Inhaltverzeichnis

1 License Terms

It is strictly prohibited to reproduce the program or the accompanying written material, in whole or in part, in its original or modified form.

All our data carriers are thoroughly checked for viruses and errors before shipping. BOENIGK-electronics assumes no liability for any damages that may result from the operation of our programs (including, but not limited to, data loss, machine damage, material damage, personal injury, additional expenses, or incorrect results).
Under no circumstances shall BOENIGK-electronics be held liable to the licensee or third parties for compensation of any damages or consequential damages, such as fines or penalties, nor for compensation claims arising from loss of profits, business interruptions, loss of business information, or other financial losses caused by the use or attempted use of the software and hardware.

The **electronic license registration** must not be removed or modified. All copies must exactly match the original supplied version.

Microsoft, Windows, Autodesk, AutoCAD, Adobe, Hewlett-Packard, and other product and company names mentioned in this book are registered trademarks of their respective owners.

2 Basics

The **cncGraF 8** software controls CNC machines with up to six axes.

Control is handled via the smc5d controller. Among other tasks, this controller performs timer functions and outputs step and direction signals to drive up to seven stepper/servo motors (up to six axes plus a dual drive for X/Y). Its inputs and outputs can control various external devices such as spindles, pumps, and pneumatic systems. Inputs are used for home/reference switches, emergency stop switches, and other automation tasks.

Several controller variants are available, ranging from the entry-level smc5d-p32 to an industrial network version for control cabinets. For more information, please visit our website at cncgraf.com.

cncGraF 8 is available in two editions. **cncGraF 8**Standard is an affordable alternative to the higher-end cncGraF 8 Professional. You can upgrade from Standard to Professional at any time. To see a tabular comparison of the editions, open the main menu and choose **Help → Differences Between Versions**.

About this manual
The cncGraF 8 software offers many settings and advanced features. During development we made every effort, with great care, to provide our customers with the broadest possible range of functionality. In some cases, this documentation may not cover every function or detail. If you have suggestions for improving this documentation, please email us at info@boenigk-electronics.com. Thank you!

Text marked with the warning symbol indicates potential hazards. The info symbol highlights important information.

This documentation has been automatically translated. It may contain errors or inaccuracies. The official version is the original German text.

2.1 What Can cncGraF 8 Do?

cncGraF 8 features a modern, clear, and intuitive graphical user interface, where all important functions can be accessed either via icons or keyboard shortcuts. The CAD-like interface represents the machine workspace. Within this area, elements such as drawing data, the workpiece, the zero point, parking positions, or measuring positions are displayed.

Typical applications of cncGraF 8 include:

- Milling
- Drilling
- Cutting foils
- Engraving
- Probing
- Plotting
- Digitizing
- Automation
- Laser cutting

2.2 What's New?

The new cncGraF 8 software comes with many additional features. This page provides a quick overview:

- Optimized user interface for more intuitive operation
- 3D probe for higher precision and productivity
- Multiple measurement routines for improved accuracy
- Customizable "Manual Move" dialog
- Spindle warm-up: extend the lifetime of your spindle
- Extended G-code functionality: IF statements and R parameters for greater flexibility
- Tool palette: simple editing of DXF files
- Angle head support for bevel or inclined cuts
- Laser cutting
- Control of up to six axes

2.3 What Else Do You Need?

In addition to the cncGraF 8 software, you will need the following hardware and software components:

- Operating system: Microsoft Windows 10/11 (64-bit version)
- Microsoft .NET Framework v4.8
- Processor: Intel i3 or higher
- Graphics card with OpenGL support
- RAM: 8 GB
- Hard disk space: 500 MB free
- Controller: smc5d or smc5d-m4

cncGraF 8 is built on Microsoft .NET technology and requires Microsoft .NET Framework v4.8. The .NET Framework is available free of charge and can be downloaded from the official Microsoft website.

2.4 Installation

Please read this page carefully before starting the installation!

Installing cncGraF 8

1. Run the **cncgraf8.setup.exe** file to install cncGraF 8. The cncGraF 8 installer will guide you step by step through the installation process.

The cncGraF 8 control requires the **smc5d** controller.

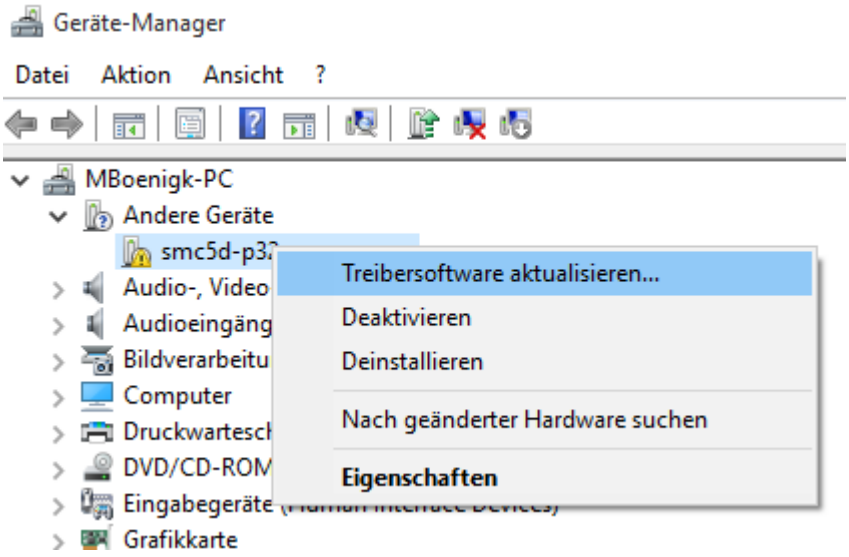
Installing the smc5d USB Driver

The USB driver can be installed automatically. To do this, select **Install USB driver** in the setup wizard at the end of the cncGraF 8 installation. Then follow the on-screen instructions.

Manual driver installation

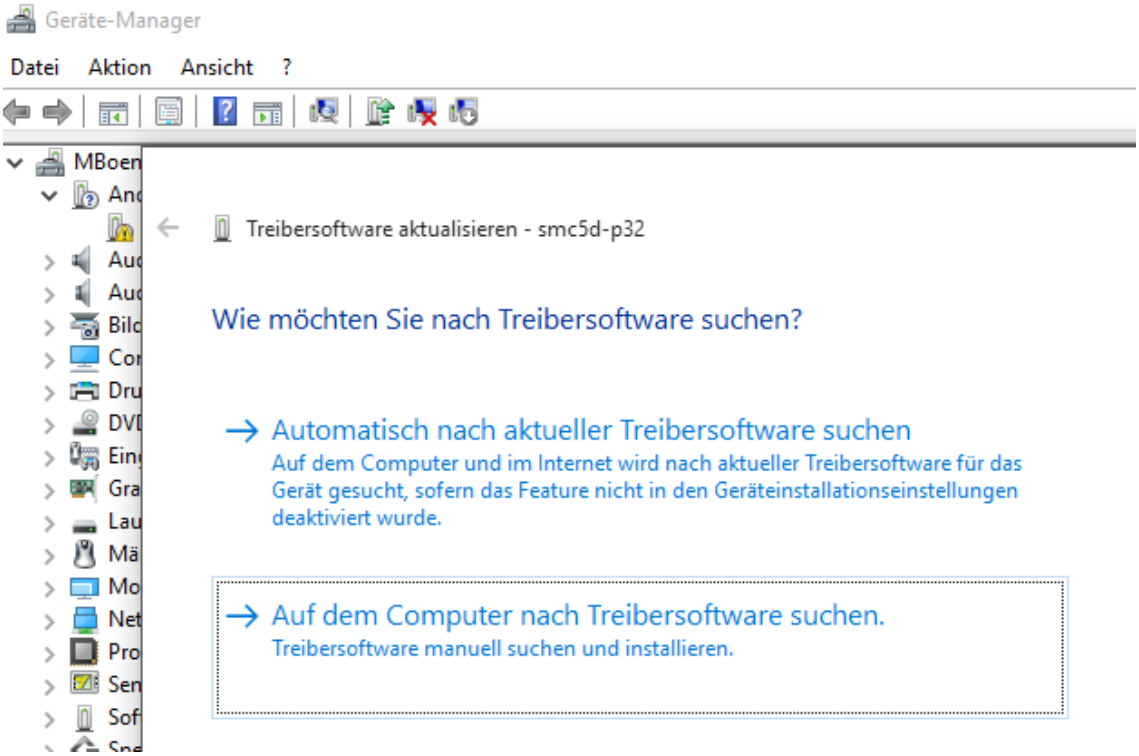
Connect the controller to your PC with a USB cable. Windows will then attempt to locate the appropriate driver automatically.

If Windows cannot find the driver automatically
Open the Windows Device Manager (Start > Control Panel > Device Manager). The smc5d controller will appear under Other Devices as not installed.

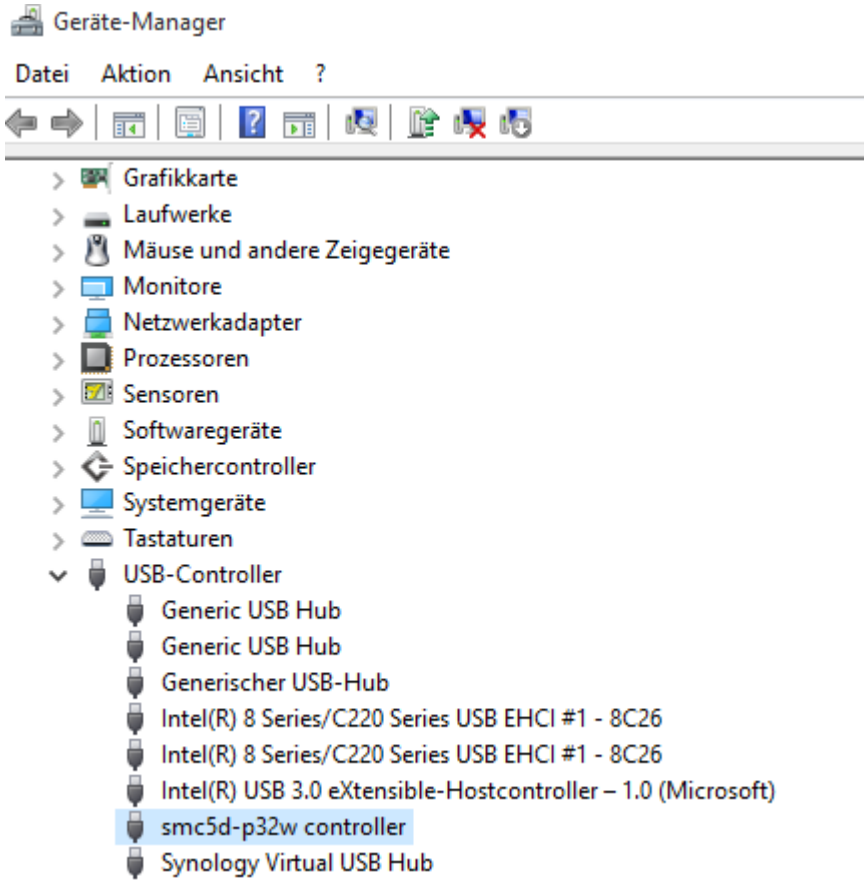


Right-click on the controller entry smc5d-p32 and select Update Driver Software.... A dialog will open with the title How do you want to search for driver software? Choose Browse my

computer for driver software. You can then select the folder where the driver is located. The driver is included in the cncGraF 8 installation directory under ...\driver. Click Next to continue.



The driver installation will now proceed and complete.
Once installed, the smc5d driver will appear in Device Manager under the name smc5d-p32w controller. The smc5d controller is now ready for use.



2.5 Starting cncGraF 8

You can start cncGraF 8 either by clicking the desktop icon or through the Start menu. Running multiple instances of cncGraF 8 is only possible when using multiple controllers. At startup, a file name and directory can also be passed as command-line parameters. Example: cncp7.exe c:\directory\file.plt

"Welcome" Window
When the program is launched for the first time, the Welcome window appears.

This window informs the user that no machine settings are currently available, which are required for the proper operation of the CNC control.
The machine file (system backup file with the extension *.SBF*) can already be specified in this window. Alternatively, the settings can be restored later via the menu **Settings → System Settings → Restore**. For more details, see the section **Basics → Settings**.

A detailed description of all machine parameters is provided in the chapter **Machine Parameters**.

2.6 License

If no license is available, a license notification will appear in the upper-right corner of the main window.

The license includes a 30-day trial period. After this period, it can be extended or unlocked with a license key. Activation is also possible before the trial period expires.

To activate the smc5d or smc5d-m4 controller, you will need a license file from your machine manufacturer.

Once you have received the license key, proceed as follows:

1. Open cncGraF 8
2. Left-click on the message in the upper-right corner: **"The license will expire in x days! Please enter the license file here."**
3. Select the corresponding license file with the *.lic* extension and click **Open**.

The controller is now activated.


Since the license is stored directly on the controller, it does not need to be re-entered if you change PCs or reinstall the software.

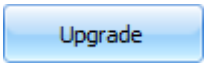
2.7 Upgrade to cncGraF 8 Pro

cncGraF 8 is available in two editions. An upgrade from Standard to Professional can be performed at any time. The following steps are required to complete the upgrade:

1. To upgrade, you will need cncGraF 8 Standard as well as the upgrade file cnc_xx-xxxx-xxxxxxx.upgrade. The name of the upgrade file corresponds to the controller's serial number (example: a controller with serial number 15-1215-4466070 uses the upgrade file cnc_15-1215-4466070.upgrade).
   The smc5d controller must be connected to the PC in order to perform the upgrade.

2. The upgrade is initiated via the main menu command Help → Upgrade to cncGraF 8 Pro...
3. If the menu item **Upgrade to cncGraF 8 Pro...** is not available,cncGraF 8 Pro is already running. In that case, switch to cncGraF 8 Standard by selecting **Help → Version Selection**.

4. Clicking **Upgrade to cncGraF 8 Pro...** opens the *Software Upgrade* dialog. In the *File* field, you must specify the upgrade file. To do this, click the button to browse for the upgrade file.

5. Click the **Start** button 

to begin the upgrade process.

Warning: During the upgrade process, neither the controller nor the PC must be disconnected from power!

Once the upgrade is complete, switch to the cncGraF 8 Professional edition by selecting **Help → Version Selection**. The upgrade is now finished. The upgrade file is no longer required and may be deleted.

2.8 Settings

Due to its complexity, cncGraF 8 offers a wide range of program settings and machine parameters. All these settings are centrally stored in the file *config.xml*. Tool settings are stored in the files *vectors.xml* and *drilltools.xml*. All configuration files are located in the directory ... *\Users\[BENUTZERNAME]\AppData\Roaming\cncGraF8*.

Backing up settings

Important: o prevent data loss, it is strongly recommended to back up your settings regularly to an external storage device. All settings can be saved via the menu **Settings → System Settings → Backup...**

Restoring settings
Saved settings can be restored from the menu **Settings → System Settings → Restore...**

2.9 User Interface Layout

cncGraF 8 provides a user interface and menu structure largely based on standard Windows applications. The interface consists of the main menu, toolbars, and various windows. Its structure is as follows:

- The main menu gives access to all program functions. The most important functions can also be accessed via the toolbars.
- The toolbars contain functions exclusively related to CNC control. Functions related to data processing (e.g., editing drawings) are also represented by icons and are located in the **Tool Palette** window.
- All windows can be moved, hidden, or displayed. To show a window, go to the main menu under **Window** and check the desired entry.
- In the 2D window, the machine area is displayed by default as a white rectangle. It also shows the current position of the CNC machine, the loaded milling file (for example, a G-code file), and other CNC elements such as the zero point, park point, measurement point, and tool changer.
- The user interface can be reset to its default layout via **View → Reset Toolbars** in the main menu.

3 Machine Parameters

Warning: For the CNC machine to operate correctly, the machine parameters must be configured. Missing or incorrect parameters may result in malfunction and can even cause damage.

There are several ways to set up the machine parameters:

Load settings:
If an SBF2 or SBF8 configuration file (provided by the machine manufacturer or from a backup) already exists, the settings can be restored by selecting this file from the menu **Settings → System Settings → Restore**. This also applies to the *config.xml* file. For more details, see the chapter **Settings**.

Setup Wizard for Machine Parameters:
Machine parameters can be configured with the help of this wizard. Please note that the wizard only performs basic configuration. Parameters may need to be fine-tuned manually or additional

parameters configured afterward. The wizard can be accessed via **Settings → Machine Parameters → Setup Wizard**.

Manual configuration of machine parameters:
A step-by-step explanation of the individual parameters can be found in this chapter. As additional guidance, our YouTube channel youtube.com/@cncgraf provides a series of videos explaining manual CNC machine setup.

Switching between different machine types:
To allow quick switching between different machine types, machine parameters can be saved and loaded separately. All machine parameters are stored in XML format in the cncGraF 8 *machine* subdirectory. They can be saved or loaded via **Settings → Machine Parameters → Open/Save**.

Importing machine parameters:
Individual machine parameters can be imported or exported through the menu **Settings → Machine Parameters → Open**.

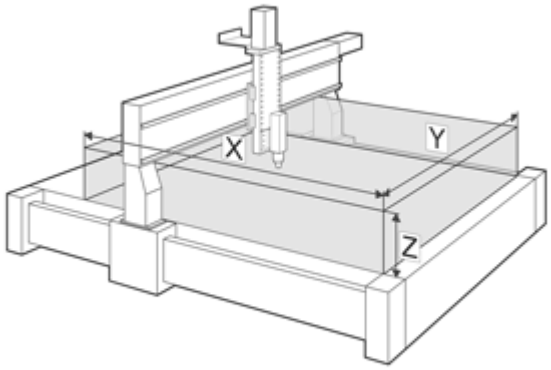


Figure: CNC milling machine and axis assignment

3.1 A First Overview

The *Machine Parameters* dialog contains all settings related to the functionality of the CNC machine. The machine parameters can be accessed from the main menu via **Settings → Machine Parameters → Adjust...** or by pressing the [F8] key. On the left side, a navigation bar is provided. The parameters are logically grouped into the following categories: *General*, *Axes*, *Pin Assignment*, *Speed*, *Sensors*, *Tool Changer*, and *Extras*.

Protecting machine parameters with a password
The machine parameters, as well as other important settings, can be protected against unauthorized changes by using a password. This password can be set up via **Settings → Machine Parameters → Password**.

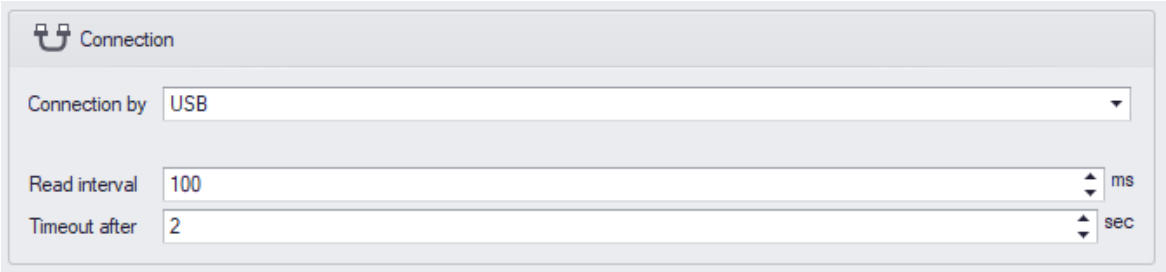

It is advisable to define a password hint (a phrase or keyword). If you ever forget your password, this hint will serve as a reminder.

3.2 Connection

To set up the CNC machine, a USB connection to the controller must first be established. This requires installation of the USB driver. A detailed guide to installing the USB driver can be found in the **Installation** chapter. Afterward, select *USB* as the connection type.

Read Interval
The cncGraF 8 software reads the machine status (axis positions, output states, etc.) from the smc5d controller at fixed intervals specified in milliseconds. Increasing the read interval value lengthens these cycles, reducing system load.

Timeout
In case of communication issues, cncGraF 8 attempts to maintain the connection with the smc5d controller. If the software does not receive a response from the controller within the preset time (for example: 2 seconds, see figure), the process is aborted and an error message is displayed indicating that the connection was lost.



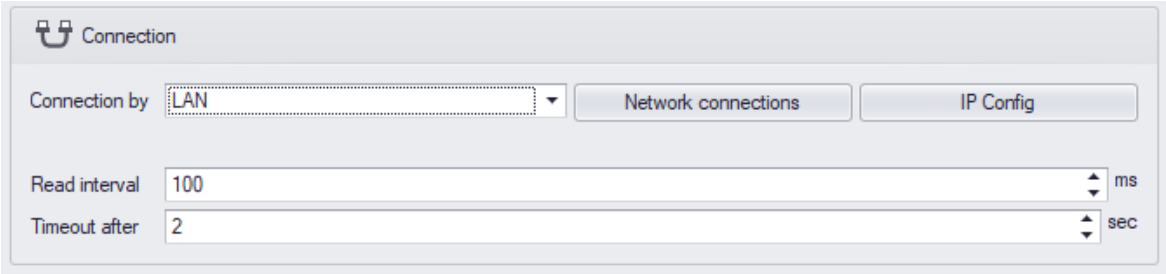
In cases of severe interference, the disturbance may affect the connection to such an extent that cncGraF 8 freezes. In such situations, disconnecting the USB connection (unplugging the cable) can help.

