section and then accelerated again.

Since acceleration transverse to the direction of motion produces a deviation from the ideal path rather than parallel to it, and this can leave visible marks on the milled surface (chatter marks or waves), the KinetiC-NC software provides additional constraint on lateral acceleration. If in the machine parameter dialog on the page "Extras" the check mark in front of "CV-mode acceleration" is set, a value for the lateral acceleration, which is smaller than the permissible acceleration of the final drives, can be entered. This value is valid only in CV mode in the rounded corners. For starting and braking at the beginning and end of the contour as well as for rapid traverse, the maximum possible acceleration can continue to be used.

The picture shows the G-Code specified, theoretically exact black course, as it would be driven in Exact-Stop mode. The green and blue curves show the curve actually driven in Constant Velocity mode with different track tolerance settings.

## **17 Post processors**

### **17.1 What is it?**

"Post processor" is the part of the CAM software that is responsible for the output of the G-code to the machine, in this case to the KinetiC-NC software. Although the structure of G code programs is more or less standardized according to DIN / ISO, every machine control has its own special features, and not every software supports all commands. For the frequently used CAM software "ConstruCAM" already exists a ready post processor. If you are using a different CAM software, customizations are generally needed.

The following chapter will help you to find the right post processor for your application or to create one yourself.

First of all, check the Benezan Electronics download page, or ask the retailer, where you purchased the software if there is already a ready-made post-processor for your CAM software. If not, a post-processor for ordinary G-code (standard DIN / ISO or Generic DIN / ISO) is a good starting point. In the following, the special features of the KinetiC NC control are briefly listed. A detailed explanation of the commands can be found in chapter „G-Code Programming“.

### **17.2 Program start**

It is important that at the beginning of the program there is a line with a "%" as the first character. This is the starting marker. All lines before that are ignored, even if they are not in brackets (like a comment).

### **17.3 initialization**

G-code is a modal language, i.e. a command or mode remains active until changed. This also applies across program boundaries and sometimes even from switching off the machine to the next power on. If it is important that certain modes within the program must be activated or deactivated, the corresponding commands should be written to the program header, i.e. after the "%" and before the first move command.

However, it may also be useful in certain cases to deliberately not set a mode in the program so that the previously made setting of the user is maintained. The user can e.g. manually select a specific workpiece zero point (G54 to G59) before starting a program. If a G54 were in the program, the previously made selection would be destroyed.

## 17.4 Typical commands

for initialization (without claim to completeness) are the following:

G17	Layer selection XY for circular arcs
G40	Switch off radius compensation
G43 / G49	Switch length compensation on / off
G90	absolute dimensions
G94	Feed in mm/min
G97	Spindle speed in 1 / min
G54..G59	Select workpiece zero point
G61/G62 or G64	Select Exact stop or constant velocity mode

## 17.5 Formatting

When formatting the lines, characters and numbers, note the following:

- The line numbering (N) is optional. If used, N must be at the beginning of the line.
- Commands must be written in capital letters. Between commands (in front of letters) may be any spaces or tabs. Within commands, on the other hand, there must not be any spaces (correct: G1 X100, wrong: g 1 X 100).
- Of commands that switch modes, several may be grouped in one line (e.g., G17, G40, G90). Of the travel commands (e.g., G1, G2), on the other hand, only one may occur per line.
- M commands with the exception of the program sequence control commands (M2, M30, M99) may appear together with G commands in one line. This is not advised. An extra line for each command is clearer.
- Numbers may have any number of decimal places (including zero). More than 4 decimal places (corresponds to 0.1µm) are usually not useful.
- Leading zeros are ignored. Signs are optional.

## 17.6 Homing

Performing a reference run(homing) at the beginning of each program is usually not useful and only wastes time. The controller independently monitors that a homing run is performed once before the first program start after switching on the machine

## 17.7 Tool change

If the machine has an automatic tool changer, or several different tools are used in the program, and you want to change them manually, you can do it with a T1 M6 (for tool # 1), for example. It is recommended to turn off the spindle and coolant before each tool change, and to reset commands that depend on tool parameters (G41 to G43) after the tool change. Since the CAM program normally does not know where the tool magazine or tool change position is, a safe strategy for starting the next machining point should be chosen after each tool change, otherwise the controller may run diagonally along the shortest path, which is a Collision with the workpiece can result.

## 17.8 Arcs

By default, for arcs (G2 and G3), the center point is specified relative to the starting point with the I and J parameters. (This can optionally also be changed to absolute coordinates in the machine parameters. A circular command can travel a maximum of one revolution, in which case the starting coordinate is the same as the target coordinate.

If, in addition, a coordinate is given perpendicular to the circle plane (e.g. Z for circular arcs in the XY plane), then a helix is driven. Helix paths with more than one turn have to be split into several commands.

The switching of the plane takes place automatically, i. if I and K are used instead of I and J, the circle lies in the XZ plane. The specification of G17 / G18 / G19 is allowed

## 17.9 End of program

At the end of the program, at least the tool should be moved out of the material (Z to safety height) and then an M2 or M30 should be available. Although not absolutely necessary, it is good style to first turn off all the aggregates used in the program (M5, M9).

Also useful is the approach of a parking position to allow easy removal of the workpiece. It is advisable to use the G28 command, which uses the parking position stored in the machine parameters. An explicit specification of the coordinates (G0 ...) is also possible but may possibly lead to undesirable results depending on the selected zero point.

## 18 G-Code Programming

CNC machining sequences are programmed in accordance with DIN / ISO 66025 or RS274, colloquially known as "G code".

A program consists of a header, which may contain any text, the start character "%" and a main part, which in turn is divided into individual sentences (lines).

Each sentence consists of one or more words, which in turn are composed of an address letter and a number.

### 18.1 Address letters

character	Meaning	Unit
<b>A</b>	Rotation axis parallel to X	°
<b>B</b>	Rotation axis parallel to Y	°
<b>C</b>	Rotation axis parallel to Z	°
<b>D</b>	Diameter for radius compensation	mm
<b>F</b>	Feed rate (feed)	mm/min
<b>G</b>	Traversing or coordinate command	-
<b>H</b>	dwel	s
<b>I</b>	Additional parameters for X direction	mm
<b>J</b>	Additional parameters for Y direction	mm
<b>K</b>	Additional parameters for Z direction	mm
<b>L</b>	Special command (parameter memory)	-
<b>M</b>	control command	-
<b>N</b>	Block number (line number)	-
<b>O</b>	Program number for subprogram definition (label)	-
<b>P</b>	Program number or parameter (P0..P9) for drilling cycles or subroutine call	-
<b>Q</b>	optional parameter for drilling cycles	-
<b>R</b>	Parameters for drilling cycles (retract height) or radius for circular arc	mm

<b>S</b>	Speed for tool spindle	1/min
<b>T</b>	Tool number (Tool)	-
<b>U</b>	Second X-axis	mm
<b>V</b>	Second Y-axis	mm
<b>W</b>	Second Z axis	mm
<b>X</b>	X-axis (main axis linear)	mm
<b>Y</b>	Y-axis (main axis linear)	mm
<b>Z</b>	Z-axis (main axis linear)	mm

Table 1: Address letters

## 18.2 DIN/ISO Commands

<b>Command</b>	<b>Meaning</b>	
<b>G0</b>	Rapid to point	
<b>G1</b>	Straight interpolation	
<b>G2</b>	Circular interpolation clockwise	
<b>G3</b>	Circular interpolation counterclockwise	
<b>G4</b>	Wait (dwell time in s)	
<b>G17-19</b>	Select plane for circular arcs (G17 = XY, G18 = XZ, G19 = YZ)	
<b>G20</b>	imperial unit for coordinates (inches)	
<b>G21</b>	metric unit for coordinates (mm)	
<b>G28</b>	Rapid traverse to the parking position or tool change position	
<b>G40</b>	Cancel radius compensation	
<b>G41</b>	Radius correction in direction of travel left (right hand)	
<b>G42</b>	Radius correction in direction of travel right (left hand)	
<b>G43</b>	Activate tool length compensation	
<b>G49</b>	Cancel tool length compensation	
<b>G50</b>	Cancel scale change	

<b>G52</b>	Approach reference point (like G74)	
<b>G53</b>	Cancel zero offset	
<b>G54-59</b>	Activate zero offset from memory 1-6	
<b>G61</b>	Exact-stop mode	
<b>G62, G64</b>	Constant Velocity mode (polygon smoothing, corner rounding)	
<b>G69</b>	Cancel coordinate system	
<b>G73</b>	Drilling with chip breaking	
<b>G74</b>	Approach reference point	
<b>G79</b>	Tool length measurement	
<b>G80..89</b>	Drilling	
<b>G90</b>	absolute coordinates	
<b>G91</b>	relative coordinates	
<b>G92</b>	Set coordinates (changes only offset, does not drive)	
<b>M0</b>	Programmed stop (wait, continue after confirmation)	
<b>M2</b>	End of program (all aggregates off)	
<b>M3</b>	Switch on spindle motor (forward)	
<b>M4</b>	Switch on the spindle motor (backwards)	
<b>M5</b>	Spindle motor stop	
<b>M6</b>	Tool change (automatic or manual)	
<b>M7</b>	Switch on spray cooling	
<b>M8</b>	Coolant pump or chip extraction on	
<b>M9</b>	Cooling or chip extraction	
<b>M10</b>	Clamp round table	
<b>M11</b>	Round table release clamping	
<b>M14</b>	Puncture knife (pen down)	
<b>M15</b>	Lifting the knife (pen up)	

<b>M30</b>	End of program with rewind	
<b>M41-44</b>	Select gear stage 1-4	
<b>M60</b>	Workpiece change	
<b>M66</b>	Tool change (explicitly manual)	
<b>M98</b>	Subprogram call (P = program number)	
<b>M99</b>	Return from subroutine	

### 18.3 special commands

<b>command</b>	meaning	
<b>=</b>	Assignment (calculation formula)	
<b>#0.999</b>	Variable (numeric value buffer)	
<b>#I</b>	Query input	
<b>#O</b>	Set output	
<b>IF THEN</b>	Conditional execution	
<b>UNTIL</b>	Termination condition for G1 or G4	
<b>REPEAT</b>	Loop (repeat)	
<b>CALL</b>	Subroutine call with file name and label	
<b>PRINT</b>	Text output on screen	
<b>SKIP</b>	Jump to label (only forward)	
<b>ASKBOOL</b>	Waiting for confirmation or yes / no decision	
<b>ASKINT</b>	Wait for input of a (whole) number	
<b>ASKFLT</b>	Waiting for a number (decimal fraction)	

## 18.4 G-commands

### **G0: Rapid traverse to point**

The machine moves the axes (X, Y, Z, and A, B, C, if present) as fast as possible to the given coordinates. Axes whose coordinates are not specified remain at the previous position. A special track shape is not guaranteed at G0.

