

Translation



# Kinetic-NC User Manual

Software: Kinetic-NC with CncPOD and 2 LPT-Interfaces

Version: 1.74

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



powered by CNC-STEP.de

From 10.11.2018

# User Manual



## Short description

This manual is intended for users and machine operators. It explains the installation of the hardware and software, the initial commissioning and the basic steps in the daily handling of the machine. It is assumed that you already own a working milling machine or engraving machine that you have purchased together with the CNC control. In this case, the setting of the machine parameters is very easy and is limited to the selection of a pre-set parameters. The control can also be individually adapted to a large number of different machines. However, this requires the exact knowledge of the drive mechanism and the electrical wiring. For advanced users who want to run the control on their own designs or machines of other manufacturers, refer to the Reference Manual, which describes parameter setting and macro programming.

## INHALTSVERZEICHNIS

1	General Information .....	12
1.1	Icon description .....	13
1.2	Safety instructions .....	13
1.2.1	Safety symbols .....	13
1.3	System Requirements .....	14
1.4	Real-time requirements.....	14
2	Installation of „CncPOD“.....	16
2.1	Power supply .....	16
2.2	LPT-Ports.....	16
2.3	Network connection.....	17
3	Installation Software .....	19
3.1	Installation WinPCAP .....	19
3.1.1	License conditions.....	19
3.2	Installation main program KinetiC-NC .....	20
3.2.1	Install Updates.....	21
4	Initial Start-up .....	22
4.1	User information.....	22
4.2	Machine Selection -> Plug & Play .....	23
4.3	Establish connection .....	24
4.4	Demo mode.....	25
4.5	User login .....	25
4.5.1	No access?.....	26
4.6	Parameter settings.....	26
4.7	Individual adaptation .....	27
4.8	Function control.....	27
4.9	Reference switch.....	29
4.10	Test outputs .....	30
4.11	Create backup.....	31
4.12	Installation of additional options .....	31

---

4.13	Firmware Update .....	32
4.13.1	Are you sure? .....	33
4.13.2	Reconnect .....	33
5	Short introduction.....	34
5.1	Starting the Software .....	34
5.2	Log in as a user .....	35
5.2.1	Release of the functions .....	36
5.3	Distribution of the screen .....	37
5.4	The coordinates display .....	38
5.5	Load file.....	38
5.6	Errors .....	39
5.7	The 3D display.....	39
6	The operation.....	40
6.1	Set zero point .....	40
6.1.1	Where to put the zero point .....	40
6.1.2	The start up.....	40
6.2	Reference movement .....	41
6.3	Speed setting.....	41
6.4	The zero point symbol .....	42
6.4.1	Zero point and material .....	42
6.5	Slow feed .....	42
6.6	Presentation of the path .....	42
7	The user interface .....	43
7.1	The program screen .....	43
7.1.1	Graphic display .....	44
7.1.2	Mouse operation .....	44
7.1.3	The NC program window.....	45
7.1.4	The control elements .....	45
7.1.4.1	The symbols in detail .....	46
7.1.5	The feeds .....	47

---

7.1.6	Tools and track settings.....	47
7.1.7	Outputs and tool spindle.....	48
7.1.8	The coordinates display.....	48
7.1.9	Offsets .....	48
7.1.10	Set the coordinates .....	48
7.1.11	Feed and Speed display .....	49
7.1.12	Emergency stop and alarm .....	49
7.1.13	Message window .....	50
7.2	The setup screen .....	51
7.2.1	The jog buttons.....	51
7.2.2	Rounding .....	52
7.2.3	Zeroing display.....	52
7.2.4	The MDI.....	52
7.2.5	Softlimits .....	53
7.3	The diagnostic screen .....	54
7.3.1	Inputs and outputs .....	55
7.3.2	Lookahead-display .....	55
7.3.3	Hour meter.....	55
7.3.4	Part counter .....	55
7.4	The special screen.....	56
7.4.1	Application examples.....	56
8	The file menu .....	57
8.1	Opening NC-file .....	57
8.1.1	File extensions.....	57
8.1.2	Error while opening.....	58
8.2	Recently used files.....	58
8.3	Create NC file .....	59
8.4	Import drawing .....	60
8.4.1	Detailed description of the import filter.....	60
8.5	Import settings .....	60

---

8.5.1	Partial import.....	60
8.5.2	Restore a backup .....	61
8.5.3	Refurbishment .....	61
8.5.4	Install options .....	61
8.6	Export settings .....	62
8.6.1	Create backup.....	62
8.6.2	Select file.....	62
8.6.3	Create individual configurations .....	63
8.7	Exit.....	63
9	The configuration menu .....	64
9.1	Machine .....	64
9.2	Offsets.....	65
9.2.1	Copy Offset .....	65
9.3	Tools .....	66
9.3.1	Do I need a tool list? .....	66
9.3.2	Radius compensation and length compensation.....	66
9.3.3	Add new tools.....	67
9.3.4	Assign names.....	67
9.4	Hardware .....	68
9.5	Hotkeys.....	68
9.5.1	Assign keys .....	68
9.5.2	Jog-keys.....	69
9.6	Macros .....	69
9.6.1	Macro directory .....	69
9.7	User.....	70
9.7.1	Which password? .....	70
9.7.2	Forgot Password.....	70
9.8	Language .....	70
9.9	Calibration - PWM output .....	71
9.9.1	Testing.....	71

---

9.9.2	Laser Applications.....	71
9.10	GUI-Settings .....	72
9.10.1	Change G-code representation .....	72
9.10.2	Change power settings .....	73
9.10.3	Accuracy vs. resolution .....	73
10	The user-menu.....	74
10.1	Login.....	74
10.1.1	Automatic login.....	74
10.1.2	No access?.....	74
10.1.3	Lock the operation.....	75
10.2	Logout.....	75
11	The Support-Menu .....	76
11.1	User Registration data.....	76
11.1.1	Why a registration? .....	76
11.2	Updates .....	76
11.3	Send request .....	79
11.3.1	No Internet?.....	80
11.3.2	Check if everything has been sent.....	81
11.4	Logfile Options .....	81
11.4.1	Support .....	81
11.4.2	Important: Reset the log options .....	81
12	The help-menu.....	82
12.1	User manual .....	82
12.2	Reference manual.....	82
12.2.1	Environment .....	82
12.3	Help.....	83
12.3.1	Search.....	83
12.4	Information about.....	84
13	Typical work processes .....	85
13.1	Processing plate/sheet material .....	85

---

13.1.1	Advantages .....	85
13.1.2	Loading program .....	85
13.1.3	The message window .....	87
13.1.4	Error messages .....	87
13.1.5	Presentation in the graphic .....	88
13.1.6	Set zero point.....	88
13.1.7	Place the material .....	89
13.1.8	Homing.....	89
13.1.9	Zero adjustment.....	89
13.1.10	Starting the program .....	90
13.1.11	Adjust feed.....	90
13.1.12	Problems? How to stop? .....	91
13.1.13	Continue again .....	91
13.1.14	Start program at any point .....	92
13.1.15	Additional tip.....	92
13.1.16	Execution time .....	92
13.2	Statistical functions .....	93
13.2.1	Running-time meter.....	93
13.2.2	Parts counter .....	93
13.2.3	Tool path settings .....	93
13.3	Other material .....	94
13.3.1	Position of the workpiece .....	94
13.3.2	Probe.....	94
13.3.3	3D-probe.....	94
13.3.4	Camera-centering .....	95
13.4	Working with height probe .....	95
13.4.1	What is a height probe? .....	95
13.4.2	The procedure.....	95
13.4.3	Connection .....	95
13.4.4	Testing before use .....	96



---

13.4.5	Height of the measuring probe .....	97
13.5	Tool management.....	98
13.5.1	Without tool change .....	98
13.5.1.1	Distribution of the program .....	98
13.5.2	Manual tool change (direct) .....	99
13.5.3	Directly used tools.....	99
13.6	Tool length sensor.....	99
13.6.1	Without tool length sensor .....	99
13.6.2	With tool length sensor .....	100
13.6.3	G40/G43 Mode.....	101
13.7	Manual change with tool holders.....	101
13.7.1	Advantage.....	101
13.7.2	How is measured? .....	101
13.7.3	Simple entry in the list .....	102
13.8	Automatic tool change (direct) .....	102
13.8.1	Magazine types.....	102
13.8.2	The M6 and G79 macro .....	102
13.9	Automatic change with tool holders .....	103
13.10	Rotary axis and cylindrical engraving .....	103
13.10.1	Cylindrical engraving .....	104
13.10.2	Simply without CAM .....	104
13.10.3	Where is the rotary axis mounted? .....	104
13.10.4	Insert workpiece .....	105
13.10.5	Probing the workpiece .....	105
13.10.6	Load NC program.....	105
13.10.7	Turn off drum engraving .....	105
13.11	Tangential knife cutting .....	106
13.11.1	What is it?.....	106
13.11.2	Import of option.....	106
13.11.3	Backup.....	106

13.11.4	Checking the knife.....	106
13.11.5	Reference switch and homing.....	107
13.11.6	No CAM program necessary .....	107
13.11.7	Sharp corners or not.....	107
13.11.8	Clean drawings .....	107
13.11.9	Control of the drawing.....	108
13.11.10	Presentation in the graphic.....	108
14	Troubleshooting .....	109
14.1	Functions locked .....	109
14.2	Connection interrupted.....	110
14.3	Error messages .....	111
14.4	The machine does not react as expected .....	111
14.5	Security door .....	111
14.6	Maintenance mode .....	112
15	Import filter .....	113
15.1	DXF-Import filter.....	113
15.1.1	What is imported? .....	113
15.1.2	The milling direction.....	113
15.1.3	The import process:.....	114
15.1.4	Tab: Layer defaults.....	114
15.1.5	Tab: Global Settings.....	115
15.1.6	Settings for milling.....	116
15.1.7	Settings for drilling.....	116
15.1.8	Tab: Settings.....	117
15.1.9	Z-values of material and machine .....	117
15.1.10	Material top: .....	117
15.1.11	General settings: .....	118
15.1.12	Layer-Details (Layer List):.....	119
15.1.13	Advanced Editing / Path Details:.....	119
15.1.14	Double lines .....	120

---

15.1.15	Navigate the graphic .....	120
15.1.16	Optimize the drawing file .....	120
15.1.17	The conversion: .....	120
15.1.18	Save the file .....	121
15.1.19	Special features when importing from DXF files.....	121
15.1.20	Note for experienced users .....	121
15.1.21	No existing import data or message "Error while parsing the DXF file, wrong version?"	122
15.2	HPGL-Import filter .....	123
15.2.1	Short description .....	123
15.2.2	Language supported: .....	123
15.2.3	The import process .....	123
15.2.4	Output file .....	124
15.2.5	Customizing the import process .....	124
15.2.6	Z-depth .....	125
15.2.7	Tools.....	125
15.2.8	Feed.....	125
15.2.9	Zero Offset .....	125
15.2.10	General settings .....	126
16	Customer service .....	127

## 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



## 1.3 System Requirements



To operate the CncPOD and Kinetic-NC control software, you need at least the following equipment:

- A PC with Windows operating system (XP, Windows7 or later), laptops, netbooks or similar devices are not recommended
- At least 1GHz processor frequency, min. 1GB of RAM and 2GB of free hard disk space
- A graphics card (or onboard chipset) with 3D hardware acceleration
- A network interface according to IEEE802.3 with min. 10MBit (10Base-T), if the network is also to be used for file, internet or other services, min. 100MBit (100Base-TX) and additionally a network switch or a second interface
- A 5V power adapter with mini USB cable
- A CNC milling machine or engraving machine with stepper motor drives or servos with step / direction signals and associated control electronics (e.g. Zero 3)
- A network cable (Cat 5 or 6) with RJ45 connectors for connecting the CncPOD to the PC
- One SUBD-25 cable (male to female) to connect the CncPOD to the stepper motor controller

## 1.4 Real-time requirements



In order for the PC to be used as a control computer, it must be "real-time capable" to a limited extent, i.e. can respond reliably to signals from the machine within the shortest possible reaction time. With Kinetic-NC control, all time-critical signals are generated with external hardware - the CncPod - instead of directly with the PC. The requirements are therefore not as high as with a CNC software without hardware support such as Mach3 with operation via the PC's internal LPT port. Nevertheless, a few things should be considered to ensure a smooth working environment.

## **Desktop PC or laptop?**

As a control computer, if possible, a desktop PC or similar device (industrial, panel or box PC) should be used and no laptop or similar portable, battery-powered device. Although laptops tend to have decent processing power these days, they are designed for low power consumption.

Unexpectedly switching to energy-saving mode may result in sudden program delays. It requires a good knowledge of the operating system to disable all energy-saving options, and to make a portable device operational for CNC operation.

## **Virus Scanner**

Another problem can be virus scanners or automatic updates, which are started automatically at a certain time, and cause sudden network load or changes to the operating system. While surfing the Internet or transferring files on the corporate network itself does not interfere with CNC operation, the background actions of an anti-virus program that it causes can cause problems. It is therefore recommended to disable problematic programs such as virus scanners, expensive screen savers and the like, and to refrain from surfing the Internet while the CNC machine is running.

## **No step losses**

Nevertheless, if an unexpected delay should occur, this will not be a problem for the Kinetic-NC controller. While with other programs such as Mach3 even a delay of a few microseconds can have fatal consequences (step losses, possible collisions), the machine with the Kinetic-NC simply stops briefly (with the correct braking ramp, of course) and drives normally once the problem has been resolved. This is usually harmless but can still lead to unwanted effects in certain cases (chatter marks in case of metal or burns in woodworking or plasma cutting).

## **2 Installation of „CncPOD“**

The following description of the installation procedure assumes that the machine and the rest of the electronics (PC and Zero3 control, for example) are already operational.

### **2.1 Power supply**

The CncPod requires a 5V power supply with USB port for power supply. To connect the power supply to the CncPOD, a USB cable with a mini-USB connector is used. Please always use an extra power adapter, and do not connect the CncPOD to the PC with a USB cable. Although this also works for test purposes, the galvanic isolation of the network connection would be ineffective, making the controller unnecessarily susceptible to electromagnetic interference. The USB port is for power only. A data transfer with USB is not provided.

### **2.2 LPT-Ports**

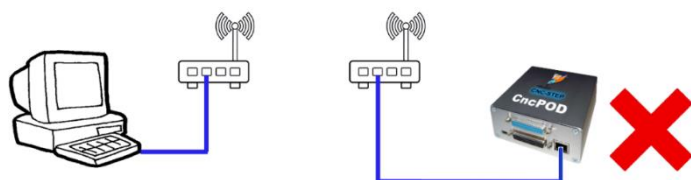
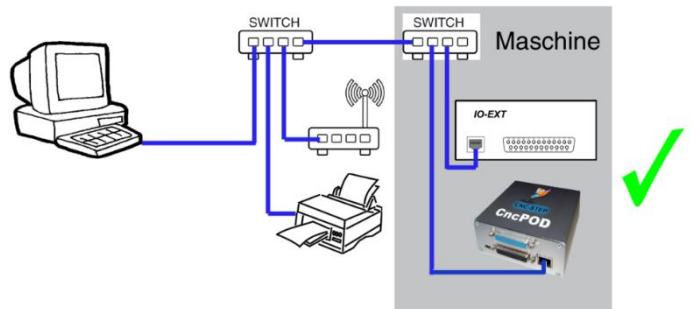
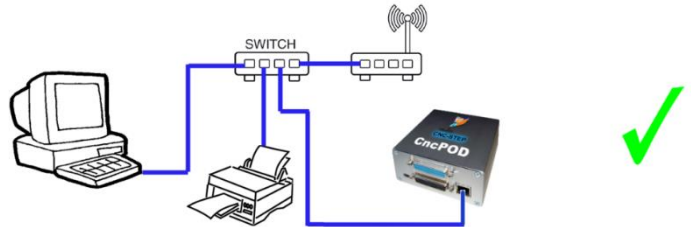
A 25-pin SUB-D cable connects the CncPod to the machine's control electronics (Zero3 or Raptor cabinet, for example). At port 1, the step signals of the first 4 axes and the switch and relay signals are available. At port 2, extensions such as Tool changer, frequency converter or a laser module can be connected.



## 2.3 Network connection

The CncPod is connected to the PC via the network connection for data transmission. Unlike competitor products, you do not need a dedicated PC port, so you do not need to install a second network card in the PC, but you can use the existing network infrastructure, and you can also share the CncPOD with other devices on a network. Connect switch. If there is a larger home or office network with multiple PCs, the following rules should be followed to avoid bottlenecks:

- The connection to the PC should always have the fastest transmission rate in the network. So if the PC has a GBit interface and the DSL router only 100MBit, then the switch should also 1GBit, and the CncPOD be connected directly to this switch. This ensures that there is no bottleneck, and data transfer between other PCs does not slow down communication with the CncPod.



- Only "unmanaged switches" may be used. Routers with "intelligent" functions (firewalls, proxy ...) that filter logs do not work. Of course, this only applies to the connection path from the control PC to the CncPod. Outside this route, any devices may be connected to the network.
- Wireless connections are not supported, as well as modems or external USB network adapters that "tunnel" Ethernet over other protocols and are associated with time delays. (This again applies only to the PC to CncPOD link)

Windows 8 and 10, and especially portable devices, may experience problems because the operating system shuts off the network connection after a few minutes of inactivity (sleep mode). This can be prevented if the CncPod is connected via a switch even if it is the only device.



The second switch on the penultimate picture on the right is therefore permitted because only devices of the CNC machine are connected to it. Thus, the connection between the two switches cannot be overloaded by foreign traffic and become a bottleneck.

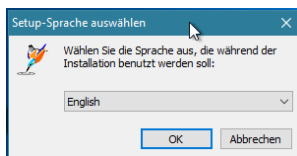
As already described in the chapter "System requirements - Real-time requirements", it is very important for the control computer that sufficient computing power is available at all times, and the PC is not slowed down by background programs. Therefore, you should make sure,

- to remove all unnecessary programs
- to disable automatic updates
- If it is a laptop (expressly not recommended), always power the device and disable all energy-saving options
- Not to use a virus scanner. If this is unavoidable, make sure the KinetiC-NC software's anti-virus software grants free access to the network interface and does not perform any elaborate scans while the machine is running.
- During the installation (especially with WinPcap) prompts may come through the firewall or the virus scanner. In any case, you must allow access.

## 3 Installation Software

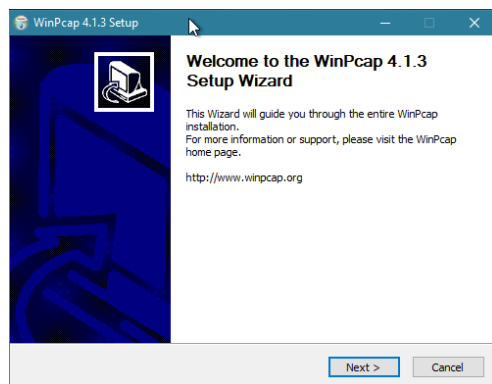
### Starting the Installation

The software is usually delivered on a CD or a USB stick. When you insert the CD into the drive, the installation starts automatically. For a USB stick or if the automatic start does not work, or you have downloaded the software from the Internet, please start the program "KinetiC-NC\_Setup.exe" manually.



After starting the setup program, you will be prompted to select the language of the installation wizard. Confirm your selection with OK. *(Depends on the language of your Windows OS this first dialog can be in a different language)*

### 3.1 Installation WinPCAP



You will first be prompted to install "WinPCAP". This is a kind of universal driver software for network controllers and is required for communication of the KinetiC-NC software with the CncPod.

Start the installation of WinPcap by pressing the "Next>" button.

#### 3.1.1 License conditions

You must agree to the license terms of the WinPcap software by pressing the "I Agree" button to continue the installation.

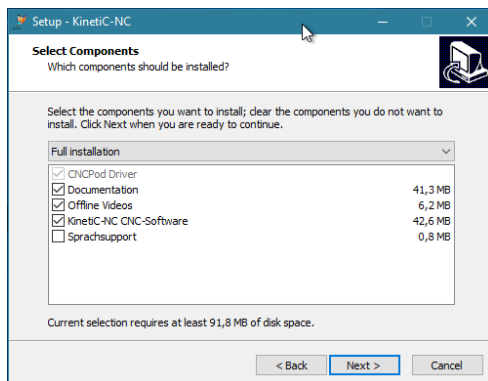
Please make sure the tick in the box is checked (as in the picture above). If this is not the case, the KinetiC-NC software will not work properly.

You have successfully completed the installation of WinPcap. Confirm this with the button "Finish".

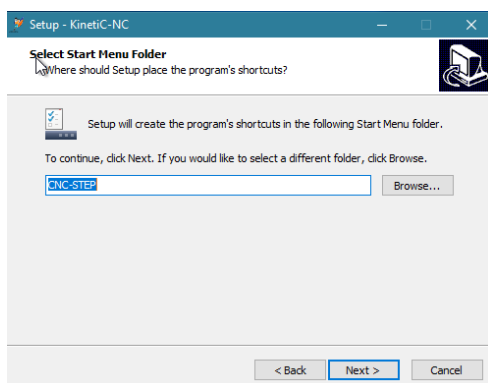
Continue the installation by confirming the "Next" button.

After having read and understood the license agreement, you accept it by ticking the option "I accept the agreement" and confirm with the "Next" button.

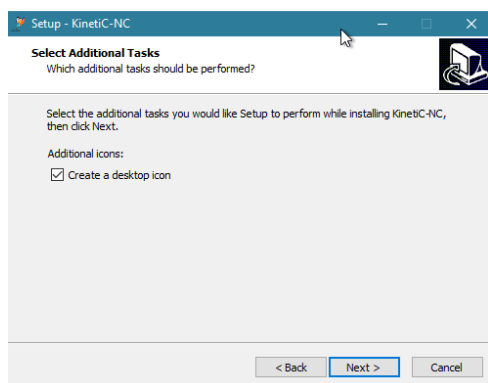
## 3.2 Installation main program KinetiC-NC



In the following dialog you can select the type of installation. It is recommended not to change the settings. However, if you would like to contribute to the further development of the program and speak a native language that is not yet supported, you may also like to install the language support. This allows you to create your own translations of the software and share it with other users.



The KinetiC-NC software is installed in the suggested folder. If you would like to rename the entry, you can do so now.



By ticking the box, you can install an icon on the desktop to quickly start the software.

The installer is now fully configured, and you are ready to install the software.

### 3.2.1 Install Updates

If you are updating an existing installation, you may be asked if existing files should be overwritten. Select "No" if you want to keep existing settings or "Yes" if you want to reset all parameters to the factory settings.

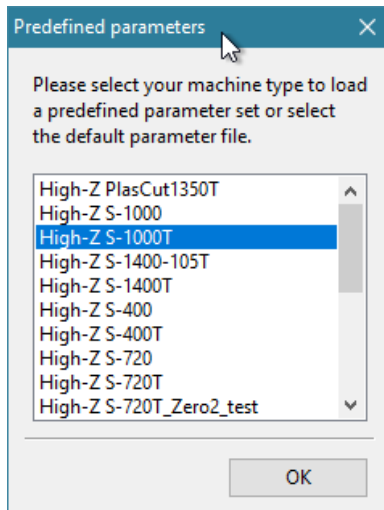
Although normally no settings (unsolicited) are overwritten when installing an update, it is strongly advised to back up the machine parameters before each update. This ensures that you can return to the old state in the event that something does not work as expected.

After completing the installation wizard, there will be a new "Kinetic-NC" entry in the Windows Start menu and, if selected, an icon on the desktop.

## 4 Initial Start-up

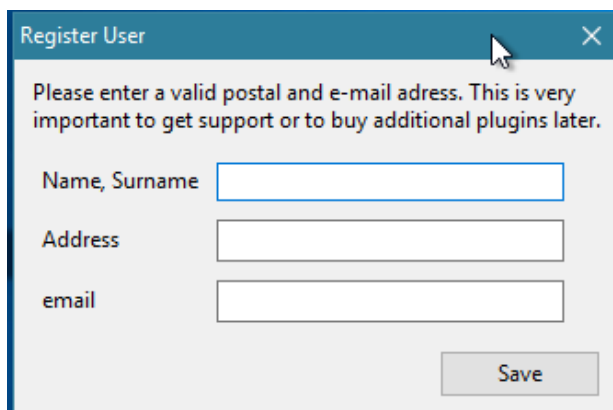
The first time you start the software, you will be asked for the language first .

Then a list of known machines is displayed, for which there are already predefined parameter.



### 4.1 User information

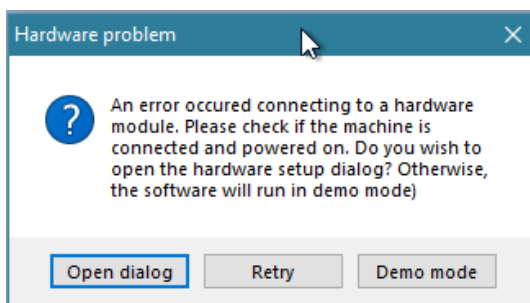
During the first start of the software its necessary to enter the user details. This can be important if you need support or you want to add additional options.



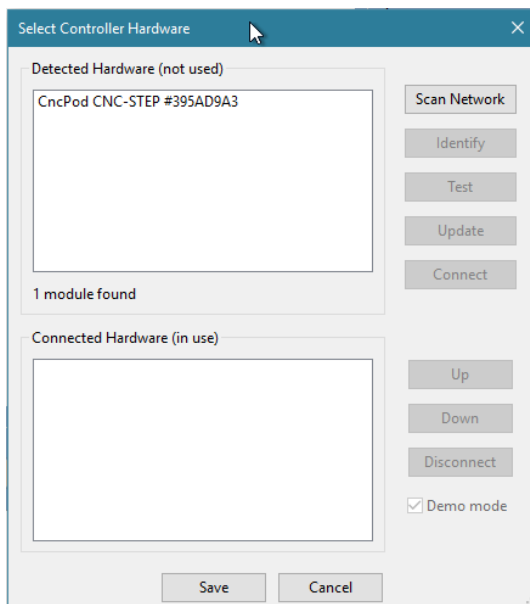
## 4.2 Machine Selection -> Plug & Play

If you purchased the software together with a machine, chose the machine type that best suits yours. Later, you will no longer need to perform any manual parameter settings, except perhaps to install additional purchased options.

If your machine does not match any of the models in the list, please select "CncPod Default Settings". You then have to enter at least the most important parameters such as travel distances, maximum speeds and resolutions of the axes themselves. For special machines or self-construction, refer to the optional Reference Manual for Advanced Users, as the detailed description of all setting options would go beyond the scope of this manual. In the following, only the most important features are explained.

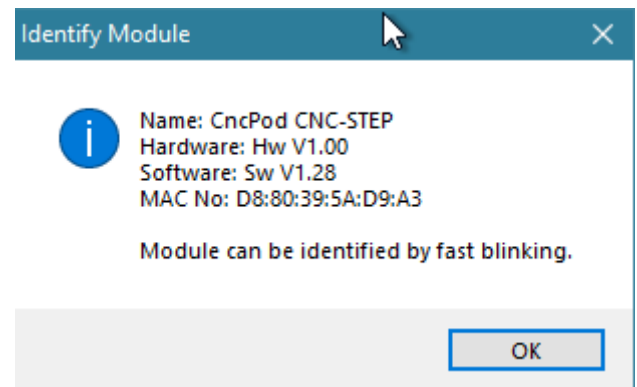
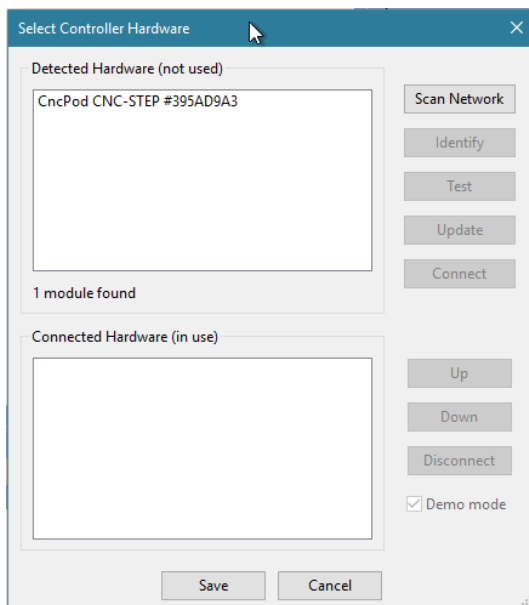


After selecting the machine model, the main window of the software opens. It is usually recommended to always turn on the machine before starting the software. If this has not already been done or there are problems with the connection, the following dialog appears:



Turn the machine on and click on the button "Scan Network". The hardware selection dialog will then appear as shown on the bottom right. This is necessary because the CncPod does not have an "exclusive" port but shares the infrastructure of the possibly larger computer network. It is possible that several machines are connected in a network, or that a machine in addition to the CncPod also uses other hardware modules. Each module has a unique number (MAC address), so that the software can remember the assignment of "its" modules.

If everything is connected correctly, in the upper part of the window under "detected hardware (unused)" the CncPod with a number similar to the one in the picture should be displayed. If nothing is displayed there, check that the CncPod with the network cable is properly connected to the PC and is powered by the USB power supply. At least the right, green LED on the network connection of the CncPOD should light up (connection status). With "Search net" you can redisplay the list. If several CncPODs are listed, which can be found on several existing machines in a larger network, you can use "Identification" to find out which one is the right one. The currently selected CncPOD is to be recognized by its flashing left, yellow Traffic LED.



### 4.3 Establish connection

After selecting CncPod in the list, you can connect using "connect". The CncPOD is then displayed in the lower part under "Connected hardware (used)".



## 4.4 Demo mode

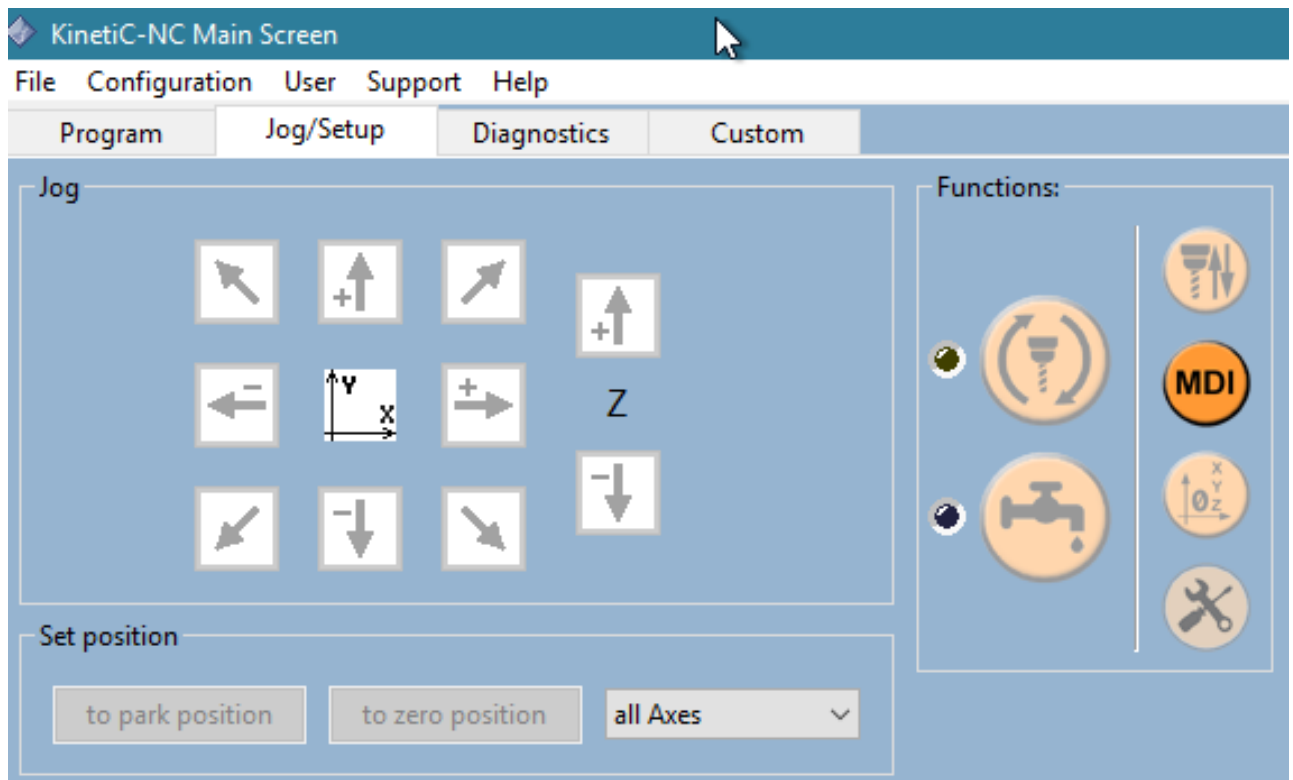
As long as the bottom list is empty, the software can only be run in demo mode. In this case, you can try almost all the features of the software, but it will not do any machine movement, just simulate it on the screen. Of course, a hardware should be connected for the subsequent commissioning, and the demo mode should be deactivated.

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. If you are asked to do so, please follow the instructions in chapter "*Updating the machine firmware*".