Frequency
Some control electronics require a clock signal for enabling. To activate the clock signal, the checkbox for the output must be selected. The clock signal can be set in a range from 1 to 25 kHz.

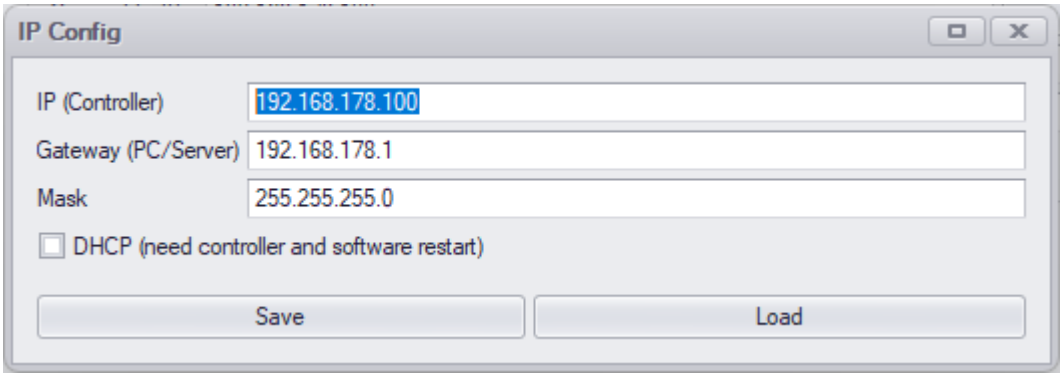
Multiple Instances
Instances make it possible to connect multiple controllers to a single PC via USB, allowing one PC to control several CNC machines simultaneously. Up to four controllers can be operated at the same time. To identify each controller, their serial numbers must be entered. Multiple instances with the corresponding serial numbers allow cncGraF 8 to be launched multiple times.

You can find the serial number by opening cncGraF 8 and selecting **Help → About...** from the menu.

LAN Connection
After establishing a USB connection, the smc5d-m4 controller can be switched to Ethernet. To do this, select *LAN* as the connection type.



In the *IP Config* dialog (opened via the **IP Config** button), you can configure the IP address, gateway, subnet mask, and DHCP settings.



The parameters in the *IP Config* dialog must first be transferred to the controller via USB!

There are two methods to connect the controller via LAN:

Connection through a server:
Automatic connection using a hostname (DHCP) or a fixed IP address.

Direct connection to the PC (recommended):
A fixed TCP/IP address must be configured on the PC (Windows). Click the **Network Connections** button to go directly to the Windows network settings. Select the Ethernet icon of your network card. Open the context menu with a right-click and choose **Properties**. Then select **Internet Protocol Version 4 (TCP/IPv4)** and enter the fixed IP address of the PC.

A video tutorial for LAN setup is available on our YouTube channel: https://youtu.be/bKFt-ieSk5c

3.3 Slave Connection

Expansion with additional controllers
This option allows the connection of additional controllers, which can be controlled via macros. Macros can be called at various points in the machining process, enabling the CNC machine to be integrated into automation workflows and allowing additional external devices to be incorporated into the process.

In the *Master S/N* field, enter the serial number of the CNC machine. The list of slave controllers, referred to here as *Slave Controllers*, contains the additional controllers. These slave controllers are controlled via macros.

Example of a macro

```
string theSN = "01-1015-8164615";
Macro theMacro = new Macro(this.Context.Parent);

De.Boenigk.SMC5D.Basics.Connector
theConnector = theMacro.GetSlave(theSN);

if(theConnector == null) {
   theMacro.MessageBox("Controller " + theSN + " nicht gefunden!");
   return;
}

De.Boenigk.SMC5D.Basics.Switch theSwitch =
new De.Boenigk.SMC5D.Basics.Switch(theConnector);

theSwitch.SetPin(Output.Con2Pin1, true); // Ausgang 1 auf Slave-Controller schalten

if(theConnector.IsInputOn(Input.Con2Pin7, true)) { // Eingang 7 auf Slave-Controller prüfen
   theMacro.MessageBox("Ausgang 7 ist an");
   return;
}
```

 For more information on macros, see the chapter **Macros**.

3.4 Power Stages

In the *Power Stages* menu, several parameters are available to ensure that as many power stages/servos as possible can correctly interpret the clock signals from the smc5d controller. Some power stages already provide predefined parameter sets that only need to be selected. For power stages without predefined parameters, the settings must be configured manually.

- **Power Stage**: Select predefined parameters for your power stage.
- **Name:** Name or designation of the power stage/servo.
- **Frequency:** The smc5d controller can output step frequencies of up to 300 kHz (depending on controller type) for interpolated movements across 6 axes plus dual-drive X/Y. At 150 kHz, the pulse length is 3.333 µs. If the pulse length is too short, it may cause malfunctions in the power stage. In this case, the frequency must be reduced to ensure a sufficient pulse length.

- **Invert Clock:** Depending on the power stage, the clock signal may start with 0 or 1. An incorrect setting can lead to lost steps.

- **Step/Direction Delay:** Some power stages (drivers) require a short delay after a change in direction. An incorrect setting can also lead to lost steps.

Always check the documentation of your power stages to determine the maximum frequency and the allowable step/ direction delay.

3.5 Axes

The cncGraF 8 software and the smc5d controller allow control of up to six axes (X, Y, Z, A, B, C) using the Cartesian coordinate system. The origin of the X, Y, and Z axes is located at the lower left. After homing, the X and Y counters for the absolute machine coordinates are set to zero/axis length (see *Reference Point*), and the Z counter is set to the configured Z-axis length. *Note: When the Z-axis moves downward, the Z counter value decreases.*

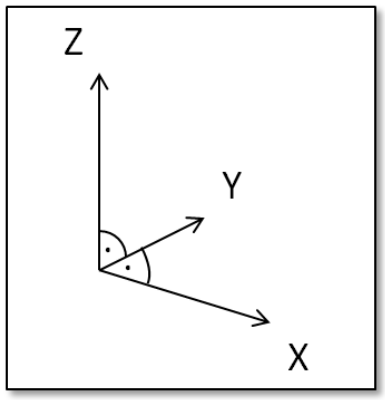


Figure: Cartesian coordinate system

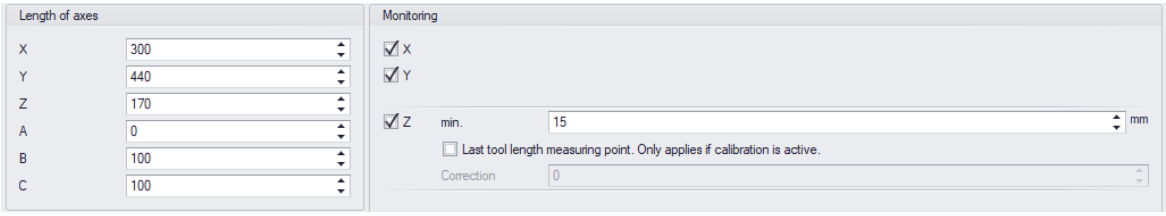
Before configuring the axes and additional parameters, make sure your machine is aligned to match your own working position. This will simplify the setup process. The *Axes* section provides instructions for configuring your machine's axes.

3.5.1 Travel Range and Travel Range Monitoring

Travel Range

In the *Travel Range* section, the working area of the machine is defined. The values entered here correspond to the maximum area that the machine can process. These values can either be taken from the machine's manual or determined by measurement.

If you need to measure the travel range manually, keep in mind that you must not only measure the pure axis length. You also need to consider the spindle clamping device — the corresponding values must be subtracted from the total axis length.



Travel Range Monitoring

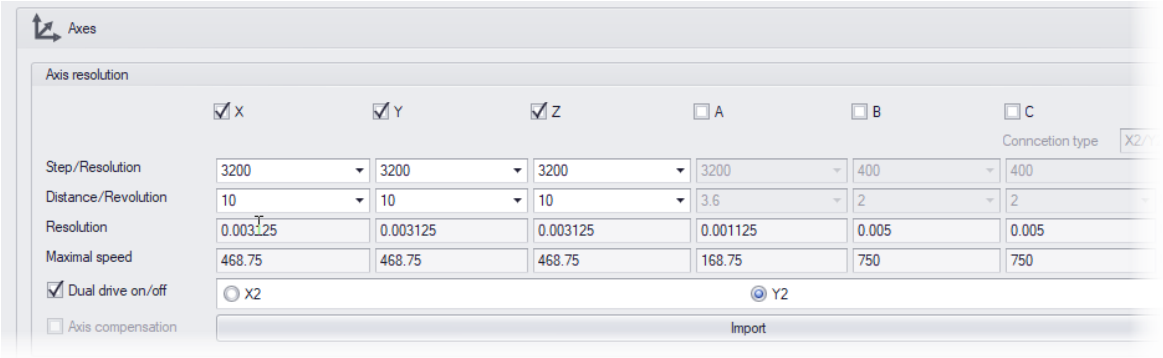
Travel range monitoring provides software-based supervision of each axis. This function prevents the machine from moving beyond its maximum travel limits, which could otherwise cause damage. In this example, for the X and Y axes, the monitored range extends from a minimum of 0 mm up to a maximum of 300 mm for X and 440 mm for Y. For the Z-axis, however, a minimum value (Z min.) must be defined to ensure that the spindle head does not crash into the worktable (here: min. 15 mm).

In addition, the reference switches are used for monitoring. By specifying the current tool length, the variable monitoring of the Z-axis minimum is activated (option **Last Tool Length Measurement Point**). Each new tool length is subtracted from the tool length specified in the input field to determine the new minimum Z height.


However, travel range monitoring will only work reliably if the machine positioning is accurate (i.e., no steps are lost). Further information on monitoring can be found in the chapters **Options → Messages**, **Safety Settings for Tool Changer**, and **Safety Zones**.

3.5.2 Dual Drive

The dual drive allows the X- or Y-axis to be operated with two motors. Both motors receive the same step and direction signals. Activation is done in **Machine Parameters → Axes** for X/X2 or Y/Y2.



Homing

- For X/X2, two separate homing switches are required.
- The switches are configured under **Machine Parameters → Pin Assignment**.

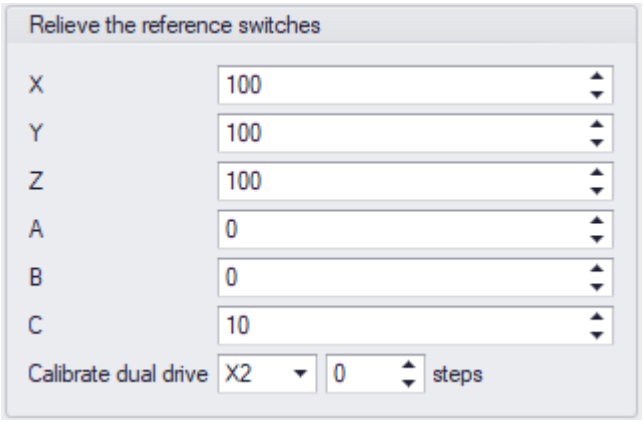
Sequence

- Both motors move simultaneously until the first switch is triggered.
- After that, the second axis continues to move until the second switch is also triggered.

Note: The homing switches should be placed as close to each other as possible.

Bridge Alignment

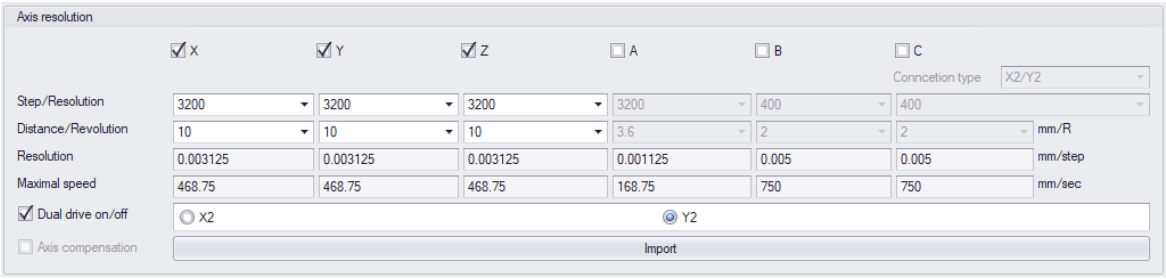
Since machine bridges are often not perfectly straight, a small correction distance can be specified for fine adjustment. This allows the bridge to be aligned precisely.



3.5.3 Axis Resolution

The first step in setup is to determine the axis resolution. This varies depending on the machine, since different types of axis drives (e.g., lead screw, timing belt, gearbox) and drivers are used. To calculate the axis resolution, you need the number of steps per revolution as well as the travel distance per revolution. These values can be found in your machine's user manual.

The number of steps per revolution is defined on the stepper motor driver (or servo), e.g., 1600 steps/rev. This value must then be entered into the corresponding field. The travel distance per revolution is the distance in millimeters that the lead screw, timing belt, or gearbox moves the axis in one full revolution.



After the values have been entered, cncGraF 8 calculates the axis resolution (mm/step) and the maximum possible speed (mm/sec) of the machine, displaying them below the corresponding fields. Using the checkboxes next to the axis labels, each axis can be enabled or disabled. In a standard three-axis configuration, only X, Y, and Z are active. The A, B, and C axes can be used for special tasks such as rotary or swivel movements.

To ensure optimal axis interpolation (simultaneous movement of multiple axes), all axes should have the same axis resolution. This improves the smoothness of machine operation.

Even if the CNC machine cannot mechanically achieve very high resolution, it is still recommended to set the axis resolution as high as possible — for example, 6400 steps per revolution with a travel distance of 10 mm per revolution. This results in a resolution of 0.0015625 mm per step. Such a high resolution makes it possible to select a finer arc resolution for the drawing 47.

Dual Drive

The X or Y axis can be operated with two motors. Enable the dual drive option and select the axis to be driven by two motors. Additional settings for operating an axis with two motors can be found in the chapter **Pin Assignment**.

Axis Compensation

The axis compensation function is used to correct inaccuracies in the lead screw, timing belt, or gearbox. First, the inaccuracy along the X and Y axes must be determined. For this purpose, a defined distance (e.g., every 200 mm) is traveled along the X-axis. The actual deviation (actual distance – target distance, here 200 mm) is calculated and stored line by line in an ASCII file.

Example of an ASCII file:
0,005
0,00425
-0,00025
-0,004875
-0,009

Both the X and Y axes must be measured in this way. Each ASCII file contains the values for a single axis, so two separate files are required. These files must be provided along with the actual travel distance, in this case 200 mm.

 **Warning:** Axis compensation is still in the testing phase!

3.5.4 Homing

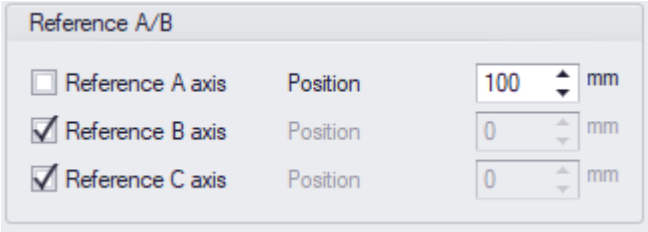
Reference Point
First, you need to define where the reference switches are located and in which direction the axes must move to reach the reference point. As shown in the diagram, there are four possible reference points (A, B, C, and D), each representing a corner of the working area. **Example**: If the reference point is at the lower left, with coordinates X=0 and Y=0, then reference point A must be selected.

Switch Release (Reference Switch Back-Off)
The *Reference Switch Back-Off* settings ensure that the switch is completely released (not activated) after being triggered. The required values can either be found in the technical specifications or determined experimentally (the distance from the switching point until the switch is fully released).

Safety Distance for "Accelerated Homing"
In the next step, the safety distance for accelerated homing is defined. In this mode, the machine moves at rapid speed (maximum speed) toward the reference points. The value entered defines the position at which the rapid movement stops and the slower approach to the reference point begins. In the example shown, the machine moves on the X- and Y-axes to within 20 mm of the zero position, and on the Z-axis to within 20 mm of Z-max.

After cncGraF 8 is started for the first time, a normal homing cycle without acceleration must be performed, since the program does not yet know the axis positions. Accelerated homing can then be enabled in the *Homing* dialog (key [F12]).

Disable Homing for A/B/C Axes
Homing for the A/B/C axes can be disabled. In the figure below, homing for the A axis is deactivated. Instead, the A axis position is set to 100 mm.

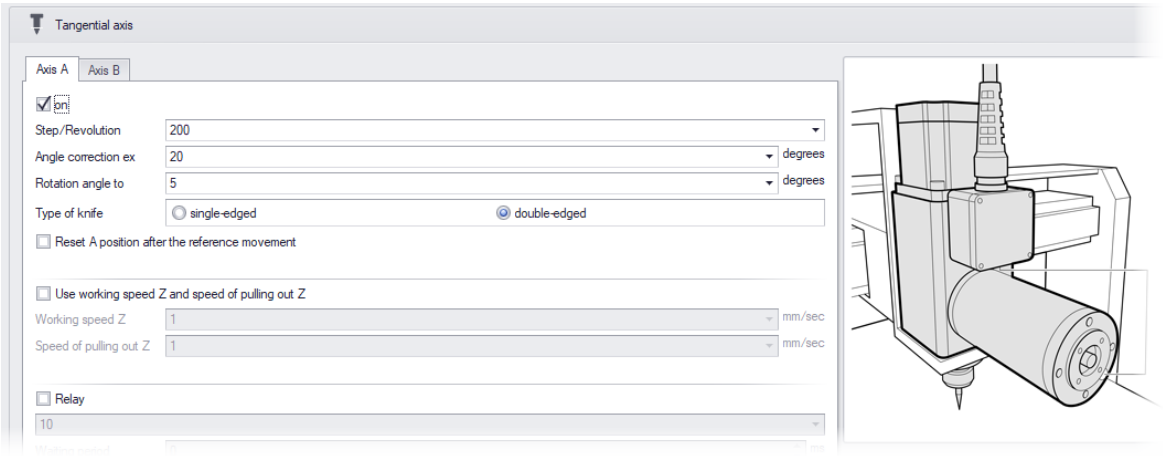


Once these values are defined, the inputs for the reference switches as well as the travel directions must be configured. This is done in the **Pin Assignment** menu.

3.5.5 Tangential Axis

For cutting materials such as leather or cardboard, a tangential knife is used. This knife automatically aligns itself with the cutting direction of the material using a stepper or servo motor. Both single- and double-edged knives can be used, although double-edged knives are recommended.

Using a tangential knife requires a 4-axis control. In addition to the X, Y, and Z axes, an extra stepper motor is needed to control the knife. Before the tangential axis can be activated, the A axis must be enabled in the menu **Machine Parameters → Axes**.



Axis A / Axis B
cncGraF 8 can control up to two tangential axes. The settings for each tangential axis are configured under the **Axis A** and **Axis B** tabs.

Steps per Revolution

Specifies the number of stepper motor steps required for one full revolution. For more details, see the chapter **Axis Resolution**.

Lift-Off Angle
This value defines the angle in degrees between two vectors at which the tangential knife can remain in the material while rotating. If this angle is exceeded, the knife is lifted out of the material, aligned with the cutting direction, and then plunged back into the material.

 To avoid damaging the material, the angle should not exceed 45 degrees. However, if the angle is set too small, the machining process may take significantly longer. Therefore, a balanced compromise should be chosen.

Rotation Angle Limit
If the angle is less than or equal to the value defined as *Rotation Angle Limit*, the knife rotates during motion until the desired travel direction is reached. If the vector angle exceeds the specified value, the knife stops and is aligned accordingly. The motion then continues (with the knife remaining in the material). To avoid damaging the material, only a small angle — up to about 10 degrees — should be entered.

Use Feed and Lift Speed
This function overrides the rapid speed of the Z-axis and allows separate speeds to be set for plunging and lifting. The Z feed speed defines the knife's plunge speed into the material, while the lift speed defines the speed at which the knife is retracted from the material.

Reset A/B Position after Homing
For the knife to operate correctly, it must be aligned as precisely as possible along the X-axis (parallel to X) after homing. The alignment can be performed manually by hand or via software. The software-based alignment has the advantage of greater accuracy. To align the knife using software, proceed as follows:

The function **Reference Switch Back-Off A/B** is used for alignment. A travel distance is specified in steps, ensuring that the knife is aligned exactly parallel to the X-axis.


The more precise the alignment, the better the cutting results. To verify alignment, run a simple drawing consisting of two horizontal lines.