**Example:**

```
G0 X100 Y50 Z-20
```

moves at maximum speed to the room coordinates (100; 50; -20). The axes of rotation (A, B, C, if present) are not moved.

### **G1: Linear interpolation**

The machine travels at the currently set feed rate F in a linear path to the specified target coordinates. If no F is specified in the current block, the last set value applies. "Linear path" means that the traversing path when driving exclusively the axes X, Y and Z is a straight line. With simultaneous movement of the axes of rotation, the movement is proportional, i.e. when the linear axes have travelled 50% of the way, the axes of rotation have covered 50% of the specified angle. Depending on the currently active mode (tool length compensation, axis tracking), however, the path travelled by the tool may no longer be a straight line.

**Example:**

```
G1 X100 A30
```

moves the X axis to the position 100mm and at the same time the rotation axis A to the position 30°. Y, Z and B, C-axes (if available) remain at the previous position.

### **G2: Circular interpolation clockwise**

The tool describes a circular arc clockwise when looking from above at the selected coordinate plane. Either the XY, the YZ, or the ZX plane can be selected. The selection is made by specifying the additional parameters I, J or K. Always exactly two of these three parameters must be specified. The selection of the level with the commands G17 to G19 is not necessary and is ignored. The coordinates X, Y and Z determine the end point of the circular arc, the center point is indicated by I, J and K relative to the starting point (current position).

**Example:**

```
G1 X50 Y10 Z0  
G2 X50 Y110 I0 J50
```

moves a semicircle in the XY plane centered at X = 50 Y = 60. The X50 in the second line could also be omitted. For a full circle, the starting point is equal to the end point and X and Y can be omitted. Simultaneous driving of the rotary axes is not allowed. The circle center may be outside the permissible travel range of the milling machine, but the circular path itself must of course be completely within.

**G3: Circular interpolation counterclockwise**

This command works in the same way as G2, except that the arc is just counterclockwise. Note that G2 will result in counter-rotating and G3 synchronous milling when the spindle is turning clockwise.

**Radius format**

As an alternative to specifying the circle center point with I, J or K, the circular arc can also be defined via the R parameter. Positive values of R produce the shortest possible (<180 °) circular arc to the target point with radius R. Negative values of R produce the longest possible (> 180 °) circular arc with radius -R. It is recommended not to define circular arcs that are exactly or nearly equal to 180 ° with R, as this can cause a large inaccuracy in the position of the center. Semicircles should therefore be given either in the IJK form or divided into two quarters.

**Example:**

```
G0 X0 Y0  
G1 Y90  
G2 X10 Y100 R10  
G1 X100
```

A rounded, right angle is driven.

## G2, G3: helical interpolation

By specifying the third coordinate perpendicular to the circle plane (Z in the example), a circular arc can be converted into a helix (spiral path). This is e.g. when thread milling or for spiral immersion in milling cutters without center cutting needed. If more than one revolution is to be run, simply several sections can be specified one behind the other. The controller combines these and drives them without interruption.

### Example:

```
G1 X50 Y10 Z0  
G2 X50 Y110 Z-5 I0 J50  
G2 I0 J-50 Z-15
```

There are a total of 1½ turns, with a pitch of 10mm per revolution milled.

## G4: Dwell time

The controller waits the time specified in H (in seconds) before the next command is executed. This is useful when, for example, when reversing the direction of threading or when operating pneumatic tensioning devices, a certain amount of run-out or actuation time must be awaited.

### Example:

```
G4 H1.5
```

waiting 1.5 seconds. If G4 is to be called several times at the same time, it is sufficient to specify the time (H) the first time. To ensure compatibility with certain controllers from other manufacturers, the parameter P can be used to specify the delay time. However, the P must necessarily be in the same line to avoid confusion.

The G4 command can be combined with the UNTIL command to wait, for example, for certain switch states (see "Error: reference not found" page error: reference not found).

## G20: Unit Inch

The unit for linear coordinates is switched to inches. This affects only the coordinates used in the NC program, the screen continues to use the unit specified in the **Machine parameters**.

## **G21: Unit Millimeter**

The linear coordinate unit is switched to millimeters (mm). As with G20, the unit of the screen display remains unchanged.

## **G28: Approach park position**

With rapid traverse speed, the system moves to the parking position stored in the parameters. This position can be set in the offset dialog via the menu item "Configuration -> Offsets" and is also used for the manual tool change. If the parking position is higher than the current position (should normally apply), then the Z axis is moved first, then the remaining axes, so that the workpiece and the tool are not damaged.

## **G52, G74: Approach reference point**

The control carries out a so-called reference travel for all axes, i.e. all axes travel one after the other to the limit switch until it is actuated, and then again just far enough away from it that it is free again. For compatibility with other controllers, both G74 and G52 can be used. These commands are normally not needed because the controller will self-home before running the first program after power up.

Repeated reference travel at each program start does not increase the accuracy of the machine, but may even worsen it, such as when the switching point of the limit switches shifts due to heating. Also, because of the unnecessary loss of time, it is therefore recommended not to use these commands. After a possible step loss due to a collision or overload, the reference travel can also be triggered manually via the button on the diagnostics page.

Normally, the G52 or G74 command is used without parameters. A reference run is then carried out successively for all axes. However, it is also possible to select specific axes (so-called "selective reference travel") by specifying the respective address letters with a zero coordinate after the command.

### **Example:**

```
G74 Z0 A0
```

performs homing only with the Z and A axes, X and Y remain at their current position. Regardless of the order, the Z axis is always retracted first, so that any tool that may be in the workpiece is not damaged. If a certain sequence of axes is to be adhered to, several separate G74 or G52 commands must be used in consecutive lines.

### **G40, G41, G42: Activate / cancel tool radius compensation**

With G41, the radius compensation in the direction of travel is switched to the left, with G42 in the direction of travel to the right. When the compensation is switched on, the tool path is moved parallel to the specified radius by the radius entered in the tool list.

In order for the displacement of the web does not lead to an uncontrolled movement at the start and end of the train, a defined approach and departure movement is required. The approach movement is the first G command (G1, G2 or G3) after G41 or G42. The starting point of this line or arc still has the non-shifted coordinates, i.e. the final coordinates of the last travel movement before the G41 / G42 command. The end point of the approach movement is already shifted by the cutter radius. The starting movement must start outside the material and have at least the length of the cutter wheel. If the approach movement is a circular arc, it should proceed as far as possible tangent-continuous in the subsequent movement. In this case, the control software tries to move the midpoint so that the moved arc remains tangent-continuous.

The down movement is the first G command after the G40. The movement starts with shifted coordinates (end point of the last move command before G40) and ends with non-shifted coordinates. Just like the starting movement, the downwards movement must be at least the length of the cutter wheel.

With active radius compensation, the tool path is displayed twice in the graphical preview. A dashed grey line represents the programmed workpiece contour. The green, solid line is the actual trajectory of the tool tip center.

### **G43, G49: Activate / cancel tool length compensation**

For all drive commands following G43, the length of the current tool is added to the Z-coordinate. The tool length is in the tool list. With G49 the length compensation is cancelled, i.e. no more tool length is added to all subsequent commands.

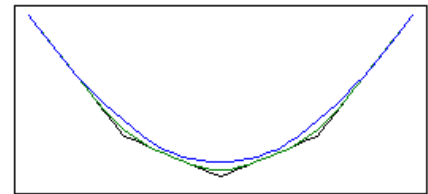
The use of the length compensation can lead to confusion in the graphical preview of the tool path, as there always the path of the tool tip is displayed. If the same workpiece surface is machined with different tools, the same height will be displayed in the graph, although the machine will of course use different Z heights for different tool lengths so that the tip will travel at the same Z height.

## G53-G59: Zero offsets

The commands G54 to G59 activate the corresponding zero memory of the machine, G53 restores the original state (the fixed machine zero). The zero-point memories can be used to place the zero point of the coordinate at any point in the processing space, usually at a corner or the center of the workpiece.

## G62, G64: Edge rounding

With the G62 or G64 command, the automatic edge rounding (CV mode) is turned on. Traverses that consist of many line segments (black curve) are automatically rounded off by inserting arcs at the corners. Strictly speaking, no circular arcs but parabolas are driven, but this makes virtually no



difference at small angles. In the machine parameters, the max. Path deviation (chord error) can be adjusted. The control completes the track only so far that the max. Path deviation is not exceeded. The smaller the permissible deviation, the more precisely the curves are "extended" (green curve), which, depending on the acceleration parameters, may mean that the speed must be reduced. The higher the permissible deviation, the more the curves are "cut" (blue curve), which leads to a "softer" course and fewer braking and acceleration processes.

## G61:

turns off edge rounding (exact stop mode). All corner points of the line segments are approached exactly, which requires a deceleration to a standstill and re-accelerating.

## G79: Tool length measurement

This invokes the subroutine (macro "G79.txt") for automatic tool length measurement. The implementation is machine dependent.

## G73, G81-G89: Drilling cycles

These commands are defined depending on the machine. The associated macros are called. The drilling cycles G73 and G81 to G83 are defined by default, the rest can be extended by editing the macro.

command	designation	explanation
<b>G81</b>	simple drilling	Z-movement to target depth Z, then single lifting to return height
<b>G82</b>	face drill	Z movement to target depth Z, short dwell time for free cutting of the plane surface
<b>G83</b>	deep hole drilling	Step by step drilling to target depth ("peck drill"), between the steps complete excavation to relax
<b>G73</b>	drilling with chip breaking	As G83 upper without complete lifting, instead only short feed stop between because steps to break the chip
<b>G84</b>	tapping	(Macro not defined)
<b>G85</b>	reaming	(Macro not defined)
<b>G86</b>	boring	(Macro not defined)
<b>G87</b>	back boring	(Macro not defined)
<b>G88</b>	Lowering with free cutting	(Macro not defined)
<b>G89</b>	reaming with free cutting	(Macro not defined)
<b>G80</b>	cancel cycle	(Optional)

### Example:

```
G0 X10 X20 Z10
G83 Z-30 R5 Q2 P0.3
X20
X30
G80
```

Three equal holes are drilled with depth Z-30, each at the X coordinates X10, X20 and X30. The retraction height after drilling is Z = 5, the increment between the decanting strokes is Q = 2mm. At the lowest point, P = 0.3s is stopped.