The installation is now complete, and you can *log in* for the first time.

## 4.5 User login

Not all functions of the software are allowed at any time. It may happen that certain buttons and menu items are locked. In the picture below, for example, the machine movement, spindle and coolant function and tool change and homing are disabled ("grayed out").



All machine movements are of course blocked as long as the emergency stop is active. Regardless of the condition of the machine, there is another reason why functions may be limited.

### 4.5.1 No access?

If you do not have access to certain buttons or menu items, make sure which user you are currently logged in with. This is indicated in the status bar at the bottom of the main window (e.g., "User: Default User"). If there is "User: logged off", you must first log in to have access to machine functions.

To do this, open the login dialog via the menu-> user-> login ... A small dialog opens in which the desired user can be selected. In the Password field, the corresponding password is entered and confirmed with "Enter".

The default user has sufficient rights to work with the machine. By default, he has not given a password, so you just have to confirm with "Enter" to close the dialog and log in. Users without a password will automatically be logged in the next time the program starts without the dialog opening. The default user may not change any settings and machine parameters. The user "Client-Admin" has the beginning password "1234". This administrator has the right to change machine settings. On request, users can be assigned passwords to protect the machine against misuse (see chapter *User management*).

If you leave the workplace without switching off the machine, you can also log out for security reasons by clicking "Menu-> User-> Logout ...".

The user is then logged off and the machine can no longer be operated.

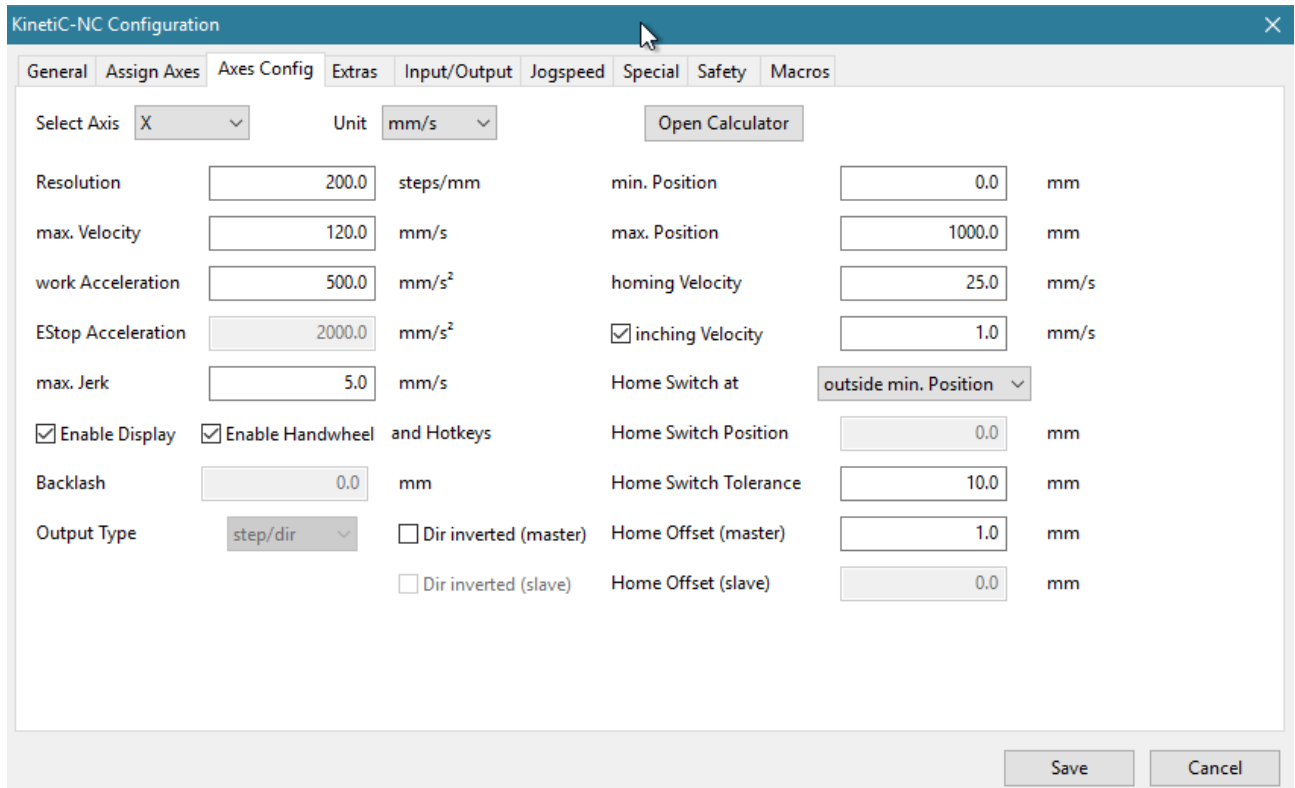
## 4.6 Parameter settings

If you purchased the software together with a finished machine and selected the appropriate model as described above when you first started the software, no further parameter setting is required, and you can skip this chapter.

In case you have made a mistake in the selection, or you want to adapt the software later to another machine, you can also undo the selection and load the parameter set for another machine model. Open the import dialog (File menu -> Import settings) and then the file selection dialog by pressing the "Default parameter ..." button.

## 4.7 Individual adaptation

If your machine does not meet the standard parameters, you must now enter the parameters manually. If you have not received information from the manufacturer of the machine, contact support or (only if you are really comfortable with it) follow the instructions in the Controller *Reference Manual*.



KinetiC-NC Configuration

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

Select Axis X Unit mm/s Open Calculator

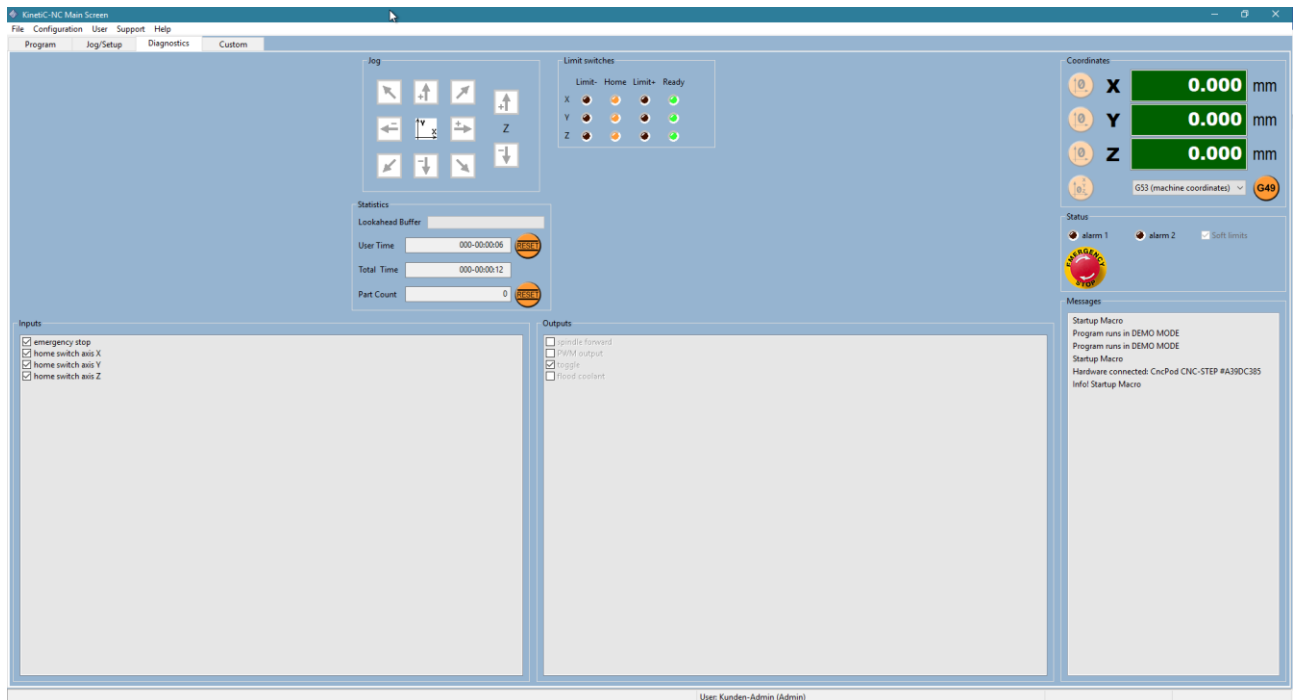
Resolution	200.0	steps/mm	min. Position	0.0	mm
max. Velocity	120.0	mm/s	max. Position	1000.0	mm
work Acceleration	500.0	mm/s <sup>2</sup>	homing Velocity	25.0	mm/s
EStop Acceleration	2000.0	mm/s <sup>2</sup>	<input checked="" type="checkbox"/> inching Velocity	1.0	mm/s
max. Jerk	5.0	mm/s	Home Switch at	outside min. Position	
<input checked="" type="checkbox"/> Enable Display	<input checked="" type="checkbox"/> Enable Handwheel and Hotkeys		Home Switch Position	0.0	mm
Backlash	0.0	mm	Home Switch Tolerance	10.0	mm
Output Type	step/dir	<input type="checkbox"/> Dir inverted (master)	Home Offset (master)	1.0	mm
		<input type="checkbox"/> Dir inverted (slave)	Home Offset (slave)	0.0	mm

Save Cancel

## 4.8 Function control



In the main window, switch to the "Diagnostics" page. First, you should test the function of the emergency stop button. When the switch is pressed, the symbol with the red / yellow button labelled "Emergency Stop" must appear on the right in the "Status" field. After unlocking the switch, it must disappear and an "OK" symbol will appear. If your machine has security doors or a high frequency spindle with a frequency converter, other symbols may appear depending on their condition.



If the DEMO symbol appears instead, there is (still) no connection to the hardware. Please check this in the menu "**Configuration -> Hardware**". The symbol with the plug drawn on the far right indicates that a connection existed but was subsequently disconnected. This may not only be due to the cable connection itself but also to the operating system (anti-virus, firewall, energy-saving mode, see also chapter "**Installing the CncPOD**").

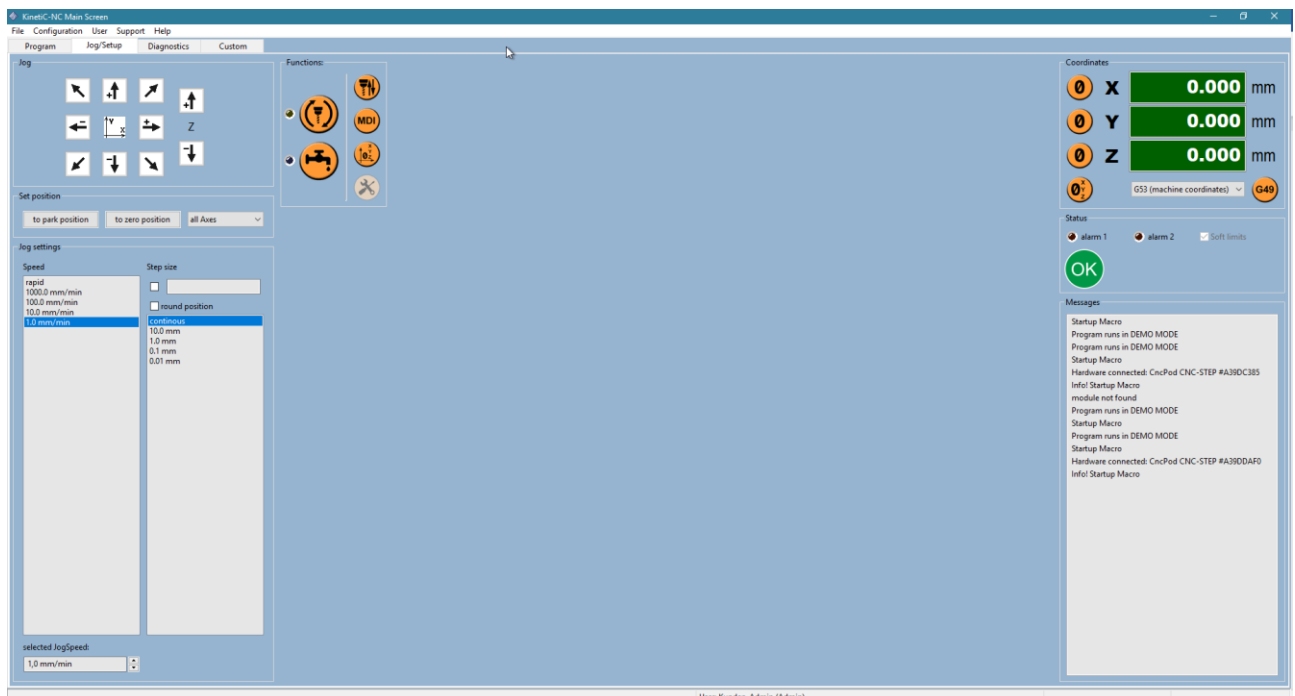
The triangular warning labels indicate a problem with the safety door, the frequency converter or other external equipment. For information about how to resolve the issues in these cases, refer to the instructions in the appropriate extension. If the "Emergency Stop" symbol does not disappear, check whether all emergency stop switches are actually unlocked and whether all connections to the controller and machine are plugged in correctly.

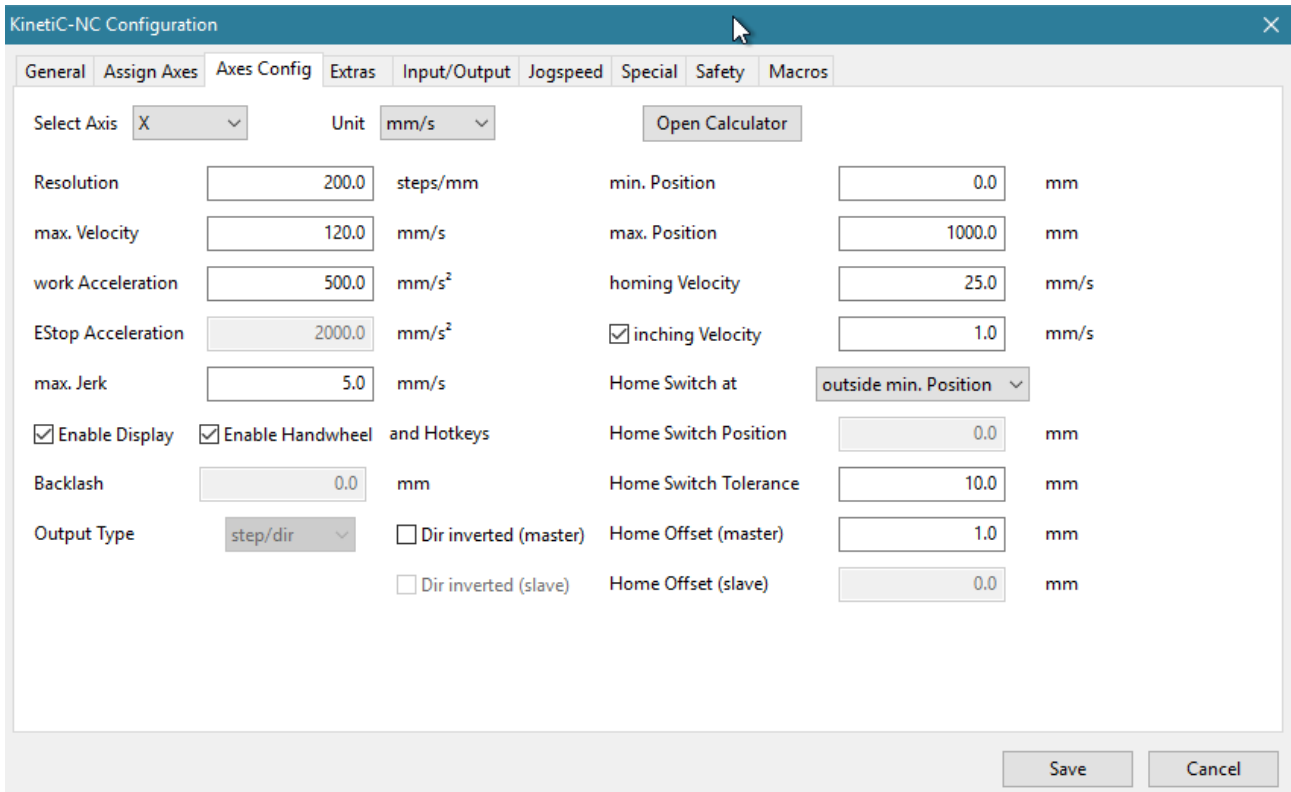
If the lists of inputs and outputs on the diagnostics page (large picture above) are empty, this can be remedied by resetting the input / output pin assignment. The default assignment of the inputs and outputs can be restored with Menu -> File -> Import Settings by selecting the default parameters, deactivating all check marks except "Inputs / Outputs" and then loading with "Open".

## 4.9 Reference switch

Next, please check the function of the reference switch. If the switches are not actuated, i.e. the axes are within the normal travel range, the indicator lights "Ref." On the diagnostics page must be off. You have to get started if you operate the switches by hand. For inductive switches you have to hold a piece of metal in front of the switch, for reed switches a magnet. Now it must be checked whether the directions of rotation of the drive motors are correct.

Go to the page "Setup". When pressing the arrow buttons, the associated axis must move in the right direction. Depending on the position of the machine and the line of sight of the operator, the actual direction (front / rear, right / left) may differ from the direction of the arrows. However, the coordinate display counting mode must be up when the tool moves away from the home switch and vice versa. With the Z axis, the tool moves up, that is, away from the workpiece when the coordinate display counts up.





KinetiC-NC Configuration

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

Select Axis X Unit mm/s Open Calculator

Resolution	200.0	steps/mm	min. Position	0.0	mm
max. Velocity	120.0	mm/s	max. Position	1000.0	mm
work Acceleration	500.0	mm/s <sup>2</sup>	homing Velocity	25.0	mm/s
EStop Acceleration	2000.0	mm/s <sup>2</sup>	<input checked="" type="checkbox"/> inching Velocity	1.0	mm/s
max. Jerk	5.0	mm/s	Home Switch at	outside min. Position	
<input checked="" type="checkbox"/> Enable Display	<input checked="" type="checkbox"/> Enable Handwheel and Hotkeys		Home Switch Position	0.0	mm
Backlash	0.0	mm	Home Switch Tolerance	10.0	mm
Output Type	step/dir	<input type="checkbox"/> Dir inverted (master)	Home Offset (master)	1.0	mm
		<input type="checkbox"/> Dir inverted (slave)	Home Offset (slave)	0.0	mm

Save Cancel

If a drive moves in the wrong direction, this can be corrected with "Inverted direction" in the axis parameters.

If the reference switches are working and the directions of all drives are correct, you can start a homing run with the "All Ref" button on the diagnostics page. If you have to cancel the homing because there is still something wrong, you can either do this by pressing the button again or pressing the ESC key.

## 4.10 Test outputs

Finally, you can test the function of the spindle and coolant outlets (or dust extraction). To do this, click on the corresponding boxes in the "Outputs" field on the diagnostics page, e.g. "Spindle forward" or "coolant".

## 4.11 Create backup



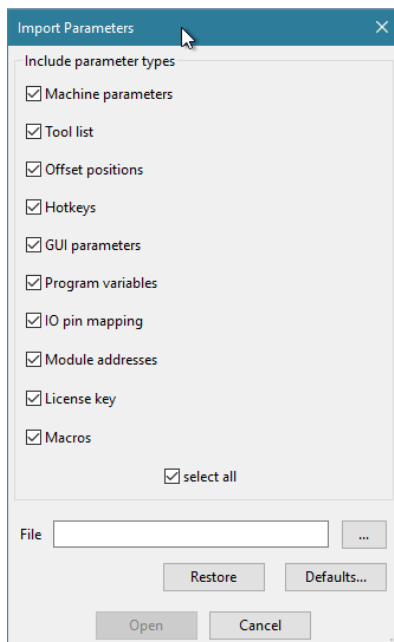
After you have set all parameters, it is advisable to make a backup of the settings. In the event that data loss occurs due to damage to the PC, or you make erroneous changes, the original parameters can then be easily recovered, and you do not have to redo all settings.

Details can be found in the chapter "*The File Menu*".

## 4.12 Installation of additional options

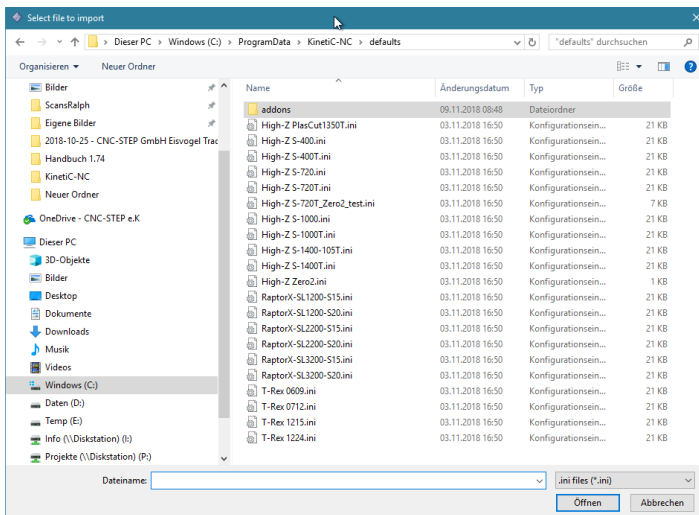


If, in addition to the basic equipment, you have purchased additional options for your machine, it may be that the machine parameters deviate from the standard settings and must be supplemented. Please refer to the instructions in the option to see if this is required.



In this case, a file is attached, or a link specified where you can download the file. You can import this file by selecting "Import settings" in the File menu.

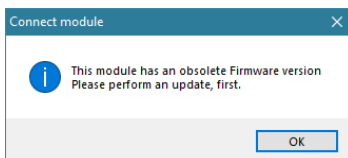
If you click on the button with the three dots to the right of the file name, you can select the file.



Then press "Open" in the file selection dialog, and then again "Open" in the import dialog, and then again "Open" in the import dialog. Make sure that in the user management the rights to use such an additional option can be set separately for each user.

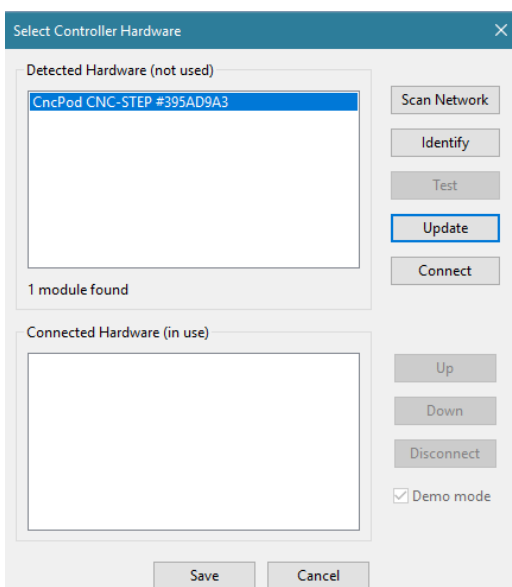
## 4.13 Firmware Update

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 happens automatically.



First open the hardware dialog via menu-> configuration-> hardware.

The following dialog opens:

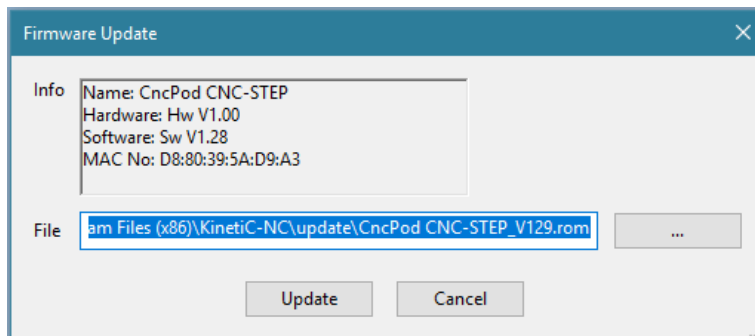


The module to be updated must be in the upper part of the list. If this is not the case, but the module is in the lower part, as in the picture on the right, select the module and then confirm the "Disconnect" button. This disconnects the machine from the software and the module is placed in the top list as unused.

Click on the button "Update" to start the update of the module.

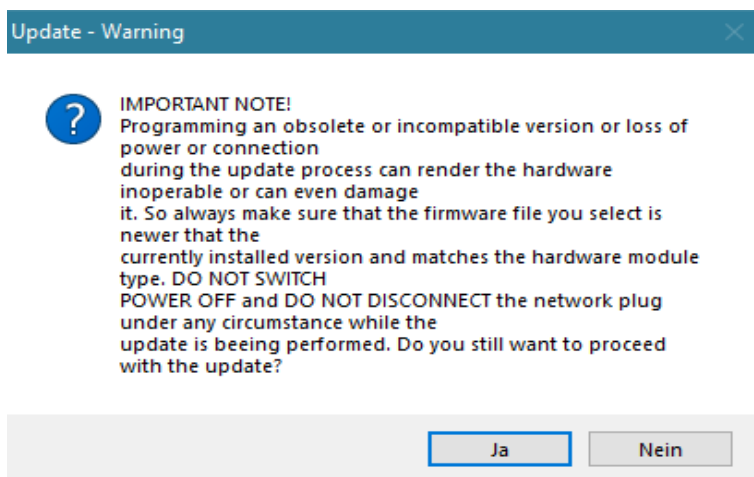


In the upper part of the dialog box you can make sure once again whether you have also selected the correct module. Then select an update file by clicking on the button with the three dots. Choose the one with the highest version number that matches the hardware module unless you have received different instructions from support.



#### 4.13.1 Are you sure?

A warning message appears. Please read this carefully and only confirm with "yes" if you are absolutely sure that the connection cannot be interrupted during the update. In particular, never carry out updates with laptops, notebooks or other battery-powered devices, as these could unintentionally turn off the network connection (power-saving mode).



During the update, do not turn off the machine or disconnect the network cable. This could irreparably damage the module.

#### 4.13.2 Reconnect

All you have to do is follow the instructions on the screen. After transferring the firmware file, it is important to wait until the alternating flashing of the CncPOD's LEDs stops. Only then restart the CNCPOD by briefly disconnecting and reconnecting the power supply (USB).

Now select the module again and connect it by confirming the button "Connect".

After the hardware module is successfully connected, click on "Save"

The hardware dialog closes and the message "Hardware connected: CncPod ..." should appear in the lower right part of the main window.

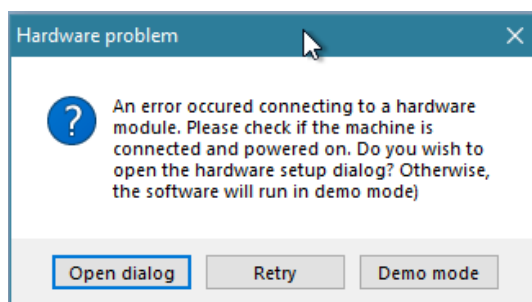
## 5 Short introduction

This chapter explains the most important steps required to load and execute a CNC program in the "fast run".

The design of the graphical interface of the software may differ from the design of the illustrations in this manual. It deliberately chose a "classic design" with few colors to optimize the contrast and readability of the manual.

### 5.1 Starting the Software

It is best to turn on the machine before starting the software. Otherwise, you will receive an error message because it will automatically attempt to connect to the controller at startup. If you forgot to turn it on, you can easily make up for it and click "Retry" a few seconds later.



## 5.2 Log in as a user

The software supports the ability to restrict access to functions depending on the logged in user. At the bottom of the screen in the middle of the status line you can see which user you are currently logged in with. As a logged-off user, you are not authorized to move the machine or start programs. The possibilities are very limited, which is also visible through the gray buttons and menus.



## 5.2.1 Release of the functions



If certain functions are inaccessible (grayed out) after starting the software, there may be two causes:

- The emergency stop of the machine is activated. This can be recognized by the symbol "**Emergency Stop**". No programs or movements can then be started, and no outputs can be switched on. If your machine has security doors or alarm signals from external devices, this may also lock machine functions. This is indicated by a different symbol instead of "Emergency Stop". Only with "OK" all functions are enabled.
- **No user is logged in** or the current user is not authorized for certain functions.

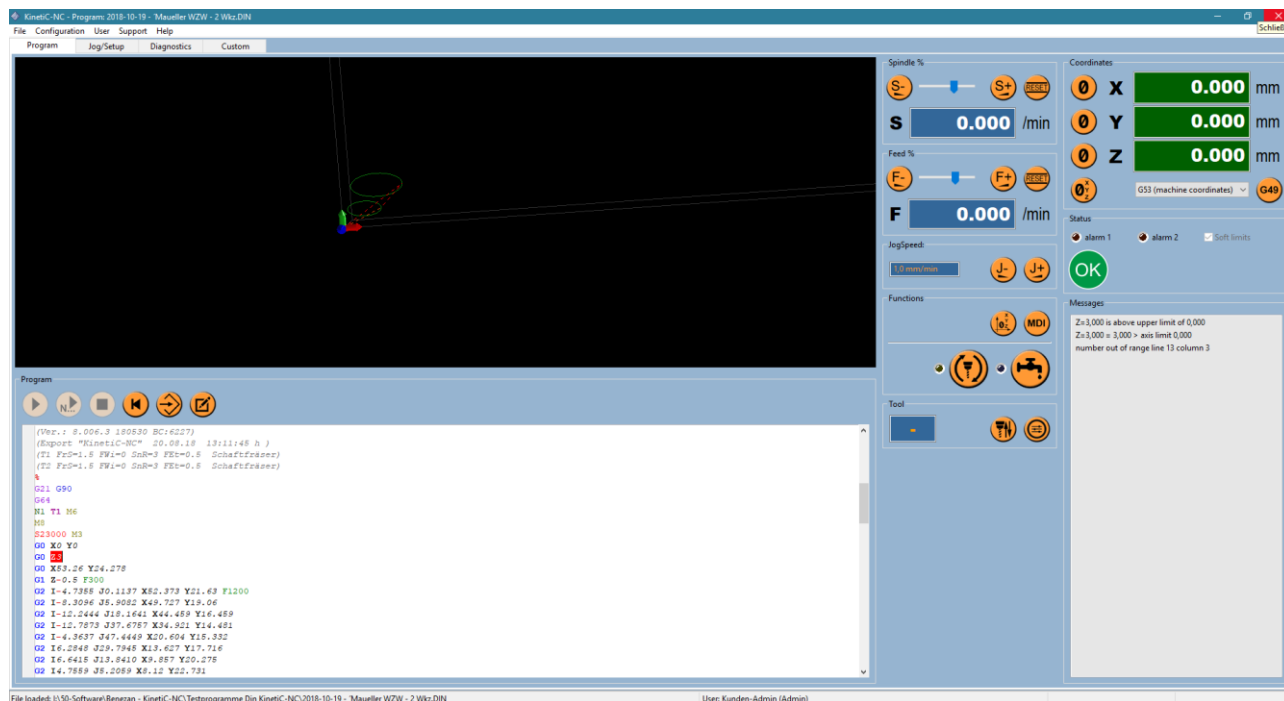
Username: Kunden-Admin  
Password:   
Login

If "User logged out" is displayed in the status bar, or if you need to access functions that require special authorization, you must log in as the user. To do this, select in menu-> user-> login.

Access to the machine parameters (Configuration - Machine menu) is e.g. only allowed to the administrator. You can log in as Administrator by selecting "Kunden-Admin". In the delivery state the password is "1234". The default user does not require a password.

## 5.3 Distribution of the screen

To keep the screen as clear as possible, the main window is divided into four pages. The tabs at the top of the title and menu bar (Program, Setup, Diagnostics and Special) allow you to switch between the pages. The exact meaning of all individual elements is explained in detail in the chapter *"The user interface"*.

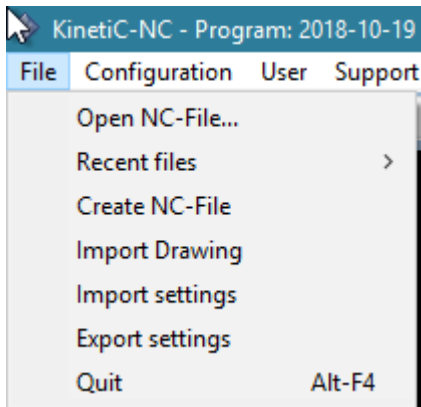


## 5.4 The coordinates display

Common to all four sides is the **coordinate display** (white numbers on a blue background, top right) and the **text window** in the bottom right for the messages.

On the main page („*Program*“) is the **graphic display** on the top left, where after the start first the company logo appears. After loading the program, the tool path is displayed here. Underneath is the **window for the program text**, which is empty after the program starts.