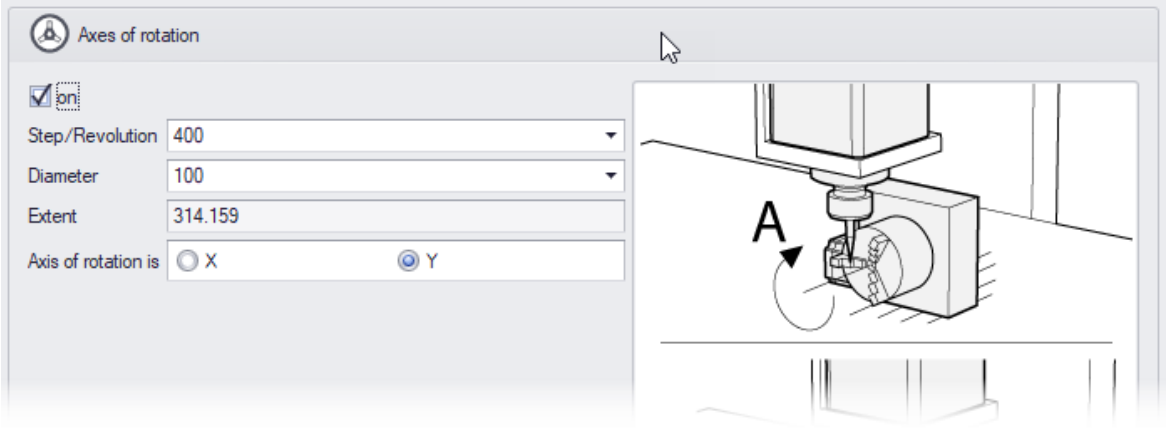
The function **Reset A/B Position after Homing** must be enabled so that the *A/B axis* is reset to zero after back-off (equivalent to homing without subsequent back-off).

Switch Outputs
Here, outputs can be switched either globally for all tools or tool-specific. This is required for Z-axes without a stepper or servo motor, which are lowered and raised via an output.

3.5.6 Rotary Axis

Depending on the number of axes supported by the controller, the rotary axis can be used in two different operating modes:

4-Axis Control
By enabling the 4th axis, it can be defined as a rotary axis. The rotary axis can be mounted on the worktable either in the X or Y direction.



If the G-code file (DIN 66025) contains coordinates for all four axes (4D machining), then the choice of axis direction only affects how the rotary axis is displayed on the screen.

3-Axis Control
With 3-axis control, the 4th axis must be disabled. The rotary axis is connected to the X and Y outputs by swapping the plugs.

Symbolic Representation of the Rotary Axis
Once the rotary axis has been enabled in **Machine Parameters → Axes → Rotary Axis**, a line of length $U = D \times P$ is displayed, where D represents the diameter of the material. From the starting point of this line, a second line is drawn at a right angle, representing the rotary axis. A third, dashed line marks the circumference of the material.

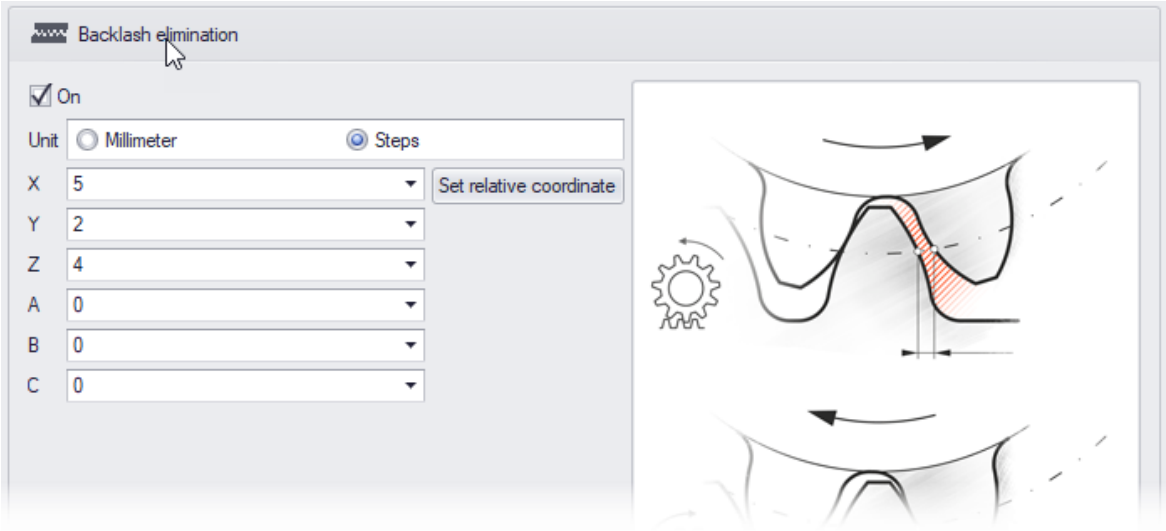
3.5.7 Z2

The Z2 function allows CNC machines to operate with a dual Z-axis.

First, the axis for Z2 operation must be enabled in the **Axes** menu. Then, select the appropriate operating mode for the axis designated as Z2. After that, the distance between the Z and Z2 axes in the X or Y direction must be specified. The Z2 axis can also be used in combination with the tangential axis.

3.5.8 Backlash

Backlash refers to unwanted play (slack) between the screw and nut, or between the pinion and rack of a mechanical drive. It may occur when the travel direction is reversed. To compensate for possible backlash in the machine, the corresponding values can be entered in this dialog. The settings can be specified either in millimeters or in steps.



3.6 Pin Assignment

In the *Pin Assignment* section, functions such as spindle, collet, reference switches, buttons, travel direction, and many other features controlled via inputs and outputs are configured.

The pin assignment settings can be found in the menu under **Settings → Machine Parameters → Adjust → Pin Assignment**.

If the CNC machine has not yet been fully configured, it is recommended to begin this chapter with the configuration of the *reference switches*. The homing procedure must always be performed first, since otherwise cncGraF 8 does not know the machine's position.

3.6.1 Reference Switches

In the *Pin Assignment* section, the settings for reference switches and travel directions are defined, among others.

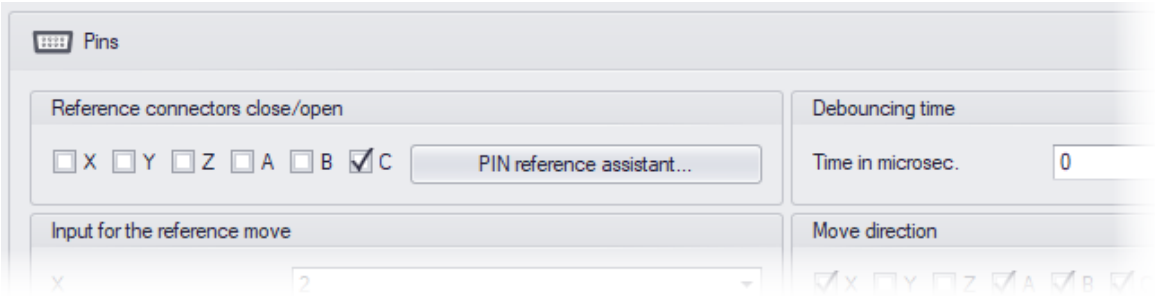
1. Preparations for the first homing cycle

First, the inputs of the reference switches should be checked. From the main menu, select **View → Set Reference Point** to set the reference point. This step is required to enable axis movement. The reference switches must be in the *NOT PRESSED* state, meaning the switches should be free. If an axis is positioned on a reference switch, it can be moved away from the switch using the control keys in **Manual Move** (note that the travel direction of the axes may not yet be correctly configured).

To check the inputs of the reference switches, click the green/blue USB/Ethernet icon in the status bar at the bottom (see figure).

The window for monitoring inputs and outputs will open. Press each reference switch one after the other. If the status of an input changes from *gray* to *green* or vice versa, the corresponding pin has been identified. Enter the pin number for each axis in the menu **Machine Parameters → Pin Assignment → Inputs for Reference Switches**.

Are the reference switches "normally open" or "normally closed"?
Another important step is to check whether the reference switches are configured as *normally open* or *normally closed* (the switches should still be in the *NOT PRESSED* state). To determine this, you can start the Reference Switch Wizard. The wizard automatically detects whether the switch is *normally open* or *normally closed* and sets the corresponding checkbox.
Example: In the figure below, the checkbox for the Z-axis was set by the wizard.



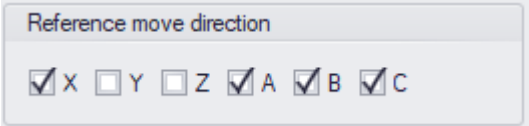
A *normally open* switch interrupts the circuit when activated, while a *normally closed* switch closes the circuit when activated.

Homing Sequence
Next, the *Homing Sequence* is defined — that is, the order in which the axes are moved during the homing cycle. It is recommended to use the sequences **ZXYAB** or **ZYXAB**. Variants such as **XZYAB** or **YZXAB** are not advisable, since the Z-axis would not be released first, creating a risk of the cutter colliding with the workpiece.

2. First Homing Cycle

The homing cycle can be executed from the main menu via **Move → Home...** or by pressing the [F12] key.

Attention: Pay close attention to the travel direction during the homing cycle! If an axis moves in the wrong direction, stop immediately, invert the homing direction in **Machine Parameters → Pin Assignment** (see figure), and restart the homing cycle.



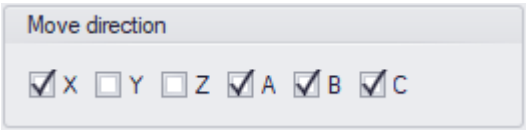
3. Final Check of Homing Settings
The homing cycle consists of two steps:

- First, the machine moves toward the reference switch at homing speed.

The homing speed and acceleration must be set so that the machine can stop behind the reference switch, since the braking distance is limited.

- Once the reference switch is triggered, the axis stops and then moves away from the switch at start/stop speed until the switch is released again.

4. Check Travel Direction
Next, the travel direction of the X, Y, and Z axes must be verified. To do this, move each axis in the **Manual Move** window. If an axis moves in the wrong direction, the orientation can be corrected in **Machine Parameters → Pin Assignment → Travel Direction** by inverting the checkbox for the corresponding axis.



3.6.2 Defining Inputs and Outputs

Once the reference switch configuration has been completed, the outputs for the spindle, pump, and collet can be defined. In addition, the inputs for the tool length sensor, zero-point sensor (X, Y, and Z axes), and depth probe can be configured.

Invert Inputs
In some cases, inputs need to be inverted because a switching state operates in reverse. A good example is the tool length sensor. It should switch *off* when pressed — meaning that in its unpressed state the input is active (the input indicator lights up).

The advantage of this configuration is that if the sensor fails, the machine will not continue to move onto the sensor and risk damaging it. The software assumes the switch is already pressed. In this case, the Z-axis will move slowly upward until it reaches the reference switch, and then stop.

Debounce Time
Debounce time refers to the delay applied to ignore false signals caused by the mechanical bouncing of a switch. When a switch is pressed, vibrations may cause the contact to briefly open and close multiple times before stabilizing. This disturbance is called *bouncing*. The debounce time ensures that only stable signals are recognized as valid switch states. Debounce time is specified in microseconds and defines the shortest switching time that will be accepted as valid, effectively filtering out noise.

 **Warning:** The higher the debounce time, the less precise the switching point becomes. Therefore, the debounce time should be kept as short as possible.

Invert Con2/3/4
The smc5d controller has several outputs on the Con2, Con3, and Con4 interfaces. When the controller is powered on, these outputs are set to a predefined state.
Example: If a pump connected to Output 2 starts immediately when the controller or the entire system is switched on, this output must be inverted in the **Invert Con** section. This prevents the pump from being activated automatically at startup.

3.6.3 Hood and Emergency Stop

Under **Machine Parameters → Pin Assignment → Hood and Emergency Stop**, safety functions are defined. There are two areas: *Hood/Door/Light Curtain* and *Emergency Stop*. All functions are controlled via inputs.

Hood/Door/Light Curtain
In this area, machine hoods, doors, and light curtains are monitored through inputs. For example, a hood can be secured via an input. The **Setup Mode** option allows slow manual movement of the CNC machine while the hood is open.

Emergency Stop
Here, any number of inputs can be defined as error signals. The **Emergency Stop Settings** dialog is divided into three sections:

1. **Input Settings:** Defines the input used for the error signal. To ignore faulty input signals, you can configure how many checks are required before the input is considered active, as well as the startup delay (in seconds after beginning the machining process).

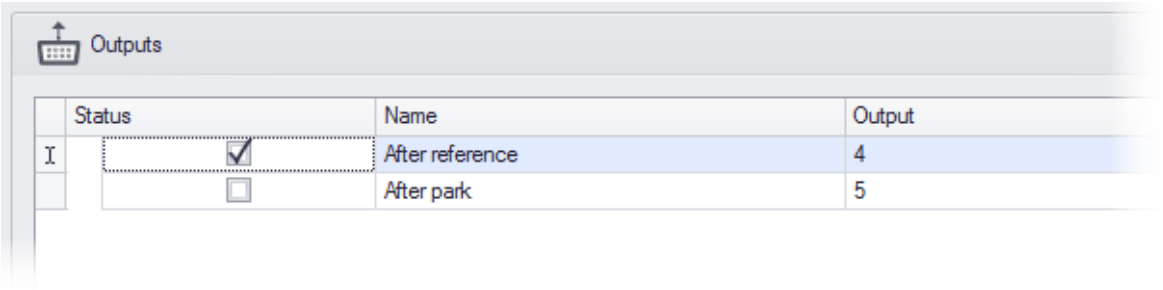
2. **Filter in Automatic Mode:** Determines when an error is recognized as valid. Various options are available, such as *Error only for a specific tool number* or *Error only if an additional output is active*. Example: If Input 21 switches off (inverted) while Tool 1 is in use (milling process) and Output 14 is active, the process will be aborted and an error message will be displayed.

3. **Action**: In the final section, you define which action is triggered when the error condition (consisting of input and filter) is met. The process will be aborted and a specified error message will be displayed on the screen. Optionally, an output — for example, to flash an external red warning light — can be activated, or the machining speed can be reduced.

Warning: Since *Hood and Emergency Stop* is a purely software-based solution, it does not provide the same level of safety as a hardware-based solution. A hardware solution should therefore always be preferred.

3.6.4 Outputs

In the **Outputs** menu, you can specify whether an output should be activated at the end of the homing cycle or after moving to the park position. In the figure below, the setting *After Homing* is active.



3.7 Speeds

In this section, the speeds for rapid traverse, homing, and start/stop movements are defined. The speed settings can be found in the menu under **Settings → Machine Parameters → Adjust → Speeds**.

It is recommended to set the rapid traverse speed first, then the start/stop speed, and finally the homing speed.

3.7.1 Rapid Traverse Speed

The rapid traverse speed is the maximum speed at which the machine can move. If the machine comes from a third-party supplier, the maximum speed can usually be found in the manufacturer's manual. If this information is not available, the maximum rapid traverse speed must be determined through testing.

When is the maximum possible speed reached?
The maximum speed is reached when the machine begins to lose steps outside the tolerance range of less than 0.1 mm (see also the section *Check for Lost Steps*).

3.7.2 Start/Stop Speed

The start/stop speed defines the speed at which the machine begins to move and comes to a stop. This value must be set according to the performance of your machine. It should not be set too high, as this can lead to lost steps. Some machines can handle a starting speed of 8 mm/sec, but this is an exception. Therefore, it is recommended to choose a lower value. The achievable speed depends on the stability and rigidity of your machine.


Once the appropriate values have been entered, you can continue with **Homing Speed**.

3.7.3 Homing Speed and Acceleration

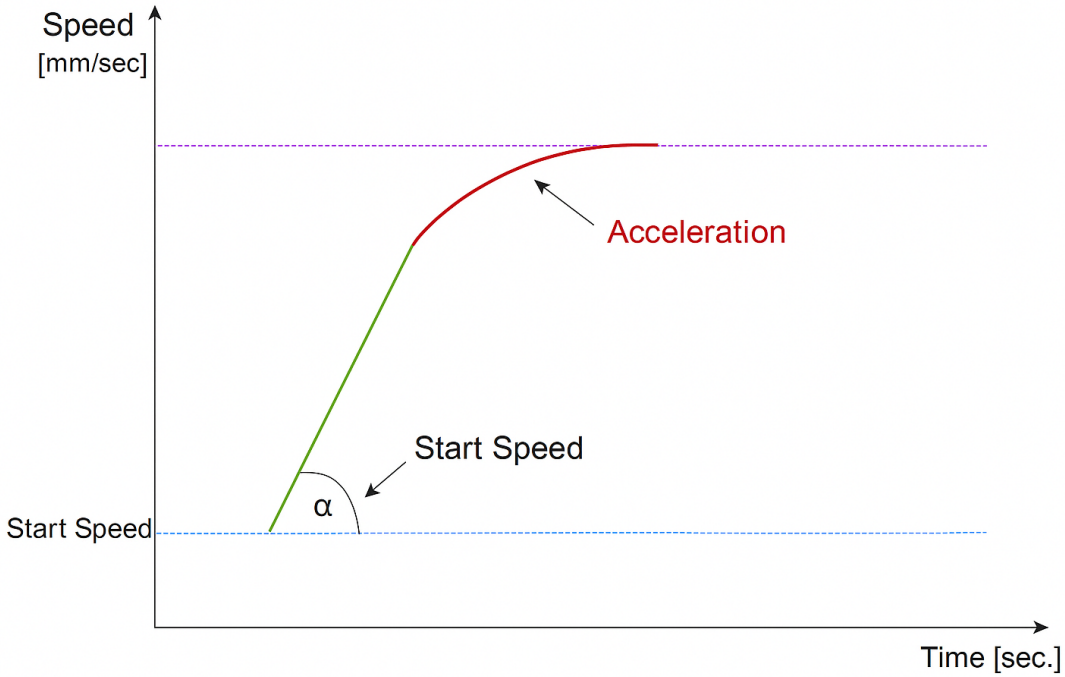
The homing speed should not be set too high, since the axis must be able to come to a stop in time after passing the reference switch. It is recommended to start with a low homing speed.

Enter the values in the field shown below:

Acceleration

Once the corresponding values have been entered, the ramp for the homing cycle must be defined. It is recommended to select the highest possible ramp setting so that the braking distance after the reference switch remains as short as possible, allowing for a higher homing speed (shorter waiting time).

The acceleration can be adjusted using the slider. The higher the value displayed in %, the steeper the acceleration ramp — resulting in a faster process. In addition, the ramp can be influenced by selecting the ramp *profile type*. Changing the type affects the acceleration and deceleration paths. By default, the type is set to Level 1 (on a scale of 1 to 6). The higher the selected value, the steeper the ramp. However, changing the type is only recommended for very stable machines capable of relatively high maximum speeds (e.g., 500 mm/sec). For most cases, it is recommended to use Type 1.



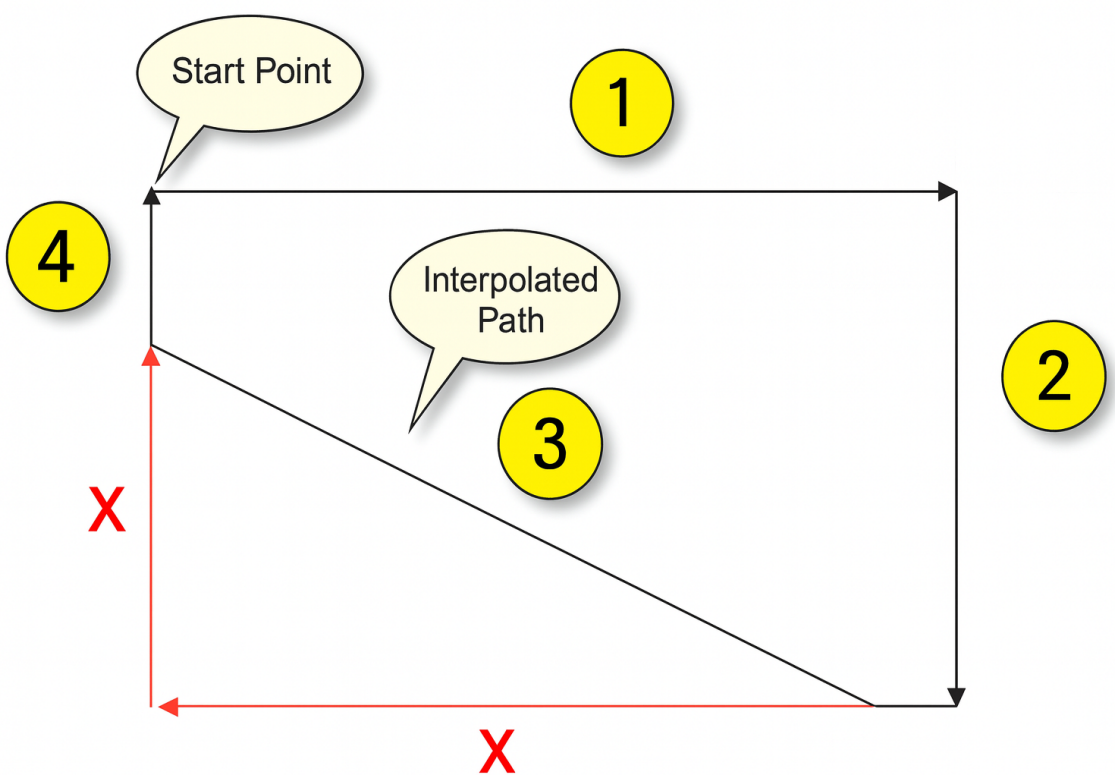
How to Check if the Acceleration Ramp is Set Correctly
To verify this, perform a homing cycle. If the machine hits the mechanical limit (clearly audible), the slider value must be increased and/or the homing speed reduced. Repeat this process until the optimal speed is reached.