### **G90: absolute coordinates**

The following coordinates are absolute. This is the default.

### **G91: relative coordinates**

It is switched to the relative coordinate mode. All values for position values X, Y, Z, A, B, etc.) are added to the previous value. This should be used with caution and preferably only in subprograms if the same movement (shape, hole pattern ...) should be repeated several times at different positions (independent of the starting position). It must not be forgotten to return to absolute mode (G90) at the end.

#### **Example:**

```
G90 G0 X50 Y70  
G91 G1 X10 Y10
```

It is first driven to the absolute position X50 Y70 and then to the absolute position X60 Y80. Clearer and less error prone is the formula notation (e.g., "X = X + 10"), which is not compatible with the DIN / ISO standard.

## 18.5 Other G-commands

Although the following G commands are known to the controller, they are ignored because they only enable a setting that is always valid anyway. These commands are only available for compatibility because, for example, many post processors basically write them to the beginning of a CNC program.

Command	function	Is ignored because ...
<b>G17</b>	Layer selection XY for circular arcs	Level is detected by parameters I and J.
<b>G18</b>	Layer selection ZX for circular arcs	Level is detected by parameters I and K.
<b>G19</b>	Level selection YZ for circular arcs	Level is detected by parameters J and K.
<b>G50</b>	Cancel scale change	Scale scaling is not supported
<b>G69</b>	Cancel coordinate system	Rotate Coordinate System is not supported
<b>G94</b>	Feed in mm / min	Feed in mm / rev. (G95) only for lathes
<b>G97</b>	Speed in 1 / min	Speed in m / min (G96) only for lathes

Table 2: Other commands (no effect)

There is also a second set of commands that has not yet been explained because they are machine dependent and can be defined by the user.

## 19 M-Commands

These commands are used to control the program sequence or the switching of additional units.

### **M0: Programmed stop**

The controller stops the program execution and waits for a manual confirmation of the operator. The M0 command is useful for debugging a new program, or when performing manual control measurements or visual checks between multiple processing steps.

### **M2: end of program**

The running program is terminated at this point and all units (tool spindle, coolant) are switched off. After M2 standing sentences (lines) are no longer executed. See also M30 (End with rewinding)

### **M3: Switch on spindle motor (forward)**

The tool spindle motor is turned on in the normal, clockwise direction. The speed can be adjusted with the parameter S in revolutions per minute (only with existing frequency converter whose setpoint speed signal is controlled with the PWM or analogue signal). If no S is specified, the last set speed applies.

#### **Example:**

```
M3 S6000
```

The spindle speed can only be changed without prior shutdown of the motor by specifying a new S value.

### **M4: Switch on the spindle motor (backwards)**

The tool spindle motor is turned on in the opposite direction of rotation (anti-clockwise) if allowed by the machine. As with M3, the speed can be selected with S1.

### **M5: Spindle motor stop**

The tool spindle motor is turned on in the opposite direction of rotation (anti-clockwise) if allowed by the machine. As with M3, the speed can be selected with S1.

**M6: Perform tool change**

Performs an automatic tool change. The tool number must be specified with T.

**Example:**

T3 M6

replaces the currently clamped tool with the number 3. The actual action performed depends on the machine parameters. Depending on whether manual or automatic tool change has been selected, either the macro M66 or M6 will be executed. The tool change macros are dealt with in more detail in the chapter Macro Programming

**M7: Switch on spray cooling**

The spray cooling (or minimum quantity lubrication) is switched on.

**M8: Coolant pump or chip extraction on**

With this command, the first auxiliary unit (depending on the type of machine coolant pump or suction) is switched on.

**M9: Cooling or chip extraction off**

Switches off the first auxiliary unit (depending on the machine type) and the spray cooling (if available). At the end of the program (M2 or M30), all units are switched off automatically, even if M9 has been forgotten.

**M14: Immerse knife**

Immerses the knife ( tangential knife cutting) into the material.

**M15: Lift knife**

Lifts the knife out of the material when cutting with tangential knife

**M30: End of program with rewinding (restart)**

Cancels the program execution like M2. The operator can also restart the program with a keystroke. This is useful in mass production, if you want to process several identical parts one after the other.

## **M66: Manually tool change**

A manual tool change is explicitly performed, even if the machine has an automatic tool changer

## **M98: A subprogram call**

The subprogram is called with the number given after "P". The subprogram is marked with an O number.

### **Example:**

```
M98 P1234  
M2
```

```
O1234 :  
G91 G1 X10  
G90  
M99
```

Alternatively, the command "CALL" can be used for the subprogram call. In addition, a file name and any identifier (not just a number) can be specified as the destination. This makes it possible to build subroutine libraries that contain repetitive processing steps, such as different hole patterns for connectors in front panels.

### **Example:**

```
CALL "SUBD9@connector.txt"
```

"SUBD9" is the name of the subprogram (jump target, label), "connector.txt" the file name. It is first searched in the current directory. If the file is not found there, it is also searched in the current macro directory, which is set in the Macro-Dialog .

## **M99 Return from subprogram**

It will be returned to the spot after the last M98 in the main program. Alternatively, the command "RETURN" can be used.

## **Other M-commands**

There are also other M commands that have not yet been explained because they are machine-dependent and can be defined by the user (see Chapter „Macroprogramming“). These include: M10, M11, M41-44, M60

## 19.1 Special commands

The KinetiC NC control has a few more additions to the standard commands. For example, variables can be calculated in order to make positions dependent on parameters, for example:

**G1 X=100 + 20\*T**

These special commands or language constructs are described in detail in the chapter „Macro-programming“ Although they can also be used in normal G-code programs (main program). For compatibility reasons, however, they are mainly used in so-called macros.

## 19.2 Compact code

The KinetiC-NC control also allows special techniques to be used to minimize the memory requirements of the program. For this purpose, the spaces between the words, or G commands, which are repeated in several successive lines (sentences), can be omitted.

### Example:

extensively	Shortened
M3 S5000	
G0 X50 Y20 Z5	G0X50Y20Z5M3S5000
G1 Z-2	G1Z-2
G1 X60 Y20	X60
G1 X60 Y22	Y22
G1 X50 Y22	X50
G0 Z5	G0Z5

Commands that do not trigger a travel movement but only set one mode (eg G90 or G40) may be several in the same line. Regardless of the order in the program text, the mode commands, then M commands and last-moving G commands are always executed first.

This format is mainly generated by CAD systems (post-processors). For handwritten programs, this approach is not recommended, because the clarity suffers greatly and the files are synonymous usually not so long that the space savings worthwhile.

## 20 Macro-Programming

### 20.1 What is it?

The Kinetic-NC controller can be flexibly adapted to different machines by programming so-called macros. Macros are small programs written in an extended G-code programming language and executed whenever special commands such as automatic tool change (M6) are executed. The macro defines the behavior of the controller for these commands. In addition, there are macros that are always executed at certain times or events, for example the startup macro when starting the control software. Then there are freely definable user macros with which frequently required command sequences can be programmed, which can be started by the user interface at the push of a button.

### 20.2 Where can I find the macros?

All macros are stored in the directory "%Programdata%\Kinetic-NC\Macros\" as text files. The actual path of this directory depends on the version of Windows you are using. For Windows7 this is usually "C:\ProgramData\Kinetic-NC\Macros\", for Windows XP "C:\Documents and Settings\All Users\Application Data\Kinetic-NC\Macros\". The last part of the path ("macros") can be changed in the macro dialog to easily switch between different macro variants. The following tables provide an overview of all macros and their intended use.

### 20.3 Re-definable commands and drilling cycles

Re-definable commands are special M or G codes that are replaced by a subroutine call (macro). As a result, the behavior of the controller can be flexibly adapted to the conditions or special features of the machine. For example, whenever the spindle (M3 or M4) is switched on, the blocking air can also be switched on, or a condition (tool magazine closed and tool changer locked) can be checked. In drilling cycles such as deep hole drilling, a single G83 instruction is translated into a sequence of multiple G0 and G1 instructions.

command	Descriptions
<b>M3</b>	Spindle motor on (clockwise)
<b>M4</b>	Spindle motor on (anti-clockwise)
<b>M5</b>	Spindle motor stop
<b>M6</b>	Automatic tool change
<b>M7</b>	Spray cooling on

<b>M8</b>	Slump cooling on
<b>M9</b>	Cooling off
<b>M10</b>	Clamping
<b>M11</b>	Clamp off
<b>M14</b>	Immerse tangential knife (pen down)
<b>M15</b>	Lift tangential knife (pen up)
<b>M41</b>	Gear stage 1
<b>M42</b>	Gear stage 2
<b>M43</b>	Gear stage 3
<b>M44</b>	Gear stage 4
<b>M60</b>	Workpiece change (pallet changer)
<b>M66</b>	Manual tool change
<b>G73</b>	Drilling with chip breaking
<b>G74</b>	homing
<b>G79</b>	Tool length measurement
<b>G81</b>	Easy drilling
<b>G82</b>	piloted counterbore
<b>G83</b>	Deep hole drilling (with chips)
<b>G84</b>	tapping
<b>G85</b>	Rub
<b>G86</b>	boring
<b>G87</b>	lowering
<b>G88</b>	Drill Cycle No. 8
<b>G89</b>	Drill Cycle No. 9

## 20.4 Event Macros

These macros are always called at a specific time.

<b>PenUp</b>	Lift tool (tangential knife cutting)
<b>PenDown</b>	Immerse tool (tangential knife cutting)
<b>Startup</b>	Starting the machine (KinetiC-NC software)
<b>Shutdown</b>	Switch off the machine (exit software)
<b>PrgStart</b>	NC program start
<b>PrgEnd</b>	NC program end (normal, without error)
<b>PrgStop</b>	NC program cancellation (error or interruption)
<b>PrgResume</b>	Continue NC program

## 20.5 User-Macros

User macros are programs that can be started by pressing the buttons on the macro page of the user interface. Frequently required processes, such as the warm-up of the spindle, can be defined here.