## 5.5 Load file



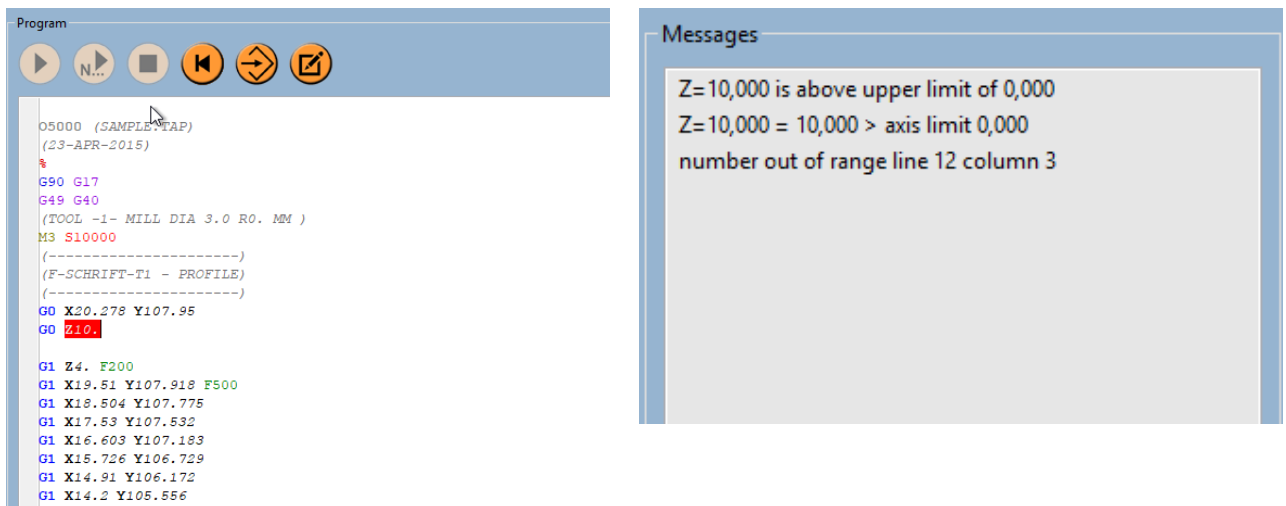
At the top of the title bar is the menu bar.

Select „*File->Open NC file*“, to open the file dialog. Select the sample file "Sample.din" or another G-code file of your choice. Depending on the Windows version, you will find the example files in the folder "My Documents\KinetiC-NC" or "Libraries\Documents\KinetiC-NC".

If you have ever opened a file, you can open it again from the list of *recently used files*.

## 5.6 Errors

The program will now be displayed on the main page ("Program") in the text window at the bottom left. In the graphics window on the top left, the tool path appears, if it fits into the editing room. If not, or if other errors have been detected, an error message appears in the message window at the bottom right.



For example, it may indicate that "number out of range ..." is displayed because the zero point is not yet in the correct location and parts of the tool path are outside of the processing space. It is important to recognize this already before the machine would hit a stop during machining.

## 5.7 The 3D display

Above the NC program, the working area of the machine is displayed.

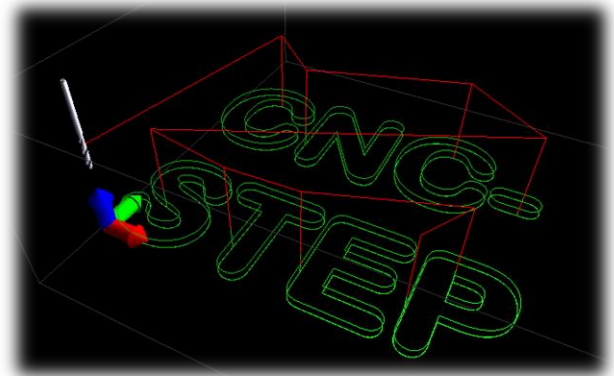
## 6 The operation

By **dragging the mouse** pointer while holding down the left key in the graphics window, you can move the tool path.

While keeping the **right mouse button pressed**, they spin the path in the room, which means they change the viewing angle to the 3D display.

**Turn the mouse wheel** to zoom in or zoom out.

By **double-clicking on the left mouse button**, you can switch between the following display modes:



- Zoom on the whole machine. The center of rotation is at the center of the machine
- Zoom on the workpiece, the center of rotation is in the middle of the workpiece.

### 6.1 Set zero point

#### 6.1.1 Where to put the zero point

Before a program can be started, it is necessary to tell the control software where the workpiece is located on the machine. This is done by setting the zero point. Where the zero point is on the workpiece depends on the drawing. The "Sample.din" file contains the zero point in the lower left corner and at the level of the drawing. For round or symmetrical parts, it may also be that the zero point is in the center. In the Z direction, the zero point is usually placed on the upper edge of the workpiece so that positive Z values are above the material, the negative ones indicate the penetration depth into the material. For some applications, it may also be useful to place the zero point on the bottom of the material, for example, to make the programs and settings regardless of the material thickness.

The following description assumes that the zero point in the drawing is on top of the material. The topic will be discussed in more detail later. This example is only about basic understanding.

#### 6.1.2 The start up

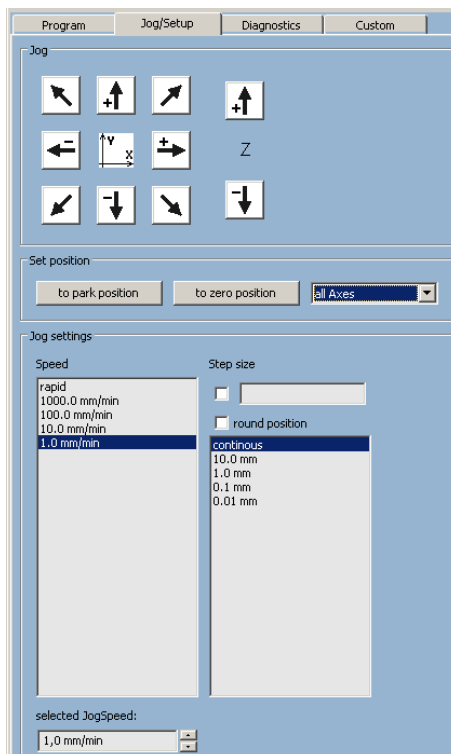
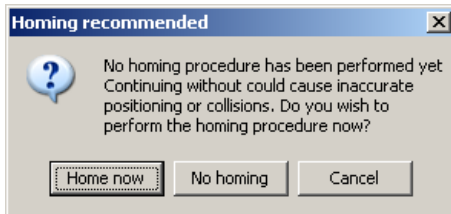
Switch to the "Setup" page, where the buttons with the arrows are located.

Clamp a workpiece to practice on the machine's working surface, such as a wooden plate. Briefly press the arrow pointing to the right.



## 6.2 Reference movement

If no homing has been performed yet, a dialog with a warning appears.



Select "Now Homing" to perform homing. Then use the arrow buttons in the X and Y direction so that the tool is above the workpiece.

Now you can move the tool down until it is about 10mm above the top. Now the speed should be reduced.

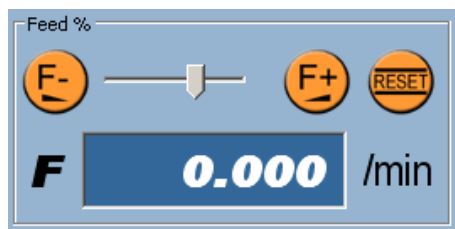
In the list of speeds, click "100.0 mm / min."

Carefully continue to drive down until the tool is only about one millimeter from the workpiece. Reduce the speed further to 10 mm / min.

Now put a piece of paper on the workpiece. Continue to drive slowly down until the paper stops moving and is clamped under the tool.

Now press the "0" button next to the Z coordinate display. The coordinate jumps to zero.

## 6.3 Speed setting



Set the speed back to 1000mm / min and drive Z up a few millimeters. Now you can use X and Y to move the tool over the left (X) lower (Y) corner, and also set these two axes to zero.

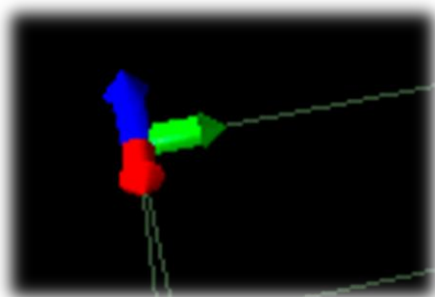
Now the X and Y coordinates should be zero and the Z coordinate should be a few millimeters (positive).

Be careful when approaching the workpiece, the machine table or any other object. You can cancel the movement at any time by releasing the button. With hard material and sensitive tools just a few 1 / 100mm too much can lead to tool breakage. Therefore, it is best to use a workpiece made of flexible material such as wood or hard foam for practicing.

## 6.4 The zero point symbol

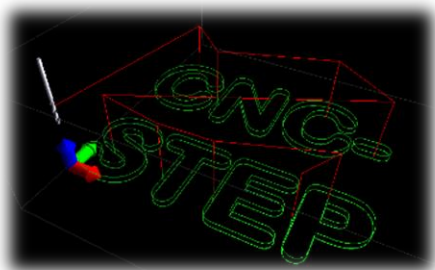
When a program is loaded, the graphic display is updated with the tool path each time the zero point is changed. The position of the zero point in the engine room is indicated by the "tripod" of three colored arrows.

### 6.4.1 Zero point and material





After the zero point is set correctly, the entire toolpath should be displayed in the graphics window, and no further error message will appear in the message window.

Before running a program for the first time, it is advisable to set the feed factor ("feed override", slider in the middle, to the right of the graphic display) to approx. 10%.





So, you have enough time to react if the machine does not run as intended, and a collision threatens.

## 6.5 Slow feed

Set the slider "Feed" 10% and then press "Start Program"  on the far left. The machine starts moving and moves to the first letter "C". During the first pass the contour is driven 4mm above the material. If everything is correct, you can set the slider back to 100% or press the "Reset" button  to the right of the slider. The machine now runs at the full, programmed speed, in the example 500mm / min. On the second run Z = 0 is driven, just above the material. In a real milling job, of course, a negative Z value would be inscribed, this is just a demonstration.

## 6.6 Presentation of the path

During execution, the already travelled distance in the graph is highlighted in light blue. You can let the program through to the end, or at any time press with "program stop"  Cancel. "Rewind"  and return to the beginning of the program and also clear the display of the distance traveled.

## 7 The user interface

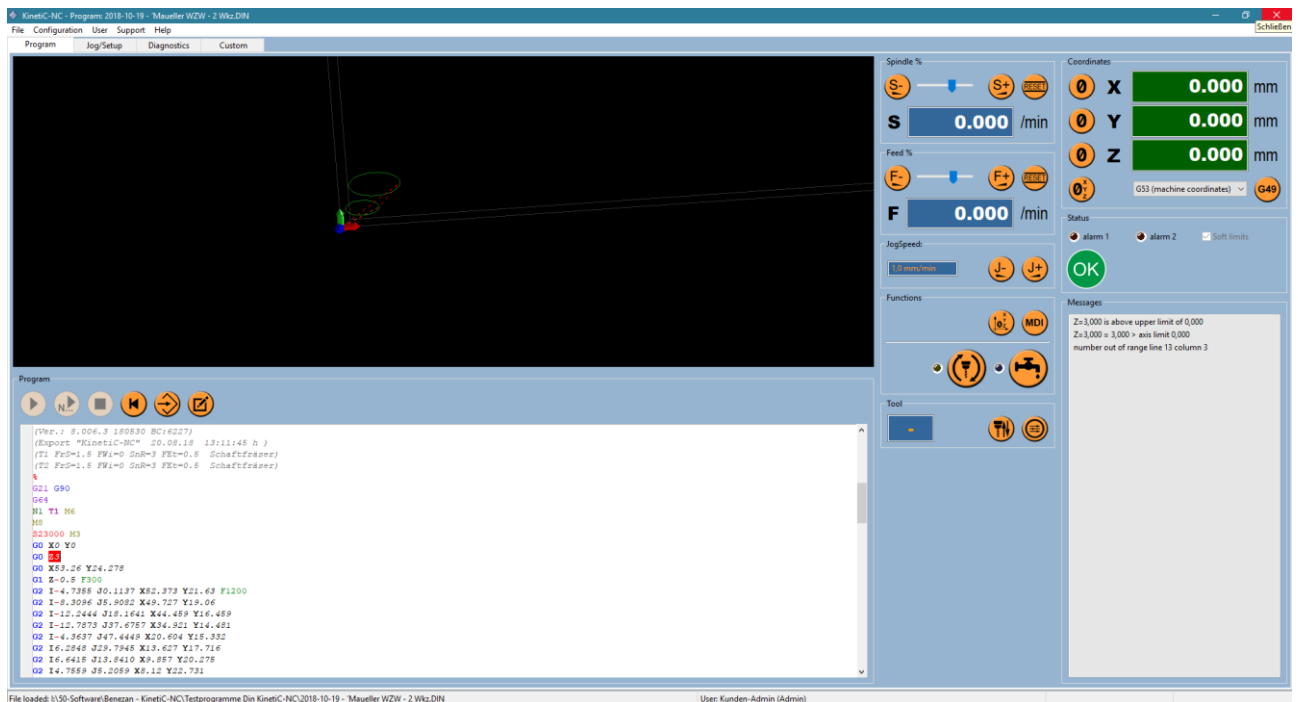
The user interface is divided into 4 clear screen areas and the menu at the top of the screen.

### Simple it safer

The design of these areas was deliberately chosen to be simple and to handle all functions while working on the machine.

A consciously overcrowded and artistically playful surface has been deliberately omitted in order to offer the user a clear, clean structure in order to exclude the risk of operating errors due to confusion.

### 7.1 The program screen

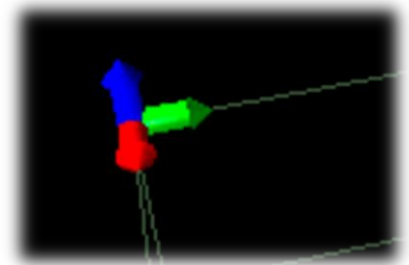


On the first page is the graphic display of the tool path, the text display of the NC program and the most important control elements to start and control the program sequence.

### 7.1.1 Graphic display

In the upper left window, the tool and its path are displayed in three-dimensional space after an NC program has been loaded.

The "tripod" with the red, green and blue arrow represents the position of the coordinate origin. Rapid traverse sections are shown in dashed red, milling paths with green lines (if several tools are used, possibly in different colors).



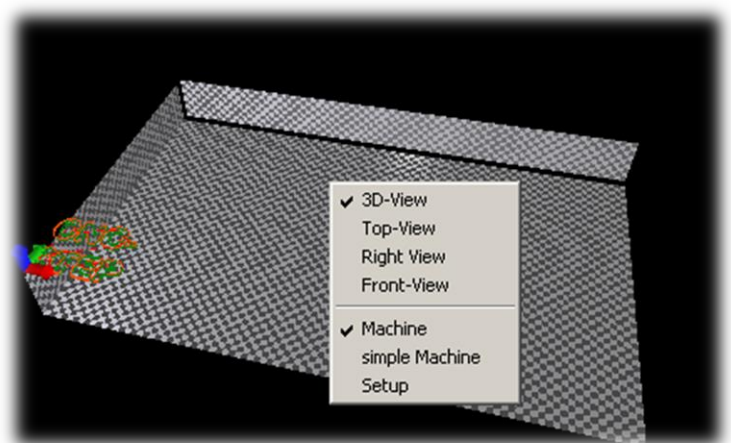
### 7.1.2 Mouse operation

After clicking with the left mouse button in the window, you can:

Move the display up / down and left / right by dragging the mouse while holding down the button.

- Use the **mouse wheel** to move closer or farther away from the subject (zoom).
- Dragging while holding down the **right mouse button** turns the viewing angle.
- **Double-click with the left mouse button** to center the object in the window and set the center of rotation in the workpiece, double-click sets the zoom so that the complete machine room is displayed, which is represented by a cuboid of thin, gray lines. Also, the center of rotation is set in the middle of the working space.
- **Double-clicking with the right mouse button** opens a small menu.

The "**Machine**" option displays the working space graphically. If this option is deselected, only the workpiece (if a program is loaded) and the zero point are displayed. The "**Simple Machine**" option draws the workspace as thin, gray lines. If this option is deselected, the machine workspace is displayed as a checkerboard pattern.



### 7.1.3 The NC program window



Below the graphics window is the text display for the NC program.

In the window you can scroll with the mouse or the cursor keys to look at the whole program or to go to a certain line.

If you click on a certain line, the button "**Start from here**" will activate and you can continue a program from this point.

### 7.1.4 The control elements

Below, the other controls are briefly explained. If one of the icons should later be unclear, you can leave a short **help text** ("**Tooltip**") by leaving the mouse pointer over the button.

First the display of the symbols is explained by the "**Edit**" -button:



Normal, active button that can be pressed.



Normal, active button that can be pressed, which has the current focus.



Currently pressed button.



Disabled button that cannot be pressed. Cause can be missing user rights, or the function is currently not possible.

### 7.1.4.1 The symbols in detail



Program start, this will start the currently loaded program.



Program start from currently selected line. This can be used to resume a program, which previously was e.g. was cancelled.



Stop the program. This will stop the currently running program.



Rewind the program. With this button you are right back at the beginning of the first line of the program.



Load program. This button is identical to Menu-> File-> Open NC File



Edit program. This button opens an editor in which the current NC program can be edited.



Sets the coordinate to the right to zero (0). The coordinate refers to the selected, current offset (G53-G59)



Sets all coordinates to zero (0). The coordinates refer to the selected, current offset (G53-G59)



Carries out a homing run with all axes.



Opens the MDI input line. This allows individual G commands to be executed, e.g. G0X100 to move the X-axis to position 100.



Activates or deactivates the tool length compensation G49.

### 7.1.5 The feeds

In the upper right column are the feed override sliders (feed%) and the spindle speed override (spindle%). This allows you to change the execution speed between 0 (standstill) and 150% or the spindle speed (if your machine has a spindle with frequency converter) while the program is running.



With these buttons you can change the current feed.

With the ESC key, you can immediately zero the feed override to temporarily stop program execution. This has the same effect as zeroing the feed slider. The button "Reset" resumes execution at normal speed (100%). To indicate the cause for the standstill more conspicuously, the field is colored red at 0% feed.

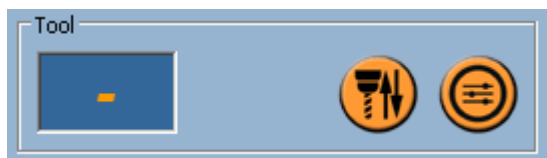


These buttons can be used to change the spindle speed.



These buttons decrease or increase the set jog speed, which is shown in the display between these buttons.

### 7.1.6 Tools and track settings



Opens the tool change dialog.



Opens a window in which many settings for the machining strategy can be changed. See *Tool-Path-Settings*.

### 7.1.7 Outputs and tool spindle



Switches the tool spindle in the corresponding direction of rotation. Depending on the machine type, only one of these buttons may be displayed.



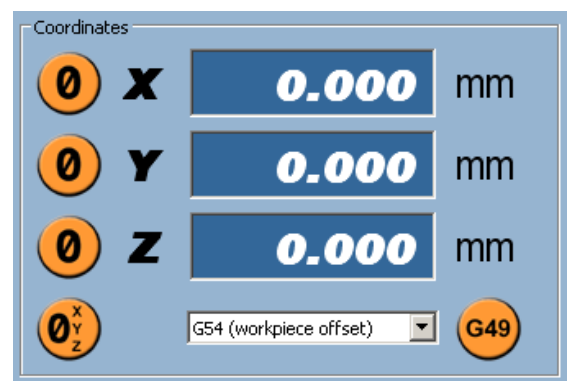
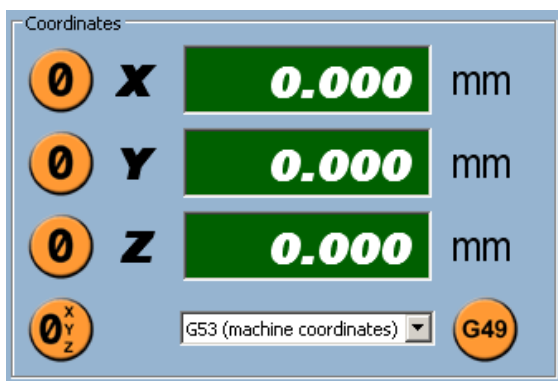
Turn on flood or spray cooling. Depending on the machine type, it may be that only one of the two types of cooling exists.

### 7.1.8 The coordinates display

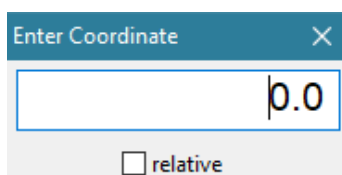
In the middle column, the coordinate displays are located at the top, where the positions of the axes are displayed. The position values always refer to the current zero point, which can be selected with the selection button below.

### 7.1.9 Offsets

For G53 the coordinates are absolute (machine zero) and are shown in white on green, all other choices are workpiece related offsets and are displayed in white on blue. All workpiece offsets G54 to G59 are permanently stored on the hard disk of the computer and are retained even after the software has been exited and restarted.



### 7.1.10 Set the coordinates

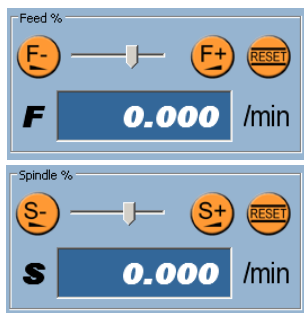


By double-clicking with the left mouse button on a number, a coordinate can be entered manually. The selection "relative" means that the number input is added to the current coordinate.

For example, the coordinate Z is currently at a value of 50,000. A manual input of 30,000 would set the coordinate to a value of 30,000, and if the option "relative" would be used, an input of 30,000 would set the coordinate to a value of 80,000.



### 7.1.11 Feed and Speed display



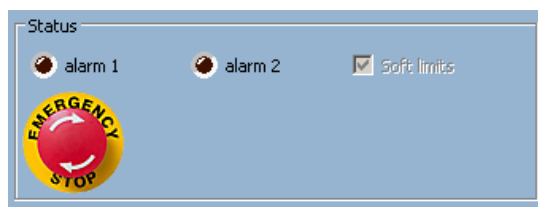
Under the spindle speed and feed controller, the current spindle speed (S) and the current feed speed (F) are also displayed. These are usually displayed in white on blue. If the option "Replace F / S global" is selected in the, these are displayed in white on turquoise.

### 7.1.12 Emergency stop and alarm

Right, below the coordinate display is the status display.

When the machine is running or ready for movement or program execution, an OK icon will appear there. When the emergency stop button is pressed, the hardware is not plugged in or powered down, or another fault condition is present, the "Emergency Stop" - or other warning icon will appear.

If the DEMO symbol appears, there is no connection to the hardware. The triangular warning labels indicate a problem with the safety door, the frequency converter or other external equipment.



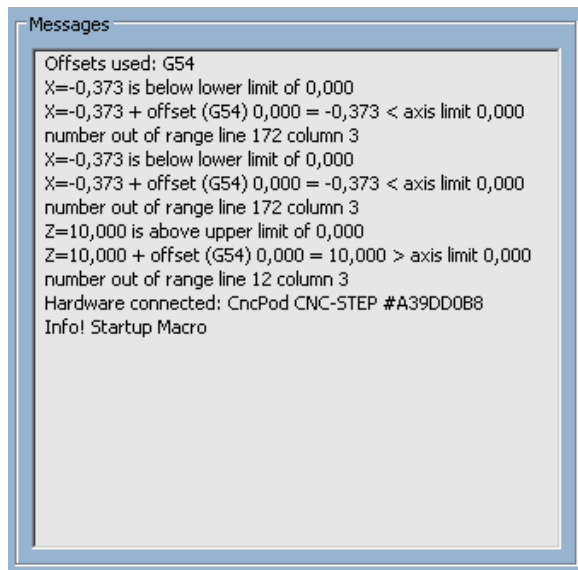
For information about how to resolve the issues in these cases, refer to the instructions in the appropriate extension.



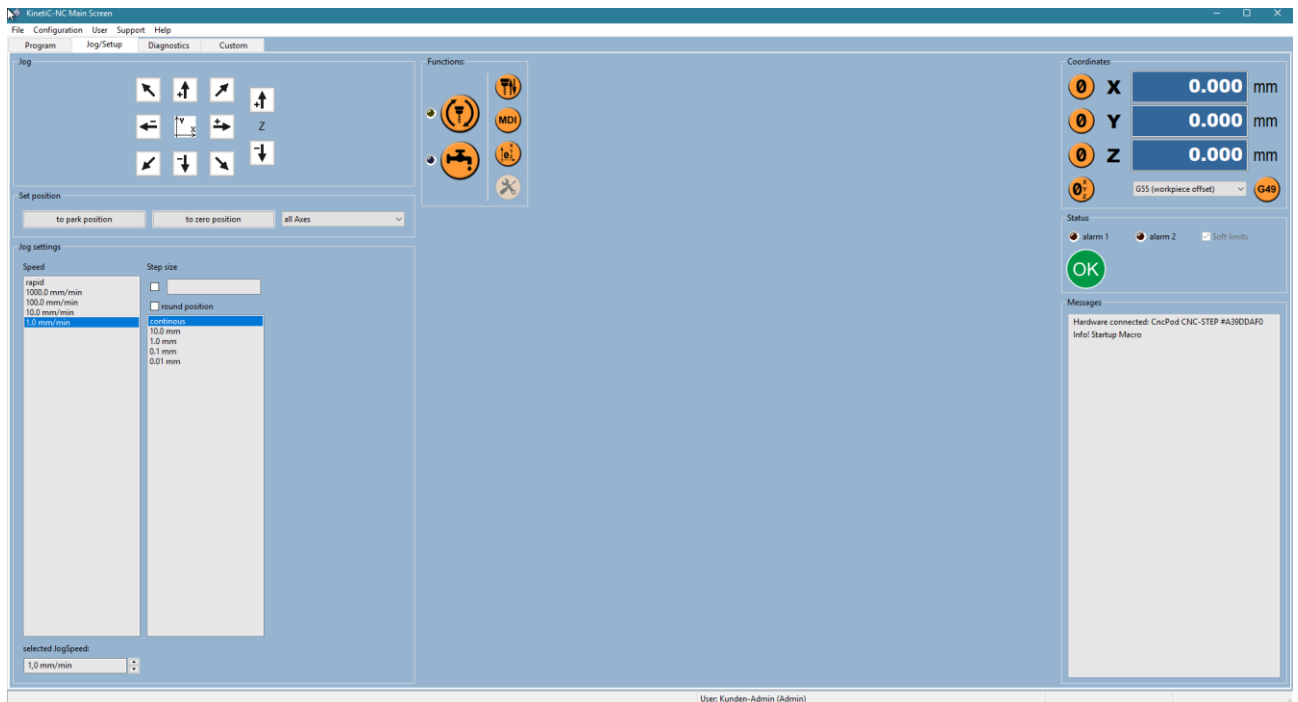
**Attention:** The emergency stop symbol is only an indication. You cannot trigger an emergency stop by clicking on the symbol. This was deliberately done because a software-triggered halt is not secure, and valuable seconds are lost in an emergency, because searching with the mouse pointer takes much too long. In case of imminent danger please use the "real" emergency stop switch on the machine or on the control.

### 7.1.13 Message window

The "Messages" window at the bottom right shows information about the program flow or error messages. The messages can be deleted by right-clicking on the window.



## 7.2 The setup screen



### 7.2.1 The jog buttons

This page is intended for setup. Just as with the program page, there is the coordinate display on the right, the message texts in the bottom right corner and the emergency stop display in the middle. Instead of the program text and graphics window, however, controls for manually moving the axes (jog) are available. The buttons with the arrows start directly a drive of the associated axis in the appropriate direction. The speed and step size can be selected using the selection buttons below. "Continuous" means the axis is moving as long as the arrow button is pressed and stops when released. For finite step sizes, and when "Position rounding" is active, the axis travels a maximum until the next position value rounded to the step size (with increments of "10mm", the axis travels a maximum of, for example, 481 to 490mm). The arrow button still has to be pressed for the axis to move, or in other words, if you release the button before the step size expires, the axis will stop immediately. This is to prevent a collision, if one has misjudged the free distance.

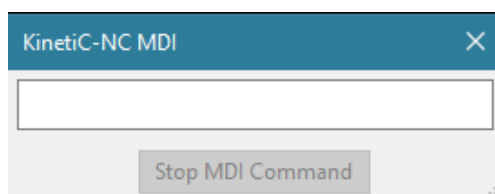
## 7.2.2 Rounding

If "Position rounding" is not active, the maximum distance travelled is the selected distance beginning with the start position, for example, with increments of 10mm from 481 to 491mm. In the input field directly under "step size", any number can be entered that is not in the list, eg 3mm. Incidentally, the axes can always be moved manually at any time, as long as no program is running, if you are on another page, using the hotkeys. By default, these are the arrow keys on the keypad of the keyboard. The buttons on the Setup page, however, give you better control over the speed and can, for example, gently touch the edges of the workpiece in small increments.

## 7.2.3 Zeroing display

With the zero buttons next to the coordinates or by entering a number after double-clicking on the coordinate, you can again change the workpiece zero point. By clicking on the buttons "to park position" and "to zero point" on the left side under the arrow buttons, the machine moves directly to the zero point or the parking position. If you want the Z-height to remain constant, you can also choose "only X + Y" instead of "all axes". Attention use this function only if you are sure that the way to it is free! If a collision threatens, you can stop the movement by pressing the button again, or of course with the emergency stop button.

## 7.2.4 The MDI



The "MDI" button at the top center opens a window with a text entry field. Here you can enter G-code commands directly. This can be useful for approaching certain locations or entering commands that enable a particular mode or function.

If a G-command has been typed in and confirmed, it can be aborted by pressing the "Stop MDI command" button.

While a command is being processed, the next command can be typed. This will be executed immediately after the previous command has been completed.

## Repeat

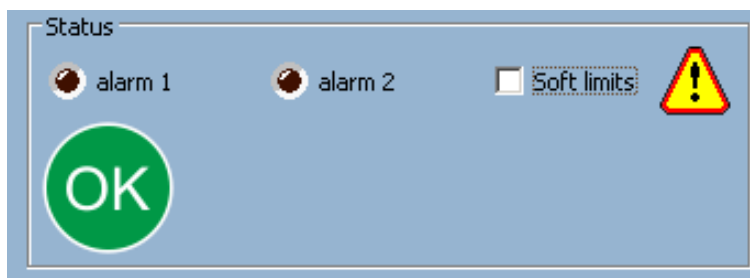
By using the cursor keys up, you can scroll through the list of already entered commands to repeat faster and safer commands.



**Attention**, it is explicitly warned against misusing the MDI input to type programs in "real time". Too easily mistype it and risk a collision. Pay close attention to what you typed in and prefer to check it again before pressing Enter, especially on G0 commands.

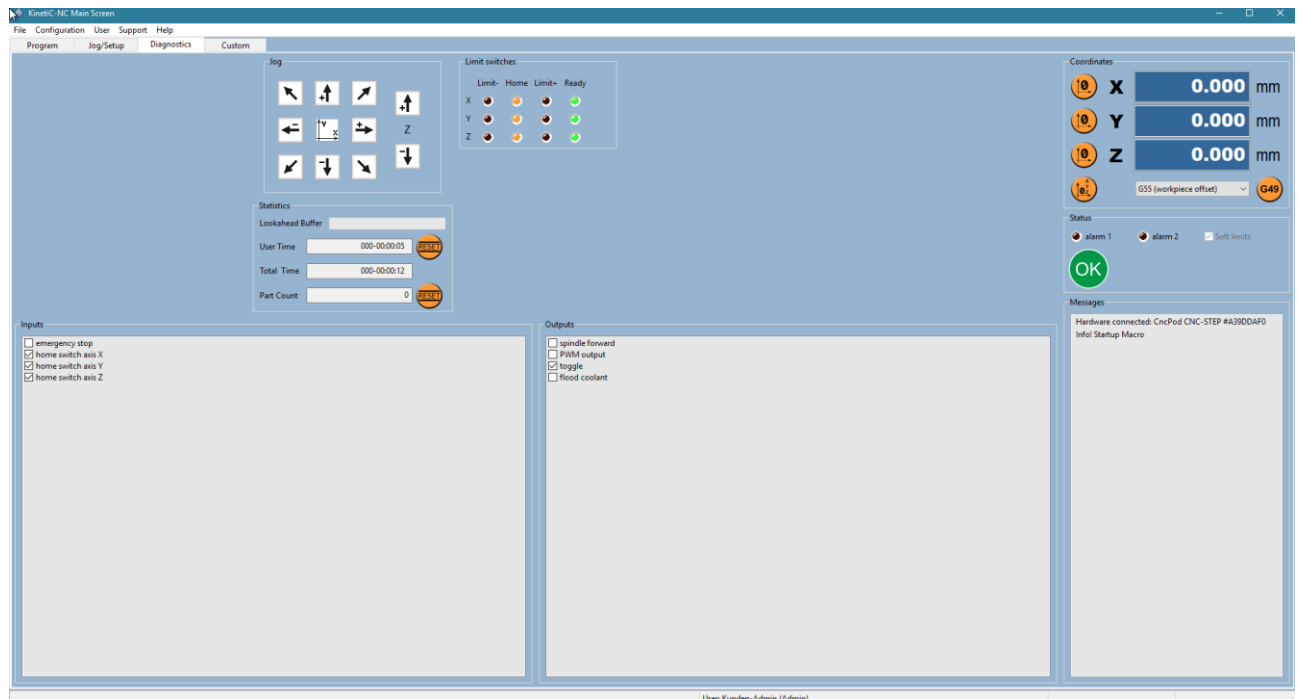
## 7.2.5 Softlimits

By default, the soft limits are always on, that is, after homing has been performed, the machine "knows" where it stands and how long the axes travel. It is therefore automatically always stopped before the hitting the stopping point. If the tick is removed before "Soft Limits", this check is switched off, which is normally only required when testing the machine, if the reference switches do not (yet) function properly. With soft limits off, you should only drive slowly and carefully. If the check mark is greyed out, deactivating the soft limits is prohibited (see Machine Configuration Dialog: Security - Force Soft Limits)



## 7.3 The diagnostic screen

In addition to the known coordinate display and the message window, the status of the inputs and outputs are displayed on this page. The states of the reference and (if present) limit switches are visible as signal lamps at the top center. To the left of the coordinate display are buttons for triggering a reference run. Either the selected axis or all axes one after the other move slowly to their reference switch to find the position of the zero point.