Afterward, the acceleration value for milling, drilling, or moving can be set separately. This value may differ from that of the homing cycle and depends on the rigidity of your machine.

Reduce Jerk
With "poor" data, such as circles with coarse resolution, and at high feed rates, the machine may accelerate and decelerate repeatedly in quick succession. This behavior can cause jerking. Activate the **Reduce Jerk** option to achieve smoother motion.

Although the **Reduce Jerk** option improves motion smoothness, it also increases the machining time.

Constant Feed Rate
This option affects the machine's behavior during diagonal (interpolated) movements or circular motions, where two motors are always driven simultaneously. During diagonal travel, the distance covered is longer than in movements along only the X or Y axis. For example, if the axis speed is set to 20 mm/sec, the machine will attempt to maintain this value.



Function Active:
When this option is enabled, the controller adjusts the motor speeds so that the feed rate remains constant at 20 mm/sec, even along interpolated paths (Vector 3).

Function Inactive:
When constant feed rate is disabled, longer interpolated paths (Vector 3) are traversed in the same time as shorter paths, resulting in a higher actual travel speed.

3.7.4 Probing Speeds

The probing speeds define how fast the machine approaches the 3D probe, the tool length sensor, and the sensor for scanning three-dimensional objects. To prevent damage, the speeds for the 3D probe and the tool length sensor should be set low, since the braking distance is short.

3.8 3D/Surface Probe

In the *3D/Surface Probe* section, the calibration values determined during the automatic calibration of the 3D probe are displayed. These values should not be changed unless there are deviations — for example, in the ball diameter of the 3D probe — which may need to be adjusted manually.

 **Warning:** When probing surfaces, a 3D probe with a ball tip must not be used, as this can cause inaccuracies. In such cases, a pointed tip must be used.

3.9 Automatic Tool Changer

An automatic tool changer is a device for picking up and releasing tools. The cncGraF 8 software distinguishes between two variants: the *Z-axis tool changer* and the *X/Y-axis tool changer*, which differ in their travel paths. In addition, rotating tool changers can also be operated, controlled by an additional stepper motor. The tool changer settings can be found in the menu under **Settings → Machine Parameters → Adjust → Tool Changer**.

 Apart from these two variants, cncGraF 8 also supports other types of tool changers via the macro function. For more information, see the section **Macro Editor**.

3.9.1 Configuring the Tool Changer

Setting up the Z-Axis Tool Changer

Properties
Under *Properties*, the following parameters are defined: dwell time, insertion speed, lift speed, Z-travel, and Z-height. In addition, you can specify whether a homing cycle should be executed after each pick-up or release operation. You can also configure the tool numbering to start at T0.

- **Dwell Time**
  Defines how long the spindle should remain above the tool position before the next action is executed. This is the time the collet requires to grip or release the tool. The value may vary depending on the manufacturer. If no information is provided, the time should be determined experimentally.

- **Insertion Speed**
  Determines the speed at which the tool is moved into the tool changer. This value should be kept as low as possible to avoid potential damage to the tool or changer and to allow the machine enough time to decelerate.

- **Lift Speed**
  Defines the speed at which the tool is removed from the changer. This can be higher than the insertion speed.

- **Z-Travel**
  Specifies the distance along the Z-axis where the tool is moved into the changer at reduced speed until it comes to a stop. This value is determined by manually moving the machine to the start position of the slow approach in Z, then setting the relative Z-value in the main window to zero. Continue moving the Z-axis until the end position of the tool in the holder is reached. The value displayed under *RZ* corresponds to the *Z-travel* value.

- **Z-Height**
  Defines the safe clearance height in the Z-axis used when approaching the tool. This value cannot be set lower than the Z-value entered under *Position of the First Tool*. If a lower value is entered, cncGraF 8 automatically resets it to the Z-value of *Position of the First Tool*.

Under *Properties*, you can also configure the display color for occupied and free tool positions, and specify whether a homing cycle should be performed before placing a tool back. This option is useful if the machine has lost steps during machining and can no longer approach the tool changer position correctly.
If the checkbox for T0 is enabled, tool numbering begins at T0 (whereas in DIN-ISO files tools usually start with T1). Another option is *Close Collet*: when enabled, the collet closes after each pick-up and release operation, preventing continuous air flow.

- **Variable Placeholders**
  This option allows arbitrary tool numbers to be assigned to a tool changer placeholder. The tool number is entered in the *T/P* field of the *Distance* table.

- **Blow-Off**
  Specifies which output is used for the *Blow-Off* function. You can configure whether the function runs permanently or for a specified period of time. *Blow-Off* is intended to remove material residues from the collet, but it can also be used to control a cover for the tool changer.

- **Position of the First Tool**
  Here, the position of the first tool in the holder is defined. To determine the values: move the machine to the exact position of the first tool holder (in Z, to the start position of the slow approach). Then left-click **Set Current Position**. The X, Y, and Z coordinates are automatically entered into the respective fields.

- **Distance**
  Click **Add** to add the first tool to the tool list (the X, Y, and Z values remain at 0). Next, add another tool position in the same way, but this time enter a value (the center-to-center distance from Tool 1 to the next tool) in X or Y, depending on the orientation of the tool changer. A deviation in Z can also be corrected by entering a value. Repeat this process until the maximum number of tool positions is reached. After the parameters are entered, the tool changer appears in the main window's workspace. Tool positions can also be edited as absolute positions by enabling the **Show as Absolute Position** option.

Setting up the X/Y Tool Changer

The setup of the X/Y tool changer is identical to that of the Z-axis tool changer, with the only difference being that the Z-travel is not specified. Instead, the X/Y-travel is entered in the *X/Y Distance* field, since the tool change takes place from the side rather than from above along the Z-axis.

3.9.2 Safety Settings

Tool Monitoring
The tool monitoring parameters ensure that a tool is correctly picked up or released. If the spindle is equipped with an automatic collet that provides a monitoring contact, this can be connected to the smc5d controller, and the corresponding input can be specified here. Monitoring can be performed either with the option **Check if a tool is present** or **Check if the tool has been released**.

Another monitoring method is **Check if the tool has been released using a tool length sensor**. In this case, the machine moves to the tool length sensor down to a defined *minimum Z-height*. If the tool length sensor is triggered, it means the tool has not been released, and an error message is displayed.

Relative Travel
The values under **Relative travel to/from the tool changer** define the safe X- and Y-distances that the machine should move before and after picking up or releasing a tool. The same applies to the values under **Relative travel to/from tool measurement**.

Spindle Stopped
A spindle that activates a contact when it stops can be monitored by cncGraF 8. In this case, a waiting window is displayed until the corresponding input is triggered.

Tool Changer and Tool Measurement Outside the Machine Area
The parameters **Allow tool length measurement outside the machine area** and **Move to position at end of operation** make it possible to place the tool changer and the tool length sensor outside the machine's working area. The value for *Move to position at end of operation* specifies the location the machine should move to after a tool change or tool measurement, ensuring that the CNC machine is back inside the working area once the operation is completed.

3.10 Swivel Arm

The swivel arm allows the surface probe to be automatically lowered or raised.

The swivel arm settings can be found in the menu **Settings → Machine Parameters → Adjust → Extras**. The swivel arm is controlled via an output. The dwell time defines how long it takes for the swivel arm to fully move down or up. An input can be used to check whether the swivel

arm is in the lower or upper position. The swivel arm can be used both for **Surface Probing** and **Zero-Point Detection**.

3.11 Vacuum Table

Sections of the vacuum table can be automatically switched on or off during the milling process. The corresponding settings are located in the menu **Settings → Machine Parameters → Adjust → Extras**.

In the table, multiple vacuum table sections can be defined, each with associated outputs. These sections can also be displayed in the 2D view for verification (**View → Vacuum Table**).

Control Modes
The vacuum table can be controlled in different ways:

- **Permanent:** All vacuum table outputs are switched on at the beginning of the milling process and remain on until the end.
- **Area Activation:** When the cutter enters a vacuum table section, that section is switched on (output enabled) and remains on until the process is complete.
- **Variable Area Control:** A section is switched on as soon as the cutter enters it and switched off again once the cutter leaves the area.
- **Permanent – Only Used Sections:** All required outputs (depending on the drawing data) are switched on at the beginning of the milling process and remain on until the end.

3.12 Final Check

This section outlines some final checks for your machine and its settings. If all machine parameters are configured correctly but lost steps or inaccurate machining results still occur, the cause may be an incorrect step signal configuration of the drivers or insufficient power supply to the motors.

Here are some suggestions for troubleshooting such cases:

- Check the driver timing
- Test the motors
- Verify the axis resolution
- Check for lost steps

3.12.1 Check Driver Timing

Stepper motor drivers typically use the **Active Low** configuration for clocking the step and direction (DIR) signals. However, some drivers operate with an **Active High** configuration. The cncGraF 8 software is set to **Active Low** by default.

The step pulses are represented graphically as follows:

Depending on the driver used, the checkbox **Invert Clocking** must be set in the menu **Settings → Machine Parameters → Adjust → Drivers**. If the checkbox is not set, **Active Low** (illustration with 0) is active. If the checkbox is set, the clocking changes to **Active High** (illustration with 1).

Warning: It is essential that all drivers operate with the same clocking mode. Using different modes may result in misinterpretation of the signals.

Modern drivers often also allow switching the clocking mode directly via a physical switch on the driver itself.

3.12.2 Motor Test

Current reduction is a function that prevents stepper motors from overheating and being damaged while at standstill. This function can be enabled in the menu **Settings → Machine Parameters → Adjust → Pin Assignment**, where a shut-off delay can also be defined.

To check whether current reduction is working correctly, allow the motors to remain idle for a while and then measure their temperature. If the motors are "cool," current reduction is functioning properly. If not, the function must be inverted by enabling the checkbox **Reduce Motor Current (Pin 8)** under *Con2* in the menu.

In the previous section *Driver Timing*, incorrect clocking was mentioned as one possible cause of lost steps. If the clocking is configured correctly, another cause of lost steps may be insufficient motor supply voltage. In this case, you should verify whether the drivers are providing the correct voltage for the motors.

3.12.3 Check Axis Resolution

To verify the axis resolution, proceed as follows:

X and Y Axes:

1. Place a sheet of paper on the machine's worktable and mark the starting point of the cutter.

2. Using the manual machine control (**Manual Move**) in the cncGraF 8 main window, enable **Relative Travel** and enter a distance of 100 mm (or another desired value) in one of the fields. Then click the control icon for the axis you want to test.
3. The machine will move the specified distance of 100 mm.
4. Mark the cutter's end position on the paper and measure the distance between the two points.
5. If the measured distance equals the desired 100 mm, the axis resolution is correctly set. If not, the axis resolution must be adjusted.

Z Axis:

1. Move the Z-axis to Z-max and measure the distance from the machine table to the Z-spindle holder.
2. Using the manual machine control (**Manual Move**) in the cncGraF 8 main window, enable **Relative Travel** and enter a distance of 100 mm (or another value). Then click the Z control icon to start the movement.
3. Once the machine has reached the end position, measure the distance again from the machine table to the Z-spindle holder.
4. If the difference between the start and end measurements matches the entered distance, the Z-axis is correctly calibrated. If not, the Z-axis resolution must be adjusted accordingly.

3.12.4 Check for Lost Steps

How do I check for lost steps?

1. Load a file to be processed. The file should be large enough for the CNC machine to run for about 30 minutes.
2. Set the Z zero point high enough so the CNC machine runs "in the air."
3. Perform a homing cycle.
4. Click the **Milling/Drilling** icon and let the machine process the entire file.
5. After the file has been fully executed, start **Check Position...** (**Menu → Move → Check Position...**). A homing cycle will be performed and the results displayed.
6. If the position data of any axis is shown in red (outside the tolerance > 0.1 mm), this indicates lost steps.

Lost steps can have various causes, such as mechanical issues, incorrectly configured drivers, excessive speeds, or even electrical interference. All of these factors should be checked.

4 Introduction to File Processing

The cncGraF 8 software supports various file formats, including DIN 66025 (G-code), Isel NCP, as well as 2D formats such as HPGL, DXF, PostScript, Adobe Illustrator, Excellon, Sieb & Maier 1000, and Sieb & Maier 3000.

Processing G-code Files
G-code files are typical machine files for professional CNC machines and are usually multidimensional (typically 3D, but 4D up to 6D is also possible). These files are generally generated by CAD/CAM software, but can also be programmed manually using the integrated text editor. This approach allows for the production of complex parts with virtually no limitations, although it is relatively complicated for beginners.

Processing 2D Files
Formats such as DXF and HPGL are 2D files that can be easily created with any CAD software. Since these files are not directly designed for CNC machining, they require some post-processing. cncGraF 8 provides a tool palette that allows for straightforward editing, including tool compensation, defining milling order, creating pockets with islands, and other simple functions. This approach offers an easy way to quickly produce 2.5D parts. The advantage lies in its simplicity, although it is suitable only for simple 2.5D parts.

Arc Resolution
Most file formats such as G-code, DXF, or HPGL contain arc and circle commands. However, the smc5d controller does not support arc commands for technical reasons (the conversion would be too computationally intensive for the controller and would drastically reduce its frequency and speed). Therefore, these commands are converted into polylines. The arc resolution can either be selected in the dialog that appears when opening the file, or later in the menu **Settings → Options → File**.


The arc resolution should be set neither too high nor too low. It should not exceed the axis resolution of the CNC machine. An arc resolution about two to three times finer than the axis resolution of the CNC machine is recommended. For this reason, it is advisable to set the resolution to *Medium* or *High*.


Even if the CNC machine cannot mechanically achieve a very high resolution, it is still recommended to configure the axis resolution as high as possible — for example, 6400 steps per revolution with a travel of 10 mm per revolution. This results in a resolution of 0.0015625 mm per step. Such a high resolution allows the use of a finer arc resolution.

“Open File” Dialog

Files can be opened via **File → Open** in the menu. The file type *.* displays all files. In this case, cncGraF 8 automatically detects the file type. If the file type is not correctly recognized, the specific type must be selected manually — for example, for G-code files: *DIN 66025; ISEL NCP (.nc; .ncp)*.

 There are two different windows available for opening files: the standard Windows *Open File* dialog and the cncGraF 8 *Open File* dialog with file preview. The desired window can be selected under **Settings → Options → File → Properties**. The standard Windows *Open File* dialog has the advantage of being faster, since it is integrated into the Windows system.

4.1 G-code and the Postprocessor

Standardized G-code according to DIN 66025 is the standard format for professional machine tools in the 2½D and 3D range. cncGraF 8 supports this standard as fully as possible and is therefore compatible with all common CAD/CAM programs.

Postprocessor (PP)
The postprocessor (PP) defines how the G-code is generated for a specific CNC machine. Although G-code is based on the DIN 66025 standard, differences in interpretation exist among different CNC machines. These differences are described in the postprocessor. In CAD/CAM programs, postprocessors can be customized to precisely adapt the G-code to the requirements of the machine.

Overview of G-functions
- G00 51: Rapid traverse
- G01 51: Linear move with feed rate
- G02 52: Circular interpolation clockwise
- G03 52: Circular interpolation counterclockwise
- G04 53: Dwell time
- **G28**: Move to Z reference point
- **G28P**: Move to Z reference point and XY park position
- G40 54: Cancel tool radius compensation
- G41 54: Activate tool radius compensation (tool left of contour)
- G42 54: Activate tool radius compensation (tool right of contour)
- G53 55: Cancel work offsets
- G54-G59 55: Select work offset
- G60 55: Re-select last canceled work offset
- G70 56: Units in inches
- G71 56: Units in millimeters
- G80 57: Cancel drilling cycle
- G81 57: Drilling cycle: simple drilling
- G82 57: Drilling cycle: simple drilling with dwell at bottom
- G73/83 57: Drilling cycle: peck drilling (fast/deep-hole with chip breaking)
- G84 59: Tapping cycle (right-hand and left-hand threads)
- G87 59: Circular pocket cycle
- G90 60: Absolute programming
- G91 60: Incremental programming
- G98 57: Return to initial plane in drilling cycle
- G99 57: Return to reference plane in drilling cycle

Overview of M-functions

- **M00**: Program stop (program resumes after key press)
- **M03**: Spindle on (clockwise)
- **M04**: Spindle on (counterclockwise)
  Note: Counterclockwise rotation is switched via a relay in the “DIN 66025 Properties” dialog.
- **M05**: Spindle off
- **M08**: Coolant on
- **M09**: Coolant off
- **M20**: Plasma torch off
- **M21**: Plasma torch on
- **M30**: End of program
- **M48/M49** 62: Feed and spindle speed override enabled/disabled
- **M995-M998** 62: Call a macro inside a DIN 66025 file

Comments

- **(....)** Comments in round brackets.
  Example: `N01 G00 Z10 (Position Z10 anfahren)`
- **[....]** Comments in square brackets.
  Example: `N01 G00 Z10 [Position Z10 anfahren]`
- **;** Comments after a semicolon.

Example: `N01 G00 Z10 ;Position Z10 anfahren`

Other Commands

- **F**: Feed rate in mm/min
- **N**: Block number
- **T**: Tool number
- **S**: Spindle speed
- **D**: Tool diameter in millimeters (must be used together with tool number)
  Example: `T1 D2.5`

Special Commands

- **M99 P** 61: Subprograms (*modified!*)
- **G25-G26** 53: Loop
- **R-Parameter** 63: Variable
- **IF-Anweisung** 64: Conditional execution

Structure of a Block according to DIN 66025

No.	Function/Condition	Coordinates	Spindle Speed	Feed Rate	Tool	Miscellaneous
N	**G**	**X Y Z A**	**S**	**F**	**T**	**M**

All commands can be written in lowercase and without spaces using any standard text editor. The line number **Nxx** is optional and can be omitted, as it has no impact on program execution.

4.1.1 G00/G01 – Rapid Traverse and Linear Feed

With **G00**, the machine moves in **rapid traverse**, i.e., at maximum speed. This motion is typically used for positioning moves where the tool is not cutting (air moves). With **G01**, the programmed path is executed at the specified **feed rate**.

Parameters for G00 and G01

- **X** – Position on the X-axis
- **Y** – Position on the Y-axis
- **Z** – Position on the Z-axis
- **A** – Position on the 4th axis (the letter assigned to the 4th axis can be configured in *DIN66025 Properties*)
- **B** – Position on the 5th axis (the letter assigned to the 5th axis can be configured in *DIN66025 Properties*)
- **F** – Feed rate
- **S** – Spindle speed

Example

N01 **G00** Z10 (Move to position Z10)
N02 **G00** X100 Y10 (Move to position X100 Y10)
N03 **M3 M8** (Start spindle with M3 and coolant pump with M8)
N04 **G01** Z-2 F50 (Feed 2 mm into the workpiece at 50 mm/min)

N05 **G01** X100 Y100 F100
N06 **G01** X20 Y20
N07 **G01** X100 Y10

N08 **G00** Z10 **M5 M9** (Retract to Z10, stop spindle with M5 and coolant pump with M9)

N09 **M30** (End program with M30)

4.1.2 G02/G03 – Circular Interpolation

- **G02** is used to move along an arc in a **clockwise** direction.
- **G03** is used to move along an arc in a **counterclockwise** direction.

The **start point** of the arc is always the current tool position. The **end point** of the arc is specified with the **X, Y, and Z coordinates** in the G02/G03 command. The **circle center** is defined by the offsets **I, J, and K**.

Parameters for G02 and G03

X – End coordinate in X direction

Y – End coordinate in Y direction

I – Circle center offset in X direction

J – Circle center offset in Y direction

K – Circle center offset in Z direction

Example 1 – Circular arc in XY-plane

```
N01 G00 X0 Y0 Z10 (Move to position X0 Y0 Z10)
N02 M3 (Start spindle with M3)

; Feed to X10 Y10 at 100 mm/min
N03 G01 X10 Y10 F100

; Feed into the workpiece 2 mm at 50 mm/min
N04 G01 Z-2 F50

; Circular arc from X10 Y10 to X30 Y10
; with circle center at I20 J10
N05 G02 I20 J10 X30 Y10

; Retract tool by 10 mm
N07 G00 Z10
N08 M30  (End program with M30)
```

Example 2 – Thread milling

This program mills a thread with a pitch of 2 mm and a depth of 6 mm (3 turns). For a full circle, the start point and end point are identical.

```
N01 G0 X500 Y500
N02 G0 Z-5
N03 G2 I550 J500 Z-7 F60
N04 G2 I550 J500 Z-9
```

```
N05 G2 I550 J500 Z-11
N06 G0 Z0
M07 M30     (Activate M7, end program with M30)
```

4.1.3 G04 – Dwell Time

The **G04** command programs a dwell, i.e., a pause in machine motion. For example, **G04 H2** specifies a dwell time of **2 seconds**.