The user macros have the names "User01" to "User10". The text displayed in the "Description" column in the macro dialog and below the image icon on the "Special" page can be set to "#LABEL" in the first line of the macro.

### Example:

```
#NAME User01.txt #LABEL "Height measurement with mobile sensor"
```

The icons (icons) can also be changed. For this, an image file with the name "User01.png" (or corresponding number) and the size 64 x 64 pixels must be stored in the icon directory. This is by default the directory "%programdata%\KinetiC-NC\icons" but can be changed in the macro dialog. Icons are saved together with the macro texts in the file, which is created when "Export settings" has the option "Macros" switched on.

## 20.6 Macro-Language

The programming language of the macros is an extension of the normal G-code. Additional commands have been defined to allow, among other things, conditional execution (IF .. THEN), loops (REPEAT), variable calculation, switching of output signals, and waiting for input signals (G4 UNTIL). The following chapter first describes the individual commands before going into the individual macros.

## 20.7 Calculation formulas (assignments "=")

Deviating from the DIN or RS274 standard, the KinetiC-NC controller can specify calculation formulas for coordinates or parameters. To maintain compatibility with other controllers, it is recommended that you use this only in subroutines and macros. If a coordinate or a parameter is not specified directly (with a constant numerical value) but via a formula, this is indicated by an equal sign after the address letter.

### Example:

```
G0 Y=T*40 (Move to tool magazine position No. T)
```

With tool number  $T = 1$ , this would result in a G0 Y40, in tool no. 2 a G0 Y80 etc. Permitted arithmetic operations are + (addition), - (subtraction), \* (multiplication) and / (division). The usual calculation rules apply (point before line), if necessary may be clipped.

To the left of the equal sign must be an output (see below), a variable, or an address letter representing a coordinate or other numerical value (for example, X, A, or S but not G or M). To the right of the equal sign is the formula, which consists of address letters, arithmetic signs and, if necessary, parentheses. Constants (numeric values), variables and inputs (see below) may also be used to the right of the equals sign.

In addition to the normal arithmetic operations, the following mathematical functions are also defined. This makes it possible to convert polar coordinates or perform distance calculations.

<b>ABS()</b>	Absolute value (always positive)
<b>SQRT()</b>	square root
<b>SIN()</b>	Sine
<b>COS()</b>	cosine
<b>TAN()</b>	tangent
<b>ASIN()</b>	Arcsine (inverse of sine)
<b>ACOS()</b>	arcuscosinus
<b>ATAN()</b>	arctangent
<b>EXP()</b>	Exponential function (ex)
<b>LOG()</b>	Logarithm function (to base e)
<b>ROUND()</b>	Round to the next whole number

## 20.8 #0..999 variables

In addition to the fixed address letters assigned a numerical value (e.g., X, Y, Z, A, B, C, F, and S), there are still user-usable variables that can be used to latch numbers. User variables are identified by a number preceded by a "#" sign. Variables can be on the left (write) or right (read) of the equals sign, or IF = THEN in comparisons.

Depending on the number range, different groups of variables are distinguished. The range # 0 to # 29 may be used freely, e.g. for transferring parameters to subroutines or for saving intermediate results and temporary values. These variables are local, that is, their value is not passed to the main program when returning from a subroutine, and volatile, i.e., it is not saved to disk when the software is exited.

The range # 100 to # 899 is also freely usable, but is global and non-volatile, i.e. the value is the same in all (sub) programs and is stored on the hard disk. States that are stored in these variables are still preserved as well as after the software has been shut down and restarted.

The range # 30 to # 99 is used for internal purposes and may not be used in programs and macros. The range # 900 to # 999 is reserved for special functions. If the meaning of the variables is documented, read access is permitted. The values should only be changed with the utmost caution and if this is expressly permitted, as otherwise unforeseen behavior of the control system can be triggered.

Variables can be accessed indexed. In this case, the number of the variable is not a constant (as in # 1) but is itself a variable or formula that is calculated at runtime. This formula must be written in square brackets. For example, # [100 + # 1] stands for variable # 102 if variable # 1 is 2. It must be ensured that the index (here # 1) is an integer and the formula complies with the permissible value range. If the index originates from a user input or from a measured value, a range check should be performed beforehand with IF ... THEN.

## 20.9 #I0..999 Inputs

An "I" after the "#" sign indicates that it is not a variable (buffer) but an input (I = input). An input can only be read by reading, i. it may only be to the right of the equal sign in an assignment or only to the left of the comparison operator in a comparison. A digital input can only have the value "0" (switched off) or "1" (switched on), depending on the electrical state at the input connection. Please note that when assigning the inputs to certain connection pins, the signal can optionally be defined as low or high active. High-active signals are energized when energized (state "1") and de-energized (state "0"). For low-active signals, it is the other way round.

The numbers of the inputs and outputs are "logical" numbers and are not directly related to the actual pin or terminal numbers of the hardware. Each logical signal number can be assigned to an input or output pin in the Machine parameter-dialog This makes it possible to write macros that will work regardless of the hardware or wiring being installed. That if the wiring is changed, only the assignment of the signal numbers has to be changed, not the program text of all macros.

The range # I0 to # I99 is reserved for predefined signals with a defined meaning. Inputs # I100 to # I999 can be freely defined.

As with variables, inputs can also be accessed indexed. Example: #I [100 + # 0]. The same rules apply as described above.

## 20.10 #O0..999 Outputs

The letter "O" (not to be confused with the zero) after the "#" sign indicates that it is not a variable (buffer) but an output (O = output). An output can only be described, i. and may only be to the left of the equal sign in an assignment. A digital output can only accept the value "0" (switched off) or "1" (switched on). Outputs are always high-active, so that the safe state is clearly defined (safe = voltage-free = switched off).

The outputs also have logical signal numbers that can be assigned to arbitrary pins of the hardware. Outputs in the range # O0 to # O99 have a fixed meaning, # O100 to # O999 can be used freely.

As with variables, it is also possible to access outputs indexed. Example: #O [100 + # 0]. The same rules apply as described above.

## 20.11 Read tool parameters

### L51:

With this command, the tool parameters of the current tool are transferred from the tool list to the variables # 920 to # 929.

No.	Definition
#920	T-No. of the currently substitute tool
#921	P-No. the current tool (magazine slot)
#922	current tool type (see tool dialog, e.g. end mill = 1, drill = 2, etc.)
#923	current tool diameter (main cutting edge)
#924	2nd diameter (depending on the type)
#925	current tool length
#926	2nd length (depending on the type)
#927	Max. number of revolutions
#928	Max. Lifetime (minutes)
#929	already expired service life (minutes)

## L52: Save tool parameter

With this command the variables # 920 to # 929 are written to the tool list. This can be used to set the length and / or diameter of the current tool in the tool list to a certain value after a survey. This is actually only required as part of the G79 macro to update the tool list after a length measurement. (see chapter "Re-definable Commands, G79")

## L53: Set zero point memory

The zero-point memories (offsets) define the position of the workpiece zero points (G54 to G59) in the machine room. The zero-point memories can be set manually via the user interface, either in the offset dialog or by pressing the reset buttons at the coordinates. Alternatively, the zero memory can also be changed by macros. The coordinates of the current offset are in variables # 900 through # 908 (the X axis in # 900, Y in # 901, and so on). The number of the current zero point is # 909 (0 = G53, 1 = G54, etc.). The variables can be changed and then stored by L53 in the memory of the currently selected zero point (G54 to G59). G53 cannot be changed.



**Attention**, it is recommended to use this option only within survey macros, and if you know exactly what you are doing. An erroneous change of the zero points can lead to unforeseen behavior of the machine and to collisions.

## 20.12 IF THEN, ENDIF: Conditional execution

Within macros, such as automatic tool changes, it is often necessary to make decisions based on the state of inputs or variables. With the IF ... THEN command, program parts can be conditionally executed. Between IF and THEN stands a so-called Boolean expression, i.e. a condition in the form of a comparison of a variable or an input with a numeric expression (formula) or a combination of several conditions. The program part between THEN and ENDIF is only executed if the condition is fulfilled.

Example:

```
IF #I100=1 THEN
  M2 (abort, no tool in the magazine)
ENDIF
```

The program part between ELSE (optional) and ENDIF is executed only if the condition is not met:

```
IF T<=10 THEN (tool no 1..10)
G0 X-50 (first row of tool magazine)
ELSE (tool no. 11..20)
G0 X-100 (second row)
ENDIF
```

Permitted comparison operations are = (equal), > (greater), >= (greater or equal), < (less), <= (less than or equal to), or <> (unequal). The comparison operation consists of a variable, an address letter or an input on the left side, the comparison operator and a formula or constants on the right side. IF and THEN must be delimited by blanks from other words. Several conditions can be linked with "&" (logical AND), "|" (logical OR) or "!" (Logical NOT).

You may also use brackets.

**Example:**

```
IF (A=1) & !(B=2) THEN
```

The condition is met if A equals 1 and B is not equal to 2. If required, several IF ... ENDIF blocks may also be nested, i.e. there may be further IF ... ENDIF blocks between THEN and ENDIF or between THEN and ELSE or ELSE and ENDIF.

Programmers who are accustomed to C or similar programming languages should note that KinetiC-NC control strictly distinguishes between Boolean and numeric expressions, and there is no tacit conversion between them. Expressions such as IF # I4 THEN are not allowed because a signal input has a numeric value. A Boolean expression is obtained by comparison with a constant, or formula (with variables). Correct would be IF # I4 = 1 THEN. On the left of the compare operation no formulas or constants are allowed, but there may only be exactly one variable or exactly one input signal.

A special feature is the & -operator. It can be used both for the Boolean AND operation of two Boolean expressions and for the bitwise AND operation of two numerical values. Result is a Boolean value in both cases.

**Example:**

```
IF (X>100) & (Y<50) THEN (execute only if both conditions met)
...
IF #2&4 THEN (execute if bit 2 in variable #2 is set)
```

### 20.13 REPEAT: Loops

The REPEAT statement can be used to repeatedly execute a sequence of commands. The number of repetitions must be calculated before entering the loop. Although this number may depend on variables, it is not recalculated during loop execution, that means the value before the first execution is relevant.

#### Example:

```
REPEAT=3  
G1 X=X+10  
G4 H1  
NEXT
```

This executes the commands between REPEAT and NEXT three times, so it moves three steps of 10mm each and waits one second in between.