### 7.3.1 Inputs and outputs

At the bottom left are the lists with the inputs and outputs. Only signals that are actually assigned to a hardware pin are displayed. A set tick corresponds to an "active" input, eg a depressed switch, or an appropriate statement (air pressure is OK, drive is ready, etc.). "Active" does not necessarily have anything to do with the voltage level at the input, because inputs can also be inverted depending on the switch type (NC or NO). For the outputs, on the other hand, an existing tick always means "on" and "voltage present", a missing "off" and "no voltage". Outputs must not be inverted because otherwise the safe state (switched off) would not be clearly defined.



By clicking on the checkmarks of the outputs, they can be switched on and off. Regarding the entrances, a click logically does nothing. The PWM output can be tested by double-clicking. It then opens a small dialog box with a slider. Please use the test function with care, switching outputs may cause machine parts to move.

### 7.3.2 Lookahead-display

The bar of the lookahead buffer indicates the level of the command buffer in CncPod. This allows conclusions about the quality of the network connection and the computing load of the PC. When driving longer distances, the display should be in the upper quarter (i.e., bar to almost far right). If the buffer runs low because the connection is bad, or if the PC is slowed down by other, computationally intensive programs, the movement of the machine may stall. Although this does not lead to a loss of position, because the CncPod initiates a normal braking and acceleration ramp in this case, and then continues normally. Depending on the type of processing, unwanted effects may nevertheless occur, such as e.g. Chatter marks during milling or burns during plasma or laser cutting. In the case of deliberate waiting times (G4) or transition from rapid traverse (G0) to processing (G1 to G3), it is normal for the buffer to be almost empty for a short time.

### 7.3.3 Hour meter

The operating hour meter and the user clock show the time for how long the machine has been running.

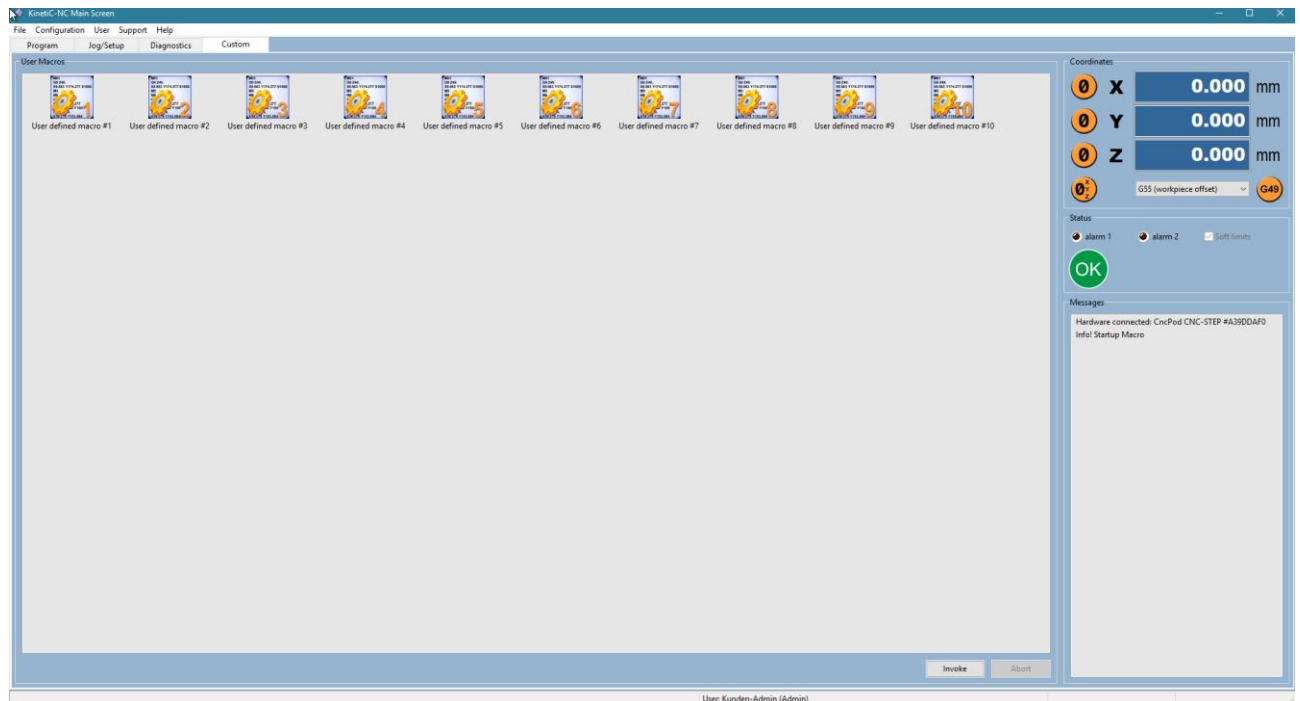
### 7.3.4 Part counter

The part counter indicates the number of complete program executions. The user clock and the parts counter can be set to zero if required, the operating hours counter not.

## 7.4 The special screen

Using the so-called user macros own functions can be realized, which are not predetermined by the software itself. You can either program the macros yourself by selecting the user macros at the bottom of the list in the Macro dialog (Configuration -> Macros menu).

Or you can install macros using the import function (menu File -> Import Settings) downloaded from the Internet or supplied with a machine accessory.



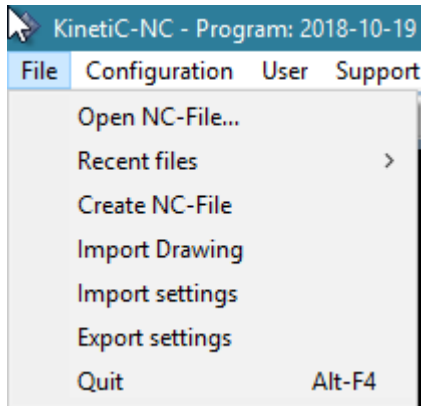
### 7.4.1 Application examples

Typical application examples for user macros are: Spindle warm up, *tool length measurement* with mobile probe, borehole center finding and much more. A more detailed description of the *Macro-programming language* can be found in the separate reference manual.



## 8 The file menu

The following chapter explains the individual elements of the menu.  
The main menu is divided into the following individual areas:



### 8.1 Opening NC-file

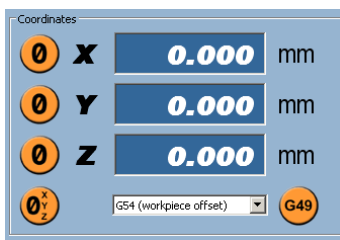
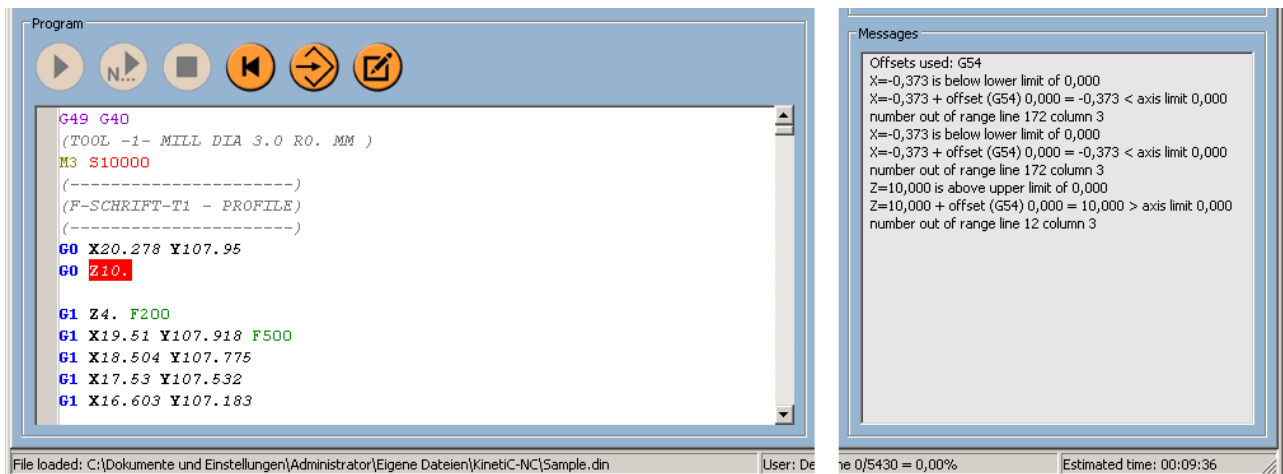
This opens a file selection dialog for loading an NC program. Alternatively, this can also be done with the "Load Program" button, which is located under the graphics window on the program page.

#### 8.1.1 File extensions

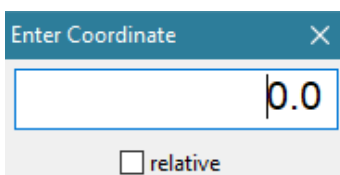


The file must contain G code according to DIN / ISO and usually has one of the endings ".din", ".iso", ".nc", ".tap", ".txt". G code contains the coordinates of the actual tool path and additional commands, such as tool spindle and coolant. Drawings created with a CAD program, on the other hand, contain only contours or areas or volumes of the workpieces and no information about the tool path. 3D drawings must first be edited with a CAM program to create the tool path. 2D drawings can be loaded with the ***Import Drawing function*** with certain restrictions.

## 8.1.2 Error while opening



After selecting the file, it is loaded and the tool path is displayed in the graphics window. If errors are detected, the structure of the tool path is aborted at this point and a message is displayed in the lower right text window. The most common error is a "number out of range", which is usually due to a false or not yet set zero point. In this case, either select the correct zero point (G54 to G59) below the coordinate display or set a new zero point.



The location of the zero point may e.g. can be changed by moving the axis to the correct position using the arrow keys (Setup page) and pressing the zero button next to the corresponding coordinate display. Alternatively, you can double-click on the Coordinate display and enter a number if you need a specific offset measure.

## 8.2 Recently used files

It often happens that you want to reload files that you have already used before. So that you do not have to remember the file names and enter them again, this submenu displays the last ten files used.

### 8.3 Create NC file

After selecting the location and entering a file name, the text editor opens and you can write an NC program by hand. The automatic coloring of the keywords leads to a clearer presentation and makes it easier to find typos.

"Save" saves the program on the data medium and automatically loads it as a preview in the graphics window.



If you still want to change the program afterwards, you can return to the text editor via the Edit button.

```
GCode Editor
1 O5000 (SAMPLE.TAP)
2 (23-APR-2015)
3 %
4 G90 G17
5 G49 G40
6 (TOOL -1- MILL DIA 3.0 R0. MM )
7 M3 S10000
8 (-----)
9 (F-SCHRIFT-T1 - PROFILE)
10 (-----)
11 G0 X20.278 Y107.95
12 G0 Z10.
13
14 G1 Z4. F200
15 G1 X19.51 Y107.918 F500
16 G1 X18.504 Y107.775
17 G1 X17.53 Y107.532
18 G1 X16.603 Y107.183
```

Save Cancel

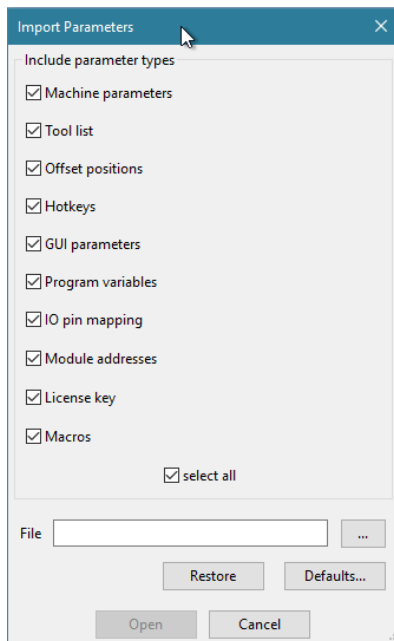
## 8.4 Import drawing

2D drawings that are in DXF or HPGL format can be imported using this function, that is means they are converted to a G-code file. This is useful for operations that do not require a CAM function, where the cutting width (cutter diameter) need not be considered, such as when engraving scripts where the lines of the drawing correspond to the web of the tool center. For all other machining operations, where the tool path has to be offset by the tool radius, as well as for the 3D machining, an external CAM program is necessary.

### 8.4.1 Detailed description of the import filter

Since the *Importfilters* sometimes offer quite extensive functions, they are dedicated to a separate chapter at the end of the manual. For example, it describes an *HPGL-Importfilter* and a *DXF-Importfilter* that are included in the standard version of this software.

## 8.5 Import settings



With this function you can load the complete machine parameters and all settings of the software or parts thereof from an external file. This can happen for several purposes:

- You want to undo changes and import a previously made backup.
- You want to install an additional feature or an update or an extension.
- You want to switch to another machine or another type of processing.
- You want to adopt settings from another, identical machine.

### 8.5.1 Partial import

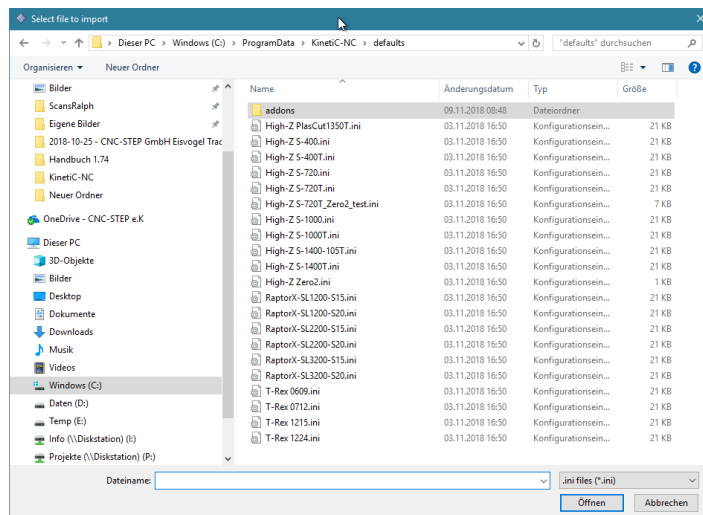
The control software not only supports complete parameter sets, but also subsets thereof. For example, if you buy a rotary axis retrospectively for a 3-axis milling machine, the seller can supply a parameter file that only contains the parameters relevant for the rotary axis, ie resolution, speed and travels of the fourth axis.

If you import this file, only these parameters will be changed. All previously made, other settings are retained.

## 8.5.2 Restore a backup

The "Restore" button takes you to the directory of the backup copies in order to restore previously made backups.

## 8.5.3 Refurbishment



If you want to rebuild a machine, or if you want to use a fresh kit from the manufacturer, go to "Standard parameters" and select your machine. With "Standard parameters" you get to the directory of the predefined parameter sets. Attention, importing one of these files overwrites all settings made with the default values. Only do this if you want to "reset" to the factory settings and, if necessary, make a backup beforehand (see below).

## 8.5.4 Install options

If you want to reinstall an option, insert the media with the options file that you have received to the product and open the file.

If you want to install an option that is part of the standard scope of the software, open the "**Default Parameters**".

The subdirectory "**Addons**" within the standard parameters contains parameter sets for extensions and additional options, such as **tangential knife cutting**. The parameters in "Addons" are subsets of the machine parameters and overwrite only the relevant parameters. All the rest stay on the old values.

If you only want to load certain parts of a configuration without overwriting existing settings, just select the desired options and then press "Open".



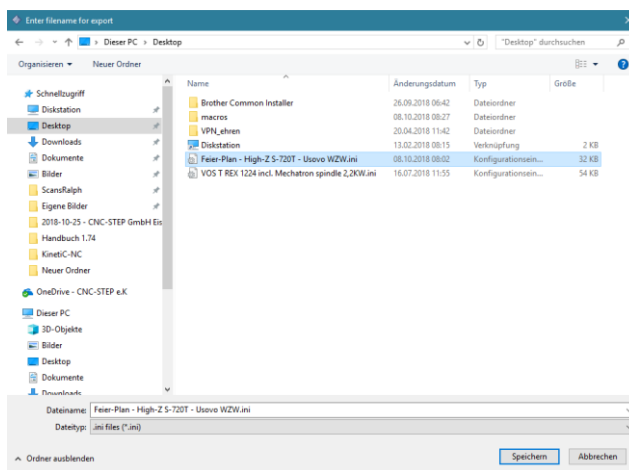
**Attention!** Before importing parameter files, make sure that they are suitable for your machine type. Incorrect machine parameters, for example, can cause drives to drive faster than expected and cause collisions. Therefore, carry out a test drive at low speed after every change of the machine parameters.

## 8.6 Export settings

With the export function you can save machine parameters and other settings of the software in a file. As with the import, this can also serve several purposes:

- You want to make a backup (backup copy) of the settings. This makes sense after the successful installation or before extensive changes to the settings.
- You use the same machine for multiple types of processing (such as milling and plasma cutting) and want to keep the settings made for later use.
- You want to transfer the settings made to another, identical machine.

### 8.6.1 Create backup



For a backup, it makes sense to select all setting categories. When transferring to another machine, however, you should definitely omit the module addresses, because otherwise the other machine would no longer recognize its own hardware modules after the import. Of course, you can only select individual categories, such as tool list or offsets.

The file is only saved when you exit the dialog shown on the right with "Save".

### 8.6.2 Select file

In the file dialog, which is opened with the button with the three dots, only the file name and storage location are selected.

If you are in demo mode, you cannot select module addresses or I / O assignments because there is no hardware connected in this case.

### 8.6.3 Create individual configurations

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.

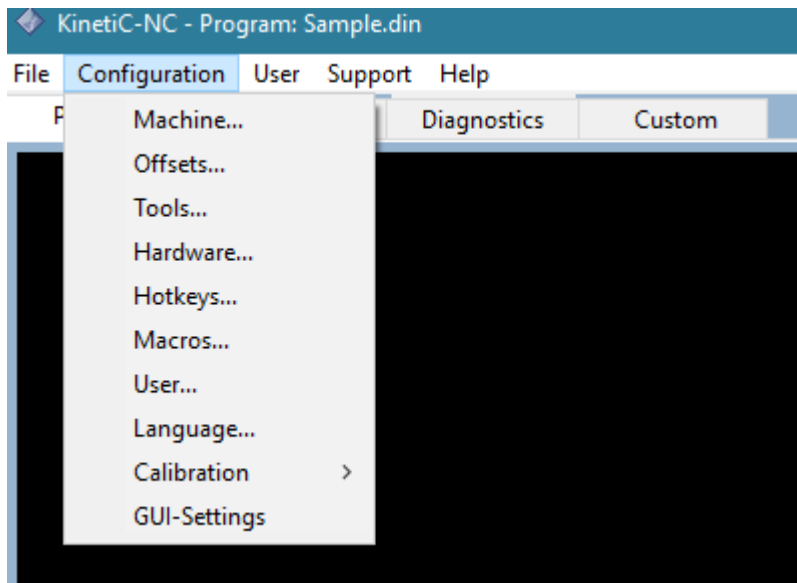
Clicking on the icon starts the software with this configuration. This is useful if you have multiple machine configurations and you want to select one at the start of the software. The software then starts directly with these settings without you having to select "import settings".

## 8.7 Exit

This menu item terminates the program, which of course is only allowed when the machine is stopped, when no NC program is currently running. Otherwise, you will be asked to stop the running program first with "Stop program". The Close Window button on the top right has the same effect.



## 9 The configuration menu



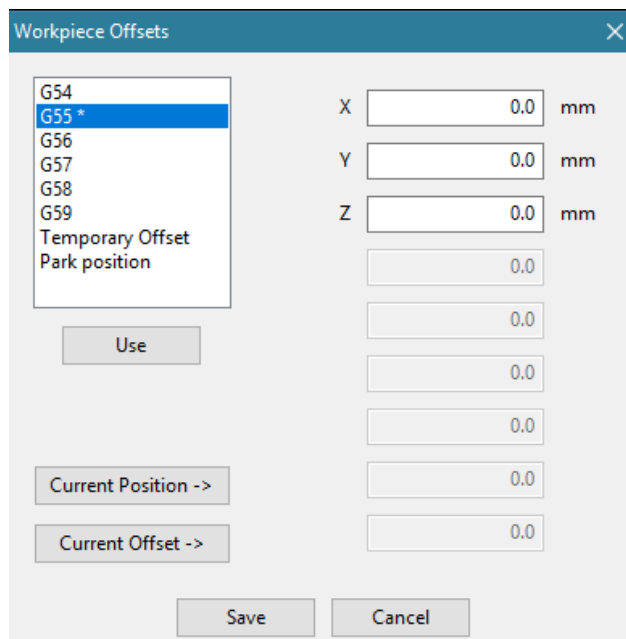
### 9.1 Machine

All parameters related to the machine are set here, such as number, resolutions and speeds of the axis drives, inputs and outputs, jog speeds and other settings. Explaining the meaning of all the parameters here is beyond the scope of this manual, and in most cases is not necessary because standard parameter sets are available for machines sold with the software. A brief introduction to the most important settings can be found in the chapter „*Initial-Start-up*“. For advanced users and designers of special machines or self-construction, please refer to the separately available *Reference manual*.

It is recommended to make a backup of the parameters before making any changes (see “*create backup*”, top).



## 9.2 Offsets



In the offset dialog, the zero offsets G54 to G59 (workpiece zero points) and the parking position are managed.

Top left is the list of offsets. The currently selected (clicked with the mouse) offset is highlighted, and its coordinates are displayed on the right. With the "activate" button, the selected offset can be activated, i.e. all subsequent programs use this offset, as long as a different offset is activated by a G53..G59 command.

With the button "Current Position", the current position of the tool is entered in the coordinates of the currently selected offset. This has the same effect as pressing "All Zero" ("0 XYZ") next to the coordinate display in the main window. In other words, after the offset has been activated with the coordinates of the current position, the coordinate display in the main window indicates the relative position ( $X = 0$   $Y = 0$   $Z = 0$  etc.).

### 9.2.1 Copy Offset

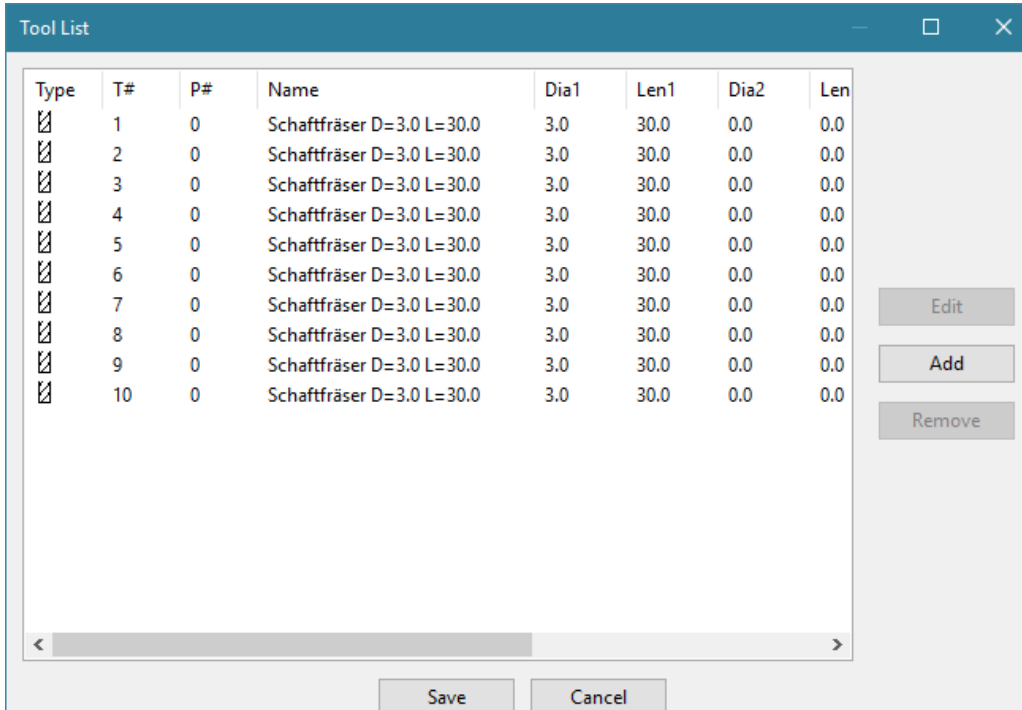
The button "Current Offset" copies the coordinates of the active offset, which is marked with an asterisk "\*" in the list, into the coordinates of the just selected offset, i.e. G56 to G54 in the example in the upper picture.

If you accidentally set an offset to incorrect coordinates, you can select "Cancel". All offsets will then be restored as they were before the dialog was called. "Save" saves the changes.

## 9.3 Tools

### 9.3.1 Do I need a tool list?

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 per program, you can always work with only one tool number (T1), and you do not need to create a tool list.



Type	T#	P#	Name	Dia1	Len1	Dia2	Len
☒	1	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	2	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	3	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	4	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	5	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	6	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	7	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	8	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	9	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0
☒	10	0	Schafffräser D=3.0 L=30.0	3.0	30.0	0.0	0.0

### 9.3.2 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.

### 9.3.3 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. Standing times 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.

The screenshot shows the 'Tool Data' dialog box. The 'Tool Type' list includes: custom, end mill (selected), drill, ball nose, V-tip, T-slot, torus, lollipop, countersink, dove tail, and reamer. The 'Name' field contains 'Schafffräser D=3.0 L=30'. The 'Number' field is 1 and the 'Place' field is 0. The 'automatic' checkbox is unchecked. The parameter fields on the right are: 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, and Path color (green). The 'OK' and 'Cancel' buttons are at the bottom.

### 9.3.4 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.

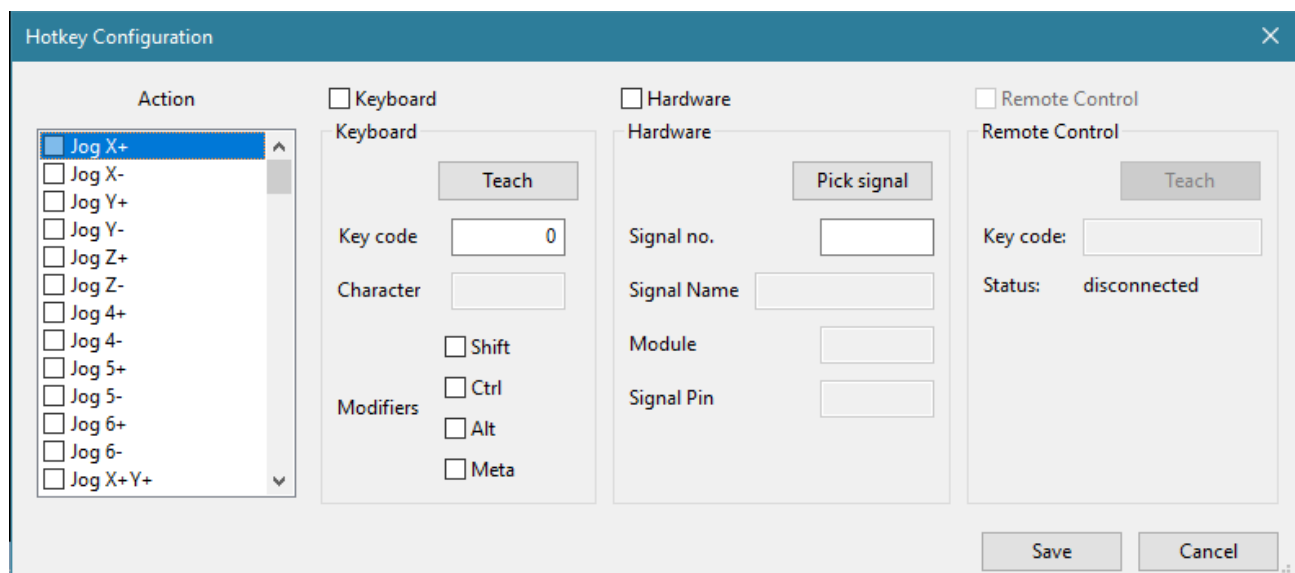
## 9.4 Hardware

The hardware dialog is used to link the PC software to the connected hardware, in this case the CNC module of the machine used. This is normally only required once and is described in the chapter „*Initial commissioning*“

Later, you only need the hardware dialog to restore the connection after a possible interruption, or if you later want to install expansion modules, have to replace a module due to a repair, or to switch to demo mode (operation without hardware).

## 9.5 Hotkeys

The control software supports so-called "Hotkeys", which means that you can trigger certain functions via the keyboard and also determine which keys should be. By default, for example, the arrow keys of the numeric keypad for manual driving (jog) are allocated so that you can change the tool position at any time without having to change to the *Setup-Page* of the main window. So that machine movements are not triggered unintentionally while typing, all Hotkeys are deactivated while text input fields are selected.



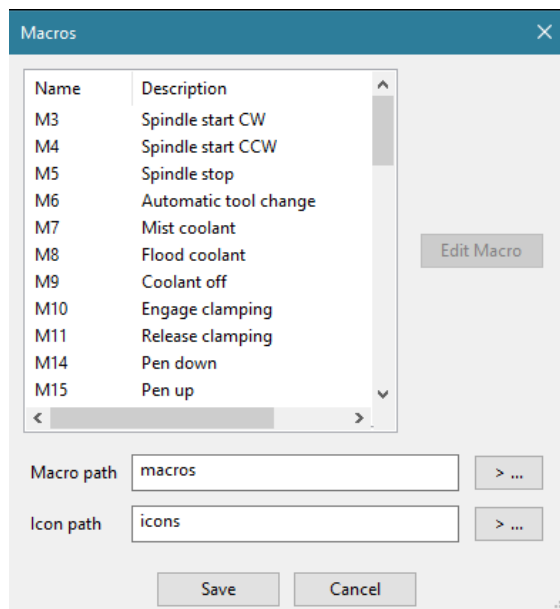
### 9.5.1 Assign keys

With the hotkey dialog the assignment of the keys can be changed. Since not all keys are assigned to a visible character, the assignment takes place via a "key code", which is difficult to remember, which is why there is a learning function. First select a function in the list on the left and set the function top right from "not occupied" to "keyboard". If the "learn" button is pressed, the code of the next key press is automatically determined and entered.

## 9.5.2 Jog-keys

For the manual drive (jog) there are a lot of possible key assignments, because there is not only one button for each direction, but also several options for the speed. The functions without speed indication leave the speed at the last set value. The functions with (fast or slow) influence the speed. For example, you can set the keys so that an arrow key without a shift key moves at the currently set speed, the same key moves in rapid traverse with the shift key, and the same key moves slowly again together with the CTRL key.

## 9.6 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 behaviour 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.

### 9.6.1 Macro directory

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. This is recommended only for advanced users and is described in the separate **reference manual**.

## 9.7 User

Logging in users is required to gain access to certain functions. This is described in „User-Login“ and "Login". This chapter explains the assignment of user rights and passwords. Of course, to be able to change user rights and passwords, you must have the corresponding rights, normally logged in as an administrator.

After selecting the currently used user, the "Change password" button becomes active. Pressing this button opens a small dialog that allows you to enter a password. After entering the current password (which can be empty if no password was assigned) you can assign a new password.

### 9.7.1 Which password?

Both inputs must be identical in order to prevent the password from being changed if it is mistyped. All numbers, letters and special characters are allowed, the password may be between 0 and 255 characters long. This means that the field may also be empty, which means there is no password assigned. If you are the only user on the machine, it may make sense not to assign a password. So you do not have to log in every time. However, if other persons have access to the machine, it makes sense to block access to prevent unauthorized manipulation.

### 9.7.2 Forgot Password



**Attention!** Be careful not to forget the password. If this has happened, only an administrator can reset the password.

If this is necessary or if you need to make further adjustments to the machine, please contact the support of CNC-STEP GmbH & Co. KG.

## 9.8 Language

Here you can change the language later. The change does not (immediately) affect windows that are already displayed, therefore it is necessary to restart the software to update all translated texts.

## 9.9 Calibration - PWM output

**PWM Output Calibration**

Output setting

Coarse

Fine

S 0.0 PWM 0.0 Voltage 0.0  M3 Output

Limits

min. S 0.0 min. PWM 0.0  PWM inverted

max. S 24000.0 max. PWM 100.0 max. V 10.0

Save Cancel

In the case of milling machines with a high-frequency spindle, the spindle speed is set using the PWM output or the laser power during laser engraving. The mapping of the S value in the NC program can be adjusted via the calibration function, it is set which minimum and maximum S value corresponds to which voltage value.