This function is typically used to allow the spindle time to reach its operating speed before cutting begins.

Parameters for G04

- **H** – Time in seconds

Example

```
; Hold the tool position for 2.5 seconds
; (spindle continues rotating)
N10 G04 H2.5
```

4.1.4 G25/G26 – Loop

With **G25** and **G26**, a loop can be programmed.

- **G25** defines the **start of the loop**.
- **G26** ends the loop.
- The parameter **Q** specifies the **number of iterations**.

Example: **Q5** → the loop contents are executed 5 times.

 **Note:** The number of iterations has changed in the newer version. For example, `Q2` now means **2 iterations**. In older program versions, it was **3 iterations**.

Parameters for G25 and G26

- **Q** – Number of iterations

Loops can be **nested arbitrarily**, but they **must not** be defined inside subprograms.

Example

```
N10 T1 M3 S2501
N20 G0 X10 Y10
N30 Z-5 M8
N31 G04 H6
N40 G91
```

N42 **G25** Q2

N44 **G25** Q4
N50 P1 (Call subprogram 1)
N51 **G26**

N60 **G0** X-65 Y10
N62 **G26**

N44 **G25** Q4
N50 P1 (Call subprogram 1)
N51 **G26**

N120 **G90**
N130 **G0** Z0 **M9**
N140 **M5**
N150 **M30** (End of program)

N160 **M99** P1 (Start of subprogram 1)
N170 **G2** I3.5 J0 Z-2 F360
N180 I3.5 J0 Z-2
N190 I3.5 J0 Z-2
N200 I3.5 J0 Z-2
N210 I3.5 J0 Z-2
N220 I3.5 J0 Z-2
N230 I3.5 J0
N240 **G1** X3
N250 **G2** I0.5 J0 Z-2 F180
N260 I0.5 J0 Z-2
N270 I0.5 J0 Z-2
N280 I0.5 J0 Z-2
N290 **G0** Z20
N290 **G0** X10
N300 **M00**
N310 **M99** P0 (End of subprogram)

4.1.5 G40/G41/G42 – Tool Radius Compensation

The cncGraF 8 software supports the commands **G40**, **G41**, and **G42**. After loading a G-code file that contains G41 or G42 commands, a **radius compensation dialog window** will appear.

Example
N10 T06 M03 S2000

```
N11 M08
N12 G00 X1.700 Y0
N13 Z10.000
N14 Z-3.000
N15 G01 G41 X4.558 Y-1.650 Z-3.488 F250
N16 G03 X5.000 Y0 Z-3.743 I1.700 J0
N17 X-5.000 Y0 Z-4.671 I0 J0 F500
N18 X5.000 Y0 Z-5.600 I0 J0
N19 X-5.000 Y0 Z-5.900 I0 J0
N20 X5.000 Y0 Z-6.200 I0 J0
N21 X-5.000 Y0 Z-6.500 I0 J0
N22 X5.000 Y0 Z-6.800 I0 J0
N23 X-5.000 Y0 Z-7.100 I0 J0
N24 X5.000 Y0 Z-7.400 I0 J0
N25 X-5.000 Y0 Z-7.700 I0 J0
N26 X5.000 Y0 Z-8.000 I0 J0
N27 X-5.000 Y0 Z-8.300 I0 J0
N28 X5.000 Y0 Z-8.600 I0 J0
N29 X-5.000 Y0 Z-8.900 I0 J0
N30 X5.000 Y0 Z-9.200 I0 J0
N31 X-5.000 Y0 Z-9.500 I0 J0
N32 X5.000 Y0 Z-9.800 I0 J0
N33 X-2.500 Y4.330 Z-10.000 I0 J0
N34 X-2.500 Y4.330 I0 J0
N35 X-3.708 Y3.122 I-.850 J1.472
N36 G01 G40 X-.850 Y1.472
N37 G00 Z-3.000
N38 G90 Z10.000
N39 M09
N40 M05
N41 M30
```

 **Warning:** The calculated toolpath compensation **must always be visually verified** in the 2D/3D view to ensure accuracy.

4.1.6 G53–G60 – Work Offsets (Zero Points)

- **G54 to G59** are used to select work offsets (zero points).

Example: G55 X20 Y10 applies an offset of 20 units in X and 10 units in Y.

- **G53** cancels the currently active work offset.
- **G60** reactivates the last canceled work offset.

Parameters for G54 to G59

- **X** – Offset in X
- **Y** – Offset in Y
- **Z** – Offset in Z

Example

```
N10 G0 T01 Z5
N20 G0 X10  Y10
N30 G1 Z-1 F10
N40 G1 X10 Y20 F20
N50 G1 X20 Y20
N60 G1 X20 Y10
N70 G1 X10 Y10
N80 G0 Z5
N90 G54 X30 (Activate work offset with X shifted by 30)
N100 G0 X10  Y10
N110 G1 Z-1 F10
N120 G1 X10 Y20 F20
N130 G1 X20 Y20
N140 G1 X20 Y10
N150 G1 X10 Y10
N160 G0 Z5
N170 G53 (Cancel work offset)
N180 M30 (End program)
```

4.1.7 G70/G71 – Units of Measurement

- **G70** sets the coordinate system to **inches**.
- **G71** sets the coordinate system to **millimeters**.

4.1.8 Drilling Cycles G73, G80–G84

This chapter describes the drilling cycles **G81, G82, G83, and G84**.

Change to G84
The **G84** command has been modified.
If you need to use the **legacy version**, enable the option **"Use old G84"** in the menu: "**Settings → Options → DIN66025** 59."

G98 / G99 – Retract Height
- **G98** – After the drilling cycle, the tool returns to the **initial (start) height**.
- **G99** – After the drilling cycle, the tool retracts only to the **R plane** (retract height), defined by parameter **R**.

G80 – Cancel Drilling Cycle
The drilling cycle is canceled with G80 or by issuing another motion command such as G00 or G01

G81 / G82 – Simple Drilling and Drilling with Dwell
These cycles are used for basic drilling (**G81**) and drilling with dwell at the bottom of the hole (**G82**).
Format: G98(G99) G81(G82) X Y Z R F (P)

Parameters for G81
- **X** – X position
- **Y** – Y position
- **Z** – Drilling depth (absolute)
- **R** – Incremental retract height, relative to the start point in Z
- **F** – Feed rate

Additional parameter for G82
- **P** – Dwell time at the hole bottom (in milliseconds, 1000 ms = 1 second)

Example: G98 G82 X10 Y10 Z-10 F300 P100

G83 – Peck Drilling (Deep-Hole Drilling with Chip Breaking)
Compared to G82, G83 includes the additional parameter Q, which defines the depth of cut per peck to break chips.

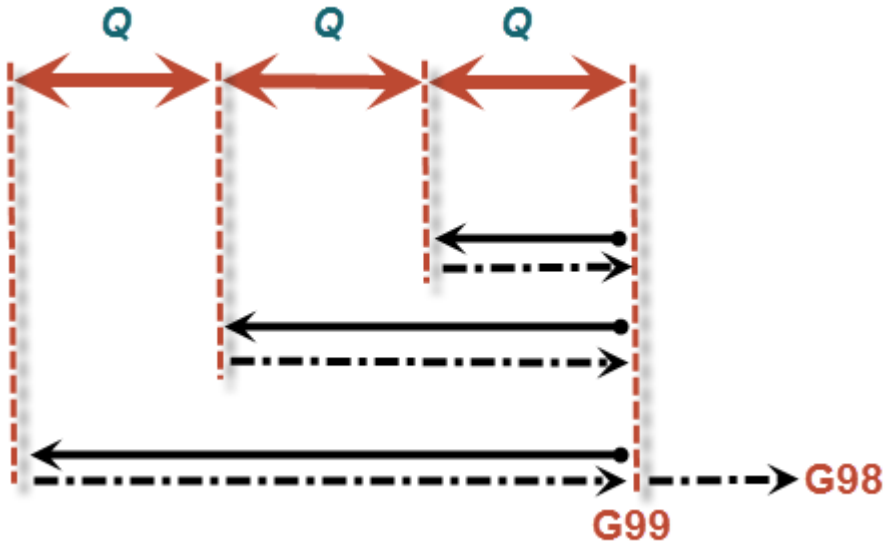


Abbildung 1: Entspänen mit Q Parameter

Format: G98(G99) G83 X Y Z R F P Q

Parameters for G83
- **X** – X position
- **Y** – Y position
- **Z** – Drilling depth (absolute)
- **R** – Incremental retract height, relative to the start point in Z
- **F** – Feed rate
- **P** – Dwell time at the hole bottom (in milliseconds)
- **Q** – Peck depth (in mm)

Example: G98 G83 X10 Y10 Z-10 F300 P100 Q2

G73 – High-Speed Peck Drilling
This cycle is similar to **G83**, but instead of retracting completely after each peck, the tool only performs a **short retract motion**. The retract distance can be configured under *Options → DIN66025*.

G84 – Tapping Cycle (Right-Hand and Left-Hand Threads)
Format: G98(G99) G84 X Y Z R P F M

Parameters for G84
- **X** – X position
- **Y** – Y position
- **Z** – Thread depth (absolute)
- **R** – Incremental retract height, relative to the start point in Z

- **P** – Dwell time at the hole bottom (in milliseconds)
- **F** – Feed rate (must match thread pitch)
- **M** – **M03** for right-hand threads, **M04** for left-hand threads

Example: G98 G84 X10 Y10 Z-10 F300 P100 M03

4.1.9 Drilling Cycle G84 (Legacy Version)

In the legacy version of **G84**, drilling is performed with **pecking (chip breaking)**. The tool drills to the specified end depth **Z** at the programmed feed rate **F**, reaching the final depth step by step through multiple incremental cuts.

Parameters for G84 (Legacy)

- **X** – X position
- **Y** – Y position
- **Z** – Final drilling depth
- **D** – Incremental step depth in Z
  If omitted (`D` or `D0`), the tool drills directly to the final depth in one pass
- **F** – Feed rate
- **H** – Safety clearance (retract height)
- **E** – Dwell time at the bottom (in seconds)

Example

```
...
...
N110 G00 G90 M03
N120 G00 X10 Y10
N130 G00 Z3
N140 G84 Z-10 D5 H3 E2 F50 (Drill a hole with pecking steps)
N150 G84 X100 (Drill another hole at X=100, Y=10 using the same parameters)
...
...
```

4.1.10 G87 – Circular Pocket

The **G87** command is used to machine a circular pocket. The coordinates **X, Y** define the **center of the pocket**.

To achieve the desired pocket dimensions, the **tool diameter D** should be specified together with the tool number (see example below, line N100). If no diameter is specified, the value is taken from the **tool library**.

The pocket is milled **from the center outward in a clockwise direction** at the specified **feed rate F** and with the incremental step **K**. The depth is cut step by step, with each pass defined by the **incremental depth D** and executed at the **Z feed rate E**. After the final depth is reached, the machine retracts to the **safety height H** and the cycle ends.

Parameters for G87

- **X** – X position (pocket center)
- **Y** – Y position (pocket center)
- **Z** – Absolute final depth
- **R** – Pocket radius
- **K** – Radial step-over (X/Y direction)
- **D** – Depth increment (Z direction)
- **F** – Feed rate (X/Y direction)
- **E** – Feed rate in Z
- **H** – Safety clearance

Example

```
...
...
N100 T1 D1 M6 S500  ; Tool 1 with a diameter of 1 mm (D1) is selected
N110 G00 G17 G90 M3
N120 G00 X50 Y50
N130 G00 Z3
N140 G87 Z-5 K1 D2 R4 F150 E50 H3
...
...
```

4.1.11 G90/G91 – Absolute or Incremental Programming

- **G90** sets **absolute programming**, meaning all subsequent coordinates are specified relative to the program's absolute zero point.
- **G91** sets **incremental (relative) programming**, where each new position is calculated relative to the current tool position (chain dimensioning).

Example

```
N08 G90 (Set absolute programming)
N09 G01 X100 Y100 (Move to absolute position X100 Y100)
; Switch to incremental programming (chain dimensioning) with G91
N10 G91
N11 G01 X10 (Move to X110 Y100)
N12 G01 X5 Y-5 (Move to X115 Y95)
```

4.1.12 M99 – Subprograms

With **M99**, up to 99 subprograms can be defined at the end of the main program (after **M30**). A subprogram definition begins with **M99 P..** and ends with **M99**. The subprogram is called using the command **P..** within the main program.

 **Note:** The syntax for calling subprograms has changed compared to earlier versions.

Example

```
N10 T1 M3 S2500
N20 G0 X21.5 Y25
N30 Z-5 M8
N40 G91
N50 P1 (Call subprogram 1)
N60 G90
N70 G0 Z0 M9
N80 X21.5 Y75
N90 Z-5 M8
N100 G91
N110 P1 (Call subprogram 1 again)
N120 G90
N130 G0 Z0 M9
N140 M5
N150 M30 (End of main program)

N160 M99 P1 (Start definition of subprogram 1)
N170 G2 I3.5 J0 Z-2 F360
N180 I3.5 J0 Z-2
N190 I3.5 J0 Z-2
N200 I3.5 J0 Z-2
N210 I3.5 J0 Z-2
N220 I3.5 J0 Z-2
N230 I3.5 J0
N240 G1 X3
N250 G2 I0.5 J0 Z-2 F180
N260 I0.5 J0 Z-2
N270 I0.5 J0 Z-2
N280 I0.5 J0 Z-2
N290 G0 Z20
N300 M00
N310 M99 (End of subprogram)
```

Subprograms can be managed directly in the integrated text editor. Open the subprogram manager via the menu: **Edit → Subprograms** in the text editor.

4.1.13 M995 to M998 – Macro Execution

The control commands (M functions) **M995 to M998** start a macro within a DIN66025 program file.

Example
N10 **G90** (Set absolute programming)
N20 **G01** X100 Y100 (Move to position X100 Y100)
N30 **M995** (Start macro)
M30

In the dialog **Settings → Link Macro**, the desired macro can be assigned to one of the commands **M995–M998**. Macros can also be added directly into a DIN66025 file using the integrated text editor.
For more information, see the chapter **Integrated Text Editor** 142.

4.1.14 M48 / M49 – Feedrate and Spindle Speed Override

M48 – Override Enabled
After calling **M48**, the operator can adjust the current override values for **feedrate (F)** and **spindle speed / cutting speed (S)** using either a potentiometer or the keyboard.

M49 – Override Disabled
With **M49**, changes to the override values for **feedrate (F)** and **spindle speed / cutting speed (S)** via potentiometer or keyboard are locked.

4.1.15 R-Parameters

What is an R-Parameter?
An **R-Parameter** is a variable (R1 to R999) that stores a numeric value. Example: `R10 = 99.567` This defines **R10** with the value `99.567`. Calling **R10** in the G-code retrieves that value.

Using R-Parameters

Arithmetic Operations
Basic operations such as addition, subtraction, multiplication, and division are supported inside an R-Parameter. Calculations follow the rule of *operator precedence* (multiplication/division before addition/subtraction). Parentheses are not supported.

Example*:*
```
R56 = 10/2 - 2*2
```
Result: 1

Using Other R-Parameters in Calculations
R-Parameters can reference each other.

Example*:*
```
R55 = 10
R56 = R55/2 - 2*2
```
Result: 1

R-Parameters in G-Code
R-Parameters can be used in multiple G-code contexts:

- **G00:** `G00 X=R10`
- **G01:** `G01 X=R10 Y=R11 + 6 / 2`
- **G02/G03 (arc commands):** `G02 I20 J20 X=R10 Y=R10`
- **G25 (loops):** `G25 Q=R4`
- **G53–G60 (work offsets):** `G54 X=R20`
- **IF statements:** `$IF R100==1`

Global Scope
R-Parameters are **global variables**. Once defined in a G-code file, they are available throughout the entire file.

Use in Subprograms
R-Parameters can also be used inside subprograms.

Status Display

The values of all R-Parameters are displayed in the **status bar of the text editor**.

Example – Loop with Parameters
R4 = 4 ; R4 defines number of iterations
R5 = 110 ; R5 defines X-position for the subprogram

G25 Q=R4 ; Loop with 4 iterations
P1 ; Call subprogram
G00 X=R5 ; Move to position defined in R5
G26 ; End loop

4.1.16 IF Statement

What is an IF statement?
The IF statement is used to evaluate a condition. If the condition is true, the corresponding G-code commands are executed. The following comparison operators are supported:

- **Equal**: (==)
- **Not equal**: (!=)
- **Greater or equal**: (>=)
- **Less or equal**: (<=)

An IF statement consists of the commands **$IF, $ENDIF, $ELSE, and $ELSEIF**. Each IF statement must appear on its own line and can be used both in the **main program** and in **subprograms**.

Syntax of the IF Statement
- `$IF` – defines the start of the condition
- `$ENDIF` – ends the condition
- `$ELSE` – executes alternative commands if the condition is not met
- `$ELSEIF` – checks an alternative condition

Examples
Example 1 – Simple IF statement
R57 = 10 ; *Define parameter R57 with the value 10*
$IF R57 == 10
; *This G-code is executed because the condition R57 == 10 is true*
$ENDIF

Example 2: IF-ELSEIF-Statement
R57 = 9
$IF R57 <= 9
; *This G-code is executed because the condition R57 <= 9 is true*
$ELSEIF R57 >= 10

```
 ; This G-code is NOT executed because R57 = 9
 ; If R57 were 10 or greater, this block would be executed
$ENDIF
```

Important Notes

- Nested IF statements are not allowed

 An IF statement must not be placed inside another IF statement. Doing so will result in an error.

Incorrect:	**Correct**:
$IF R200 == 0	$IF R200 == 0
; *G-Code*	; G-Code
$IF R1 == 1	$ENDIF
; *G-Code*	
$ENDIF	$IF R1 == 1
$ENDIF	; G-Code
	$ENDIF

4.2 DXF

The DXF format (Drawing Exchange Format) was developed by Autodesk for AutoCAD and is used for data exchange between CAD programs. cncGraF 8 currently supports the following DXF entities:

- 3DFACE
- ARC
- CIRCLE
- LINE
- LWPOLYLINE
- POINT
- POLYLINE
- SPLINE
- VERTEX
- LAYER
- ELLIPSE

 Since 3D DXF files are not suitable for milling, they are imported as **2D files**. The Z-depths are taken from the tool library.

Layers
The cncGraF 8 software can automatically assign the data of a **layer** to a tool. For this to work, the name of the layer must be specified in the tool library in the tool's **Name** field. In addition, after opening a DXF file, the **DXF Layer dialog** will appear, allowing you to manually assign layers to any tool.

4.3 HPGL

Hewlett-Packard Graphics Language (HPGL) is generated or exported by many CAD programs. cncGraF 8 supports the following HPGL commands:

Supported HPGL Functions
- **PU** – Pen Up (lift tool)
- **PD** – Pen Down (lower tool)
- **PA** – Absolute Positioning
- **PR** – Relative Positioning
- **SPx** – Select Pen / Tool number
- **AA** – Absolute Arc
- **AR** – Relative Arc
- **CI** – Circle

Example	
HPGL	**Description**
PU; **PA**100, 50; **PR**100, 50; **PD**; **CI**100; **AA**300, 200, 50; **SP**1;	Lift tool Move to absolute position X=100, Y=50 Move to relative position (new absolute position: X=200, Y=100) Lower tool Draw a circle with radius 100, counterclockwise (negative radius → clockwise) Draw a 50° arc with center at X=300, Y=200, starting from position X=100, Y=50 Select tool number 1

4.4 GRF5

The **GRF5 Interpreter** is a cncGraF 8 file format that stores the following information:

- Vectors and drilling points (all drawing elements)
- Tool library for vectors and drilling points
- Tool compensation (radius compensation)
- Units and scaling
- Material offset and workpiece zero point
- Tool processing sequences with depth-of-cut compensation
- File properties

When opening GRF5 files, cncGraF 8 checks whether the workpiece zero point saved in the GRF5 format is active. If the zero point is not found, a message is displayed prompting the user to verify the workpiece offset.

4.5 Postscript and Adobe Illustrator

cncGraF 8 can load **Postscript 3** files (*.eps) and ***Adobe Illustrator*** *files (*.ai).
Postscript is a page description language developed by Adobe. The Adobe Illustrator format is based on Postscript but provides a more limited command set.


Use Postscript or Adobe Illustrator files to import data (e.g., from CorelDRAW). To ensure that text objects from CorelDRAW are imported correctly, the option **"Export text as curves"** must be enabled.