### 20.14 UNTIL: stop condition

The UNTIL statement can be used to define an abort condition for a G1 trip or for a wait command (G4). The G1 command then moves in the direction of the target position until it has either been reached, as with G1 without a termination condition, or before the condition has been fulfilled. Similarly, the G4 command waits until either the condition is met, or the waiting time has expired. At present, only conditions with a single input signal are permissible, meaning Boolean combinations of several comparisons are not possible.

#### Example:

```
G4 H10 UNTIL #4=1
```

Waits until the input signal # 4 becomes active or a maximum of 10 seconds have elapsed.

### 20.15 SKIP: Jump to label

The SKIP command can be used to jump to a specific point in the program, which is marked with a label. In other words, the program part between SKIP and the label (jump target) is not executed. Of course, this only makes sense if the SKIP command is combined with IF...THEN. It can only be jumped forward. To program loops, REPEAT ... NEXT must be used.

**Example:**

```
SKIP label  
...  
label:  
PRINT: text output
```

With these commands, texts and variable values can be printed out in the message window. Texts and variable names are separated by a semicolon.

**Example:**

```
PRINT "current tool no = ";T  
PRINT "tool diameter = ";#923;"mm"
```

## 20.16 ASK: Dialogs for parameter input

User macros may require that the program be paused and wait for input from the user. In the simplest case, this is simply a confirmation that it can go on. But you can also enter a parameter as a number. There are three versions of the ASK command.

### 20.17 ASKBOOL „message text“ I=1

A dialog opens and the message text is displayed. Then wait for confirmation, the user must press one (of possibly several) buttons. The I parameter determines the number of choices. If I = 0, there is no selection, meaning there is only one "Cancel" button, which always terminates the program. The text is de facto an error message. If I = 1, there is an "OK" and a "Cancel" button. The user can either continue or cancel the program. If I = 2 there is a choice between "Yes", "No" or "Cancel". If "Yes" or "No", the variable # 0 returns a "1" or "0", otherwise the program is aborted.

### 20.18 ASKINT „message text“ I=min J=max

A dialog opens and the message text is displayed. The user then has the option of entering an integer in a field. There is an "OK" and a "Cancel" button. The user can either continue or cancel the program. The parameters I and J are optional limits. If omitted, there is no limit to the number range. The result is passed in variable # 0.

## 20.19 ASKFLT „message text“ I=min J=max

Just like ASKINT, but for floating-point numbers.

## 20.20 CALL: subprogram call

Similar to the **M98-command**, the CALL instruction is used to call subroutines. In addition, a file name and any identifier (not just a number) can be specified as the destination. This makes it possible to build subroutine libraries that contain repetitive processing steps, such as different hole patterns for connectors in front panels.

### Example:

```
CALL "SUBD9@connector.txt"
```

"SUBD9" is the name of the subroutine (jump target, label), "connector.txt" the file name. It is first searched in the current directory. If the file is not found there, it is also searched in the current macro directory, which is set in the *macro dialog* .

## 21 Re-definable commands

As described above, these macros are called whenever the appropriate M or G command is used.

### M3 and M4

These macros are used to switch on the spindle motor. In the simplest case, only the corresponding output # O3 or # O4 is activated. If required, additional outputs can be activated, for example for sealing air, or inputs can be checked before the spindle is started, such as whether the tool changer is locked or the pneumatic pressure is sufficiently high.

The M3 and M4 macros are given the following parameters in the variables S and # 0 to # 2:

<b>#0</b>	Actual speed (before)
<b>S</b>	Set speed (after)
<b>#1</b>	Delay time (speed independent) is s
<b>#2</b>	Ramp time (speed dependent) in s / (1000 / min)

Thus, a speed-dependent time can be waited before the program is continued to give the spindle enough time to reach the predetermined speed.

The default implementation of the M3 macro looks like this:

```
%  
#O4=0  
#O3=1  
G4 H=#1+(S-#0)*#2/1000  
RETURN
```

Instead of the calculated waiting time  $G4 H = \dots$ , it is also possible to wait for an input if, for example, the frequency inverter of the spindle motor can output a signal "Speed reached". In this case you could write:

```
G4 H20 UNTIL #I100=1  
IF #I100=0 THEN  
PRINT "error: spindle speed not matched"  
M2 (Abort)  
ENDIF
```

Since G4 always waits with UNTIL only for a maximum time (here 20s), it is advisable to additionally check whether the input actually became active or the max time was exceeded. If the machine has an automatic tool changer, it is also advisable to check before starting the spindle if the last tool change was successful, if there really is a tool in the spindle, and the clamping mechanism is properly locked.

## **M5**

This macro is used to switch off the spindle motor. The parameters are the same as above, except that the setpoint speed is always zero. Either a speed-dependent delay time can be calculated or the "standstill" input signal (# I5) can be waited for.

## **M6**

This macro performs an automatic tool change if the machine has a changer, depending on the type of changer, to carry out quite complicated actions. Unless the machine has been delivered ready with preconfigured controls, the default macro is therefore blank for security reasons and is only waiting for a manual change. Examples of the most common types of changer (linear magazine, drum magazine / revolver, chain magazine) can be found in chapter 33 "Examples".

The "Tool change" setting on the "General" page of the configuration dialog ("Configuration -> Machine" menu) determines whether the M6 (automatic) or M66 macro (manual) is executed when changing. For machines without automatic changer M6 does not need to be defined. Only M66 will always be used.

The number of the tool to be loaded and the associated magazine location are transferred in the variables T and P. The M6 macro must memorize the number of the previously substituted tool in another variable so that it can be put back in the right place. It makes sense to use a flag variable for "tool change successful" or "valid tool in spindle", which is deleted at the beginning of the change and reset without error when the macro is ended. Thus, unauthorized states can be detected and reacted accordingly. For example, with most spindle motors with changer it is forbidden to turn the spindle without inserted tool. This can then be prevented by querying the flag variable in the M3 macro.

## **M7, M8 and M9**

These macros turn the cooling or exhaust on and off. In the simplest case, only the associated output (# O7, # O8) is switched on or off. In variable # 1, the delay time set in the machine parameters is transferred, which is to be maintained before the program is resumed. If this is critical for processing, for example using an internally cooled tool, it would also be conceivable to wait for the signal from a pressure sensor.

## **M10 and M11**

These commands are normally intended for (hydraulic or pneumatic) clamping of an axis of rotation but may also be used for workpiece clamping or other auxiliary devices.

## **M14 and M15**

These macros are called when the knife is raised (M15 = Pen up) or immersed (M14 = Pen down) when cutting with the tangential knife. Depending on the transition angles of the tool path, the controller may decide to insert a retraction and puncturing movement even without programmed Z movement. This will prevent tearing of the material or breakage of the blade at too large angles in corners of the web.

The PenUp macro must remember the current Z-depth in a variable so that PenDown can return there. The set safety height is transferred in variable # 2.

## **M41 to M44**

These commands are provided for switching the transmission, as long as the main spindle has a manual transmission. In addition to operating the corresponding outputs for the gearshift mechanism, it is also possible to set variables that provide a gear ratio adapted speed calculation in the M3 macro.

## **M60**

This command is used to initiate the change of the workpiece if the machine has a pallet changer. Alternatively, in a lathe, the bar loader could be instructed to move the material one step further.

## M66

This command is used on machines with automatic tool changer to explicitly request a manual tool change. The "Tool change" setting on the "General" page of the configuration dialog ("Configuration -> Machine" menu) determines whether the M6 (automatic) or M66 macro (manual) is executed when changing. For machines without automatic changer M6 does not need to be defined. Only M66 will always be used.

## G79

This command performs an automatic measurement of the tool with a length scanner. To do this, move in the direction of the length button until it is actuated, and then calculate the tool length from the current position and enter it in the tool list with L52.

### Example:

```
G1 Z-100 F150 UNTIL #4=1 (move down until switch triggered)
#925=Z+76 (calculate length from Z position)
L52 (save value to tool list)
```

The input signal number for the switch (# 4) and the Z-values (-100mm or 76mm) are arbitrary and must of course be adapted to the machine. If, in the example, the switch is reached at Z-90, Z = 76-90 = -14 would be entered in the tool list. Of course, the desired tool must have been previously loaded with T and M6.

## 22 Drilling cycles

Commands G81 through G89 and G73 are effectively an "abbreviation" for multiple G0 and G1 commands to create a hole. With the KinetiC-NC, these cycles are not permanently programmed in the control software but can be adapted with the macros according to their own needs. Only cycles G73 and G81 to G83 are predefined by default.

### **G81**

Simple drilling: Drilling is performed with the set feed F to depth Z, then retracted to the retraction height R.

### **G82**

Lowering: In addition to the simple drilling, a dwell time P is still maintained at the target depth Z in order to cut free the end face of the hole or countersink.

### **G83**

Deep hole drilling: The drilling process is divided into several steps of depth Q. Between the steps, the drill is completely pulled out of the hole for decanting.

### **G73**

Drilling with chip breaking: Same as G83, but only pausing between steps to break the chip flow rather than immersing completely.

### **G84**

Tapping - not defined by default (macro must be added).

### **G85**

Rub - not defined by default

### **G86**

Boring - undefined by default

### **G87**

Countersink - not defined by default

**G88**

Pit countersinking - not defined by default

**G89**

Rub with trimming - not defined by default

## 23 Event-Macros

The following macros are invoked implicitly, without a special command being executed, at certain times.

### 23.1 Startup

The startup macro is executed immediately after the software is started, or the first time hardware is connected and the emergency stop is unlocked. It can be used to perform required initializations of variables or output signals, for example, or to activate central lubrication.



**Attention!** Please note that it is not permitted under the Machinery Directive to carry out movements immediately after switching on, which could pose a danger to the operator. It is therefore not advisable to carry out a reference run, for example within the startup macro. This could result in the machine starting to move in an uncontrolled manner after a power failure and subsequent return of the supply voltage.

It is better to set a flag variable that indicates that a certain operation still needs to be performed, and then query and, if necessary, catch up with the PrgStart macro. If movement of the drives in the startup macro is indispensable, the machine must have interlocks (protective doors or similar) which preclude any danger to the operating personnel.

### 23.2 Shutdown

The shutdown macro is executed when the control software is closed. This can be used to switch off additional units or bring them into a parking position.