To do this, first enter the permissible range for the S value (speed or power level) at the bottom left. The middle column shows the assigned values for the minimum and maximum duty cycle of the PWM output in percent. The value for the maximum voltage at the bottom right is just for control.

### 9.9.1 Testing

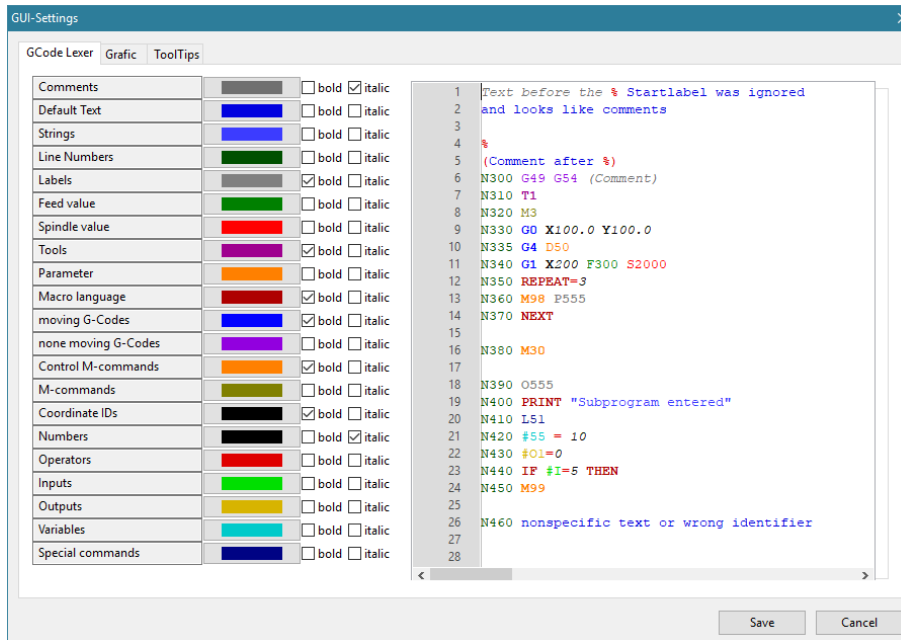
In the upper part of the window, the setting can be tested. When the M3 output is turned on, the spindle should rotate at the indicated speed, or the displayed voltage at the analog output should be measured. If this deviates, it can be adjusted by adjusting the values for min and max PWM. The arrow button to the right of the PWM values adopts the currently set value from the upper area in the input field to the left.

### 9.9.2 Laser Applications

When laser engraving, the output cannot be tested directly because a hole would burn into the material when the laser is stationary. Here, the PWM setting must be determined with a special test program. This is described in the laser unit manual.

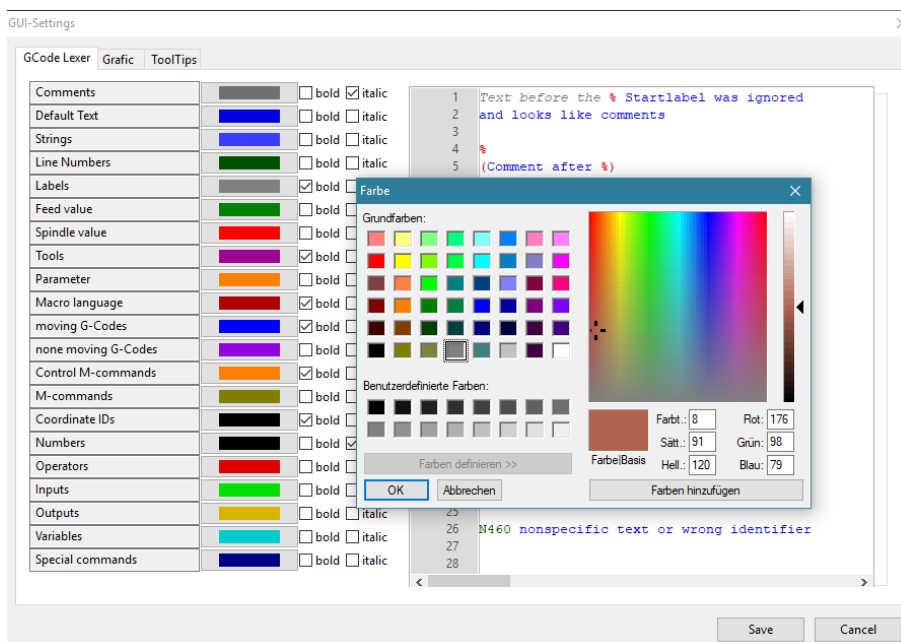
## 9.10 GUI-Settings

These settings affect the graphical user interface (GUI), i.e. 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: GCode-Lexer and Graphics.



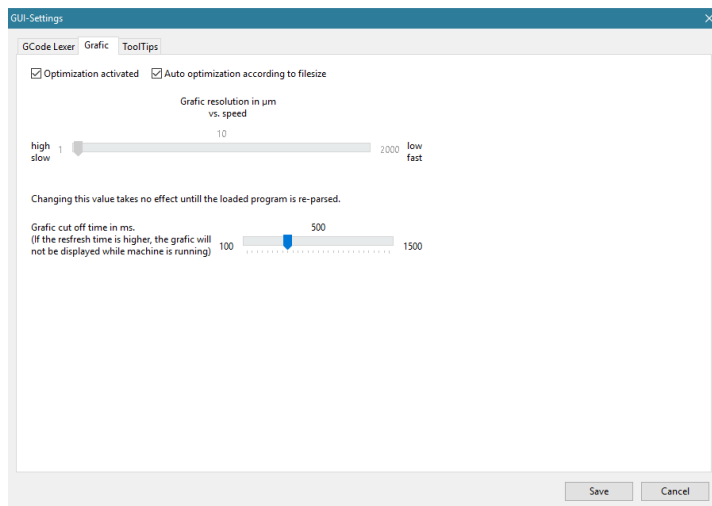
### 9.10.1 Change G-code representation

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





## 9.10.2 Change power settings



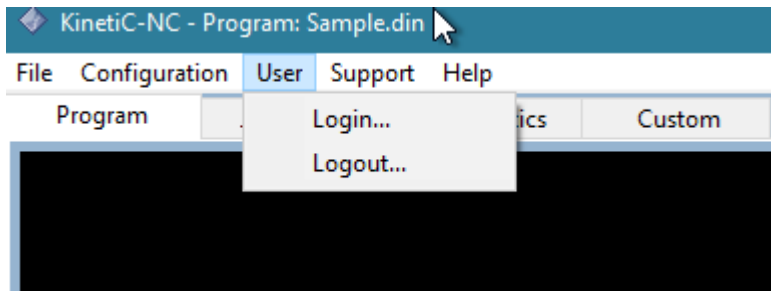
On the “Graphics” page the performance of the graphic display can be changed. For very large programs with several million line segments (depending on the performance of the PC and the graphics card from about 30MB file length) can lead to performance degradation. This is noticeable by the fact that 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 optimized. When the optimization is active, very short lines of the tool path are summarized in the graphic display.

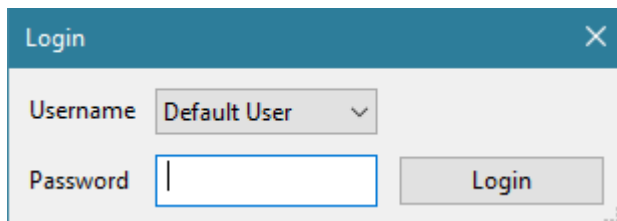
## 9.10.3 Accuracy vs. resolution

This has no effect on the accuracy of the actual tool path of the machine, but causes only 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.

## 10 The user-menu



### 10.1 Login...



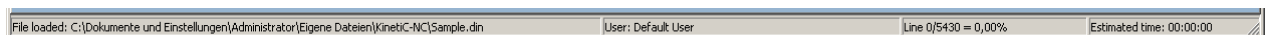
A small dialog opens in which the desired user can be selected. In the Password field, the corresponding password is entered and confirmed with "Enter".

By default, the default user has not assigned a password, so just confirm with "Enter" to close the dialog and log in.

To be able to change machine parameters, you have to log in as "Customer Admin", which has the default password of "1234".

#### 10.1.1 Automatic login

If the user has not assigned a password, the next time the program is started, it is automatically logged back in without opening this dialog



The currently logged in user is displayed in the status bar at the bottom of the main window, for example: "User: Default User" or "User Logged Out".

#### 10.1.2 No access?

So if you do not have access to certain menu items, make sure you are currently logged in with which user. Users can be assigned passwords to protect the machine from unauthorized manipulation (see Chapter user administration).

### **10.1.3 Lock the operation**

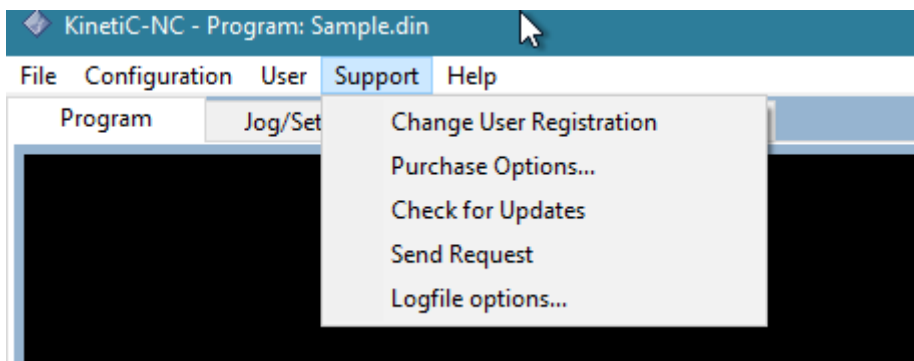
If you leave the machine without switching it off, you can also log out by clicking on "Menu-> User-> Logout".

The user is then logged off and the machine can no longer be operated.

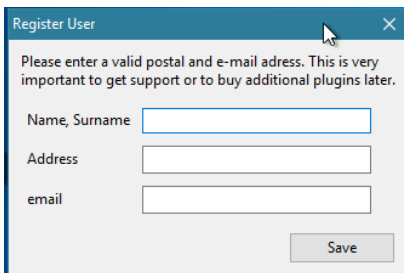
### **10.2 Logout....**

If you leave the machine without switching it off, you can also log out by clicking on "Menu-> User-> Logout". The user is then logged off and the machine can no longer be operated.

## 11 The Support-Menu



### 11.1 User Registration data



With this menu item you can open a dialog that allows you to register the software for a specific user name. Entering a license code is not required for the basic version (Economy) of the software.

#### 11.1.1 Why a registration?

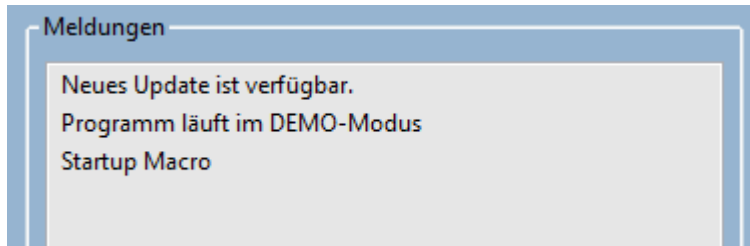
Registration may be useful if you want to use the "Send support request" feature. Entering your name and an email address is required in this case to assign the request and to contact you if necessary. Of course, this is only at your request. We do not share your information with third parties or use them for unsolicited advertising

### 11.2 Updates

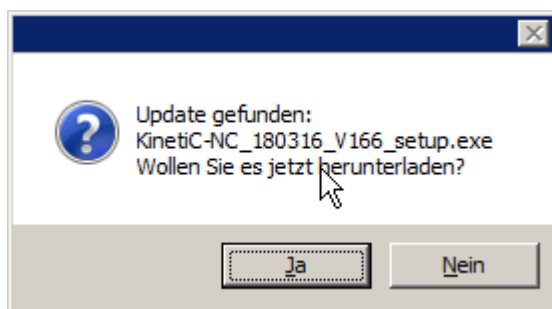


**Attention!** Unlike Internet browsers and anti-virus software, where it is always advisable to work with the latest version for security reasons, we strongly recommend that you do an update only when absolutely necessary, for example because an urgent problem has been resolved or you cannot do without a new feature. Be sure to make a backup of your settings before updating. **After the update, perform a functional check** before working on expensive workpieces to avoid any surprises due to changed behavior.

Assuming that the computer has an active internet connection, you will be informed about the availability of a new version of the program in the "Messages" window at the start of the program.



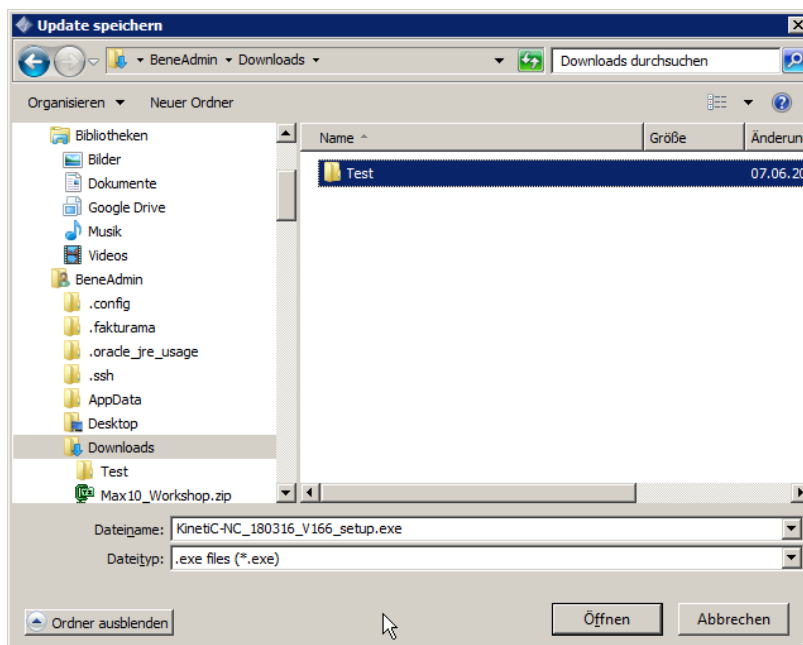
Before you start an update, it is always recommended to backup your settings. This is described in the chapter "create backup".



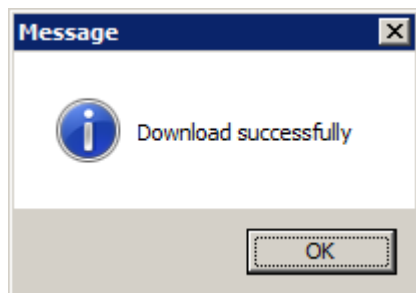
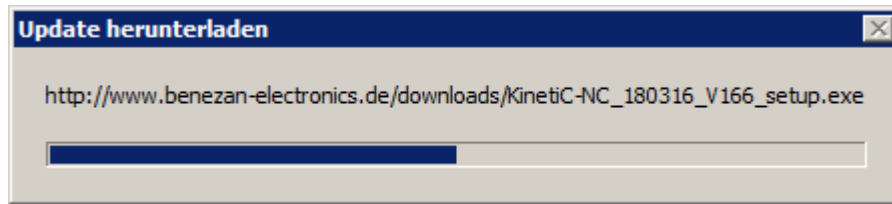
The update process is started via the menu item "Search updates"

A hint box opens, which version was found and if you want to download it now:

To save the update, choose a location on your computer that you can easily find. Ideal is, for example, the desktop, or the download directory.

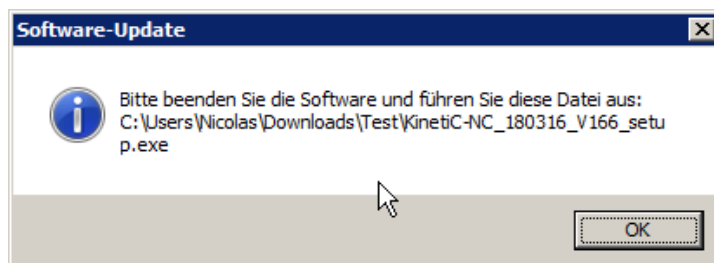


If you confirm the entry with Save, the Progress dialog opens and displays the progress of the download.



If the download was successful, you will be notified. If the download was unsuccessful, just start over again.

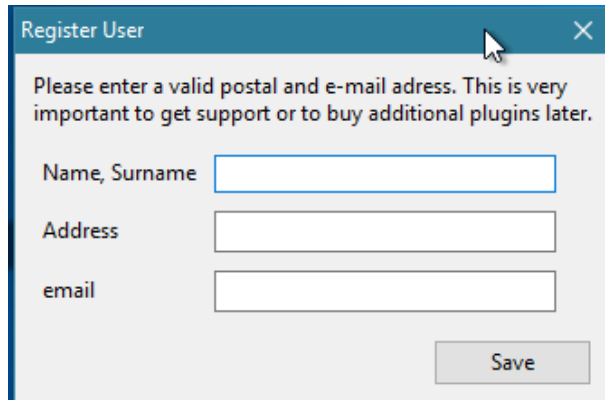
After the download, a note dialog opens with the exact path where the installation file was saved.



Now stop the program and install the update by starting (in this example) the "C:\Users\Nicolas\Downloads\Test\KinetiC-NC\_180316\_V166\_setup.exe". Follow the instructions to the *Install the Software* and follow the "Install Update" section. Do not forget the functional test!

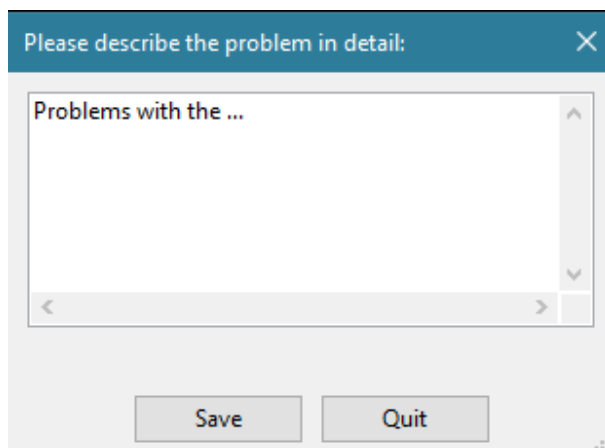
## 11.3 Send request

### Registration information



If you open the dialog for the first time and have not previously entered any license data, the license dialog opens first. This is necessary to be able to identify you as a customer in the support department. Please enter only correct data and remember that support can only contact you when all the data entered is complete.

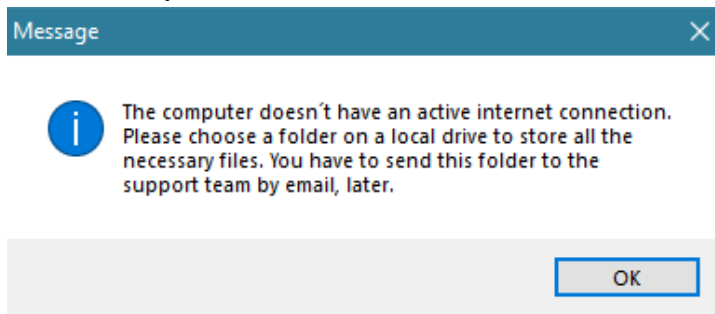
### Problem Description



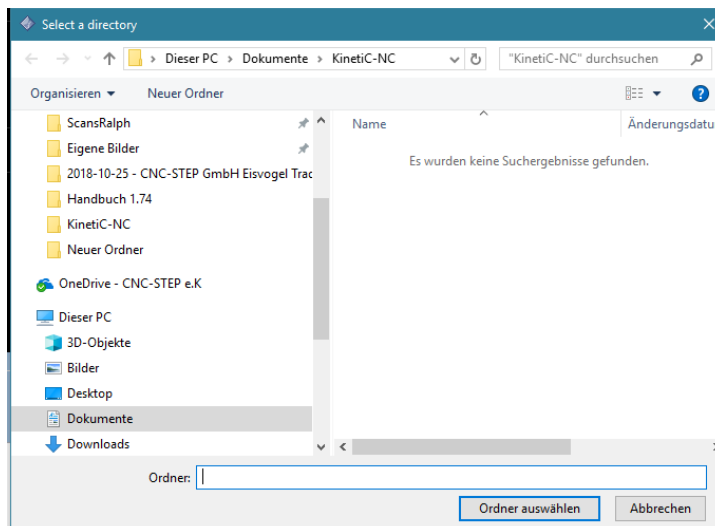
If you have problems with the software, you can use this feature to send an automatic e-mail to the support team. This contains all important information about the current state of the software and the last executed operations (log file). Apart from the machine parameters and the address entered in the license dialog, no personal data will be transmitted. The log file and parameters help the support locate the cause of the problems. Please describe your problem in as much detail as possible. Not: "machine does not work".

### 11.3.1 No Internet?

Usually, an email will be sent to support now. If the control PC does not have an Internet connection, you will be notified.

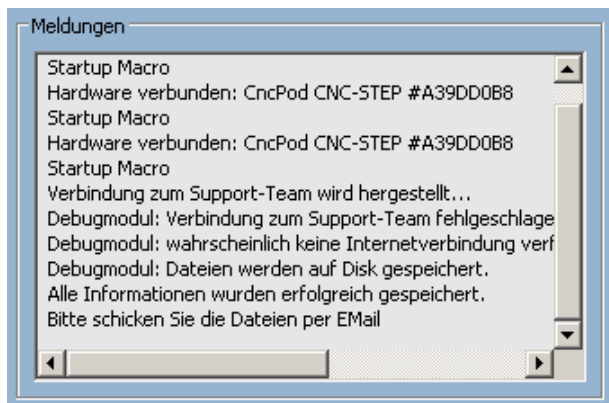


You now have the opportunity to save the information in a file, transfer it with a USB stick to another PC with Internet connection, and send it to the support via e-mail. The e-mail address of the support can be found on the dealer's website or via the help menu (info about ...).



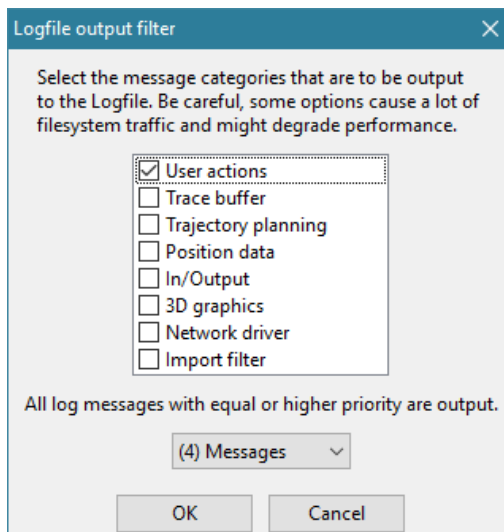


### 11.3.2 Check if everything has been sent



After successful transfer of the data or storage on a data carrier, you will be informed about this in the message window.

## 11.4 Logfile Options



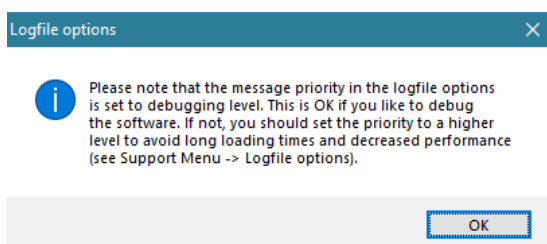
This feature allows you to determine what actions are logged in the log file and how detailed this information is. The more actions or message categories are selected, and the smaller the priority is set, the more data is written to the log file. In normal operation, the log file is not needed, and only a few categories and a high priority should be selected in order not to generate an unnecessarily high volume of data, which could possibly have a negative impact on system performance.

### 11.4.1 Support

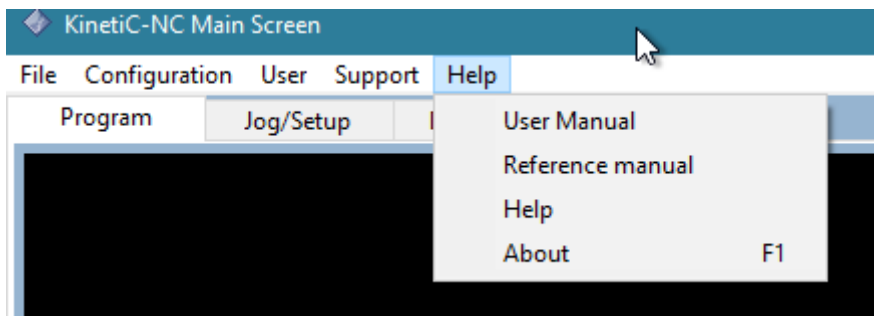
But the log file can become important when it comes to finding the root cause of problems. Therefore, you may be asked by support to enable certain log options if you report issues.

### 11.4.2 Important: Reset the log options

If, after submitting the log files to the support and fixing the problem, you reset the log files.



## 12 The help-menu



### 12.1 User manual

With this menu item you can open the PDF version of this manual directly via the software. To do this, the default PDF reader is opened. If you do not have a PDF reader installed, a warning appears that you must first install a PDF reader. For example, the Adobe Acrobat Reader from [www.adobe.com](http://www.adobe.com) or any other PDF reader.

This manual is also great for printing, if you prefer a printed manual.

### 12.2 Reference manual

With this menu item you can open the PDF version of this manual directly via the software. To do this, the default PDF reader is opened. If you do not have a PDF reader installed, a warning message will appear asking you to install a PDF reader first. For example, the Adobe Acrobat Reader from [www.adobe.com](http://www.adobe.com) or any other PDF reader.

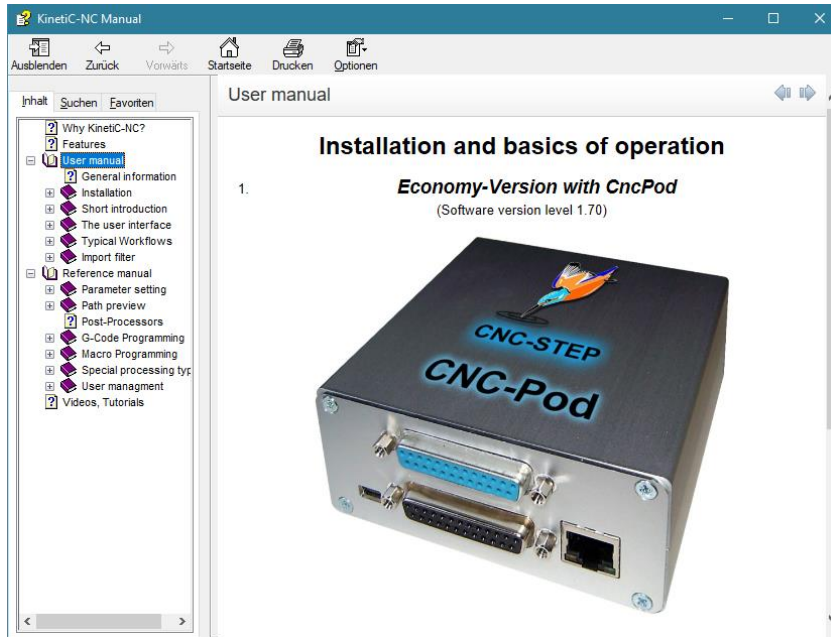
This manual is also great for printing, if you prefer a printed manual.

#### 12.2.1 Environment

Out of respect for our environment, we do not provide a printed manual about our software. As software changes and expands very quickly, new manuals would have to be constantly printed because they are never as up-to-date as electronic help. In addition, electronic help is interactive, and you can search for terms in it.

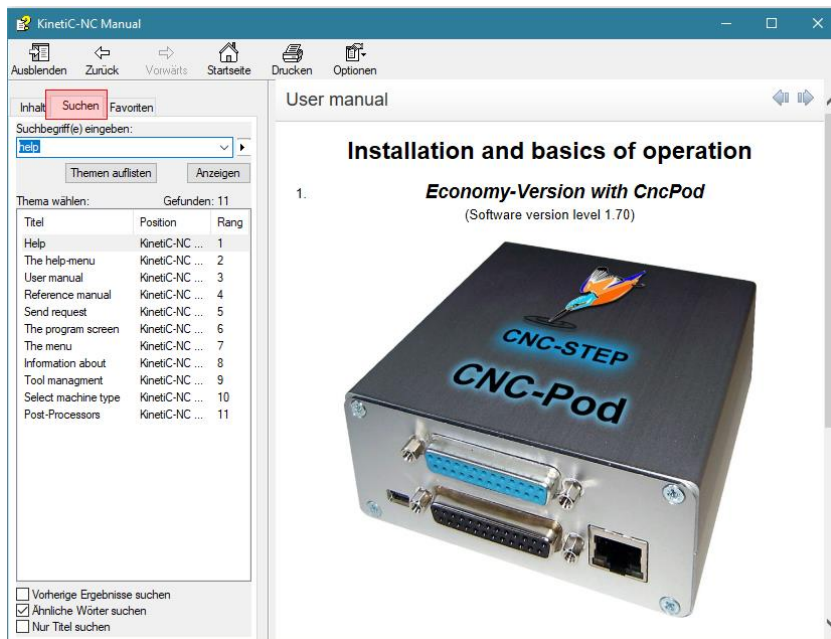
## 12.3 Help

This menu item opens the Windows interactive help for this program.



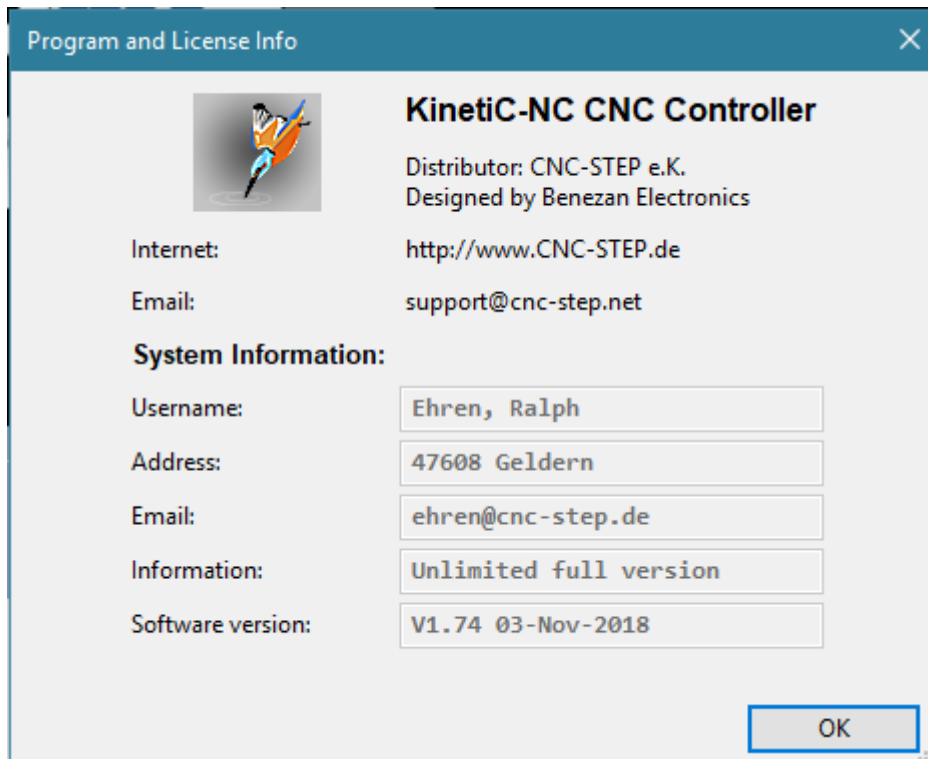
### 12.3.1 Search

With this help you have a powerful tool at your disposal. You can search for individual terms, which are then marked in the found text.



## 12.4 Information about...

Here you will see a message with information about the current software version (version number and creation date), the registered user data and the e-mail and website addresses of the manufacturer.



## 13 Typical work processes

There are many different machines, different processing methods, materials and applications, and of course different habits and preferences of the users. Of course, not all conceivable procedures can be explained in this manual. Only a few examples will be shown so that you can understand the basic procedure and adapt it to your own needs.

### 13.1 Processing plate/sheet material

The simplest case is when you use plate/sheet material to make parts that are completely milled out. This means that first holes and, if present, inner pockets are milled out. With the last processing step, the outline of the part is milled out and the part thus completely separated from the plate. From the raw material remains a thin frame and possibly leftover material. There are many applications that work on this principle.