4.6 Drill Data

cncGraF 8 can read drill data in **Excellon**, **Sieb & Maier 1000**, and **Sieb & Maier 3000** formats. Below are two examples in the **Sieb & Maier** format:

Sieb & Maier		Description
Example 1	**Example 2**	
% T01 X001Y0001 M30	% T01 X100Y10 M30	T01 selects Tool 1. Position X=100, Y=10 is reached and drilled into the material. The program ends with M30. Same operation, written in the second format.

Two Types of Drill Files

1. **First Format (with leading zeros)**
   Coordinates use a 5-digit notation, but trailing zeros are often omitted.
   - Example: `X001` actually means `X00100`, and `Y0001` means `Y00010`.

2. **Second Format (without leading zeros)**
   Coordinates are written without leading zeros after the axis letter, which results in values of different lengths.

Although the formats look different, both describe the **same machine setting**. In the **Options** dialog, you can switch between the two formats.

5 Settings

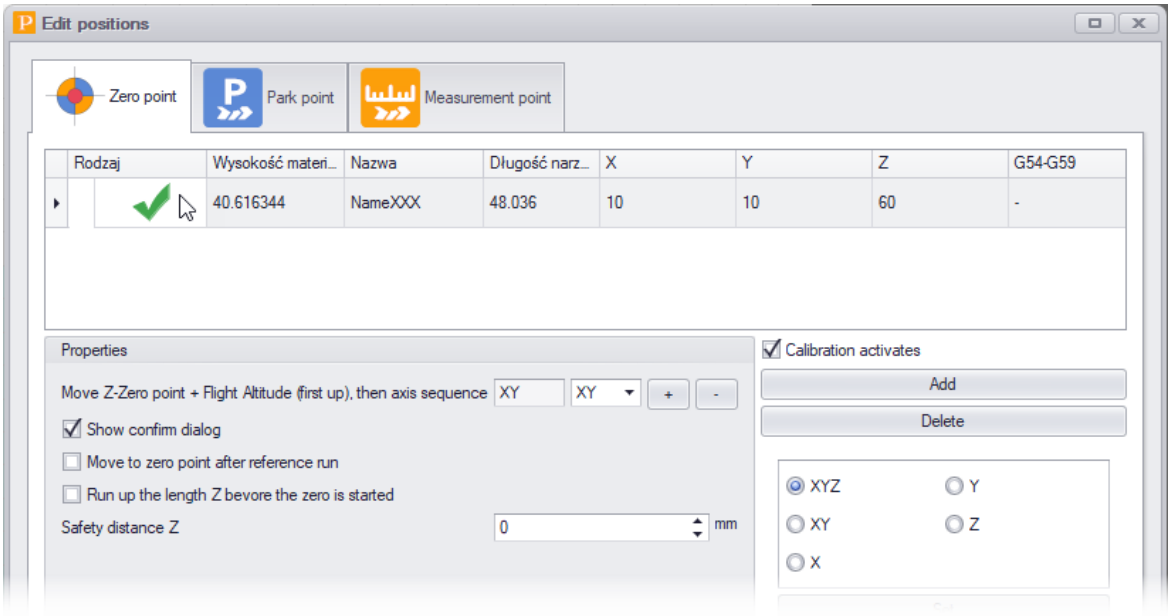
This section describes all relevant settings required for operating the machine.The **Settings** main menu provides the following functions:

- **Positions** 70: Defines zero points, parking positions, and measuring points used during machining.

- **Workpiece Parameters and Workpiece Offset** 74: Specifies the material and displays it in the 2D view.
- **Tool Library**: Used to define tools, including feed rates, Z-depths, stepdowns, and other parameters for 2D files.
- **Safety Zones**: Provides software-based monitoring of the CNC machine's position.
- **Options**: Includes various interface customization settings. File-related options can also be configured here.
- **Input Devices**: Settings for handwheel, Xbox controller, keyboard, control panel, etc.
- **Macro Editor and Macro Linking**: Extends functionality by allowing macros to be created and linked. Linked macros can be triggered automatically at different stages of the process.
- **Plugin Manager**: Enables cncGraF 8 to be extended with external software.
- **Units and Scaling**: Defines the measurement units and scaling factors.
- **Zero Point Tools**: Provides multiple methods for defining the workpiece zero point.
- **Link Settings**: Allows different configurations to be linked. A shortcut is created on the Windows desktop that starts cncGraF 8 with the selected configuration.
- **Video Positioning**: Aligns the drawing with fiducial marks, ensuring it matches the actual material position on the machine.

5.1 Positions

Before a parking position, zero point, or measuring position can be approached, it must first be created in the **Edit Positions** dialog. This dialog can be accessed from the main menu via **Settings → Positions**.



Any number of zero points, parking positions, or measuring positions can be created, but only one of each type can be active at a time. Therefore, at most one zero point, one parking position, and one measuring position will be displayed graphically.

Add Position

Clicking the **Add** button opens the **Add Position** dialog. Here, you can enter the parameters of the point (X, Y, Z, A, B coordinates, status, and name). The Z-height can also be defined by entering the material thickness. To use this option, the tool length sensor must be calibrated once. (For more details on the two types of zero points, see *Settings → Positions → Zero Points*.)

Editing Properties of Zero, Parking, and Measuring Points

Under **Properties**, you can specify how points are approached—either directly or in a chosen axis order. The option **Show confirmation dialog** enables or disables a dialog box before movement begins. You can also choose whether the machine should automatically move to the zero point after homing. The value entered in the **Z Safety Distance** field defines how far the axis retracts upward after zero point probing.

Measuring Point: Multiple Measurements

This option allows measurements to be repeated, such as recording tool lengths or probing edges with a 3D sensor. By averaging multiple measurements, higher accuracy can be achieved. Additionally, error detection is included: if a measured value deviates too much, the software automatically issues an error message and aborts the process.

Machine Position
The current position of the CNC machine can be saved by clicking **Machine Position → Set** (see figure below).

Enable or Disable Calibration
Here 72 you can switch between two types of zero points: a simple XYZ zero point or a zero point that also includes tool length and workpiece height. For more details on these two zero point types, please refer to the corresponding section.

5.1.1 Zero Points

There are two types of zero points:

1. Simple Zero Points (without tool length and workpiece height)
These zero points can only be used when calibration is disabled or not available. They do not include Z-correction and are suitable for simple jobs where only one tool is used.

2. Zero Points with Tool Length and Workpiece Height
These zero points are measured when created and include both tool length and workpiece height data. For this, the tool length sensor must be calibrated once. If the tool length sensor is not calibrated, the **Calibration** switch is inactive, and zero points with tool length are not available.

Zero points with tool length can be used for files with multiple tools.
- For **manual tool changes**, a parking position must be defined.
- For **automatic tool changers**, a parking position is not required as long as the tool to be changed is within the tool changer.

Advantages of zero points with measured tool length:
- If a tool is changed but the workpiece height remains the same, the Z-height of the zero point does not need to be manually reset. The length of the new tool is compared with the old tool, and the Z-height of the zero point is automatically adjusted.
- If the workpiece height changes but the tool length remains the same, the Z-height does not need to be manually redefined. The new workpiece height can be entered directly in the table. cncGraF 8 then calculates the new Z-height and updates the zero point accordingly.

Disadvantages of zero points with measured tool length:
- At the beginning, the zero point must always be set manually, since tool length and workpiece height are initially unknown. For automatic Z-height determination of the zero point, a 3D probe is recommended.

Z Safety Distance
Once the new Z-height for the current zero point is defined, a relative move upward from the zero point can be executed. This move is called the **Z safety distance**.

5.1.2 Measuring Points

Multiple measuring points can be managed in the **Positions** dialog, but—similar to zero points and parking positions—only one measuring point can be active at a time.

 **Important:** A measuring point consists of the three coordinates (X, Y, Z) plus the **maximum tool length**. The machine first moves to the defined X, Y, Z position and the maximum tool length before the actual measuring process begins.

 **Critical Note:** The measuring point Z + maximum tool length must be defined high enough to ensure that the longest tool fits between the starting Z-height (measurement start) and the tool length sensor.

Z-Safety Height After Tool Measurement

After the tool length has been measured, the machine automatically moves to a Z safety height:

- **With automatic tool changer:**

The values *Z position of the first tool*, *Zero Point Z*, and *Measuring Point Z* are compared. The highest Z-value plus the flight height is approached.

- **Without automatic tool changer:**

The values *Zero Point Z* and *Measuring Point Z* are compared. The highest Z-value plus the flight height is approached.

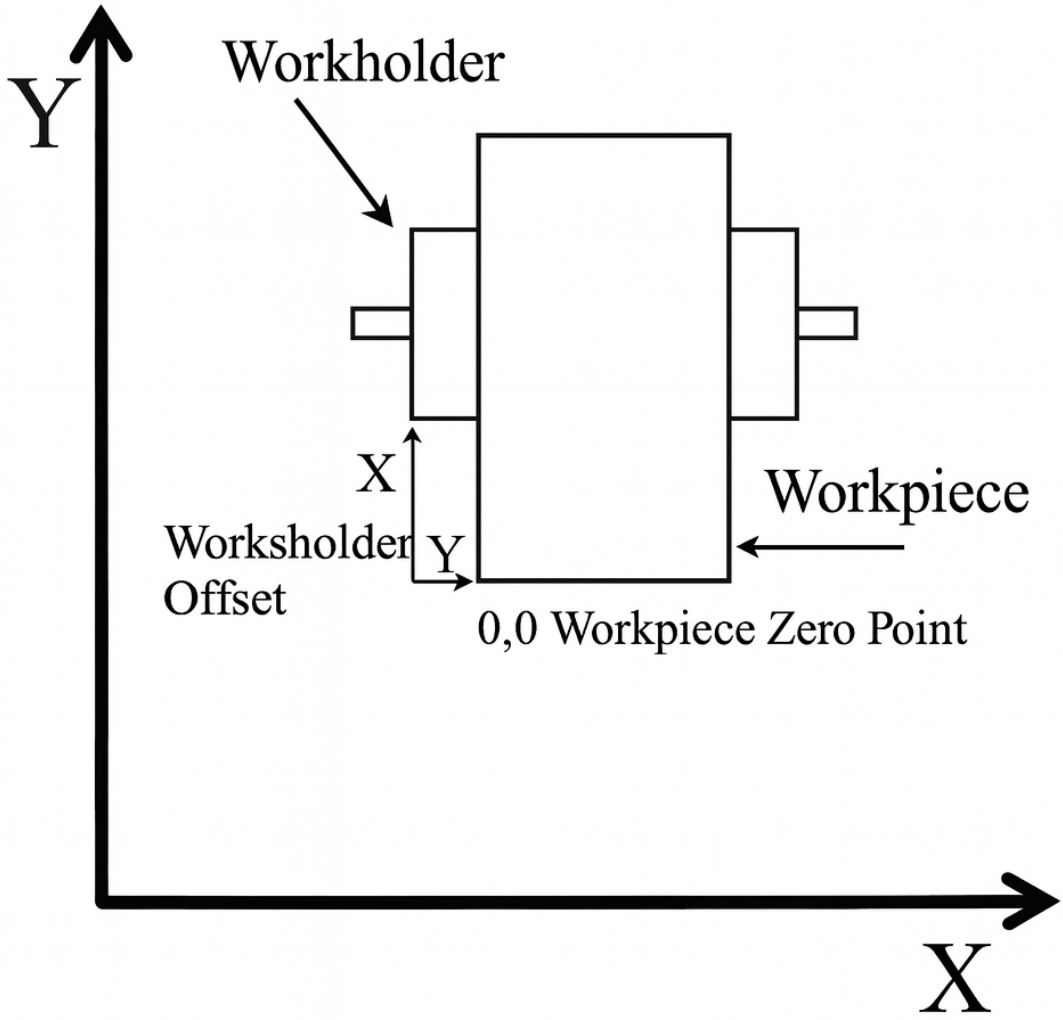
 **Collision Warning:** If the zero point is set on the underside of the material, the tool may collide with the material when moving to the measuring point due to material thickness or a low Z measuring position. To avoid collisions, enable the option "**Move to Z-axis maximum before moving to measuring point"** under *Settings* → *Positions* → *Measuring Point*.

5.2 Workpiece Parameters and Workpiece Offset

Workpiece Parameters
For visual verification, the size and position of the workpiece can be displayed graphically. The workpiece, accessible via *Settings → Workpiece Parameters*, can be aligned either with its lower-left corner or centered at the zero point. The defined workpiece dimensions in the X and Y directions are displayed in the selected color.

Workpiece Offset
The workpiece offset is the distance between the marked workpiece holder and the workpiece zero point (see figure). In the GRF format, both the workpiece offset and the workpiece zero point are stored. The workpiece zero point stored in the GRF file is only used for validation when opening the GRF file. If the zero point of the loaded drawing does not match the active workpiece zero point, the following message appears:

cncGraF 8 does not contain the drawing's workpiece zero point! Please check the workpiece offset.



If the workpiece zero point has changed, it is not necessary to update all GRF files. It is sufficient to adjust the value of the workpiece offset in the currently opened GRF file.

5.3 Change Tool Number

The tool numbers of a loaded file can be adjusted in two ways:

Directly in the drawing:

- Using a selection box and the tool palette → Tools → Tool Properties (only available for 2D files).

Via the dialog:

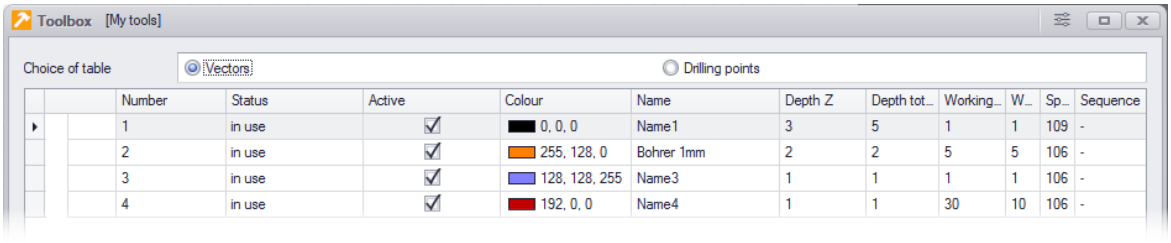
- From the main menu, go to *Settings* → *Change Tool Number* to adjust tool numbers.

5.4 Tool Library

The *Tool Library* dialog (menu: *Settings* → *Tool Library*) manages two tool lists:

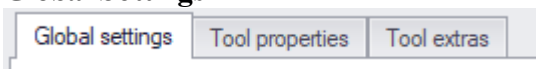
- **Vectors:** For DXF, HPGL, and optionally G-Code (DIN66025, see *Settings* → *Options* → *File* → *G-Code*).
- **Drill Points:** Only for Sieb & Maier file types.

At the top of the Tool Library window, you can enter the tool library name (e.g., *[My Tool Library]*). Clicking the icon with three horizontal lines on the right side of the window opens the tool library management menu.



Number	Status	Active	Colour	Name	Depth Z	Depth tot...	Working...	W...	Sp...	Sequence
1	in use	☑	0, 0, 0	Name1	3	5	1	1	109	-
2	in use	☑	255, 128, 0	Bohrer 1mm	2	2	5	5	106	-
3	in use	☑	128, 128, 255	Name3	1	1	1	1	106	-
4	in use	☑	192, 0, 0	Name4	1	1	30	10	106	-

Global Settings



Use Spindle Speed

The spindle speed can be controlled in 255 steps. First, enable the *Use Spindle Speed* option. By clicking the *Set Spindle Speed* icon, you can define the ranges for these steps. If the DAC-INT-10V v.02 interface is used, make sure the corresponding field is activated.

Use Tool Lengths from the Tool Library

This function reduces the number of required measuring operations.

Collet Offset

If a value is specified here, it will be subtracted from the tool length in the *Positions* dialog. In *Positions* → *Zero Points*, the actual workpiece height is then displayed. This parameter has no functional effect; it is only used for visual representation.

Use Body Length from the Tool Library

When enabled, each tool is assigned an individual Z-height via the probe. If a tool does not have a body length specified under the *Tool Properties* tab (body length = 0), the longest body length from *Positions* → *Measuring Point* will be used instead.

Tool Properties

Tool Type, Diameter, and Body Length

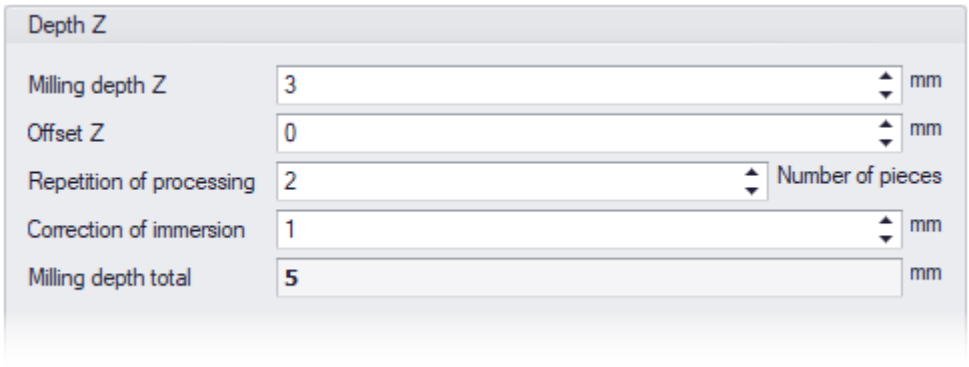
In the *Tool* section, you can enter the tool type, diameter, and body length. The diameter is used for calculating tool compensation in the *Tool Compensation* dialog. The body length is activated in the *Global Settings* section. Information such as the tool type and other parameters are required for generating 3D models.

Feed Rates, Cutting Depth (Z), and Spindle Speed
Operating parameters such as feed rate, cutting depth, and spindle speed can be assigned here. In the *Active* column of the table, tools can be enabled or disabled. Polylines assigned to inactive tools appear gray in the drawing and are not used for machining.

Cutting Depth (Z)
Since 2D files such as DXF or HPGL do not include Z-depth information, the cutting depth (Z) must be manually entered in the tool properties for each tool. The total cutting depth is composed of:

- Cutting depth (Z)
- Z-offset
- Repetitions
- Incremental depth correction

Example
A tool first plunges 3 mm into the material. The operation is then repeated twice (*Repetitions*), each time with an additional 1 mm depth. This results in a total cutting depth of 5 mm. The Z-offset shifts the Z-axis by the specified value (similar to the X/Y offset).



Switching Outputs
Each tool can be assigned outputs to switch external devices on or off.

Speeds
Since 2D files (DXF, HPGL) do not contain speed parameters, speeds must be entered manually for each tool.

For G-code files (DIN66025), the speeds defined in the tool library can also be used. To enable this, activate the option **"Use Speeds from Vector Tool Library"** in the dialog *Settings → Options → File → G-Code*.

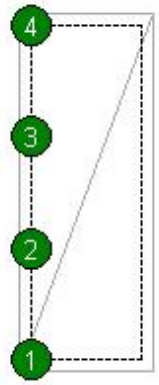
Tool Extras

Z2
Allows a tool to be redirected to the second Z-axis. For this, the X/Y offset of the Z2 axis must be specified. This function is only used for 2D files such as HPGL or DXF. For DIN66025, the data for the fourth axis (Z2) must already be included in the file.

Tangential
Allows a tool to be assigned to the A- or B-axis as a tangential axis.

5.5 Safety Zones

A safety zone (Menu: *Settings* → *Safety Zones*) protects against unauthorized actions. Any number of safety zones can be defined. A good example of applying a safety zone is the automatic tool changer. Within the tool changer's area, only tool changes hould be carried out. All other actions, such as *Manual Move* or *Jogging/Move To*, are dangerous for the automatic tool changer.



The figure on the left shows a tool changer (4 tools) protected by the *Safety Zones* function.

A safety zone is displayed as a gray rectangle with a diagonal line.

The following actions can be permitted within safety zones:

- Manual Move
- Milling/Drilling
- Tool Change
- Tool Measurement
- Move to Position

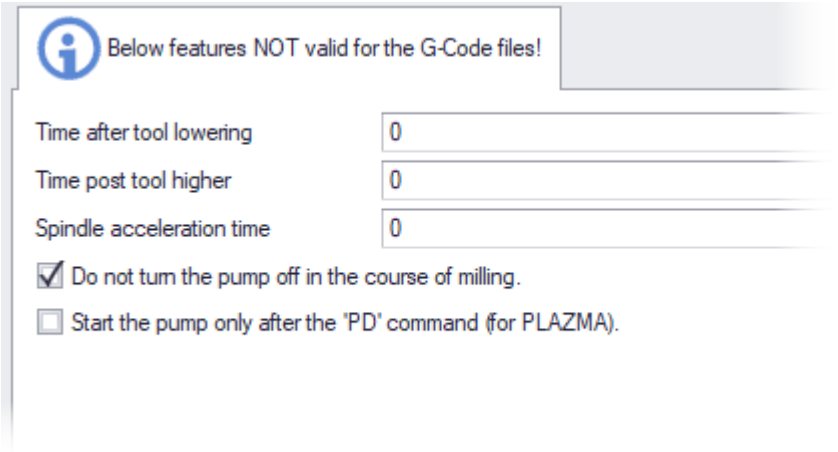
Important: Safety zones are a purely software-based solution and cannot guarantee sufficient protection if cncGraF 8 does not know the exact position of the CNC machine.