**Attention!** An execution cannot be guaranteed in all cases. In the event of a power failure, interruption of communication between the PC and machine hardware or other serious errors, controlled shutdown of the machine is no longer possible. It is therefore important that even in uncontrolled shutdown, the machine goes into a safe state by itself, and among other things vertical axes are secured by self-closing holding brakes from falling down. However, by setting a flag variable in the shutdown macro, it is possible to remember if the machine shut down properly. If not, this can be detected at the next start in the startup macro and responded accordingly.

### **23.3 PrgStart**

The PrgStart macro is executed before each program start of a loaded NC program. Individual commands, which are entered via MDI, do not count as programs.

### **23.4 PrgEnd**

The PrgEnd macro is called after every program execution, if the program was successfully completed (without error) until the end.

### **23.5 PrgStop**

This macro is executed after a program has been aborted, i.e. when the operator pressed the stop button or the program was terminated by an error.

### **23.6 PrgResume**

The resume macro is executed before a program is continued or jumped into the middle of a program, i.e. it is not executed from the beginning.

## 23.7 Examples

Although all commands and uses have been explained, programming macros is not easy when you need to start from "zero". Therefore, some examples of macros are listed and explained. The macros are not readily usable for a particular machine, because at least the approached positions, the numbers of the inputs and outputs and other parameters must be adapted to the particular machine.

In all examples, it is assumed that G53 Z0 is the highest position of the Z axis and is driven down into the negative range. If the top Z position is in the positive range, you must adjust the Z coordinates accordingly.

### 23.7.1 Automatic tool change with linear magazine

The linear magazine is the simplest case of all tool change systems. All tools have a fixed space within the processing space and can be achieved by moving the X, Y and Z axes. Optionally, the magazine can be swung into the processing room with a pneumatic cylinder, for example, or a safety door can be opened and closed.

```
#LABEL "automatic tool change"
Inputs: #I8=LuftOk, #I52=magazine open, #I53=magazine closed
Outputs: #O13=open magazine, #O14=eject tool
Parameters:
magazine position of T1: X=#965 Y=#966 Z=#967
distance T2-T1: dX=#968 dY=#969
move into magazine: dX=#970 dY=#971 dZ=#972
safety position X=#974 Y=#975 Z=#976
number of slots #973
%
PRINT "old tool=";#950;" new=";#920
IF #920=#950 THEN
  RETURN (same tool, done)
ENDIF
IF #940=0 THEN
  PRINT "error! state of the ATC is undefined."
  PRINT "Please verify all tools and execute manual"
  PRINT "tool change or 'set number only'."
  M2
ENDIF
IF #920<0 | #920>#973 THEN
```

```
PRINT "error! invalid tool number T=";#920
M2
ENDIF
#940=0 (change not completed)
G4 H0.1
IF #I8=0 THEN
  PRINT "error! low air pressure"
  M2
ENDIF
#7=#909 (remember G53..G59 origin)
G53
#0=(#950-1)*#968 (offset magazine slot X)
#1=(#950-1)*#969 (offset magazine slot Y)
G0 #976 (Z to safe position)
G0 X=#965-#970+#0 Y=#966-#971+#1 (safety distance to magazine)
#O13=1 (open magazine door)
G4 H5 UNTIL #I52=1
IF #I52=0 THEN
  PRINT "error! magazine not open"
  #909=#7
  M2
ENDIF

IF #950>0 THEN (drop old tool)
  G0 Z=#967 (to clamping height)
  G1 X=#965+#0 Y=#966+#1 F2000 (slow move sideways)
ENDIF
#O14=1 (open tool clamping)
G4 H1
G0 Z=#967-#972 (Z upward to safety level)
#0=(#920-1)*#968 (offset magazine slot X)
#1=(#920-1)*#969 (offset magazine slot Y)
IF T>0 THEN (pick new tool)
  IF #950=0 THEN
    G0 X=#965-#970+#0 Y=#966-#971+#1 (distance to magazine)
  ENDIF
  G0 X=#965+#0 Y=#966+#1 (above magazine)
  G0 Z=#967-#972 (safety distance)
  G1 Z=#967 F2000 (slowly move downward)
ENDIF
```

```
#O14=0 (close tool clamping)
G4 H1
G0 X=#965-#970+#0 Y=#966-#971+#1 (move sideways out of slot)
G0 #976 (Z to safety level)
IF #974<>0 THEN
  G0 X=#974 (X to safety distance if defined)
ENDIF
IF #975<>0 THEN
  G0 Y=#975 (Y to safety distance)
ENDIF
#O13=0 (close door)
#950=#920 (remember last tool)
#940=1 (change successfully completed)
#909=#7 (restore previous origin)
G4 H5 UNTIL #I53=1
IF #I53=0 THEN
  PRINT "error! door not closed"
  M2
ENDIF
RETURN
```

Simulation:

```
PRINT "simulated tool change T=";T
RETURN
```

If the spindle has sensors that indicate the correct end position of the clamping mechanism (fully ejected or clamped correctly), it is strongly recommended that you also query them and cancel the change in the event of incorrect conditions. Tool changer and spindle can be damaged if, for example, the clamping mechanism does not fully engage due to dirt particles. The queries were omitted in this example, because not all spindles have such sensors, and this would make the already complex macro even more confusing.

For the correct function of the macro, care must be taken that at least the signals # I52 (magazine flap open), # O13 (exit magazine flap open), # O14 (tool Release cone clamping) are defined. The position of the magazine, the distances of the individual tools, etc. are defined via the macro parameters in the Machine parameter dialog For example, if the tools in the magazine are parallel to the Y-axis, with T1 at the lowest Y-coordinate, T2 at the next largest, and so on, then the distance T2-T1 in the Y-direction (# 969) must be greater than zero and X direction (# 968) equals zero. The approach movement describes the movement in the direction of the tool retaining clip in the magazine. If the magazine is on the right side of the machine, i.e. if the tool is placed in the

magazine in the direction of larger X values, the approach movement in the X direction (# 970) must be positive, and in the Y direction (# 971) it must be zero ,

If, on the other hand, the tool is to be moved from top to bottom, the Z direction (# 972) must be negative and the X and Y directions equal 0. The safety position (# 974 to # 976) can either remain undefined (all values equal 0), if the approach movement alone is sufficient for the change. However, if a "prohibited area" is defined for the changer, it may be useful to perform additional movement after the change to leave that area.

### **23.7.2 Tool change with other magazine types**

By customizing the M6 macro, other types of magazines, such as revolver magazines, chain magazines, multi-row or multi-level magazines, and even complex strategies such as "chaotic" changers can be realized, where a particular tool is not always in the same place. However, this is associated with considerable programming effort and would go beyond the scope of this manual. On request, Benezan-Electronics can also create individual macros for special change systems against payment.

### **23.7.3 Tool length measurement with stationary probe**

With this method, a tool length sensor is installed at a fixed position in the machining area. For manual tool change without tool taper (collet directly on the spindle motor) or for automatic tool change with direct changer, a length measurement must be performed after every tool change. In these cases, the length measurement is best integrated into the changeable macro (M66 or M6).

When changing automatically with taper pins (SK, HSK or similar systems), a new length measurement only has to be made once when a new tool has been inserted in the socket, i.e. after a break or replacement due to wear. With normal, automatic change without opening the collets, the already stored length can be used from the tool table. The length measurement is in this case most conveniently created as a user macro and called only when needed.

```
#NAME G79.txt #LABEL "Tool length measurement"  
%  
#0=#960 (probe X position)  
#1=#961 (probe Y position)  
#2=#951 (lowest Z position)  
#3=#952 (approach feed rate)  
#4=#962 (offset Z)  
#5=#963 (safety position X)  
#6=#964 (safety position Y)
```

```
#7=#909 (remember G53..G59 origin)
IF #2=0 THEN
  PRINT "error! undefined macro parameters, please check
#951..#964"
  M2
ENDIF
IF T=0 THEN
  PRINT "error! no tool in spindle"
  M2
ENDIF
G53
G0 Z0
G0 X=#0 Y=#1
G1 Z=#2 F=#3 UNTIL #I4=1
IF #I4=0 THEN
  PRINT "error! probe switch not triggered"
  #909=#7
  M2
ENDIF
G1 Z=Z+2 F50 UNTIL #I4=0
IF #I4=1 THEN
  PRINT "error! probe switch not released"
  #909=#7
  M2
ENDIF
#925=Z+#4 (offset compensating probe height and collet)
PRINT " tool length = ";#925
L52 (save tool parameters)
G0 Z0 (move away from probe to avoid crash)
G0 X=#5 Y=#6
#909=#7 (restore old origin)
RETURN
```

Simulation:

```
PRINT "simulated length measurement"
PRINT " tool length = ";#925
RETURN
```

It is assumed that the length sensor is already connected to the hardware, and signal # I4 has been assigned to the corresponding pin (menu Configuration Machine -> Inputs/Outputs). The macro

---

parameters described at the beginning (# 960 etc.) must be defined (menu Configuration ->Machine -> Macros)

The X and Y coordinate of the probe position (# 960, # 961) must be entered so that the tool hits the center of the probe.

The lowest Z coordinate (# 951) indicates how far downwards maximum is travelled until the button must be pressed.

The approach speed (# 952) must be selected so that the braking distance is no longer than the travel reserve of the button (distance of the switching point to the stop), otherwise the Z-axis would not come to a standstill in time and the button will be damaged, Example: F1000 at an acceleration of  $0.5\text{m} / \text{s}^2$  results in a braking distance of 0.28mm. If in doubt, use the calculator for the axis configuration (menu -> configuration -> machine -> axis parameter -> button "open calculator")

The Z offset (# 962) should be specified so that the length is zero for the collet without tools. For example, if the collet on Z-92 is directly on the button, then # 962 = 92 should be entered.

At the end of the measurement, it should be moved to a position that prevents the probe from being damaged if a G0 Z ... command follows the survey. The exact position (# 963, # 964) is not important, it just has to be driven far enough away from the button.

### 23.7.4 height measurement with mobile probe

The measurement of the tool length can also be done with a movable probe. The tool spindle is positioned over the material and the button is placed under the tool on top of the material. During the measurement, the tool is used to move to the probe, and the workpiece zero point is set to the position of the tool tip minus the probe height. This compensates both the tool length and the thickness variations of the material at the same time.