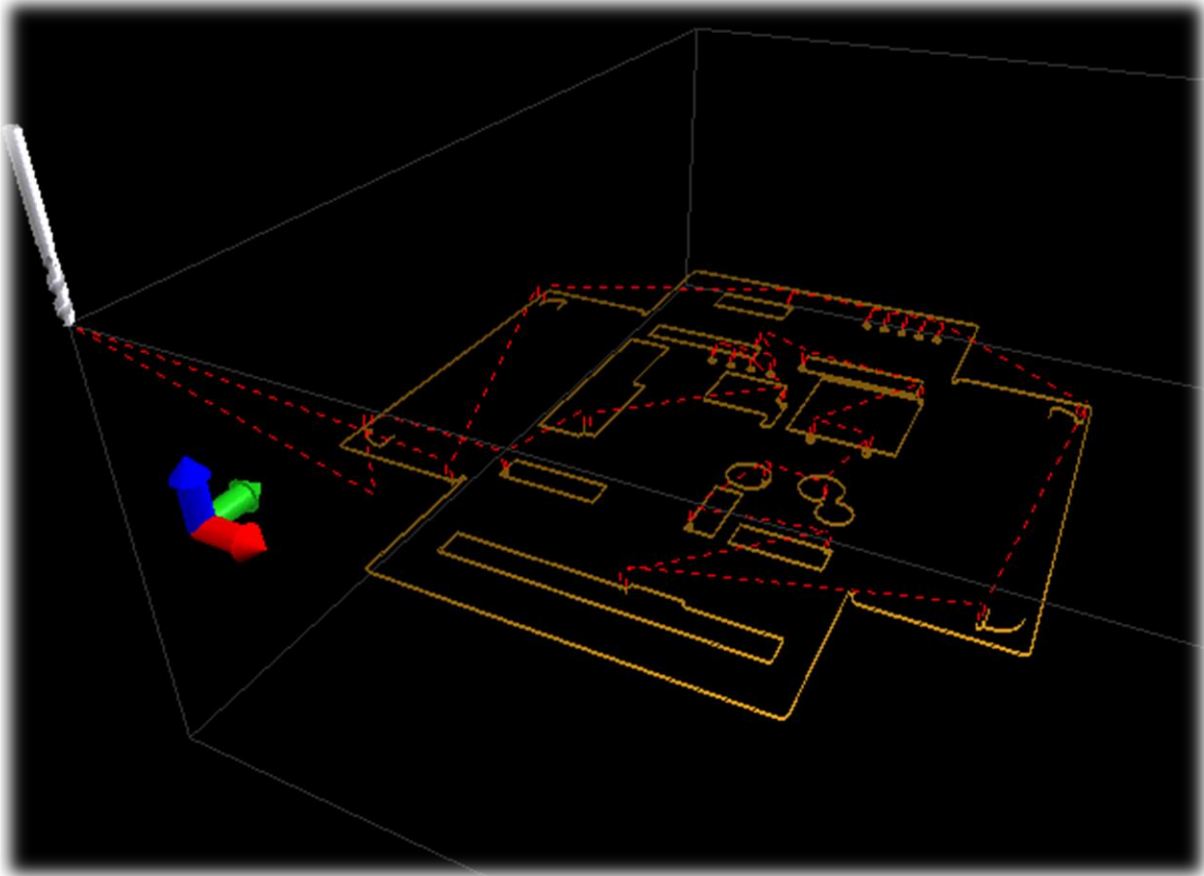
- Production of front panels for housings or sheet metal parts in general (also plasma cutting)
- Parts for modelling (plywood, fiberglass, CFK)
- Engraving signs

#### 13.1.1 Advantages

The advantage of this type of processing is that it does not depend on the exact alignment of the material. As long as the milling path is completely within the raw material, the parts will always be accurate. All you have to do is adjust the Z height exactly. If there are no blind holes or pockets, and a sacrificial plate is placed under the material, then this does not even have to be done very precisely.

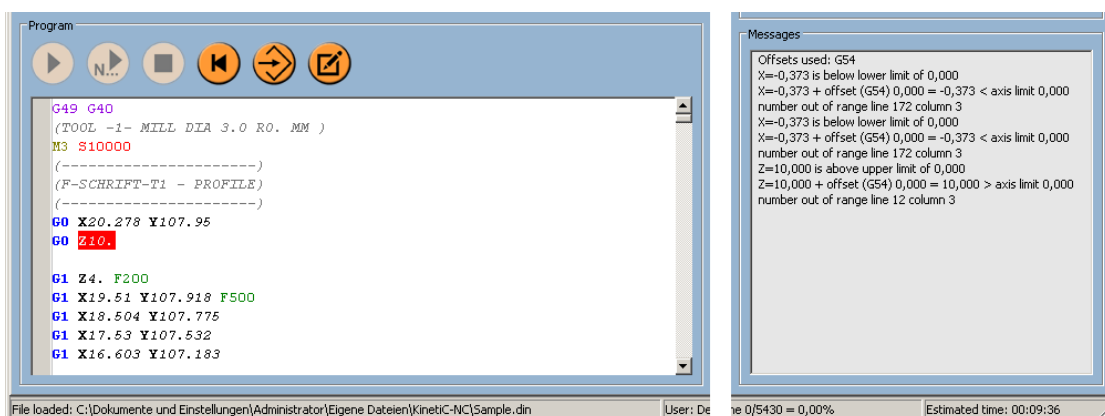
#### 13.1.2 Loading program

With „*File->Open NC file*“ you can load a program. For two-dimensional parts this is usually very fast, and a preview of the tool path is displayed in the graphics window.



The picture above shows an example of a sheet metal part with several holes and recesses. Milling paths along the contour are normally displayed in green (possibly with different tools in other colors), rapid traverse movements with dashed red lines. The position of the zero point is represented by the three arrows in red, green and blue.

The text window below shows the source code of the program. Normally, the first few lines contain comments with the name of the part and possibly further information about date of creation, tools used and more.



### 13.1.3 The message window

It may be that one or more error messages appear in the "Messages" window on the right. If "number out of range" is reported, this is not a cause for concern, just an indication that the origin of the drawing does not yet match the position of the workpiece in the machine. As a result, it may happen that part of the tool path lies outside the travel ranges of the positioning drives. In the example shown below, the tool path projects beyond the right boundary of the machine's working space, which is shown as a box with gray edges. The area of the tool path that violates the boundaries is displayed in orange.

### 13.1.4 Error messages

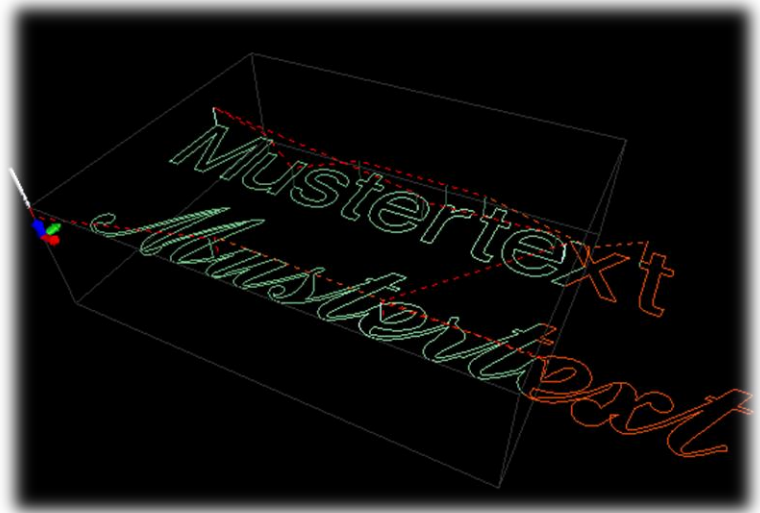
The associated error message could be for example:

```
X = 200,263 is above the upper limit of 200,0  
X = 200,263 + offset (G54) 200,00 = 400,263 > axis limit 400  
Number out of range row 30069 column 3
```

The first line shows that the coordinate in the NC program (G code in the program window) is programmed to a target position of 200,263. With the current settings, however, the X axis may only drive up to a value of 200.0. The second line gives more detailed information why this is the case: The currently selected zero point is G54. The X offset of the zero point is 200.0. The absolute upper limit for the X coordinate (set in the machine parameters) is 400.0. The X-coordinate from the program (200,263) plus the offset results in a sum that is above the limit. The third line indicates the text of the error within the program. The position will also be highlighted in red in the "Program" window on the left.

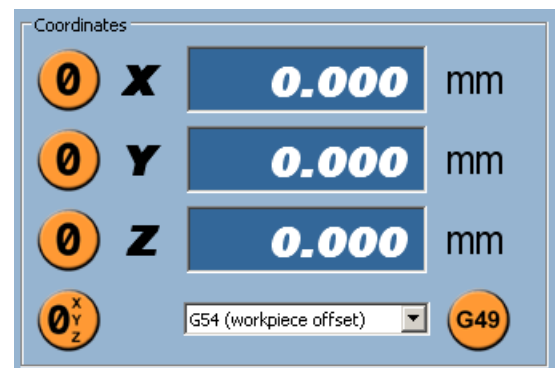
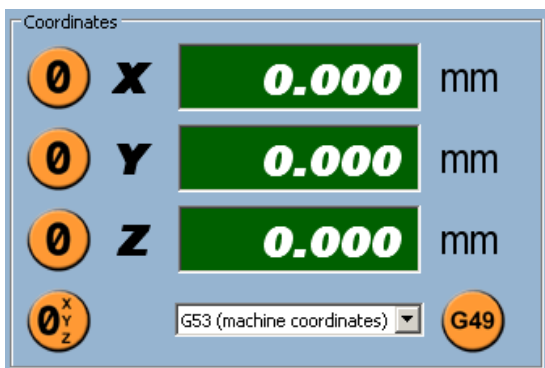
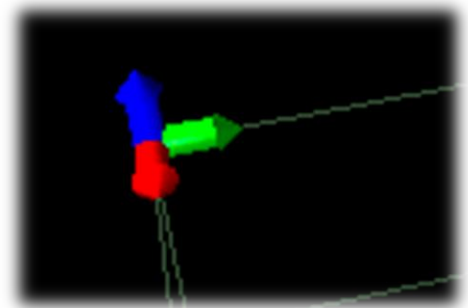
### 13.1.5 Presentation in the graphic

As you can see from the picture, the working surface protrudes to the right out of the processing area of the machine, and should be moved to the left, in the direction of smaller X-values.



### 13.1.6 Set zero point

The following description assumes that the zero point in the drawing is at the top of the material, as in the example of the image, and at the bottom left in the X / Y direction. If the coordinate display on the top right screen still has a green background color, then no workpiece offset has been selected and "G53 (Machine Coordinates)" is displayed below the coordinates.



First select an offset there, e.g. "G54 (workpiece offset)". The coordinate display changes to blue background color.



### 13.1.7 Place the material

After you have fixed the material on the table of the machine, now move with the Hotkeys (number pad) or the jog buttons on the Setup-Page to the point where the workpiece zero point should be. If you have just switched on the machine, a warning appears that no homing has yet been performed.

### 13.1.8 Homing



Select "Now homing" and then move to the desired position. The tip of the cutter should be just above the material surface and in the X / Y direction slightly inside the lower left corner of the material.



Now press the "all zero" button on the left side of the coordinate display. On the main page in the graph, the tool (light gray cylinder) is located at the point of origin (three colored arrows), and the tool path should have the desired position in the processing space, completely within the acceptable limits, as indicated by no part is marked in orange, and all lines are inside the gray box. In the message window no error message should appear anymore, only "used zero points ...".

### 13.1.9 Zero adjustment

You can now carry out a fine adjustment of the zero point, for example with the paper method described in the chapter „Setting up the zero point“.

Of course, if your work contains blind holes or pockets whose depth should be exactly right, you must now subtract the thickness of the paper from the Z coordinate of the offset. If you know this thickness (for example, using a calliper gauge), you could carefully and slowly lower the z-axis by that amount, and then press the zero Z-coordinate button again. It is better, however, not to risk a tool break, and enter the number directly. You can double-click on the Z coordinate display, and in the pop-up window that appears, for example, type "0.1". The zero point is then 0.1 mm lower than the current position of the cutter tip. This also avoids scratching the material surface.

### 13.1.10 Starting the program

If the zero point is set correctly, the graphic display shows the tool path as expected, and no error message appears in the message window, you can start the program.



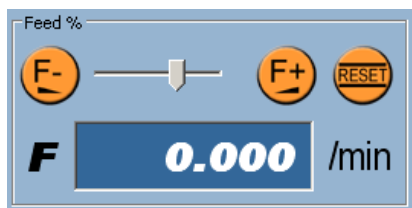
The button for this is located on the main page ("Program") on the left below the graphics window. In the event that an error message has been displayed or the text display ("Program" window below the graphic) is not at the beginning of the program, the start button may be disabled (gray). In this case, you can first click on



"Rewind", then the text display jumps to the beginning of the program and the start button becomes active again.



### 13.1.11 Adjust feed



Before starting a new program for the first time, it is recommended to set the feed override (slider "Feed%" top right) to a small value (eg 10-20%) to slow down the machine.

If you have made a mistake while setting the zero point, if the wrong tool is in the spindle, or if the machine makes an unexpected move to the wrong place, you still have time to react to an imminent collision.



Pressing the stop button stops the program, or in an emergency, of course, the emergency stop button on the machine.



If everything works as expected and without any problems, you can reset the Feed% slider to 100%, which is easiest with the Reset button.

### 13.1.12 Problems? How to stop?

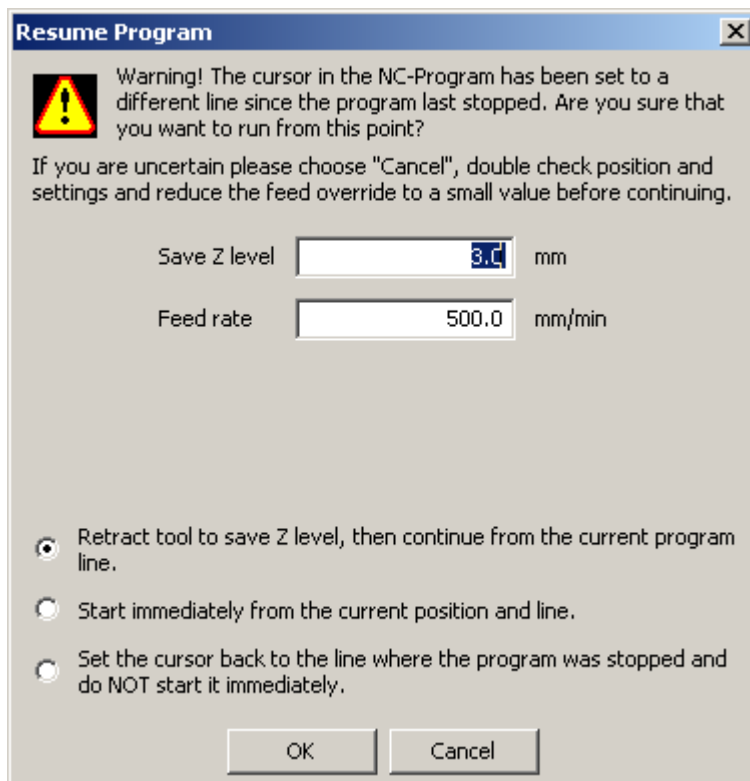
If a problem occurs during machining, for example, swirling chips around the tool, or the tool breaks off and needs to be changed, you can stop program execution at any time by pressing the Stop button. After a short delay until the spindle motor has come to a standstill, the start button is released again. After eliminating the problem, you can resume editing at the same location.

### 13.1.13 Continue again

However, after a tool break, the controller may have allowed a small part of the program to continue without the (broken) tool being engaged. In order not to omit any piece of the tool path, you would actually have to rewind this section again. Depending on how far you have travelled since the break, you can scroll up a few lines in the text window and double-click on the desired line.



The left start button is disabled (gray) and the second button to the right is activated. When starting with this button, a window now appears with a warning as shown in the picture below.



### 13.1.14 Start program at any point

Retaining a program at any point carries a certain amount of risk depending on the type of workpiece, the tool and the machining strategy. In extreme cases, such as a T-slot cutter, which is located in the middle of the material, no resuming at this point is possible at all, without damaging workpiece or tool. Since the controller cannot automatically detect such hazards, the responsibility lies with the operator in such cases. If in doubt, calculate that the tool travels diagonally to the next machining point with all axes at the same time, in other words under certain circumstances it may be that there is still raw material on the way there. If you cannot be sure that a collision will not occur, then rewind until you reach a non-hazardous location, such as a safe Z-height outside of the material.

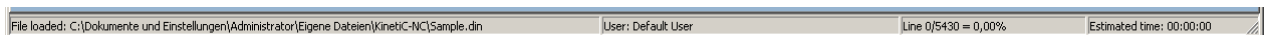
For short programs that only run for a few minutes, and if the part is not too valuable, it may not be worth continuing, and it is better to run the program completely from scratch, instead of risking another tool break.

### 13.1.15 Additional tip

The title bar at the top of the main window displays the name of the currently loaded program.



The status bar at the bottom left shows the complete pathname of the loaded program. To the right, the line number is displayed, how far the processing has progressed



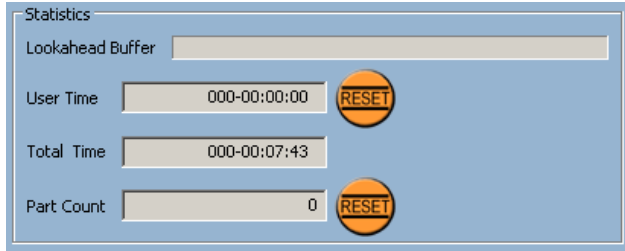
If you cancel a program, and If you want to continue the next day, make a note of this line number to find the right place to continue later.

### 13.1.16 Execution time

At the far right, the preview displays the estimated execution time after loading the program, and the elapsed time during execution.

## 13.2 Statistical functions

### 13.2.1 Running-time meter



On the diagnostics page, the hours and parts counter are displayed at the bottom of Statistics. The operating hours counter and the user clock always run while a program is running.

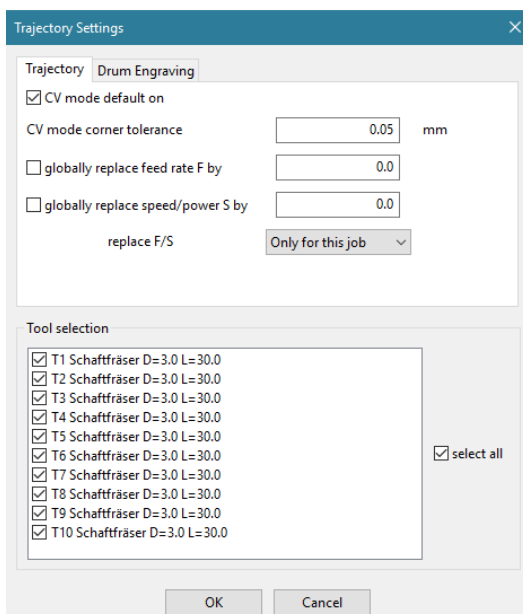
### 13.2.2 Parts counter

The part counter is incremented by one with each complete execution of a program. The user clock and the parts counter can be set to zero if necessary.

### 13.2.3 Tool path settings



Right next to the tool display (in the middle column of the main window) there is a button for the tool path settings.



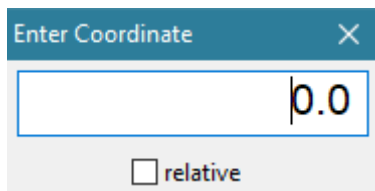
This can be useful in order to influence the path of tolerance (chord error) or feed and spindle speed afterwards, without having to change machine parameters, or having to edit the program in the text editor. If the program contains several machining steps with different tools, these are displayed below under "Tool selection". It is possible to exclude individual work steps, or to select only one of them.

## 13.3 Other material

If the workpieces do not have to be completely milled out of plate material, but you have to (re) process blocks, cylinders, castings or pre-machined workpieces, they are usually clamped in a vise or held on the machine table with clamps. If not, the entire surface is machined, the workpieces must be aligned exactly. You therefore need not only measure the Z-height, but also the position of the workpiece in the X and Y direction

### 13.3.1 Position of the workpiece

In the simplest case, this can be done again by "scratching" the surface with the tool itself. To do this, drive sideways to one edge of the workpiece until the tool just touches the edge. Now the axis of the milling cutter is half a diameter from the surface. For example, if you drove to the edge in the X-direction, and the tool diameter is 6mm, then the cutter center is 3mm from the edge.



Double-click on the X-coordinate display and enter "-3". Of course, the same procedure is necessary once again for the Y and Z coordinates.

### 13.3.2 Probe

The measuring process can be automated if you have a 3D probe. In contrast to the height sensor, the 3D button also reacts when actuated from the side and is inserted into the tool spindle. The macro "corner finder" can be used for the lateral probing of the workpiece edges.

### 13.3.3 3D-probe

The 3D probe is also suitable for measuring bores as a reference point. The tip of the 3D button must be placed within the hole. The "Center Finder" macro probes the wall of the hole in several places and determines the position of the hole center. After execution of the macro, the measuring tip is exactly in the center. The X and Y coordinates can then be set to either zero or (by double-clicking) on the known coordinates of the reference point.



### 13.3.4 Camera-centering

An alternative method for determining edge or bore positions is to use a miniature camera as a centering microscope. For this purpose, special cameras are commercially available, usually with USB port and a cylindrical shank, which can also be used in the recording of the tool spindle.

## 13.4 Working with height probe

### 13.4.1 What is a height probe?

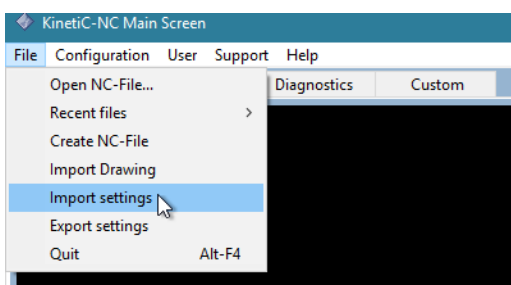
A height probe is a small device that gives an electrical signal to the controller when it is driven from above with a tool. It can either be used - permanently installed - to measure the length of the tool after a tool change. In this case we speak of a tool length sensor. Or it is used mobile to determine the height of the workpiece surface, and to set the zero point automatically. This is a significant reduction in labour compared to the direct "scratching" of the workpiece surface or the "paper method" mentioned above.



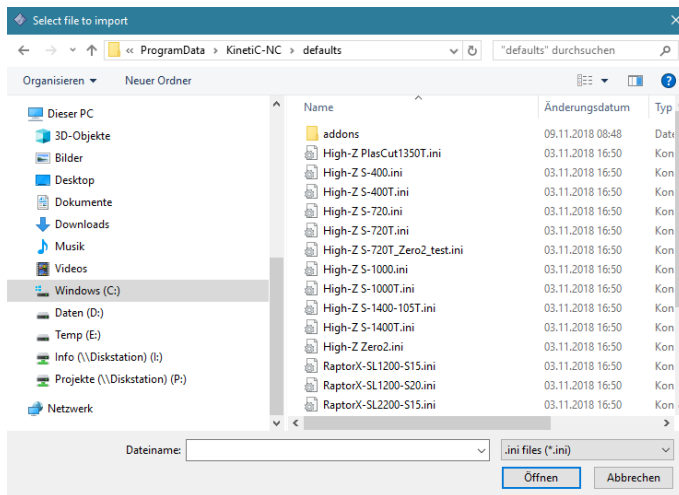
### 13.4.2 The procedure

The procedure is as follows: The button is connected to the controller and placed on the material surface. The tool is moved vertically over the button of the probe. Then a macro is called, which means a small program is started, that does the measurement automatically. The tool moves slowly on the probe until the switching point is reached. The height of the probe itself, from the top button to the bottom of the case, is subtracted from the position reached, and the result is stored as the Z coordinate of the workpiece offset. Thus, after that, the coordinate zero point is located exactly on the workpiece surface.

### 13.4.3 Connection

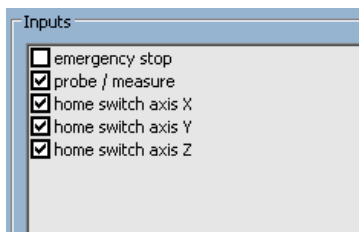


The height probe is normally connected to an extra socket on the machine or controller. For more detailed information, please refer to the documentation for your control hardware. To use the macro for height measurement, you must first import it.

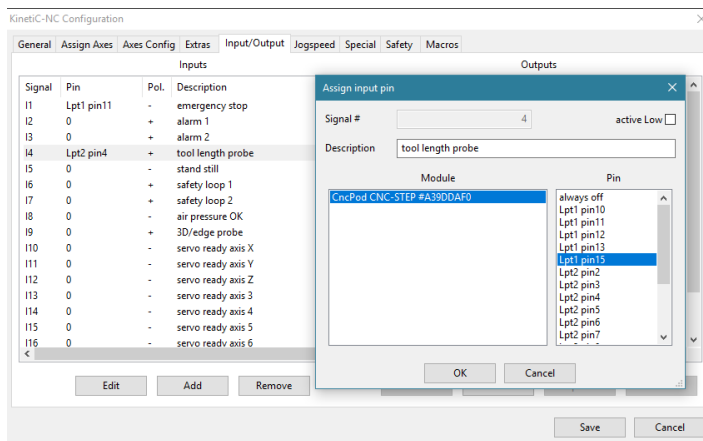


To do this, go to „**File->Import settings**” and select the file "HighZ\_LengthProbe.ini" in the "Addons" subdirectory under "Default parameters". This will both install the required macros and assign the input for the probe with the standard pin-assignment.

### 13.4.4 Testing before use



It is very important that you check the correct function of the probe before first use. To do this, please switch to the Diagnostic page.

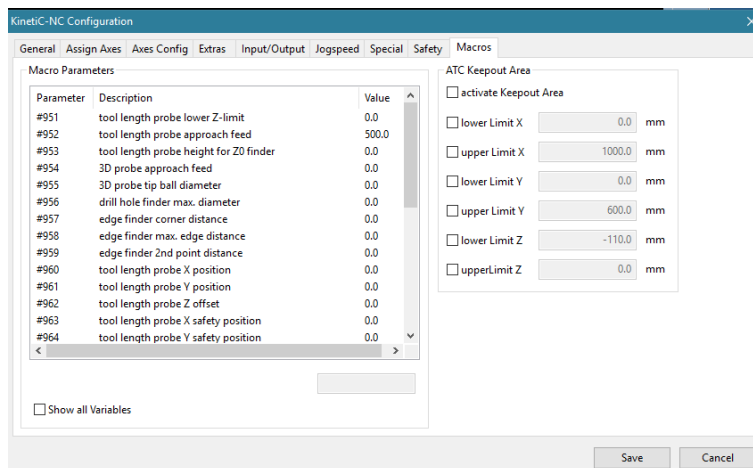


For the inputs (top right), a "tool length sensor" entry should be visible with a box to the left. In the normal state, no check mark may be displayed. Now press the button with your hand. Now a tick must be visible. If nothing changes, the probe is either not connected correctly or the signal is assigned to the wrong pin.

If the behavior is exactly the opposite (tick visible if button is not pressed), then you must invert the signal. You can do this under „**Configuration->Machine->Inputs/Outputs**” by clicking on the input "Tool length sensor" and then "Change". To invert the signal, change the state of the option "low-active", top right.

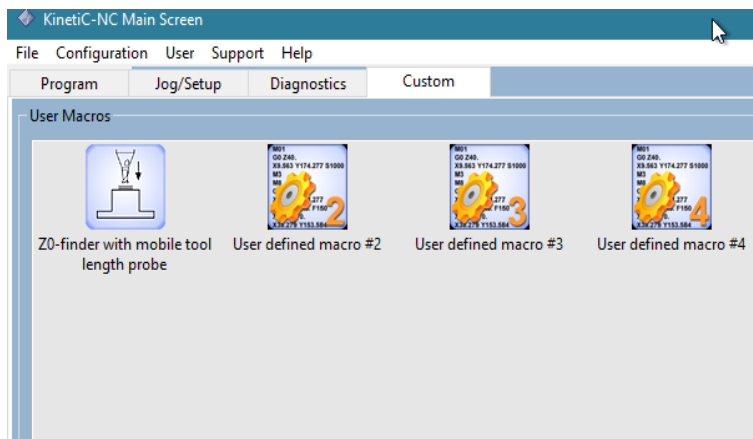


### 13.4.5 Height of the measuring probe



Now the height of the probe must be determined and entered into the machine parameters. First determine the approximate height of the probe with a caliper gauge and enter it in the configuration dialog on the page "Macros" at Variable number # 953 "Tool length sensor height for Z0 determination".

Now place the probe on the material and the tool via the probe. If you now call up the macro (double-click on "Height measurement with mobile probe" on the *Special Page*), the measuring process will be carried out.



If you now remove the probe and lower it with the tool until it just touches the material, a Zero coordinate should be displayed near Zero. The small difference - ideal would be exactly zero - comes from the fact that the real switching point of the probe is located at a slightly depressed ram, and this cannot be measured accurately with the caliper.

Therefore, the variable # 953 needs to be corrected slightly.

If a positive Z value is displayed, the offset has been set too low and you must reduce the variable by the displayed Z value. Conversely, if the Z is negative, the variable must be increased by the amount.

## 13.5 Tool management

### Why?

As soon as you need more than just one tool per workpiece, you have to think about how to make it happen that, despite different lengths, diameters and shapes, the machining is done in the right place and the right plunge depth. That means, the controller must "know" which tool is currently in the machine and how long and thick it is, etc. There are different tool change methods that require different approaches.

### 13.5.1 Without tool change

This is the simplest case. Only one tool is used per CNC program. This sounds like a drastic limitation, but it's not that bad when it comes to CNC machining, at least if you only work on thin plate material. You can, for example, to produce holes with different diameters, select a tool to match the smallest hole, and mill for the larger one circle. In this way, one often gets amazingly far with a single tool.

#### 13.5.1.1 Distribution of the program

If you need several tools in succession, the processing is divided into several programs. Each processing step is carried out separately. In between, the tool is changed manually and then the zero point is re-measured. In this case you only need a single tool (T1) in the tool list. You can enter the tool data in the list (menu "Configuration -> Tools"), but this is only relevant for the display on the screen. Since the workpiece offset is re-measured after each change, the entered tool length is irrelevant, and you do not have to worry about the tool number. The setting in „*configuration->machine->general*“ is best set to "Ignore tool change" to avoid unnecessary requests to change tools. You have to have the tool anyway before the start suitable for the program have changed.

### **13.5.2 Manual tool change (direct)**

Even if one does not have the "luxury" of an automatic tool changer, one can work off programs with several different tools. In this case, the setting in "Configuration -> Machine -> General" is set to "Tool change manual". In addition, you should create at least as many tools in the tool list as occur in the programs used, eg. T1 to T10.

### **13.5.3 Directly used tools**

If the tool spindle does not have an exchange system with extra tool holders (usually SK = steep taper, HSK = hollow shaft taper), but the tool with the shank is inserted directly into the collet chuck in the motor shaft, this is called direct change. The advantage is that this can be done with any reasonably priced tool spindle. Disadvantages are that the tool can be in a slightly different position after each change, and that with different shank diameters each time the collet must be replaced with. This requires more time, and usually the tool length must be re-measured after each change.

## **13.6 Tool length sensor**

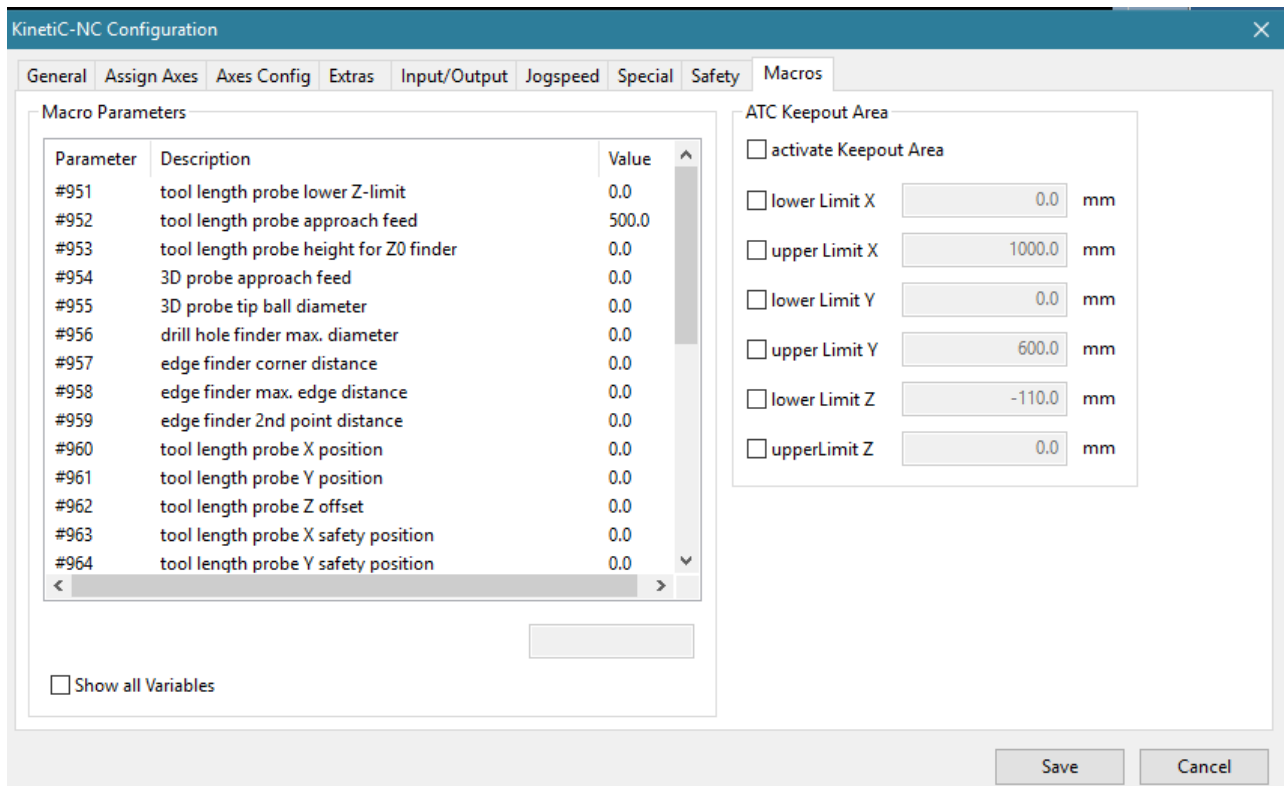
For this reason, when switching directly, it is strongly advised to purchase a tool length sensor. This is the same as the height probe described above, only fixedly mounted at a certain point on the machine. There are also combined length and height probe, where the same probe can be used for both purposes. Such a button usually has a magnetic foot, which is firmly mounted on the machine. There, the probe is used for tool length measurement. For mobile application to measure the workpiece surface height, the probe can be removed from the foot and placed on the workpiece. In contrast, if two separate buttons are used, they can nevertheless be connected to the same input of the hardware (series connection) because the buttons are never used at the same time.