5.6 Options

The Options dialog (Menu: Settings → Options) contains all program settings, organized into different sections.

File → Properties
In the File section (Menu item File), you can configure settings related to loading files. For example, under File Properties you can specify whether a file should automatically be aligned to the zero point after opening.

The lower section contains options that apply exclusively to 2D files such as DXF or HPGL (not to DIN 66025 files).



In certain cases, it is useful for the tool not to begin machining immediately after plunging into the material, but instead to pause for an adjustable time.
For this, a *dwell time after tool down* can be defined in milliseconds. If the tool should also remain briefly in the material before lifting, an additional *dwell time before tool up* (in milliseconds) can be set.

Furthermore, a *spindle startup delay* (in milliseconds) can be configured to ensure that the spindle reaches its full speed before the machining process begins.

Activate pump only after the "PD" command (Plasma/Laser)
The pump output (Machine Parameters -> Pin Assignment 74) can be used for plasma or laser units. Activation only occurs after the HPGL command *PD* (lower Z-axis). This feature is especially important for plasma cutting, as the arc remains stable only at the correct distance from the material.

Additional file-related options for G-Code (DIN 66025) are explained in the chapters G-Code Settings 82 as well as DXF, HPGL, PostScript, Sieb & Mayer, SVG 85.

Options – Overview

The *Options* dialog offers several tabs that allow you to customize the software interface and operation to your needs. The following areas are available:

General & Tool Palette
- Set software appearance (theme, colors)
- Define working directory
- Invert zoom direction in 2D view
- Show or hide functions in the tool palette

Colors & Grid
- Set colors for 2D view elements (e.g., color of rapid-move vectors)
- Configure the grid
- Define line thickness of tool compensation lines (in pixels)

Fonts, Icons & Touchscreen

- Select font and size for menus
- Adjust icon size (important for high-resolution displays)
- Customize the color of absolute and relative coordinates

3D

- Define settings for the 3D view
- Enable/disable display of the machine area and the workpiece
- Adjust color transparency with sliders

Messages

In this section, messages can be defined that appear during certain actions.
Conditions are checked and—if met—a message is displayed for the operator.

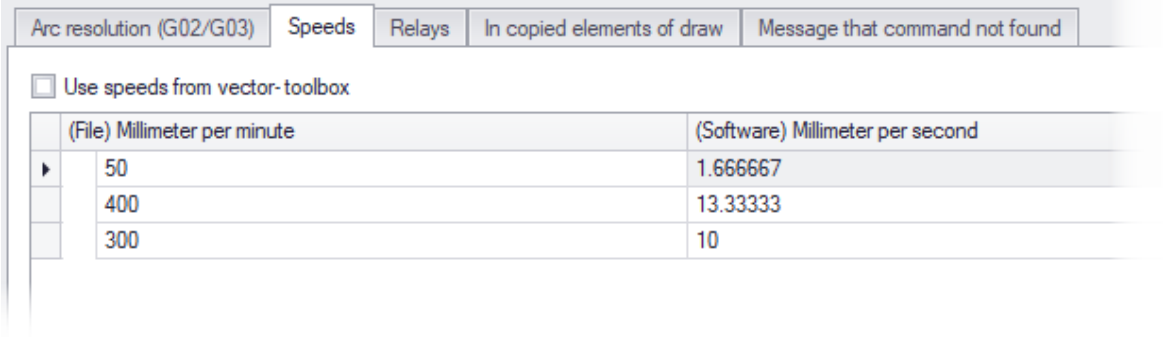
Typical use cases include safety messages such as:

- *“Check drawing position before milling”*
- *“Warning: Open collet?”*
- *“Warning: Spindle on?”*
- *“Tool length successfully measured.”*

5.6.1 G-Code Settings

For G-Code files (DIN 66025), feed rates specified with the command **"F"** are always given in **millimeters per minute**. Since cncGraF 8 internally works with speeds in **millimeters per second**, an automatic conversion is performed.
Feed rates can also be scaled by a factor or taken directly from the tool library.

- **Factor**: Multiplies the feed rates defined in the file. A value of **1** leaves the feed rates unchanged.
- To use the feed rates from the G-Code file, the option **"Use feed rates from vectors tool library"** must be **disabled**.



Display of Feed Rates: The feed rates from the G-Code file (if available) are only displayed in the table **after the file has been loaded**.

DIN 66025 – Configuration Options

- **Command G02/G03 (Circular interpolation, clockwise/counterclockwise):**
  Can be interpreted either **relative** or **absolute**.

Note: An incorrect setting may result in **misinterpretation of the arc**.

- **Z-Vector Inversion**: Reverses the direction of the Z-axis.
- **Drill Spindle Startup Time (ms)**: When starting, the machine switches on the spindle and waits for the specified time until the rated speed is reached.

This option is only required if the G-code file does not contain a **G04 H** command.

M20/M21 (Plasma Torch)

For command **M21**, a startup delay (ms) can be defined.

Drill Points
Display drill points as crosses.

Decimal Places
Defines how many decimal places are used when loading the file.

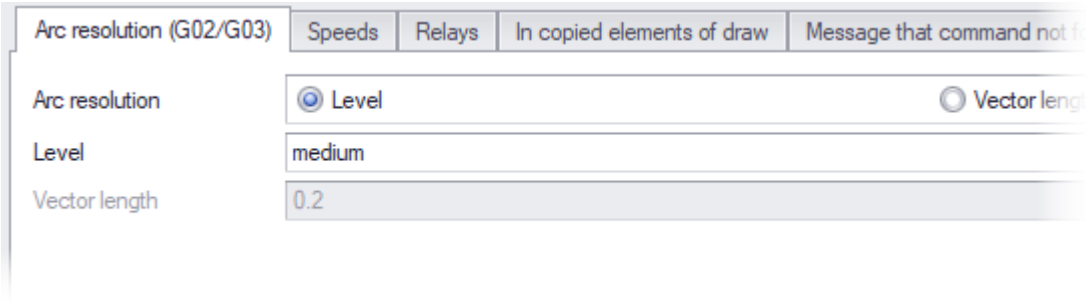
Legacy G84 Version
Enable this option to ensure compatibility with older cncGraF 8 versions.

Axis Definition
Assign custom letters for the axes.

Lift Distance for G73
Defines the short retraction distance after each peck during G73 drilling.

Arc Resolution (G02/G03)
The G-code commands **G02/G03** (circular interpolation) are converted into vectors when loaded.
This is necessary because controllers work vector-based (Step/Dir).

Since CNC machines have a defined axis resolution (set under *Machine Parameters* → *Axes* → *Steps per Revolution and Travel per Revolution*), the chosen drawing resolution should not be lower than the machine resolution.
You can visually check the arc resolution in the **2D view** by enabling: *View* → *Start and End Points of Vectors*.



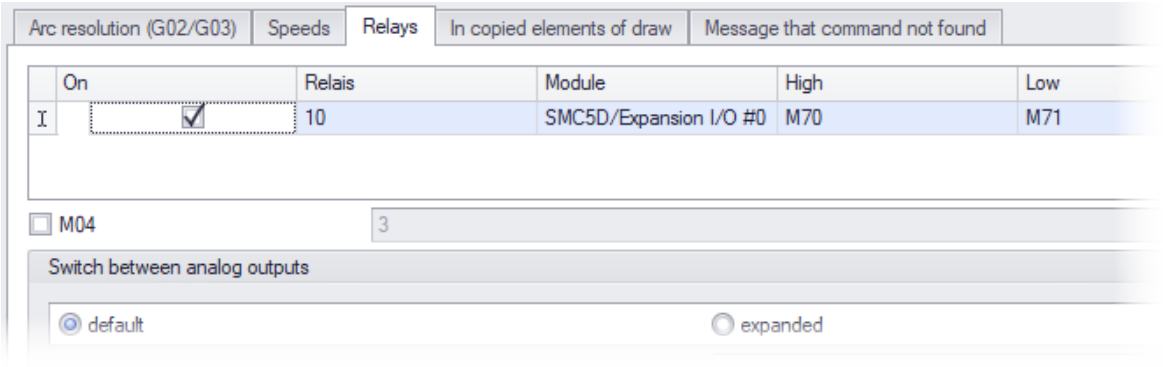
The arc resolution: **medium or high is recommended**, as it provides a good balance between accuracy and data size.

M-Commands
In the *Outputs* tab (M-commands), commands can be defined to switch outputs on or off.
With **M04**, the spindle rotation direction is set to counterclockwise. The corresponding output for counterclockwise rotation must be specified.

Since cncGraF 8 can control up to **3 analog channels (AD0 to AD3)**, these can also be switched via M-commands.

Example for switching an output:
For output 10, the commands **M70 (on)** and **M71 (off)** are defined.

If **M70** appears in the G-code, output 10 is switched on.
With **M71**, the same output is switched off again.



In copied drawing elements
This section defines how a G-code file should be copied.

Note: Keep in mind that a G-code file already represents a complete 3D description file for the CNC machine and normally should not be modified.
Furthermore, G-code files are generally difficult to edit manually. cncGraF 8, however, offers the option to copy G-code files upon customer request.

To ensure this process runs as smoothly as possible, several options are available:

1. **Remove specific commands**, e.g.:
- G04 (dwell time)
- M05 (spindle off)
2. **Automatically insert a G00 Z… command** (initial height, value defined in the input field).

Message if command not found
With this option, a user message can be defined that automatically appears when opening a G-code file if a specific G-code command (character) is not found.
The command to check is entered in the input field “Command.”
The desired message is entered in the input field below.

5.6.2 DXF, HPGL, PostScript, Sieb & Mayer, SVG

All of these file formats use the same import dialog. In this dialog, the following settings can be configured:

- **Arc resolution:** Defines how finely circles and arcs are divided into line segments. *Medium* or *High* is recommended, as this provides a good balance between accuracy and data volume.
- **Unit:** Defines the unit of measurement for the coordinates.

DXF files are usually created in millimeters. Therefore, unit **1 (mm)** is typically used.

5.7 Input Devices

The following components can be used as input devices (Menu: **"Settings → Input Devices"**):

- Keyboard (Keypad) 85
- External Control Panel 86
- Microsoft Xbox Gamepad 87
- Handwheel 89
- Industrial-Joystick 93 **(no longer supported)**

The settings for the keyboard, external control panel, and Xbox gamepad are configured in the menu **"Settings → Input Devices"**. The available functions are grouped into the categories: **Manual Move**, **Move**, **Switch Relays On/Off**, **Other Functions**, and Macro 95.

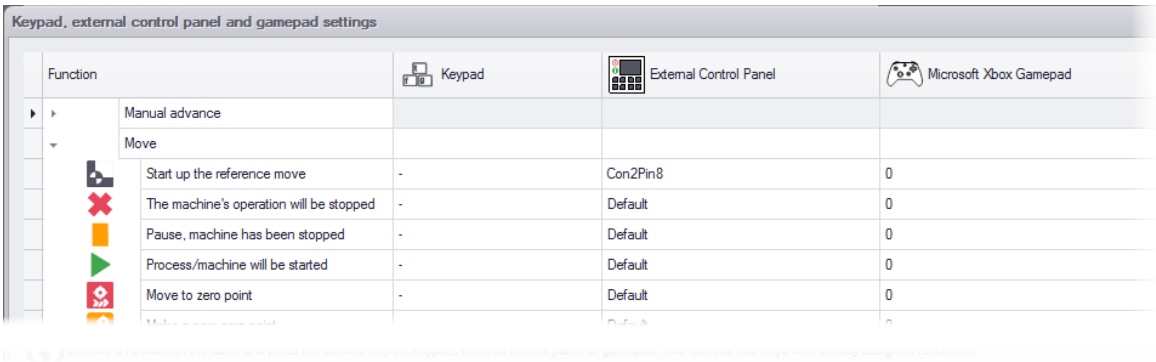


Figure: Setup menu for keypad, external control panel, and Xbox gamepad

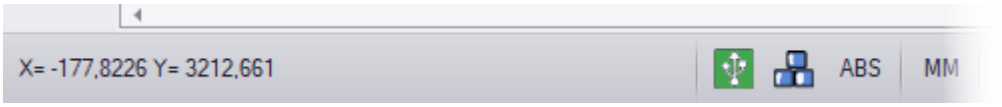
5.7.1 Keyboard

In the dialog "Configure Keyboard, External Control Panel, and Xbox Gamepad" (Menu: "Settings → Input Devices"), the keyboard can be enabled, and specific functions can be assigned to the keys of the PC keyboard or an external keypad.

Important: Since the assigned functions are executed immediately, it is strongly recommended to configure the key mapping carefully..

The keyboard can be enabled or disabled using a switch. To change the assignment, click on the desired cell in the list and then assign a key.

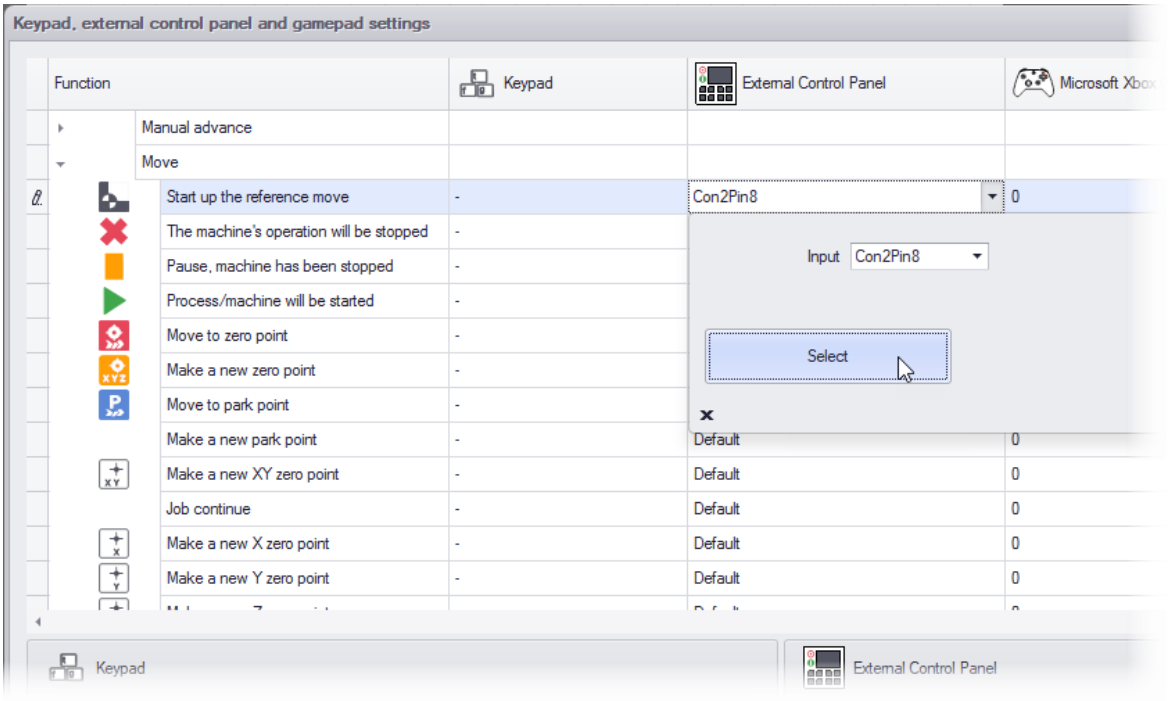
The keyboard is not permanently active. In certain dialogs, such as the text editor or manual move mode, the keys are automatically disabled to prevent unintended functions from being triggered. The keys are only active when the corresponding symbol in the status bar is displayed in **blue**.



5.7.2 External Control Panel

An external control panel can also be used to operate the machine. This is a simplified, wired remote control that is typically connected via the **Con2 interface**. Each switch or button on the panel is assigned to a pin (input) of the interface.

The control panel can be configured in the menu: **“Settings → Input Devices → Keyboard, External Control Panel, and Xbox Gamepad.”** In the **“External Control Panel”** column of the table, each pin (= button/switch) can be assigned a specific function.



Note: When using an external control panel, the signals are transmitted to the controller via switching relays. This can lead to unwanted input activation caused by interference pulses. To prevent this, the number of queries until the switching process is confirmed can be adjusted.

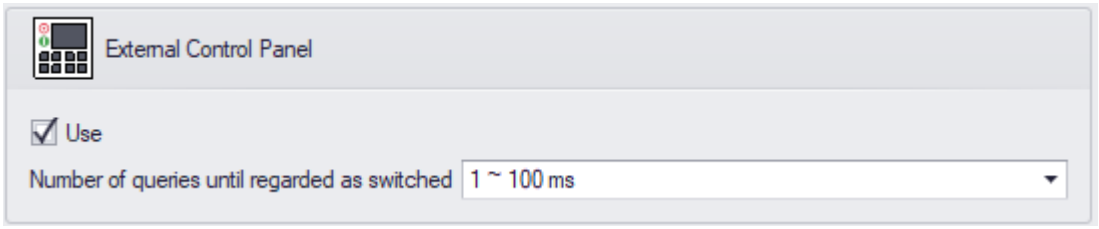


Figure: The query must last at least 100 ms before a relay is recognized as switched.

5.7.3 Xbox Gamepad

The software cncGraF 8 also supports using a Microsoft Xbox Gamepad to control the machine. This feature can be enabled or disabled via the menu:
Settings → Input Devices → Keyboard, External Control Panel, and Xbox Gamepad.

Within the dialog, the buttons and joysticks of the gamepad can be assigned to functions such as:

- Switching manual speeds
- Homing cycle (Reference move)
- Moving to the zero point

- And other operations

If the value **"-1"** is entered for a button, the assigned function is disabled.

5.7.4 Handwheel

With cncGraF 8, the machine can also be controlled via a **handwheel** (menu: **Settings → Input Devices → Handwheel**). This option provides a convenient way of manual operation and can be enabled or disabled in the menu. Control buttons can be assigned to specific functions.

In the fields **"0.1"** or **"0.01"**, the size of the incremental step is defined in millimeters.
- Turning the knob by one detent moves the machine by the selected step.
- Turning by two detents moves it by two steps, and so on.

This allows precise positioning and comfortable visual control when approaching the target position. The minimum selectable step size is limited by the machine's resolution.

Rapid Move

By turning the handwheel while simultaneously pressing one of the **"–"** or **"+"** keys, the machine moves left or right. The travel speed is controlled by the handwheel knob. Releasing the "–" or "+" button stops the machine automatically.

Special Functions (requires firmware 0.26 or higher)

The handwheel supports additional functions that can be triggered with key combinations:
- **F2 + F3 (Direction change in rapid move mode):** Change travel direction directly with the handwheel, without using the "+/–" keys.
- **F10 + F11 (Change travel direction for X/Y):** Select an axis (X or Y) and switch the travel direction.
- **F6 + F7 (Buzzer on/off):** Enable or disable the acoustic signal.
- **F3 + F4 (Encoder/Potentiometer selection):** Switch between encoder and potentiometer. This setting depends on the hardware of the handwheel and should only be modified if required.

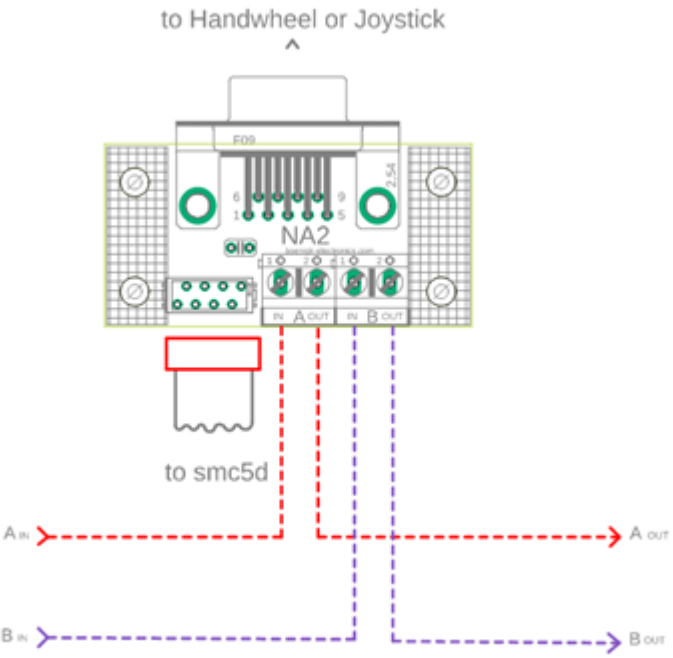
Sonderfunktionen des Handrads (benötigt Firmware 0.26 oder höher)
Das Handrad unterstützt zusätzliche Funktionen, die über Tastenkombinationen aufgerufen werden können:
- F2 + F3 (Richtungswechsel im Rapid-Move-Modus): Fahrtrichtung direkt mit dem Drehrad ändern, ohne die Tasten „+/–" zu verwenden.
- F10 + F11 (Fahrtrichtung ändern für X/Y): Auswahl einer Achse (X oder Y) und Wechsel der Fahrtrichtung.
- F6 + F7 (Buzzer Ein/Aus): Akustisches Signal ein- oder ausschalten.
- F3 + F4 (Encoder/Potentiometer-Auswahl): Umschalten zwischen Encoder und Potentiometer. Diese Einstellung ist abhängig von der Handrad-Hardware und sollte nur bei Bedarf verändert werden.