Since the material height and the tool length are recorded simultaneously, it is not sensible to enter the length in the tool list since the measured height is only valid for the current material anyway. Therefore, only the current workpiece zero point is set. It is assumed that the length sensor is already connected to the hardware, and signal # I4 has been assigned to the corresponding pin (**menu Configuration -> Machine -> Inputs/Outputs**).

```
#NAME User01.txt #LABEL "Z-finder with mobile probe"  
%  
#2=#951 (lowest Z position)  
#3=#952 (approach feed rate)  
#4=#953 (offset Z)  
IF #2=0 THEN  
  PRINT "error! undefined macro parameters, please check  
#951..#964"
```

```
M2
ENDIF
#1=#909 (remember G53..G59 origin)
IF #1=0 THEN
  PRINT "no offset G54..G59 active"
  M2
ENDIF
G53
G1 F=#3 UNTIL #I4=1 Z=#2-#925
IF #I4=0 THEN
  PRINT "error! probe switch not triggered"
  #909=#1
  M2
ENDIF
G1 F=#3/10 UNTIL #I4=0 Z=Z+2
IF #I4=1 THEN
  PRINT "error! probe switch not released"
  #909=#1
  M2
ENDIF
#0=Z
G0 Z0 (move away from probe)
#909=#1 (restore old origin)
#902=#0-#4 (offset compensating probe height)
PRINT "offset G";53+#1;" Z=";#902
L53 (save offset)
RETURN
```

```
Simulation:
PRINT "simulated height probing"
RETURN
```

The above program text is best created as one of the user macros (User01 to User10). The macro parameters described at the beginning (# 951 etc.) must be defined (menu Configuration -> Machine -> Macros):

The lowest Z coordinate (# 951) indicates how far downwards maximum is travelled until the button must be pressed.

The approach speed (# 952) must be selected so that the braking distance is not longer than the travel reserve of the button (distance of the switching point to the stop), otherwise the Z-axis would not come to a standstill in time and the button will be damaged , Example: F1000 at an acceleration

of  $0.5 \text{ m} / \text{s}^2$  results in a braking distance of 0.28mm. If in doubt, use the calculator for the axis configuration (menu -> configuration -> machine -> axis parameter -> button „open calculator“)  
The Z offset (# 953) should correspond to the effective height of the probe, i.e. the distance from the base to the actuating surface at the switching point.

## 24 Special processing types

This chapter describes machining methods that require a strategy other than normal milling and possibly additional parameter settings. The parameters are set on the "Special" page in the configuration dialog (Menu - Configuration -Machine – Special).

### 24.1 Cylindrical engraving

#### 24.1.1 What is it?

Cylindrical engraving refers to milling or engraving work in which a flat drawing is wound on the circumference of a cylinder ("drum"). For this purpose, an axis of rotation (A or B) is required, in which the workpiece is clamped.

#### 24.1.2 Works without CAM

The special feature of this function is that no special CAM program for 4-axis machining is required. Drawing and NC program will be created as if the workpiece is flat. The software automatically converts the movements in the XY plane into movements of the X and A or Y and B axes.

In the simplest case, only the workpiece diameter needs to be known. Since there are no more Y motions with an A rotation axis, all machining takes place along a line parallel to the X axis and exactly perpendicular above the center of the rotation axis (analogous to the Y axis for a B axis). In the configuration dialog on the special page, you only have to select the orientation of the axis of rotation (A parallel to X or B parallel to Y), enter the workpiece diameter and activate the option "Position will be determined from current zero point". Of course, for this to work, the workpiece zero point must be on top of the workpiece above the center of the axis of rotation.

The actual cylindrical engraving function, i.e. the conversion of the Y- into A-resp. X and B coordinates are activated from the main page via the dialog "Toolpath Settings" (button next to the tool number). There, the workpiece diameter can also be changed later, without the machine configuration dialog having to be called.

### 24.1.3 Other work with rotary axis

Cylindrical engraving is very simple but has the limitations that you can only work with a tool perpendicular to the cylinder surface and a constant workpiece diameter. If these restrictions are unacceptable, for example, if you need angled holes that do not go through the center of the workpiece, or if you want to handle intricate three-dimensional contours of different diameters on the same workpiece, you must turn off the Cylindrical Engraving feature.

In this case, the position of the rotation axis should be entered explicitly as numerical values on the special page of the configuration dialog and the option "Position is determined from current zero point" should be switched off. The position of the rotary axis is then only relevant for the correct display of the graphic on the screen and has no influence on the movements of the machine. For the correct calculation of the rotation angle and feed values, the CAM software is responsible, which must output appropriate movements of the A or B axis. In this case, the workpiece zero point may also be outside the center (perpendicular to the axis of rotation).

## 24.2 Plasma cutting

### 24.2.1 What is it?

In plasma cutting, an electric arc torch is used to melt metallic material, which is then blown out of the cutting gap with a gas nozzle.

### 24.2.2 Particularities

As a special feature, first of all, a special immersion sequence is required at the beginning of each cutting process. Secondly, because of the deflection, the distance between the nozzle and the material must be kept as constant as possible, at least for thin sheets, by adjusting the Z height during cutting (THC = Torch height control).

**The height control is not currently supported by the KinetiC-NC control, but is in preparation.**

## 24.3 Tangential knife cutting

### 24.3.1 What is it?

In this machining method, a rotatable mounted knife is always aligned at the correct angle tangential to the tool path. The knife may be either rotatable but vertically fixed or oscillating. With the fixed knife usually slides are cut, with an oscillating knife and thicker materials such as foams can be cut.

### 24.3.2 Easy Installation

For the simplest case of a ready-made machine with positioning Z-axis and stepper motor 1: 1 driven knife drive there is a file "Tangential-Knife.ini", which contains all important standard parameters for the operation of the tangential knife, and their application described in the user manual. The following chapter explains additional details that are required for different configurations.

### 24.3.3 System requirements

The rotary axis for the knife (C-axis) should be designed for high dynamics and must be able to spin freely, so it must not have a stop. For referencing (zero setting after switching on), a reference switch must be available, which must be able to pass over in both directions (e.g. cam switch or light barrier). For the rotary drive should be easy and be directly connected 1: 1 or with a small ratio with the stepper or servo motor. Worm gears with high transmission ratio are unsuitable. As a rule of thumb, the drive should be able to set the adjustable max. Buckling angle (default  $5..10^\circ$ ) to be turned in a maximum of 100ms, including acceleration and deceleration. This corresponds approximately to a minimum rotational speed of  $60 / \text{min} = 1 / \text{s} = 360^\circ / \text{s}$  and an acceleration capacity of at least  $7200^\circ / \text{s}^2$ . Well-sized drives achieve even higher values, which allows a much smoother work flow.

### 24.3.4 Transmission choice

When selecting the gear ratio, care must be taken that the number of steps per revolution of the knife is an integer. This is always the case for translations of 1: N when N is a natural number. With "crooked" gear ratios (for example 2: 5), it depends on the stepper motor used or the encoder resolution for servomotors. The combination  $\ddot{U} = 2: 5$  and encoder resolution 512 bars would be for example inadmissible, because  $512 * \ddot{U} = 204,8$  is not an integer. 500 dashes would be ok.

### 24.3.5 What is supported?

For tangential knife cutting, only 2.5D programs are used. The actual cutting movement takes place only in the XY plane. The vertical axis is used only for piercing and lifting the knife. 3D or multi-axis machining is not supported, not even the combined use of tangential and drum engraving functions. Sloping, ramped grooving (G1 with XY and Z simultaneous downward movement) is permissible.

Vertical axes, which do not have a linear positioning drive, but can only perform fixed up / down movements (pneumatics or lifting magnet) are also possible. In this case, an adjustment of the pen-up and pen-down macros is necessary.

### 24.3.6 Parameter settings

The following parameters must be set for the operation of the tangential knife. The easiest way is to import the file "Tangential-Knife.ini" first, and then change only those parameters that do not match the default values.

Page	Parameter	Description
<i>General</i>	machine type	must be set to "Knife Cutting Machine"
<i>Axes parameters Axis „C“</i>	resolution	Steps / ° multiplied by 360 ° must be an integer
	Max. speed	at least 360 ° / s
	working acceleration	at least 720 ° / s <sup>2</sup>
	minute position	always -360 °
	Max. position	always + 360 °
	Reference switch at	"Min. Position within "or" max. Position within "
	Reference switch position	Must be set so that the knife is in the 0 ° position parallel to the X axis and the cutting edge points to X +.
<i>Special</i>	Max. Angle without stopping	If the angle between two consecutive lines is greater than this value, it is not driven through continuously but is first stopped, turned, and then moved on.
	Lift when angle>	If the angle between lines is greater than this value, it is completely lifted out, rotated, then immersed again and continued.

### 24.3.7 Track preview and excavation automatic

The controller performs an automatic motion optimization and traverses continuous lines if possible in one piece. There is a distinction between braking and stopping. If the angle between successive lines is still below the permissible value for the knife, but the acceleration caused by the kink overstretches the positioning drives of the X and Y drives, then braking is slowed down and after the kink it is accelerated again without dropping the knife. If the permissible limit angle is exceeded, it is first braked, the knife lifted (PenUp), the C-axis turned to the new angle, and then the knife again pierced (PenDown) and accelerated.

The recommended value for "Max. Angle without stopping" (see table above) is 5 to 10 °. This depends on the resolution of the drawing. Ideally, the drawing should contain only tangent transitions with rounded arcs in all places where you do not want to stop. Small angles <5 ° are usually unavoidable. Too high values well above 10 ° are not recommended because the machine's response time will be slower (more time to reserve rotational movement must be scheduled). If you have problems with jerky movements and frequent stops, first check your drawing. Often, dirty transitions with gaps or overlaps between the individual lines are the cause.

The recommended value for "Lift when angle>" is 15 to 45 °. For hard or delicate materials or wide blades rather less, otherwise the material may rip or even break the blade if too large angles are rotated within the material. Elastic materials and narrow blades tolerate a larger angle of rotation. The "Max. Angle without stopping" applies regardless of the setting "Exact stop when angle>" on the "Tools" page of the configuration dialog. It is recommended that you drive in Constant-Velocity-Modus if possible, to allow a smoother, smoother motion. Of course, the parameters for the CV mode on the extras page should be set sensibly, as with other types of processing.