### **13.6.1 Without tool length sensor**

If you do not have a length sensor, it is best to use the "no tool change" method and separate the programs so that each machining step is performed separately with just one tool. Alternatively, the program can also be left "in one piece", but only one of several tools can be activated each time it is run. You can do this by switching off the "Select all" toolbox in the Tool path dialog and activating only a single tool.

## 13.6.2 With tool length sensor

To perform the length measurement automatically after every tool change, you must first install the survey macros as described in Chapter 27 "working with hight probe" After this, the position of the button must still be entered in the machine parameters, so that the controller knows where the length sensor is, what height it has, and at what speed it can be approached.

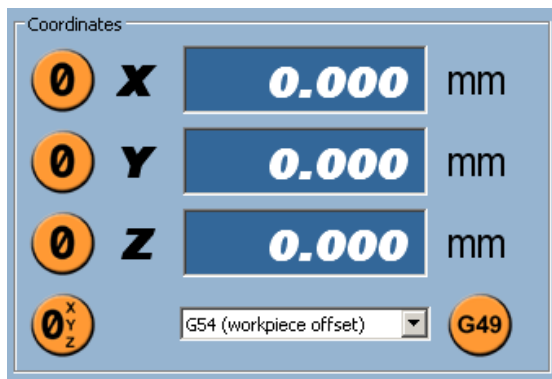


The parameters # 951, # 952 and # 960 to # 964 are used for this purpose. First enter the X and Y position where the button is mounted on the machine table. The safety position is approached after the measurement. This should be somewhere in the distance next to the button. This prevents the probe from being damaged if, after a measurement, a program is started which moves as the first movement with the Z axis all the way down.

The "offset Z-axis" (# 962) can later be adjusted so that the measured tool length corresponds to a dimension (in principle arbitrary but verifiable), for example the length of the part of the tool that protrudes from the collet. The exact value is actually insignificant, since only the difference in the length of the old and the new tool is relevant when changing tools. If a different offset is entered, all tools are measured longer or shorter, but the relevant length differences when changing tools remain the same.

Enter the appropriate values for your machine and save it. Remember to check the function of the signal input before using it for the first time by pressing the probe manually and observing the response of the input on the diagnostics page.

### 13.6.3 G40/G43 Mode



When working with several tools, it is important that the tool length compensation is switched on. Only then is the length of the current tool taken into account and added to the programmed Z value. In particular, the length compensation must also be active while you measure and set the workpiece offset (either with the height key or with the "scratch method"). Therefore, before setting up the workpiece offset, make sure that "G43" appears next to the offset by pressing the button.

## 13.7 Manual change with tool holders

The tool change can be somewhat more rational if the machine has a change system with tool holders and pneumatic or hydraulic clamping. Some machines also have a threaded pull stud that needs to be tightened with a wrench by hand. Each tool is first inserted in a tool holder (usually SK = steep taper, HSK = hollow taper or other systems). During the actual tool change, the pictures are then loaded into the tool spindle. The tool remains in "its" receptacle as long as it is not worn or damaged.

### 13.7.1 Advantage

This has the advantage that the length measurement must be performed only after the insertion of a new tool in the recording. This can be done either automatically as described above with a length sensor or manually with an extra length measuring device (in the simplest case with a calliper) outside the machine. The determined length is entered in the tool list and is then only looked up there during the actual tool change.

### 13.7.2 How is measured?

The absolute length of the tools is actually irrelevant, only the differences between the tools are considered. For example, with all tools, it is possible to measure the distance of the tip to a specific reference plane, for example the stop plane of the HSK cone. The important thing is that each time exactly the same reference plane is taken. The front of the collet nut is therefore unsuitable because its exact position depends on the tightening torque.

### **13.7.3 Simple entry in the list**

The manual change with tool holder does not require special macros or other adjustments if you enter the tool lengths manually in the tool list (Configuration -> Tools -> Change -> Length 1). If you have a length scanner, you can install and customize the G79 macro as described above. However, it is not invoked every time a tool is changed, but only when needed, either via the MDI input or by selecting the option "with length measurement" in the tool change dialog (main window, "tool change" button, left). As described above, you must also make sure that the tool length compensation (G43) is active.

## **13.8 Automatic tool change (direct)**

If the tool spindle has a pneumatically or hydraulically operated mechanism with which the collet can be opened and closed without manual intervention, an automatic change of the tool is possible. Of course, this also requires a magazine in which several tools can be stored.

### **13.8.1 Magazine types**

There are fixed (linear) magazines in which each tool is picked up at a different position by utilizing the X, Y and Z positioning of the spindle, or moving magazines (revolvers) where the pickup is always at the same X / Y / Z position, but there is an additional drive to select the tool.

Since the exact position of the tool is not always the same when changing directly into the collet, a length measurement must be carried out after every change. A length measuring probe is therefore necessary in every case. The setting in "Configuration -> Machine -> General" must be set to "Tool change automatic".

### **13.8.2 The M6 and G79 macro**

For the actual tool change, an M6 macro must be installed and adapted, for the length measurement the G79 macro. Special programming is required to program the automatic change. The Reference Guide lists examples of macros for each simple linear and revolver magazine. Even more complicated systems, for example linear magazines with several rows or revolvers with additional robot arm are conceivable. However, it is recommended that you seek professional help in programming and adjusting such changer systems.

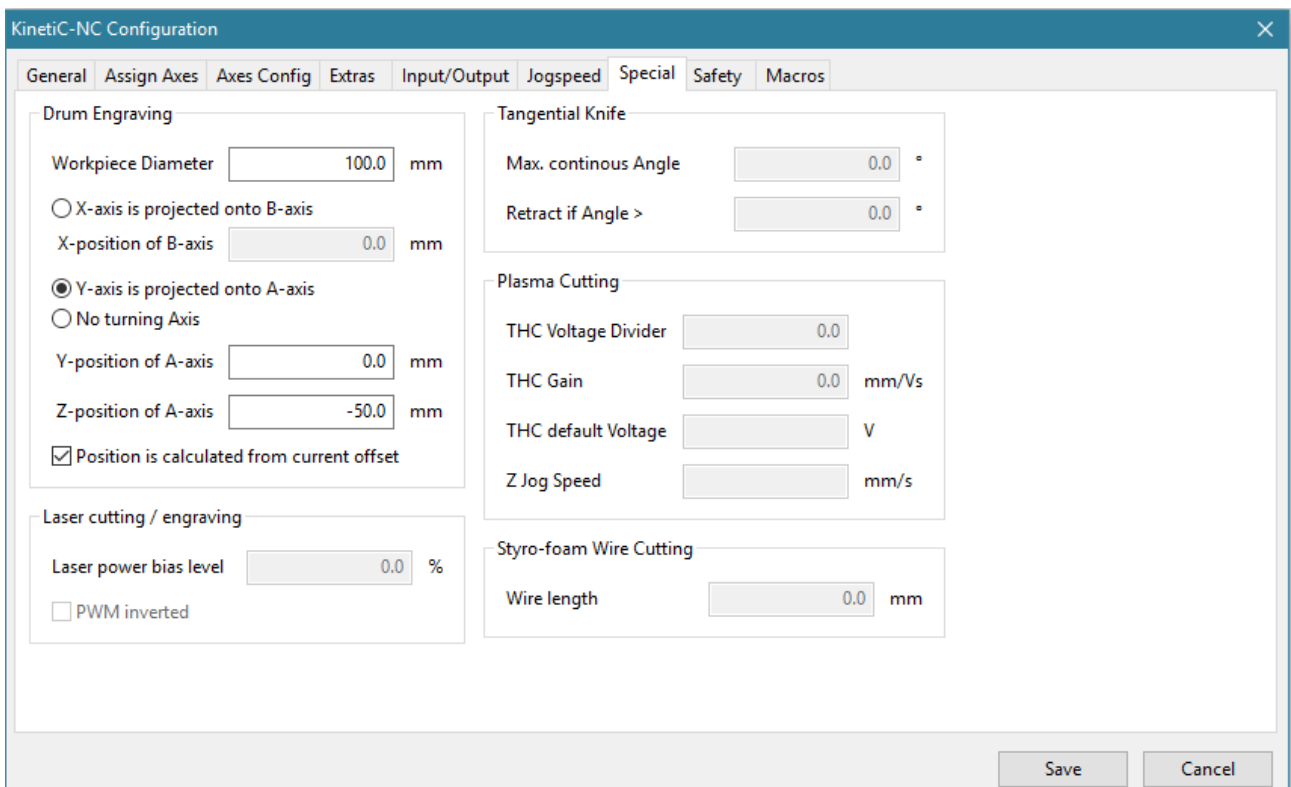
## 13.9 Automatic change with tool holders

If the tool is not inserted directly into the collet but into its own tool holder, the length measurement after each change can be dispensed with, as with manual replacement with tool holder. This results in the advantage of shorter cycle times in mass production but is also the most expensive system. As with automatic direct change, several magazine types are possible again. A length measuring probe is not absolutely necessary. If the tool lengths are measured by hand and entered in the tool list, no probe or G79 macro is required.

## 13.10 Rotary axis and cylindrical engraving

With the normal, three-axis machining (X, Y, Z), the workpiece can only be machined from one side - vertically from the top. If you also want to machine the reverse side, you will need to turn the workpiece over after the first pass and start a second pass. With a rotation axis (available as an accessory), you can rotate a workpiece at any angle and work from all sides around the circumference.

In the following description, it is assumed that the rotation axis is already connected to the controller, and the parameters (resolution, speed, etc.) in the machine parameters are set correctly. The necessary steps for the installation can be found in the separate documentation for the rotary axis.



The screenshot shows the 'KinetiC-NC Configuration' window with the 'Special' tab selected. The settings are organized into several sections:

- Drum Engraving:**
  - Workpiece Diameter: 100.0 mm
  - X-axis is projected onto B-axis
  - X-position of B-axis: 0.0 mm
  - Y-axis is projected onto A-axis
  - No turning Axis
  - Y-position of A-axis: 0.0 mm
  - Z-position of A-axis: -50.0 mm
  - Position is calculated from current offset
- Tangential Knife:**
  - Max. continuous Angle: 0.0 °
  - Retract if Angle >: 0.0 °
- Plasma Cutting:**
  - THC Voltage Divider: 0.0
  - THC Gain: 0.0 mm/Vs
  - THC default Voltage: V
  - Z Jog Speed: mm/s
- Laser cutting / engraving:**
  - Laser power bias level: 0.0 %
  - PWM inverted
- Styro-foam Wire Cutting:**
  - Wire length: 0.0 mm

Buttons for 'Save' and 'Cancel' are located at the bottom right of the window.

### 13.10.1 Cylindrical engraving

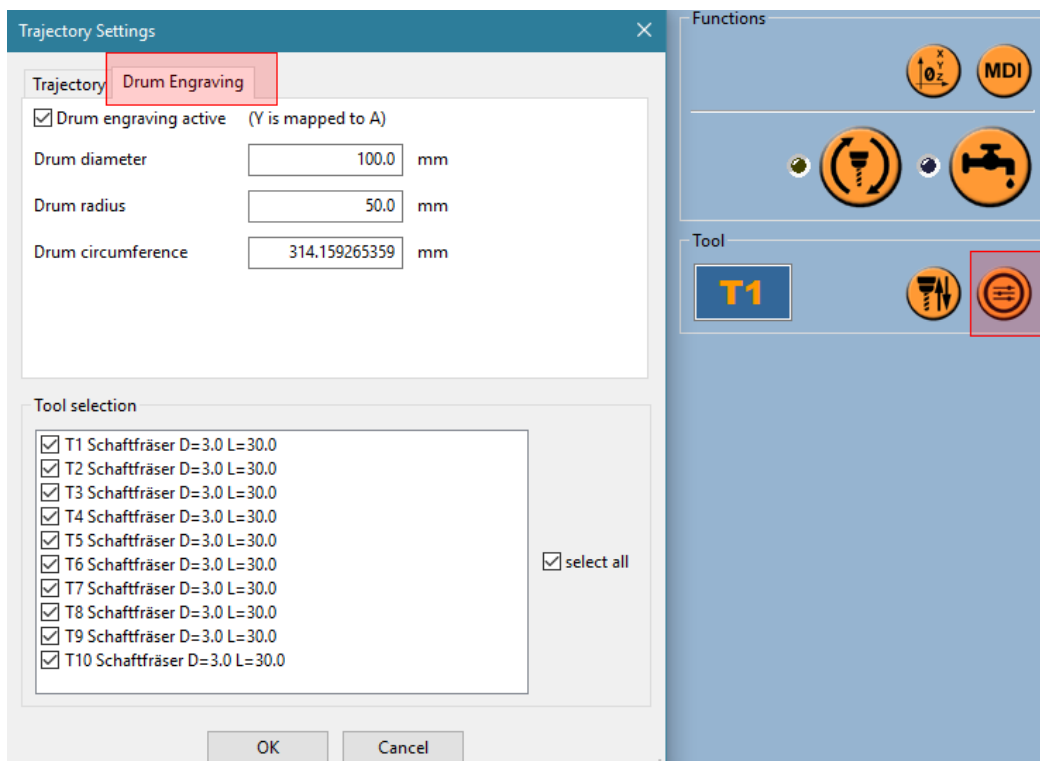
Drum engraving is the simplest machining method with a rotary axis. The normally flat working plane (XY) is "wound up" on the circumference of a cylindrical surface. If the axis of rotation is aligned parallel to the X axis, the axis of rotation (A) is moved, for example, instead of the Y axis. To avoid distortion of the image, the workpiece diameter must be known. This is necessary for the controller to calculate how many degrees of turn it takes to travel a certain distance on the surface.

### 13.10.2 Simply without CAM

For drum engraving, no special CAM program with rotary axis support is required. You can even use an existing NC program that was originally designed only for flat plate flat machining, later for drum engraving.

### 13.10.3 Where is the rotary axis mounted?

All you have to do in the machine configuration (*configuration->machine->special->drum engraving*) (see picture above) is see if the axis of rotation is parallel to the X or Y axis.) Normally the axis of rotation will be parallel along the long side of the machine In this case, select "Y-axis is projected on A-axis" and activate the option "Position is determined from current zero point" as shown below: workpiece diameter and Y and Z position of the axis of rotation are still irrelevant at the moment and will be specified later.



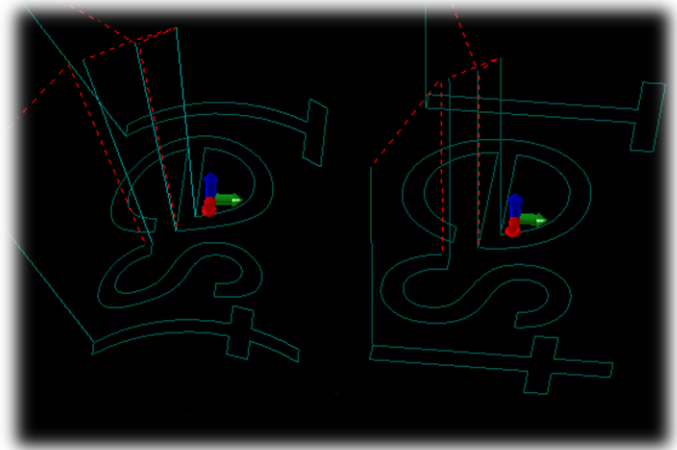


### 13.10.4 Insert workpiece

After inserting the workpiece into the rotation axis, the workpiece zero point must be placed on the material surface vertically above the center. For an A-axis of rotation parallel to the X-axis, move to the workpiece so that the tip of the cutter just touches the top of the cylinder (pipe, rod, etc.) and set the Y and Z coordinates to zero.

### 13.10.5 Probing the workpiece

In order to find exactly the center point in the Y-direction, you can either touch the workpiece on both sides laterally, or instead of the actual workpiece first clamp a cylindrical pin in the jaw chuck whose diameter matches that of the tool, pin and cutter congruent drive over each other, and first set the Y axis to zero. Then clamp the real workpiece again and set the Z coordinate to zero on the top.



### 13.10.6 Load NC program



Open now the NC Program. The tool path is initially displayed "flat", i.e. parallel to the XY plane.

Now go to the tool path dialog via the button to the right of the tool display (see picture above right). Switch to the "Drum Engraving" tab and activate the check mark "Engrave drum active". If you now enter the correct workpiece diameter (or optionally radius or circumference) and exit the dialog with "OK", the graphic is rebuilt and the tool path appears curved, wrapped around an imaginary cylinder. The picture on the left shows an engraving once with drum engraving turned on and once "flat", with drum engraving turned off.

### 13.10.7 Turn off drum engraving

If you want to process a "normal" workpiece after finishing work, you should switch off the drum engraving option again. If the train preview looks strange, or you get inexplicable error messages, this may be a sign that you have forgotten it.

## 13.11 Tangential knife cutting

### 13.11.1 What is it?

This machining method does not work with a tool that rotates at a constant speed, but rather a knife is always rotated at the correct angle tangential to the tool path with an extra positioning drive, this explains the name. The knife may be either rotatable but vertically fixed or oscillating. With the fixed knife usually slides are cut, with an oscillating knife even thicker materials such as foams can be cut.

### 13.11.2 Import of option

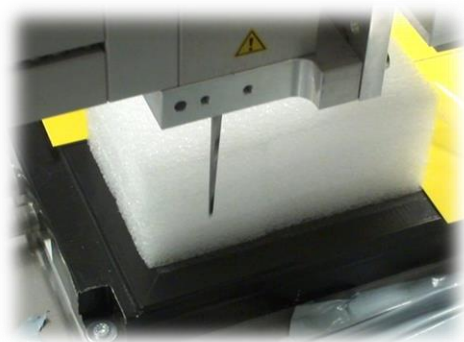
The Kinetic-NC software supports the automatic turning of the knife in the right direction, without having to explicitly specify the angle of rotation in the NC program. To do this, the machine type must be set to "Knife Cutting Machine", a rotary axis "C" must be defined, and a few other parameters must be adjusted. The easiest way to do this is to import the file "Tangential-Knife.ini", which is located in the subfolder "Addons" of the default parameters (menu File - Import Settings - Default Parameters).

There, all necessary parameters for the most common tangential knife type with stepper motor and 1: 1 gearbox are already predefined.

### 13.11.3 Backup

If the machine is also used alternately for milling, do not forget to make a backup of parameter before importing the tangential file.

### 13.11.4 Checking the knife



Before using for the first time, check the resolution of the C axis and the correct orientation of the knife. There are knife drives with different resolution. The tangential gauge EOT-2 has e.g. 200 full steps per revolution, which corresponds to a resolution of 5.55555 steps / ° in 1/10 micro step. The tangential EOT-3 has twice the resolution.

First carry out a homing run on the C axis. If the C axis coordinate display is not zero, use the C + / C rotary arrow buttons on the Setup page to move the axis to the zero position. Do not use the zero button, this would just set the coordinate to zero without rotating the knife. Now check that the blade of the knife points exactly in the X + direction. For a high-z machine this means that the knife must be aligned parallel to the long axis with the cutting edge to the right.

### **13.11.5 Reference switch and homing**

If the cutting edge is not aligned correctly, use the C + / C button to turn the knife to the correct angle. Note the displayed C coordinate and subtract it from the value "Reference switch position" in the axis parameters (Configuration - Machine - Axis parameter, select axis: C For example, if "Reference switch position" was -55 before, and the coordinate display for exact angular orientation is +3.5, then you must enter  $-55 - 3.5 = -58.5$ . Then carry out a new reference run and repeat the check of the angle.

### **13.11.6 No CAM program necessary**

For cutting with tangential knife, it is not absolutely a CAM program necessary. You can import the lines to be cut directly as an HPGL or DXF file and only need to specify the plunge depth and security height for trips over the material. Alternatively, you can also use a G-code program. This should only be 2-dimensional, this means it consists of a sequence of rapid traverse positioning (G0) above the material, an insertion movement (G1 with Z movement vertically downwards), one or more XY movements within the material (G1, G2 or G3 without Z) and a lifting movement (G1 with Z movement vertically upwards) and any number of repetitions.

### **13.11.7 Sharp corners or not**

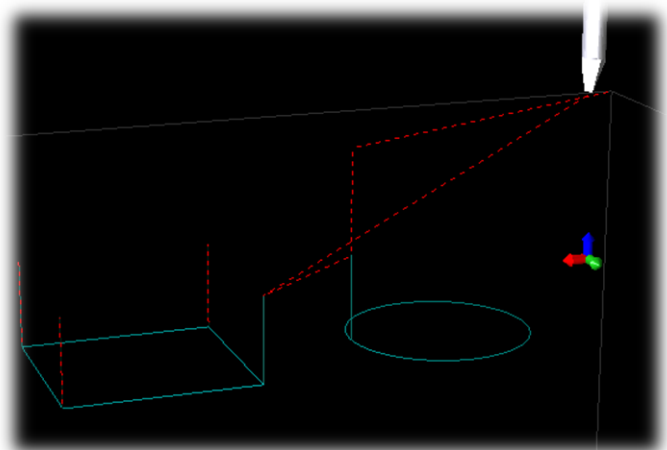
It is important that the drawing has tangent-continuous transitions at all points where liquid is to be passed through. Wherever sharp-edged lines about each other, the control automatically generates an additional and insertion movement. This is intended if, for example, a sharp-edged rectangle is to be cut and prevents the unsightly tearing of the material or even a breakage of the blade, which could happen if the blade had to be turned too large an angle within the material. If the blade is not to be taken out of the material, the corners of the rectangle must be rounded with quarter-circle arcs.

### **13.11.8 Clean drawings**

It is also important that the drawing is made carefully and that the starting points of the following lines are exactly at the end points of the preceding lines. Gaps or overlaps not only lead to jerky movements, but because of the step-like direction changes also to unnecessary excavation, turning and insertion movements. The limit values for the angles between successive lines, from which a stop or an lifting / insertion movement is inserted, can be set in the machine parameters on the special page.

### 13.11.9 Control of the drawing

However, if you experience problems with frequent stops, jerky movements, and unnecessary insertion, first check your drawing to see if the start and end points of the lines are clean and the transitions are constant tangent.



### 13.11.10 Presentation in the graphic

The points at which the knife is lifted out can be recognized as vertical lines in the path preview (see picture on the right at the corners of the rectangle and on the left at the circle). If lifting motions are present at locations on the web where they should not actually be, this is an indication that there are either tiny gaps or intersections, possibly not visible, depending on the magnification, or the transitions are not continuous, too big "kink" (angle). Excavation movements at (intentional) 90 ° corners as in the picture on the right are quite normal.

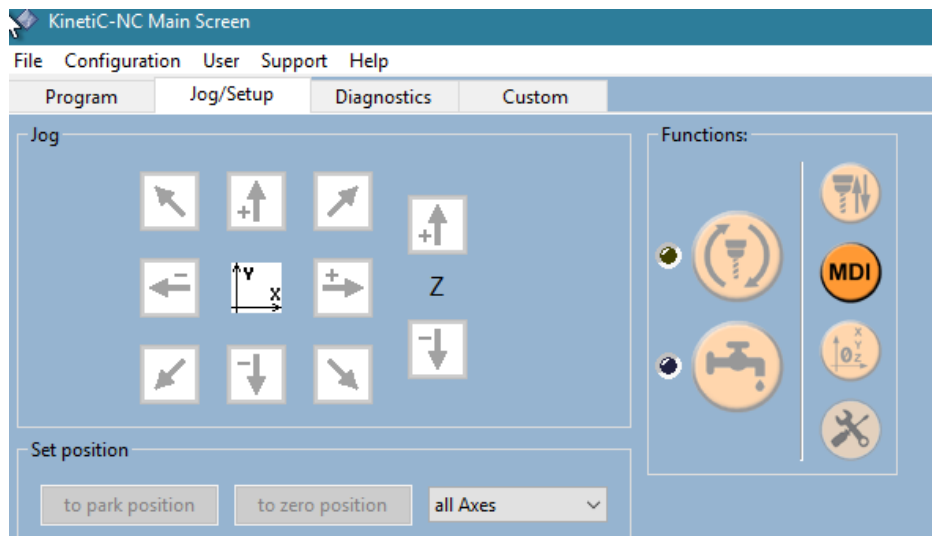
## 14 Troubleshooting

When operating a CNC machine many different problems can occur, which of course not all are mentioned here. The following sections list only those issues that can occur frequently during initial startup and for which there are standard solutions.

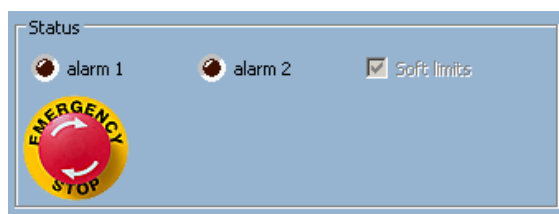
In case of unforeseen problems you can contact the Support. If possible, try to solve the problem first by using this guide.

### 14.1 Functions locked

Not all functions of the software are allowed at any time. It may happen that certain buttons and menu items are locked. In the picture below, for example, the machine movement, spindle and coolant function and tool change and homing are disabled ("greyed out").



All machine movements are of course blocked as long as the emergency stop is active. First check the symbols in the "Status" field.



There, one or more of the following symbols will appear:



With "OK" the machine is ready. "**Emergency Stop**" means that either one of the emergency stop buttons on the machine or on the control is pressed, or there is a problem with the electronics (power failure, interrupted cables, etc.). The exclamation point triangular **warning sign** indicates a problem with an external device that is signalled by an alarm signal (see "Frequency converter alarm" in the status field above). In this case, please refer to the instructions of the external extension (for example, the frequency converter) for the cause and instructions for correcting the problem.

The warning sign with the door appears if the machine is equipped with a **safety door** and it is open. This is described in more detail below.



When the **DEMO** icon appears, there is (still) no connection to the hardware. The software runs in demonstration mode and simulates the machine movement only in the 3D graphics. The right-hand plugged icon indicates that a connection existed but was subsequently disconnected (see below, "Connection interrupted")

Regardless of the state of the machine, there is another reason why functions may be limited: Some functions require special rights. In order to be able to change the machine parameters, for example, you have to be logged in as an administrator. Therefore, in the picture below "Configuration - Machine" is greyed out. (See also "*User-Login*")

## 14.2 Connection interrupted

The above icon indicates that the connection to the CncPod has been disconnected. This may not only be due to the cable connection itself but also to the operating system (anti-virus, firewall, energy-saving mode, see also chapter "*Installing CncPod*").

If this always happens at a certain time, e.g. Turning on a powerful device (compressor, exhaust) indicates electromagnetic interference. A momentary voltage fluctuation can lead to a breakdown of communication between PC and CncPod.

On the other hand, if it is idle, but always after a certain amount of time after starting the software, the power-saving mode of the PC can be the problem. For Windows 8 and 10, and especially for portable devices, the operating system may shut down the network connection after a few minutes without Internet usage. This can be prevented if the CncPod is connected via a network switch.

To re-establish the connection, first check all cable connections and the power supply, and then go to the menu "*Configuration->Hardware*".

### 14.3 Error messages

In manual mode, the machine functions flawlessly, ie in Setup-mode all axes can be moved, the spindle can be switched on and off etc. When starting a program, however, you will receive an error message in the Message window.

The most common causes are:

- **“Out-of-range number”**: The tool path is too large for the machining area of the machine, or the Workpiece zero point is in the wrong place, leaving parts of the tool path out of the range.
- **“% Character missing at the beginning of the program”** or **“unexpected character”** or **“unknown command”**: The program was created with a post processor for other control software (e.g., Mach3 or the like). Although the G-code is standardized in principle and is largely the same for all control types, not all controllers understand all commands. If you use your own CAM software that was not supplied with the machine, check our website to see if there is already a ready-made Post-processor for KinetiC-NC and your CAM software. Tips for creating or customizing your own post-processor can be found in the Reference manual.

### 14.4 The machine does not react as expected

If the machine is working properly in manual mode, means in Setup-mode all axes can be moved, the spindle switched on and off, etc., but the program does not run the expected tool path, moves too fast or too slow, or stops at certain points, may be the faulty setting of the CAM software or the wrong Post-processor has been selected (see above). Since the possibilities here are very diverse and difficult to predict, it is best to contact the support.

If you do not send an automatic support request it is best to send the program that caused the problem and your parameter settings, and describe the problem as accurately as possible, otherwise it will be difficult to quickly analyse the cause.

### 14.5 Security door



If your machine is equipped with a housing, the safety door function prevents the machine from being moved as long as the door is open and allows the user to reach into the hazardous area of the machine. Starting programs and switching on the spindle motor or other outputs is only possible with the door closed. As long as the door is open, the triangular warning symbol with the door open will be displayed in the status field.

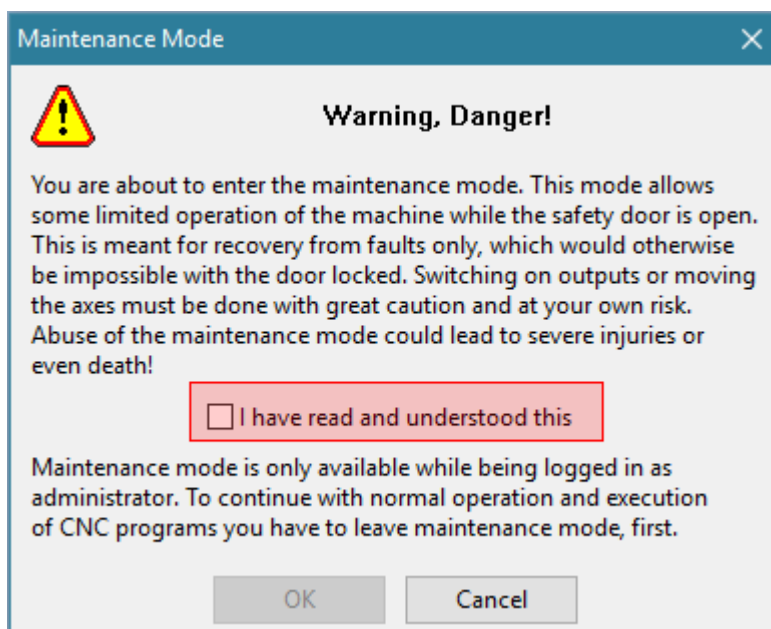
## 14.6 Maintenance mode

The safety door and the alarm function are usually designed to protect the user from injury and the machine from damage. In certain cases, however, the safety functions may block each other. Suppose there is a technical defect that triggers an alarm condition. However, the defect can only be remedied if the machine were moved a short distance to get to the location with the defect. The alarm blocks all machine movements.

In this case, there is nothing left but to break the "vicious circle" by overriding the blockage caused by the alarm condition.



This is the maintenance mode, which is activated with the button on the setup page. Since overriding the security feature involves a high level of risk, this feature is only available when logged in as an administrator. Before releasing the machine and switching outputs, the following warning message is displayed, which must be confirmed first



In order to prevent the maintenance mode from permanently bypassing the safety functions, the execution of CNC programs in maintenance mode is not permitted. In order to resume normal operation of the machine, maintenance mode must be exited first. This happens automatically when closing the security door.



## 15 Import filter

The standard version of the software contains 2 import filters. One is the HPGL import filter for importing HPGL or plot files and the other is the DXF import filter for importing the popular and popular DXF graphics format.

### 15.1 DXF-Import filter

#### Short description:

The DXF import filter is used to convert drawing data that exists in dxf format to DIN-G code for the KinetiC-NC software. The import filter reads files in DXF ASCII format.

The following elements from DXF files are read and taken into account:

- Layer (layer names are displayed)
- Line (lines with 2 vectors in 2D or 3D)
- Circle (circle with center vector in 2D or 3D and radius)
- Arc (angle)
- Polyline (as open or closed line)
- Vertex (2D or 3D, points for polyline)
- Point (2D or 3D)

#### 15.1.1 What is imported?

Not supported are B-splines (Bezier lines), also no 3D-solid data. However, all supported elements are also supported in 3D, ie with z-values.

If the elements Point or Polyline (with only one vertex) are included, they are interpreted as drilling points.

Line, Polyline, and Arc are grouped into segments if they have common start / end points. The tolerance of catching these points can be set by software and is by default at 0.001mm.

#### 15.1.2 The milling direction

The direction of the individual elements is adjusted and possibly changed. The milling direction of a single segment therefore depends on the direction in which the first element found was drawn. It is also possible to delete individual segments or specify individual Z-depths, laser power or feed rates for individual segments.