### 24.3.8 Macros and programming

As already mentioned above, the control software can also insert automatic excavating and plunge movements in places where there is no movement of the Z axis in the NC program. For this, the controller must know what to do in this case. This is shown in the macros M14 for Pen-Down and M15 for Pen-Up. The height of the Z axis to be driven in these cases is stored in variables Q (plunge depth) and R (retract height).

**Example for M14-Macro:**

```
#NAME M14.txt #LABEL "Pen down"  
%  
R=Z (remember retract height)  
#0=F (remember feed rate)  
G1 Z=Q F=P  
F=#0 (restore old feed rate)  
RETURN
```

Since a different feed value can apply to the immersing movement than to the rest of the cutting process, the F value for grooving must also be stored in another variable (P). Otherwise, the following cutting movement would be performed after the (automatically inserted) grooving with the wrong feed value. It should be noted that the M14 and M15 macros are not allowed to perform movements of the X and Y axes, because this could confuse the automation of the control software and possibly trigger recursive calls of the macros. If the vertical axis is constructed pneumatically, the macro can be realized much easier:

**Example:**

```
#NAME M14.txt #LABEL "Pen down"  
%  
#O100=1 (solenoid on)  
G4 H1 UNTIL #I100=1 (wait until stop reached)  
RETURN
```

After switching over the pneumatic valve, either a constant time can be waited (G4 H0.3) to give the pneumatic cylinder time to move. Or wait as above for the input signal of a limit switch (G4 H1 UNTIL ...).

The actual NC program for cutting the workpiece (main program) can be structured differently. The first option is to set variables Q (plunge depth) and R (retract height) once at the beginning (if there is no pneumatic Z axis), and then explicitly call macros M14 and M15 for plunge-in and punch-out. Between M14 and M15, any number of G1, G2 and G3 commands can then follow. Before M14 or after M15 is positioned in rapid traverse with G0. Z motions are then not required in the program and are automatically executed by the macros. This type of programming is similar to commercial cutting plotters. It is very clear, but has the disadvantage that, in contrast to milling, a special post processor for the CAM program is required.

**Example:**

```
%  
Q=-5 R=15 P=2000  
G0 X100.751 Y47.673  
M14 (pen down)  
G1 X144.525 F1000 (cut rectangle)  
G1 Y6.38  
G1 X100.751  
G1 Y47.673  
M15 (pen up)  
G28 (park position)  
M30
```

After switching over the pneumatic valve, either a constant time can be waited (G4 H0.3) to give the pneumatic cylinder time to move. Or wait as above for the input signal of a limit switch (G4 H1 UNTIL ...).

The actual NC program for cutting the workpiece (main program) can be structured differently. The first option is to set variables Q (plunge depth) and R (retract height) once at the beginning (if there is no pneumatic Z axis), and then explicitly call macros M14 and M15 for plunge-in and punch-out. Between M14 and M15, any number of G1, G2 and G3 commands can then follow. Before M14 or after M15 is positioned in rapid traverse with G0. Z motions are then not required in the program and are automatically executed by the macros. This type of programming is similar to commercial cutting plotters. It is very clear, but has the disadvantage that, in contrast to milling, a special post processor for the CAM program is required.

**Example:**

```
%  
S15000 M3 (oscillation on)  
G0 X100.751 Y47.673 Z15  
G1 Z-5 F2000 (pen down)  
G1 X144.525 F1000 (cut rectangle)  
G1 Y6.38  
G1 X100.751  
G1 Y47.673  
G0 Z15 (pen up)  
M5 (oscillation off)  
G28 (park position)  
M30
```

## 24.4 Oscillating knife

The drive motor for an oscillating knife is treated like a spindle motor in milling mode and can be switched on and off with M3 as usual (see example above). The usual parameter settings on the *extras-page* (min./max spindle speed etc.) apply.

## **25 User management**

The program has a comprehensive user management, which allows to limit the user rights for a variety of functions. This makes it possible, for example, to create an operator who is only allowed to start programs, nothing else. This is useful in mass production to prevent accidentally changing settings by an operator or, for example, resetting the parts counter after producing rejects.

The administration of users is, of course, only possible if the current user has the right to manage the user, normally, one must be logged in as an administrator.

### **25.1 The users**

After installing the software 3 users exist.

### **25.2 Support-Admin:**

(requires password, please contact the Support of CNC-Step GmbH & Co. KG)

has all rights. This user may create new users and assign all rights, provided they do not have their own rights. So, for example, an administrator can copy himself and restrict the rights of his copy.

This copy may then continue to copy itself but can never assign the rights that the primary administrator removed from it.

The support Admin is allowed to reset the passwords of other users. But he can also copy an administrator who no longer has this right.

### **25.3 Customer-Admin:**

This Admin needs a password. By default, this is "1234", but can be changed.

This administrator allows to customize some machine parameters. It is recommended that you switch back to the default user after changing the settings on the machine to avoid accidentally changing settings.

### **25.4 Default users:**

(after installation without password)

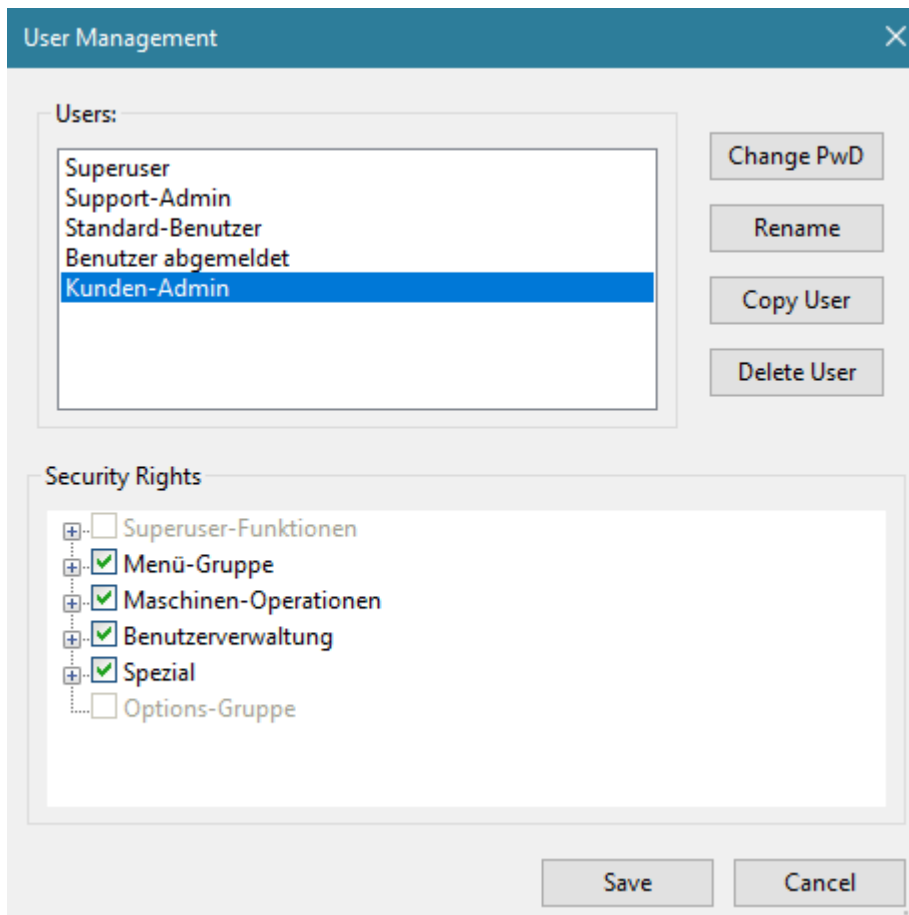
This user may operate the machine, he may not copy users or change their passwords. This user may not modify any tools, he may not change machine settings, such as speeds, accelerations, hotkeys, etc.

## 25.5 Logged out user:

(never has a password)

This user is not really a user. In this mode, the machine cannot be operated or configured. If a password-protected user logs in with an incorrect password, this "switched-off user" mode will be toggled to. The same thing happens when logging out of the user menu.

## 25.6 The administrative dialogue



The three system users that exist after installation cannot be renamed, modified, or even deleted. These users always remain in the system to quickly rebuild the entire system in the event of a failure.

### 25.6.1 Copy user

If you are logged in as an administrator, the button Copy user is activated.

To copy a user, select the desired user as a template in the "User:" window. If you have copied a user, all options for modification will be unlocked. After copying a user, the "Rename User" dialog opens automatically.

### **25.6.2 Rename user**

Copied users are given the abbreviation "(copy)" at the end to distinguish them. You can now give this user a meaningful name. However, the name must be unique, there cannot be duplicate names in the system.

### **25.6.3 Set / reset password**

After a user has been copied, he does not have a password. It is therefore advisable to assign a password. As an administrator you can also reset the passwords of other users, which makes sense if a user has forgotten his password. If you want to change your own password, you must first enter your current password to prevent anyone from unauthorizedly taking over the account by changing their password.

### **25.6.4 delete user**

Copied users can also be deleted by the administrator.

### **25.6.5 give out rights**

If a user (administrator, standard user or logged-off user) is copied, all his rights are also copied. This makes it possible to quickly create a set of identical users, or only a few changes are necessary.

The rights are listed in a tree structure. Many functions of the software can be released so sensitively. Due to the tree structure, it is also very easy to assign rights. For example, if there is a check mark in "File Menu" as in the example opposite, all options below are free. This greatly facilitates the selection and definition of the rights, since not every option has to be clicked on. For example, if you want to allow all functions except "Open MDI" from the "Special" section, then the check mark for "Special" should be removed and all options selected, except "Open MDI". When assigning the rights, however, you should be careful not to allow "nonsensical" combinations.

## 26 Customer service

For technical information please contact our customer service:

Address	CNC-STEP GmbH & Co. KG Siemensstraße 13-15 D-47608 Geldern	
Phone	+49 (0)2831/91021-50	(Mo. - Fr. 07.00 am - 03.15 pm)
Mobile	+49 (0)2831/91021-20 Only in urgent cases	(Mo. - Thu. 03.15 pm – 06.00 pm)
Fax	+49 (0)2831/91021-99	
E-Mail	support@cnc-step.de	
Web	<a href="https://www.cnc-step.com/">https://www.cnc-step.com/</a>	

If you have any questions, please contact our customer service by e-mail or telephone. We are happy to help.

Numerous suggestions and information can also be found on our website:

***<https://www.cnc-step.com/>***