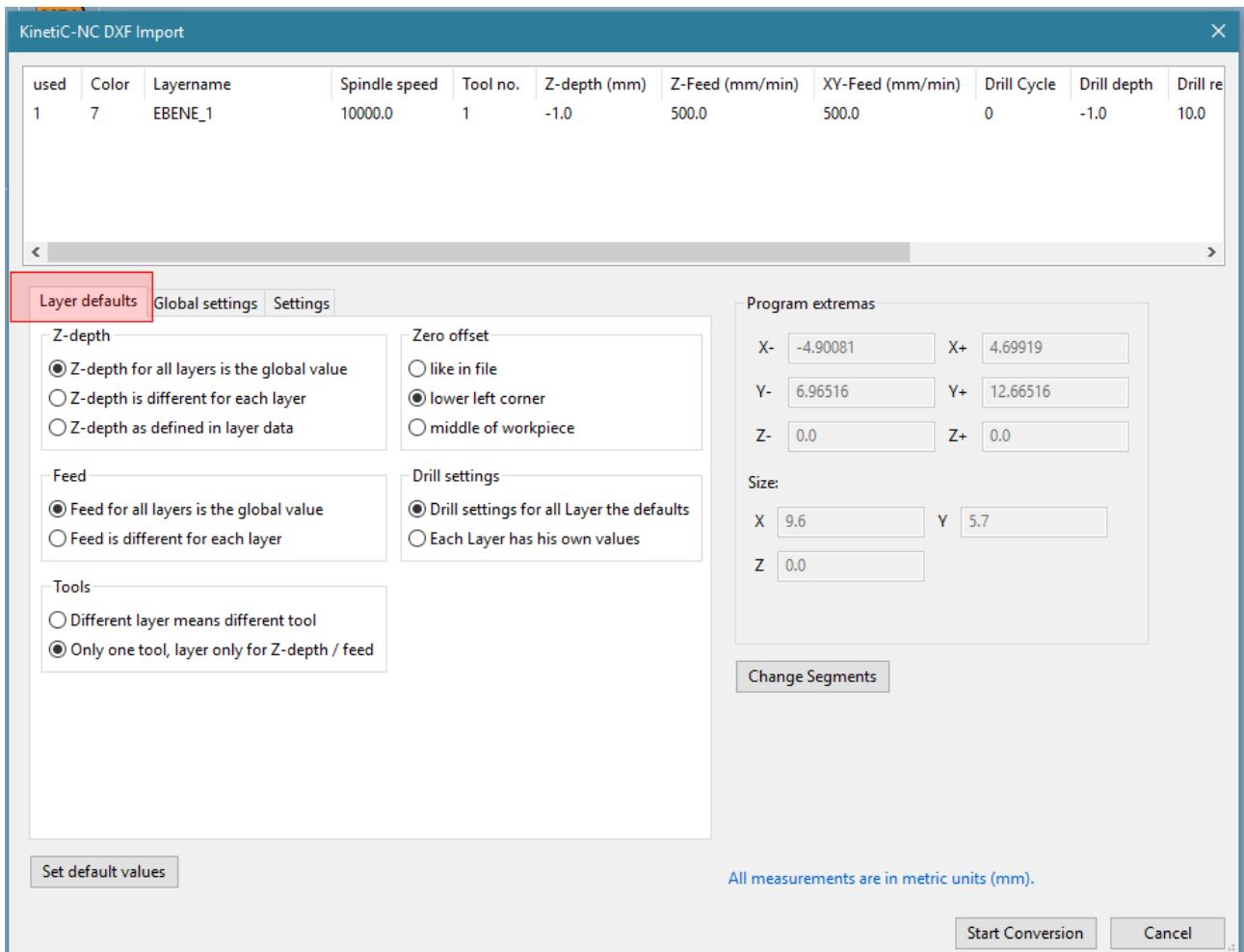
### 15.1.3 The import process:

The file to be imported is selected and opened under Menu / Import Drawing. If this file contains correct, readable DXF data, the following dialog opens:

In the upper field, the layers are displayed with data. Layers in the file that contain no evaluable data are not displayed. Layers can be activated / deactivated individually. Only active layers will be imported. The size of the data to be imported is always updated and applies only to the active layers / segments. By clicking on the tabs of the left side, the following additional settings appear:

### 15.1.4 Tab: Layer defaults

Here it is determined *which* data should be used to convert the individual layers.



used	Color	Layername	Spindle speed	Tool no.	Z-depth (mm)	Z-Feed (mm/min)	XY-Feed (mm/min)	Drill Cycle	Drill depth	Drill re
1	7	EBENE_1	10000.0	1	-1.0	500.0	500.0	0	-1.0	10.0

**Layer defaults** | Global settings | Settings

**Z-depth**

- Z-depth for all layers is the global value
- Z-depth is different for each layer
- Z-depth as defined in layer data

**Zero offset**

- like in file
- lower left corner
- middle of workpiece

**Feed**

- Feed for all layers is the global value
- Feed is different for each layer

**Drill settings**

- Drill settings for all Layer the defaults
- Each Layer has his own values

**Tools**

- Different layer means different tool
- Only one tool, layer only for Z-depth / feed

**Program extremas**

X-: -4.90081 | X+: 4.69919  
 Y-: 6.96516 | Y+: 12.66516  
 Z-: 0.0 | Z+: 0.0

Size:  
 X: 9.6 | Y: 5.7  
 Z: 0.0

Change Segments

Set default values

All measurements are in metric units (mm).

Start Conversion | Cancel

## Z-depth

- 1) For all layers, the Z depth set in the Global Setting field is used for milling / drilling.
- 2) Each layer is set with individual values.
- 3) (only active if layer contains 3D data) The Z values from the layer data are used.

## Feed

- 1) For all layers, the value specified in the Global Setting field is used.
- 2) Each layer has individual feed values.

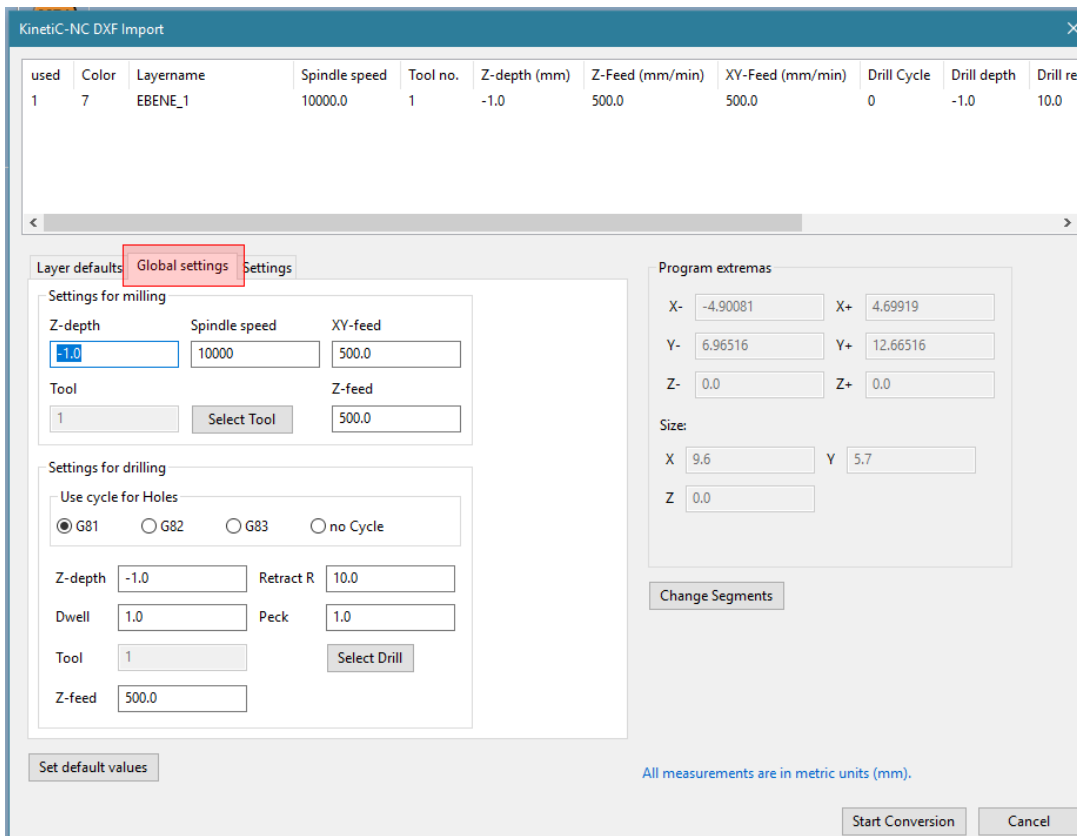
## Zero offset

- 1) The zero point is not changed but remains as specified in the file.
- 2) The zero point is placed in the lower left corner of the workpiece.
- 3) The zero point is placed in the middle of the workpiece.

## Tools

- 1) For all layers, the value specified in the Global setting field is used.
- 2) Each layer has its own tool.

## 15.1.5 Tab: Global Settings



The screenshot shows the 'KinetiC-NC DXF Import' dialog box. At the top, there is a table with the following data:

used	Color	Layername	Spindle speed	Tool no.	Z-depth (mm)	Z-Feed (mm/min)	XY-Feed (mm/min)	Drill Cycle	Drill depth	Drill re
1	7	EBENE_1	10000.0	1	-1.0	500.0	500.0	0	-1.0	10.0

Below the table, the 'Global settings' tab is active. It contains two main sections: 'Settings for milling' and 'Settings for drilling'. The 'Settings for milling' section includes fields for Z-depth (set to -1.0), Spindle speed (10000), XY-feed (500.0), Tool (1), and Z-feed (500.0). The 'Settings for drilling' section includes radio buttons for 'Use cycle for Holes' (G81 selected), Z-depth (-1.0), Retract R (10.0), Dwell (1.0), Peck (1.0), Tool (1), and Z-feed (500.0). On the right side, there is a 'Program extremas' section with fields for X-, X+, Y-, Y+, Z-, and Z+ coordinates, and a 'Size' section with X and Y dimensions. At the bottom, there are buttons for 'Set default values', 'Start Conversion', and 'Cancel'. A note at the bottom right states 'All measurements are in metric units (mm)'.

Here are the settings that are used both as default values for changing the individual values of a layer and the Global values if no individual values are set. A distinction is made between milling data and drilling data (only available if at least one layer contains drilling data in the file).

### **15.1.6 Settings for milling**

#### **Z-depth**

Value, how deep into the material to be milled / drilled.

#### **Spindle speed**

Value for the speed of the milling spindle

#### **XY feed**

Value for the G1 feed in mm / min

#### **Tool**

The tool to use from the tool list

#### **Z-feed**

G1 feed to be used for immersion in the material.

### **15.1.7 Settings for drilling**

These data can be set separately from the milling data.

#### **G81 / G82 / G83 / no cycle:**

Here you choose whether a drilling cycle should be used. If no cycle is selected, code will be generated that behaves as if G81 was selected.

#### **Z-Depth:**

Borehole depth in mm

#### **Retreat R:**

Retraction plane to which the drill is retracted before drilling or re-immersing a new hole.

#### **DWELL:**

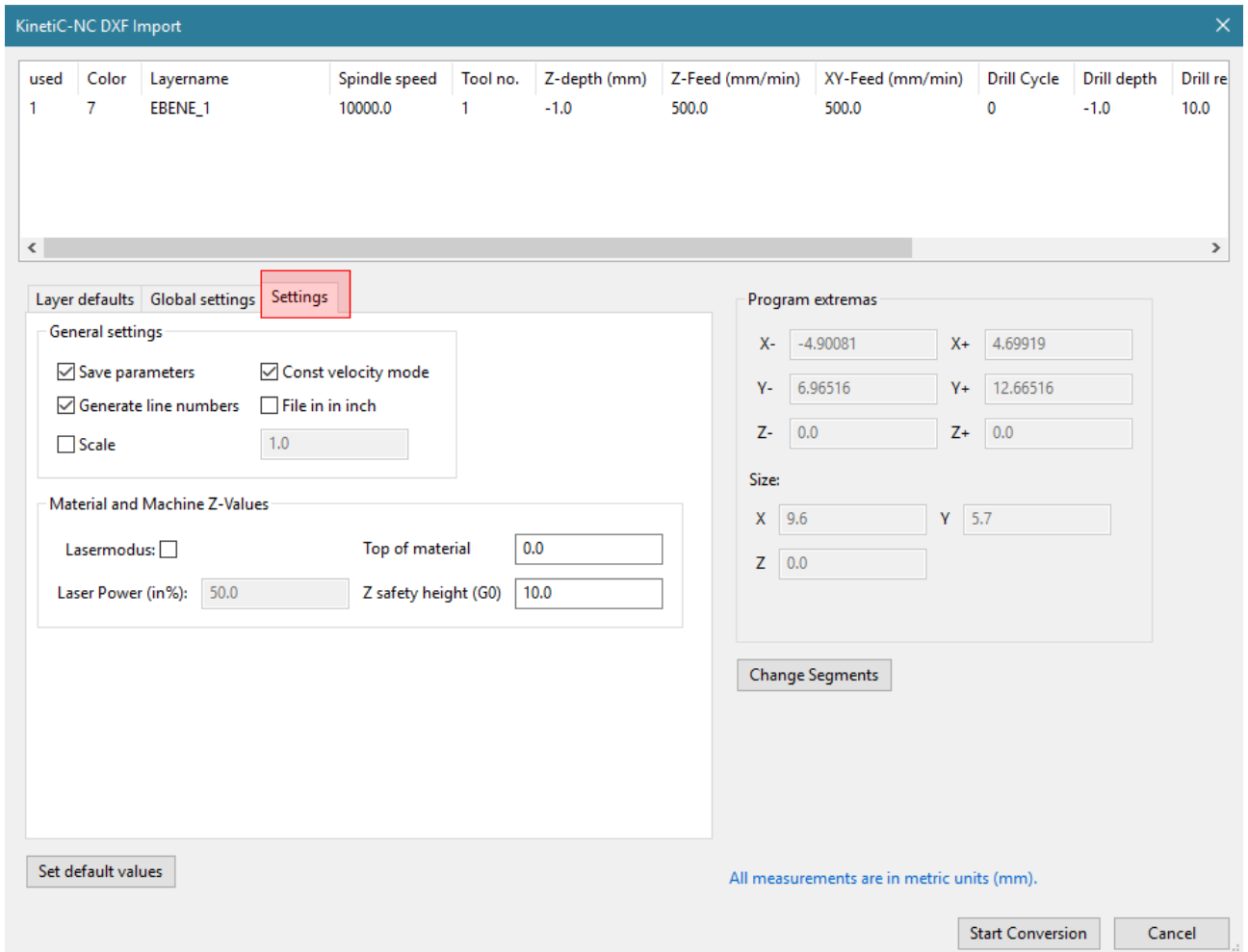
Pause time to stay at the bottom (G82)

#### **swarf removal:**

Depth to be delivered (G83)

### 15.1.8 Tab: Settings

This field displays the general data for the machine or workpiece / material.



The screenshot shows the 'KinetiC-NC DXF Import' dialog box. At the top, there is a table with the following data:

used	Color	Layername	Spindle speed	Tool no.	Z-depth (mm)	Z-Feed (mm/min)	XY-Feed (mm/min)	Drill Cycle	Drill depth	Drill re
1	7	EBENE_1	10000.0	1	-1.0	500.0	500.0	0	-1.0	10.0

Below the table, there are three tabs: 'Layer defaults', 'Global settings', and 'Settings' (which is highlighted in red). The 'Settings' tab contains the following sections:

- General settings:**
  - Save parameters
  - Const velocity mode
  - Generate line numbers
  - File in in inch
  - Scale: 1.0
- Material and Machine Z-Values:**
  - Lasermodus:
  - Top of material: 0.0
  - Laser Power (in%): 50.0
  - Z safety height (G0): 10.0
- Program extremas:**
  - X-: -4.90081, X+: 4.69919
  - Y-: 6.96516, Y+: 12.66516
  - Z-: 0.0, Z+: 0.0
  - Size: X: 9.6, Y: 5.7, Z: 0.0

At the bottom right, there is a note: 'All measurements are in metric units (mm)'. Buttons for 'Set default values', 'Change Segments', 'Start Conversion', and 'Cancel' are also visible.

### 15.1.9 Z-values of material and machine

#### 15.1.10 Material top:

Highest part of the material to be milled / drilled

#### Z-level security:

Height at which it is safe to travel at rapid traverse (G0) without colliding with the clamping device.

### **15.1.11 General settings:**

#### **Saving parameters**

If this option is selected, all settings are saved and will be recalled on the next import. The resetting of all values to factory settings is done with the button "Standard values".

#### **CV mode**

When selected, the constant velocity mode is selected instead of the exact stop mode.

#### **Create line numbers**

When selected, line numbers are output in the G code.

#### **File in inches scale**

When this box is checked, the coordinates in the file are interpreted as inches.

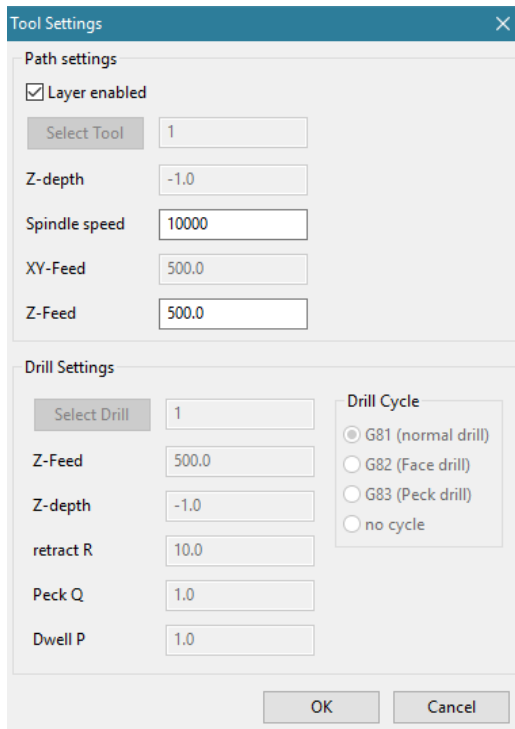
#### **scaling:**

Any value for scaling the file can be specified here. 0.5, for example, cuts the file in half, meaning that a 100x100mm template is now only 50x50mm in size, but an indication of 2.35 would scale a 100x100mm file to 235x235mm.

#### **Program extreme values**

Here the maximum coordinates of the entire program are displayed. Also, the absolute size of the workpiece is displayed. When activating / deactivating individual layers, this value is automatically recalculated. So only the dimensions of the active layers are displayed.

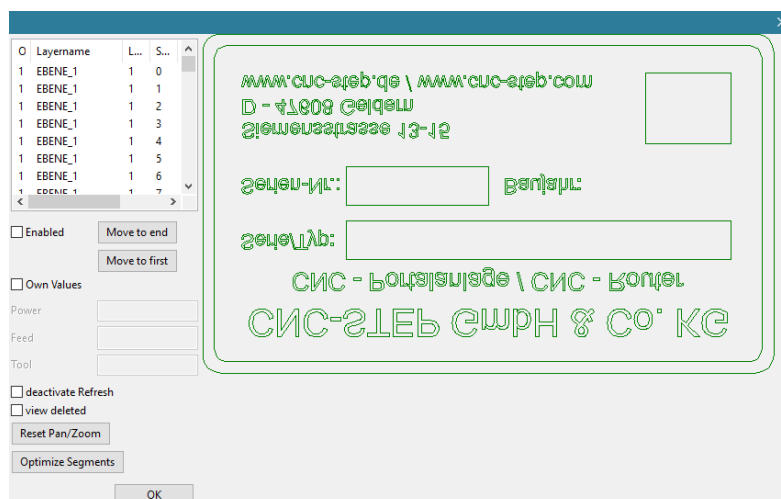
### 15.1.12 Layer-Details (Layer List):



By right-clicking with the mouse on a layer in the layer list, the global settings are assigned, or you can display and modify the set values. Thus, individual values for feed, Z depth, tool or drilling can be set for each layer. The description of the individual values is identical to the global values described above.

### 15.1.13 Advanced Editing / Path Details:

Clicking on the "Path Details" button opens the following window:



#### Segment selection

In the upper left corner is a list of all segments that contain this file. Clicking on one of these segments turns this "light green". At the same time, the fields below this list become active. By unchecking "Active" you can exclude the selected segment from the export.

### 15.1.14 Double lines

This is useful if the file contains duplicate lines or elements that should not be milled / lasered. By checking the box marked "Show deleted" only the already deactivated elements are displayed.

### 15.1.15 Navigate the graphic

While holding down the mouse button, you can move the image in the graphic, while holding down the right mouse button, you can zoom in. Click on the button "Reset Pan / Zoom" to restore the original state.

### 15.1.16 Optimize the drawing file

For files that consist of countless small line segments, it may be useful to sort the segments again and chain them together. This can be triggered by the "Optimize path" button. Unfortunately, this option is NOT reversible, so if an unexpected result arises, for example, with double lines, the import is to be repeated again.

### 15.1.17 The conversion:

After confirming the button Start conversion, the data will be converted into G-code and automatically loaded as a program. The converted data is documented for better understanding:

```
(Kinetic-NC DXF Import Filter)
(Filter Version: V1.0)
(converted from: C:\DXF\samples_dxf\hasen.dxf)
(circle mode JK relative)
%
N10 G64 (constant velocity mode)
N20 G21 (metric)
N30 G90
N40 G0 Z50 (move to secure Z-height)

(Processing vector data from layer EBENE_1)
(Tool Change)
N50 Z55
N60 M6 T1
N70 M3 S1000
N80 G0 Z50
(Processing segment: 0, containing 603 elements)
X-1.45876 Y4.08865
```



The generated code is partially optimized. This means, G commands and F commands are issued only when the command changes. Coordinates are only output if they differ from the current position. For better readability, however, the commands and parameters are separated by spaces.

### **15.1.18 Save the file**

The converted file is saved with the extension .nc in the same folder from which the .dxf file was opened. It is therefore important to ensure that the program KinetiC-NC also has write access to the disk from which the dxf file was opened (for example, not from a DC / DVD).

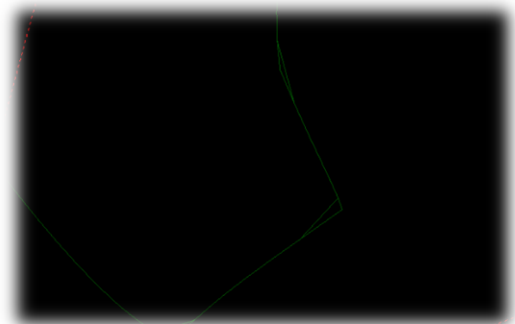
### **15.1.19 Special features when importing from DXF files.**

Polylines are usually marked as closed or not. Unfortunately, this does not always work. Therefore, the import will see if the first point matches the last one. If that is the case, the line will always be marked as closed, even if it is open in the DXF file. If the file was saved as a closed line, but the endpoints do not match, the line will be closed.

### **15.1.20 Note for experienced users**

#### **Double elements:**

Unfortunately, many programs export DXF files very inaccurate. This includes, for example, Corel-Draw. This often happens because the lines in the drawing program have a certain thickness. Then the program will export each line page, creating double lines. But as this is often done in a very inconsistent and inaccurate way, very peculiar segments emerge. These are never 100% identically congruent, but always differ slightly. The import filter does not want to decide here which segment was exported "better" or "worse". Also, it makes no sense to delete the really congruent elements and leave only the rest.



## **Remedy for bad templates**

During import, all contiguous segments are numbered and commented.

If double lines are visible when opening a file, the best course of action is as follows:

- Import the file with high feed values and small Z values for testing.
- Test the file WITHOUT machine in demo mode. You can clearly see which are the "better" segments and which should be deleted. You can make a note of the wrong segment numbers in such a test run.
- Import the file with the correct feeds and Z-values.
- Open the file with the editor and remove all previously noted, duplicate segments.

### **15.1.21 No existing import data or message "Error while parsing the DXF file, wrong version?"**

It may happen that some programs in DXF export, e.g. Disassemble fonts into splines. The Kinetic-NC import filter does not read any splines. In this case, try to save in an older file format, which still does not support splines, e.g. Autocad DXF R10 or older.

Generally, it is a good practice to always use the oldest file format available for import errors.

## 15.2 HPGL-Import filter

### 15.2.1 Short description

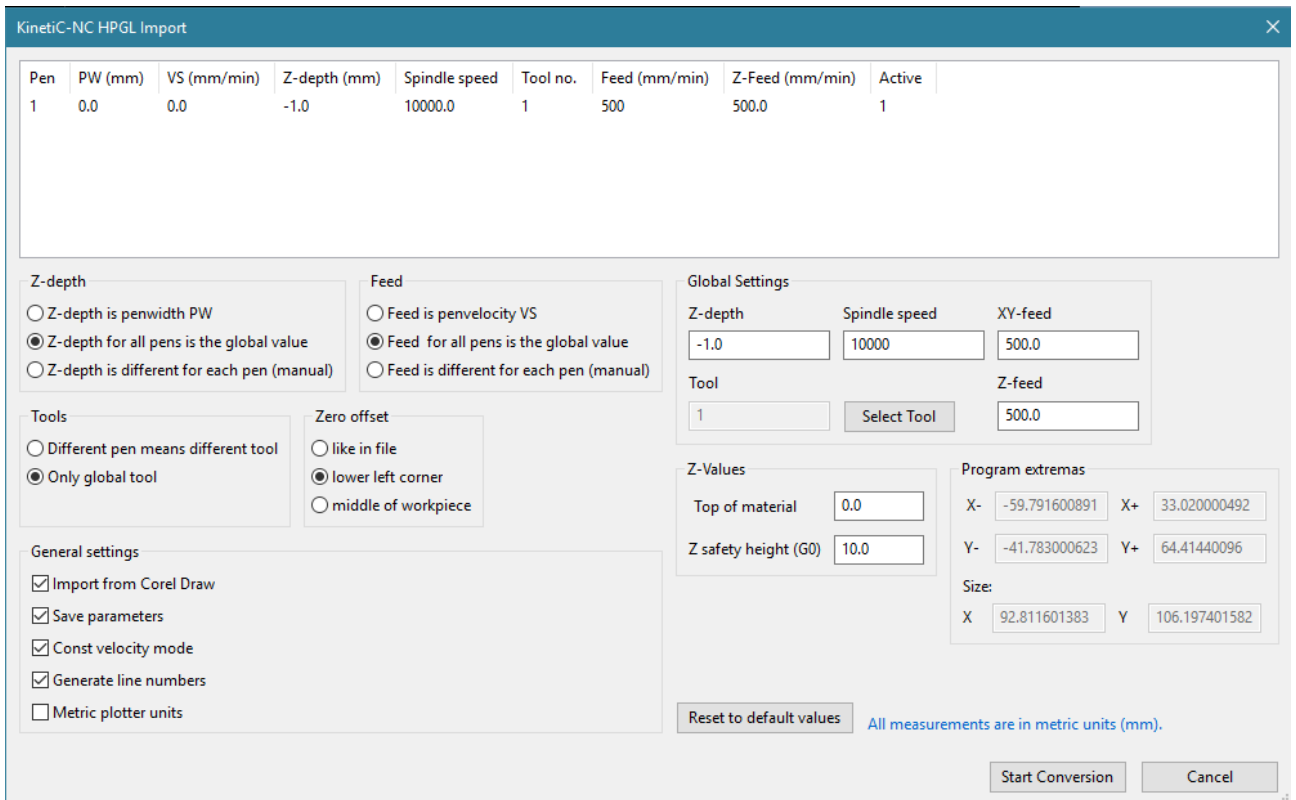
The HPGL import filter is used to convert HPGL data into G-code for the KinetiC-NC control software. HPGL data files usually have the extension .hpgl or .plt and are provided by most drawing programs. These types of files are originally intended for controlling character plotters and cutting plotters.

### 15.2.2 Language supported:

The HPGL import filter supports the following commands:  
IN, DF, IP, SC, VS, WU, PW, SP, PU, PD, PA, PR, AA, AR.  
All other HPGL commands are ignored.

### 15.2.3 The import process

To import an HPGL file, open it via "Menu-> File-> Import drawing". If the file is a valid, readable HPGL file, the following dialog opens:



Pen	PW (mm)	VS (mm/min)	Z-depth (mm)	Spindle speed	Tool no.	Feed (mm/min)	Z-Feed (mm/min)	Active
1	0.0	0.0	-1.0	10000.0	1	500	500.0	1

**Z-depth**

Z-depth is penwidth PW  
 Z-depth for all pens is the global value  
 Z-depth is different for each pen (manual)

**Feed**

Feed is penvelocity VS  
 Feed for all pens is the global value  
 Feed is different for each pen (manual)

**Tools**

Different pen means different tool  
 Only global tool

**Zero offset**

like in file  
 lower left corner  
 middle of workpiece

**Global Settings**

Z-depth: -1.0    Spindle speed: 10000    XY-feed: 500.0  
 Tool: 1    Z-feed: 500.0

**Z-Values**

Top of material: 0.0  
 Z safety height (G0): 10.0

**Program extremas**

X-: -59.791600891    X+: 33.020000492  
 Y-: -41.783000623    Y+: 64.41440096  
 Size: X: 92.811601383    Y: 106.197401582

Reset to default values    All measurements are in metric units (mm).

Start Conversion    Cancel

The top list shows all the pens used. A pen in a HPGL file is originally a tool, and each of these pens can be assigned different properties. If you do not change anything, the same settings apply to all pens, tool 1 is used for all pens, and the workpiece origin is in the lower left corner. By clicking on "Start conversion", the HPGL file is converted to G-code and loaded as an NC program in KinetiC-NC.

### 15.2.4 Output file

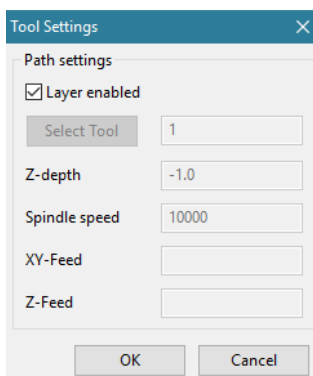
The NC file is created in the same directory as the HPGL file to be imported. Therefore, make sure that the HPGL file is in a folder with write permissions. Do not import from a CD or DVD or a folder where you do not have write permission.

```
(KinetiC-NC HPGL Import Filter)
(Filter Version: V1.0)
(converted from: C:\plotter\out\222.plt )
(circle mode JK relative)
%
N10 G64 (constant velocity mode)
N20 G21 (metrical)
N30 G90
N40 G0 Z10 (move to secure Z-hight)

(tool Change)
N50 M6 T1
N60 M3 S10000
N70 G0 Z10
N80 G0 X0.3048 Y0.254
```

The generated G code is commented on, and also contains information about the original file.

### 15.2.5 Customizing the import process



You can adapt the global values to your needs and the material. If the import file contains several pens, you can configure these individually.

### **15.2.6 Z-depth**

#### **Z-depth is pen width PW**

In the HPGL file, widths can be assigned to the pens. If your drawing program supports it, you can use these values to comfortably adjust the Z depth. Just enter the desired depth as pen width in your drawing program and select this option.

#### **Z depth different for each pen**

If you select this option, you can specify an individual depth for each pen by right-clicking on the respective pen.

### **15.2.7 Tools**

If the "Extra tool for each pen" option is selected, you can right-click on the pen in the table above to select a different tool for each pen.

### **15.2.8 Feed**

#### **Feed is pen speed VS**

In a HPGL file, individual speeds can be specified to the pens with the VS command. If your drawing program supports this, you can specify a speed for each pen when exporting. If you select this option, then the milling feed is automatically and comfortably taken over by each pin.

#### **Feed for pens is different**

With this option, an individual value can be specified for each pen. To do this, click on the desired pen in the upper list and set the feed individually.

### **15.2.9 Zero Offset**

This field allows you to specify where the zero point should be at the workpiece. This makes it possible, regardless of the information in the HPGL file, to move the zero point and thus comfortably adapt to his workpiece.

## 15.2.10 General settings

### Import from Corel-Draw

CorelDraw is a very popular paint program, but unfortunately it generates a very inaccurate export of data. If you have exported data from CorelDraw, check this box to be able to import the file correctly.

### Save parameters

If you make changes to the pens or global settings, they will be saved and automatically loaded again on the next import. If you do not want this, remove the check mark and you can change the data without saving it. By clicking on the button "Reset to default values" all settings are reset to the default values.

### CV-Mode

In CV mode, curving and cornering do not slow down until the machine stops, but, similar to motorcycling, the "curve is cut". So you get a faster, jerk-free movement, but the path is not 100% accurate traversed. The maximum permissible error can be changed in the machine settings.

### Create line numbers

The NC program thus contains line numbers. This can be useful if you want to interrupt the work and want to continue again to remember the position.

### Metric plot units

Normally, a plotter unit is 0.0254mm, that is 1/1000 inch. But there are also programs, which set the export only 0.025mm as a unit. That's 1/1016 inches. If your part is not accurate and your drawing program uses this metric base, check the "Metric plotter units" box.

## 16 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/>***