The respective keys must be held for about 2 seconds to activate or change the function.

Settings Overview

1. **Emergency Stop (NOT-AUS):** Immediately halts the machine and cuts off the power supply to the electronics.

Warning: For the electronics to be fully disconnected from the power supply, the terminals must be connected at **NA2** (see wiring diagram).

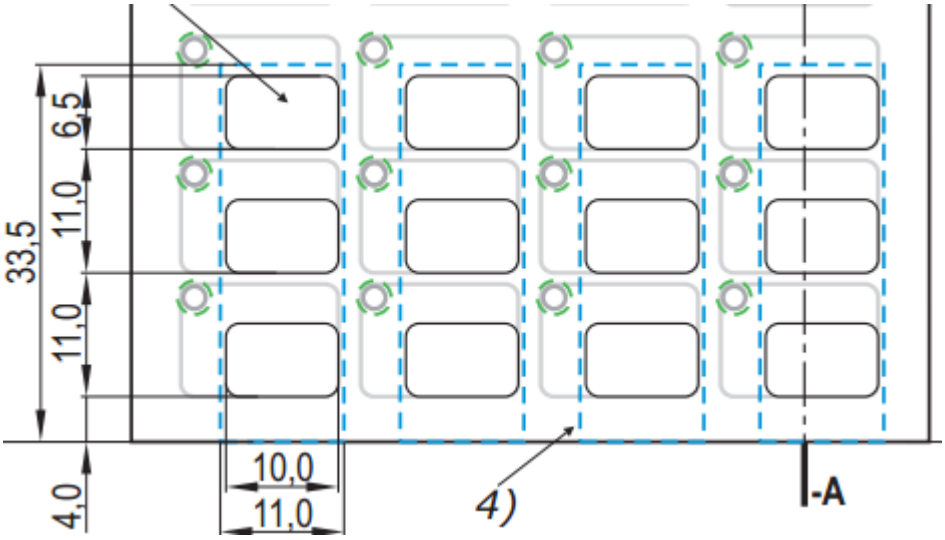


Safety Notice: If the connection is not made according to the wiring diagram, pressing the emergency stop button will **not** cut the power supply — it will only interrupt the program.

2. **Axis Selection Keys:** The last selected axis is stored.
3. **Travel of All 5 Axes:** Controlled via the rotary wheel (optical encoder).
4. Single-Step Motion Keys.
5. **Rapid Move Keys:** To enable rapid moves with the wheel, the "–" or "+" key must be pressed simultaneously.
6. **Feedrate Adjustment in Percent:** From 1% to 100%.
7. Function Confirmation Key.
8. Function Keys:

- Perform homing cycle
- Move to zero point
- Move to park position
- Spindle on/off
- Set X/Y/Z zero point
- Pump on/off
- Start job
- Resume job
- Pause
- Abort

9. **Spindle Speed Adjustment with Handwheel:** The spindle speed can be changed during machining. To adjust it, the **B axis** (see point 2) must be selected, and the wheel (see point 3) turned.
10. **Freely Programmable Function Keys (F1–F12):** These keys can be custom-labeled.



11. **LED Status:** When the emergency stop is pressed, all LEDs light up (from handwheel firmware V1.01).
12. **Buzzer Control:** Pressing **F6 + F7 simultaneously** toggles the handwheel buzzer on or off.

Dialog Window – Control

Many dialog windows can also be operated directly using the handwheel keys **OK, Pause,** and **ESC**.

5.7.5 Industrial Joystick (discontinued)

The electronic joystick is connected directly to the smc5d controller via cable. Since the machine movement is controlled directly by the controller, there are no delays in CNC positioning. The joystick includes three freely programmable buttons, which can be assigned in the input fields **“Buttons F1–F3”**. It can be activated or deactivated in the menu **Settings → Input Devices → Industrial Joystick**.

Rapid Movement

Quick motion is achieved by pressing the **X/Y** or **Z** axis selection button along with the **fire button**. The movement speed is controlled by the joystick’s orientation.

Settings Overview

1. Emergency Stop (NOT-AUS): Immediately halts the machine and cuts off the power supply to the electronics.

Warning: For the electronics to be fully disconnected from the power supply, the terminals must be connected at **NA2** (see Figure 1).

Safety Notice: If the connection is not made according to the wiring diagram, pressing the emergency stop button will **not** cut the power supply — it will only interrupt the program.

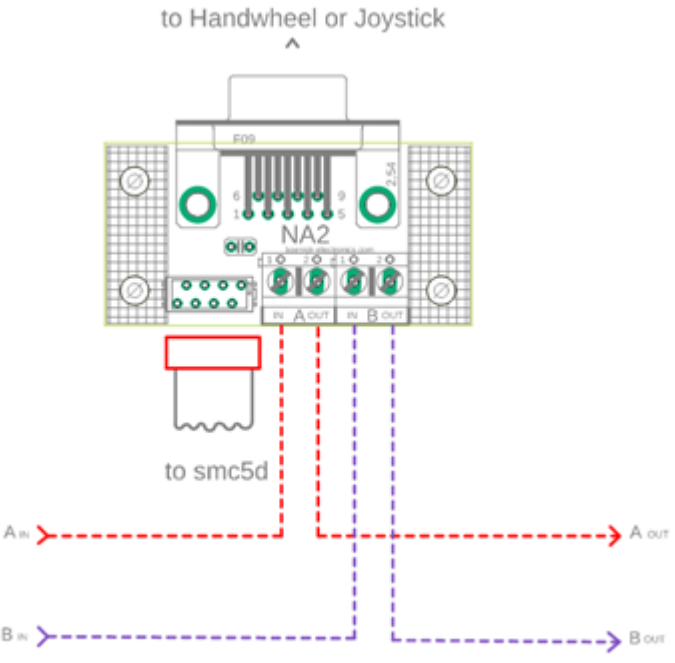


Figure 1: NA2 wiring diagram

2. By pressing the **X/Y** or **Z** button together with the fire button, the XYZ axes can be moved via the analog joystick. The movement speed is controlled by the joystick's orientation. Maximum permissible speeds can be set separately for the X/Y and Z axes.

3. Three freely programmable buttons can be assigned. By pressing **F1–F3** together with the **OK button**, the defined function is executed.

4. Joystick calibration: To calibrate, hold the **X–Y** and **Z** buttons for about 10 seconds until a confirmation tone sounds.

5. Diagonal movement sensitivity can be adjusted with sliders. If desired, both diagonal travel and the fire button function can be disabled.

5.8 Commands

The **"Commands"** function provides a simple alternative to the Macro Editor. It allows uncomplicated work sequences to be defined, which can be executed at any point during a machining process.

The linking of work sequences is done in the dialog **"Link Macro/Command…"** (main menu: **Settings → Macro/Link Macro…**).

5.9 Macro Editor

The creation and use of macros in cncGraF 8 is an important tool for extending the software with custom functions. For example, a macro can be developed to control a special tool changer, or to operate machines with automatic tool clamps that must be opened or closed. This makes it possible to implement numerous special functions that are not part of the original program but can be added via macros.

Create or open a macro file
To create or open a macro file, start the **Macro Editor** from the main menu via **Settings → Macro Editor...**. The following window will then appear:

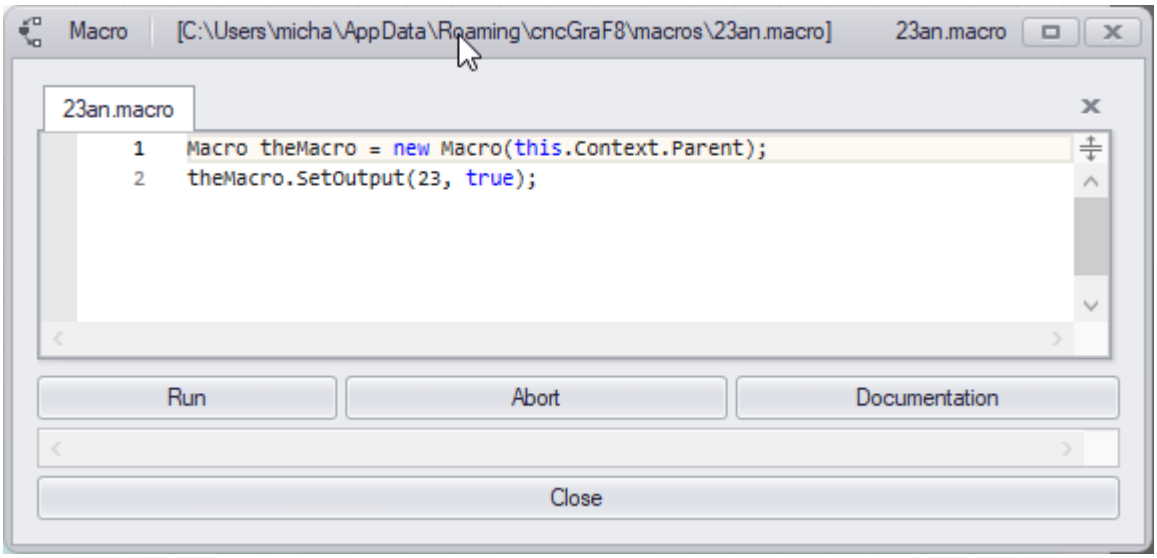


Figure 1: Macro Editor with an opened macro file

Create your own macro
A custom macro (=program) can be written in the **File** tab . . . . Basic knowledge of C# programming is required. To assist with coding, the text editor provides **autocompletion**, **automatic error detection**, and **syntax highlighting**. Autocompletion displays a dropdown function list during typing, making macro creation much easier.

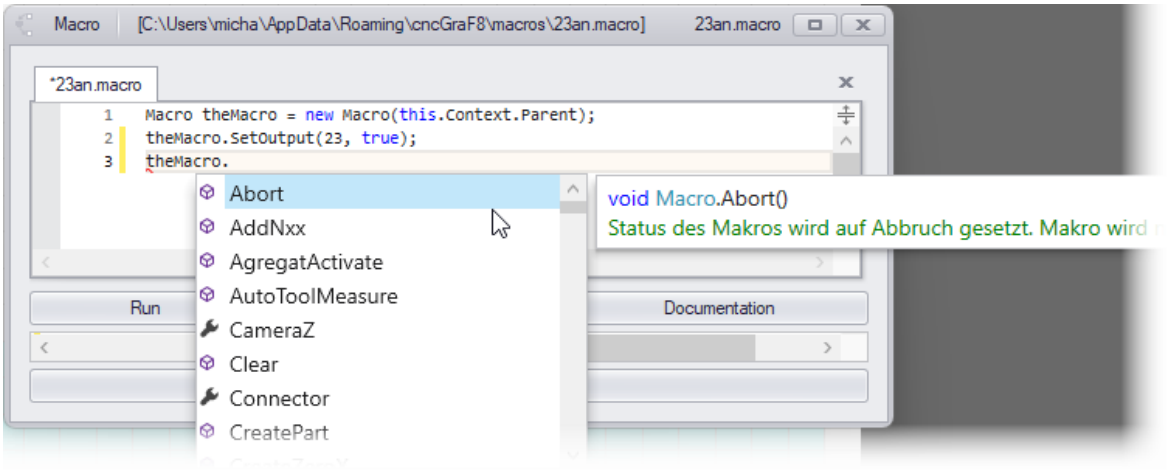


Figure 2: Autocompletion window after typing the dot (“.”).

Open an existing macro

Alternatively, an already existing macro can be opened. To do this, go to the **Macro** menu, select **Open...**, and choose the desired macro file.

Test a macro

Before a macro is integrated into a machining job, it should first be tested. The test can be started in the **Macro Editor** dialog using the **Execute Macro** button. The commands will then be processed line by line and checked for logic.

If the syntax is error-free, the message **“Execution succeeded”** will be displayed at the end of the test.

Integrating Macros/Commands into a Job

Macros or commands can be assigned to a job via the dialog **Macro/Command Link...** (Menu: **Settings → Macro/Command Link...**).

Examples of possible links:

- Job Start / Job End
- After machining
- **Homing start/end** (macro replaces standard functions)
- **Pick up/return tool** (replaces commands from machine parameters)

Assigning macros as buttons in *Manual Move*

Macros can also be linked to so-called **free buttons**. These are available as possible links in the dialog **Macro/Command Link...**.
Free buttons can be added to the **Manual Move** window via the context menu **Customize layout** using drag & drop (plug-and-play). More details can be found in the chapter *Customizing* Manual Move 125.

Important: When macros are linked to the functions listed above, certain standard functions (previously defined in the machine parameters, e.g. *return tool*) will be replaced by the macro commands. In this case, the macros take priority.

5.9.1 Macro Programming

This section explains the basics of macro programming. Every macro **must** start with the following line:

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent, "Name of the macro shown
in the status bar during execution");
```

The line above creates a macro object named `theMacro`. The parameter `this.Context.Parent` links the macro to cncGraF 8 and **must always be provided**. The second parameter is the text that appears in the program's status bar while the macro is running.

Using the `theMacro` object, you can call various functions. The following macro executes DIN 66025 (G-code) commands. Commands are added **line by line** with
`theMacro.AddNxx("...").`
`theMacro.Start();` executes all previously added commands.
`theMacro.Clear();` clears the command list.

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent, "DIN66025 Befehle
ausführen");

// Add DIN66025 commands line by line
theMacro.AddNxx("G90");
theMacro.AddNxx("G0 Z1 M03");
theMacro.AddNxx("G0 X50 Y150");
theMacro.AddNxx("G0 Z5 M05");

theMacro.Start(); // Execute the DIN66025 commands
```

You can switch outputs or query their status from a macro. The macro below turns **Output 1** on and then checks its status.

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent, "Execute DIN66025
commands");

// Turn Output 1 ON ('true'); use 'false' to switch outputs OFF
theMacro.SetOutput(1, true);

// Query the status of Output 1
if(theMacro.IsOutputOn(1)) {

      // Output 1 is ON!
      theMacro.MessageBox("Output 1 is ON!");
```

```
} else {
      // Output 1 is NOT ON!
      theMacro.MessageBox("Output 1 is NOT on!");
}
```

Just like outputs, you can also query **inputs**. The macro below checks **Input 9**:

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent);

// Query the status of Input 9. With 'true' or 'false' you can invert
the test.
if(theMacro.IsInputOn(9, true)) {

      // The IF statement evaluated to true (1),
      // meaning Input 9 is ON.
      theMacro.MessageBox("Input 9 on Con3 is ON!");
}
```

With a macro, you can **return** and **fetch** tools. (Requires a connected tool changer.)

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent, "Return last tool and
fetch Tool 1");

theMacro.ToolPutOff();
theMacro.ToolFetch(1);
```

The macro below reads the machine's **absolute coordinates** and shows them in a dialog:

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent);

var theZ = theMacro.GetAbsPosZ(); // Read machine position in Z

//  Show it in a dialog window
theMacro.MessageBox("Position der Maschine in Z beträgt: " + theZ);
```

This macro performs a **homing cycle**:

```
// Create the macro (this line must always be at the beginning)
Macro theMacro = new Macro(this.Context.Parent);
theMacro.Reference();
```

Documentation for the `Macro` object can be found in the file **macro.chm**. This file is located in the cncGraF 8 directory and can be opened from the Macro Editor.

5.10 Plugin Manager

A **plugin** is a program that attaches to a host application—here, cncGraF 8—to extend it with new features. In the **Plugin Manager** dialog, plugins can be enabled or disabled. Once a plugin is enabled, it appears under the main menu **Plugins**.

For more information on developing your own plugins (Visual Studio C# sample projects, documentation), see: ...\Users\[BENUTZERNAME]\AppData\Roaming\cncGraF8\code\csharp

5.11 Unit and Scaling

Without the correct drawing unit, the dimensions of a drawing will not match the intended size. To simplify the selection, several predefined units are available: **1 mil, 1/40 mm, 1/100 mm, and 1/1000 mm**. If none of these presets are suitable, a custom unit can be defined in the input field.


For the file type **G-Code (DIN66025)**, specifying a unit is not possible because the file always uses its **original size**. In this case, the **Scaling** dialog appears without a unit selection.

5.12 Tools for Zero Point Determination

To determine the zero point in **X/Y**, either a **camera** or a **cross laser** can be used. This function can be enabled via the menu:
Settings → Zero Point Determination.

When the corresponding icons are pressed, the zero points in **X/Y** are automatically adjusted, taking into account the preset offset between the tool (cutter) and the camera or laser.

The icons **"Set Zero Point with Laser/Camera Offset"** can be shown or hidden in the **standard toolbar**. This is done using the arrow at the end of the toolbar.

5.13 Link Settings

To avoid constantly switching between configurations, a setting can be copied via the menu:
Settings → System Settings → Link and placed as a shortcut on the desktop.

When linking, a copy of the configuration is created and associated with the desktop shortcut. This allows cncGraF 8 to be started with different configurations.

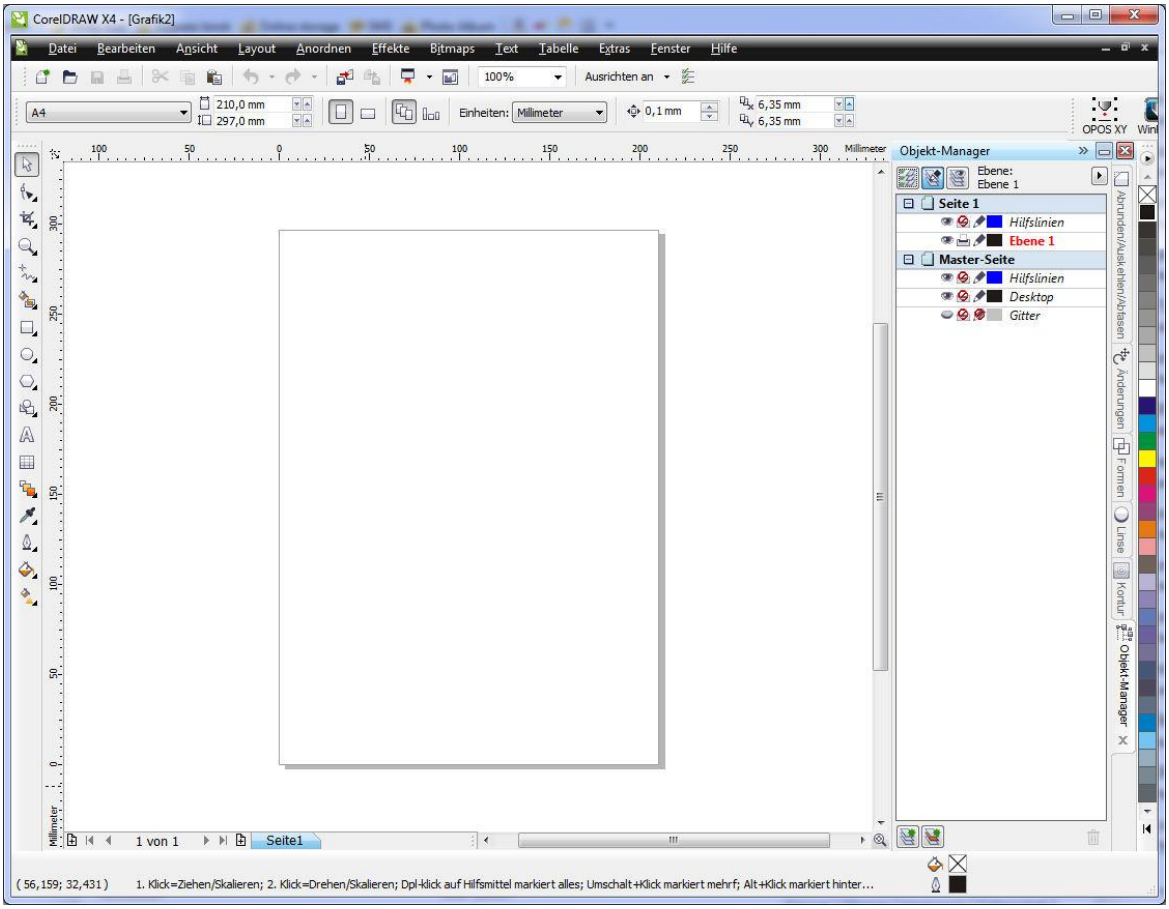
5.14 Video Positioning

Before the video positioning system in cncGraF 8 can be used, it must first be set up for the machine. The *Video Positioning* chapter is structured as follows:

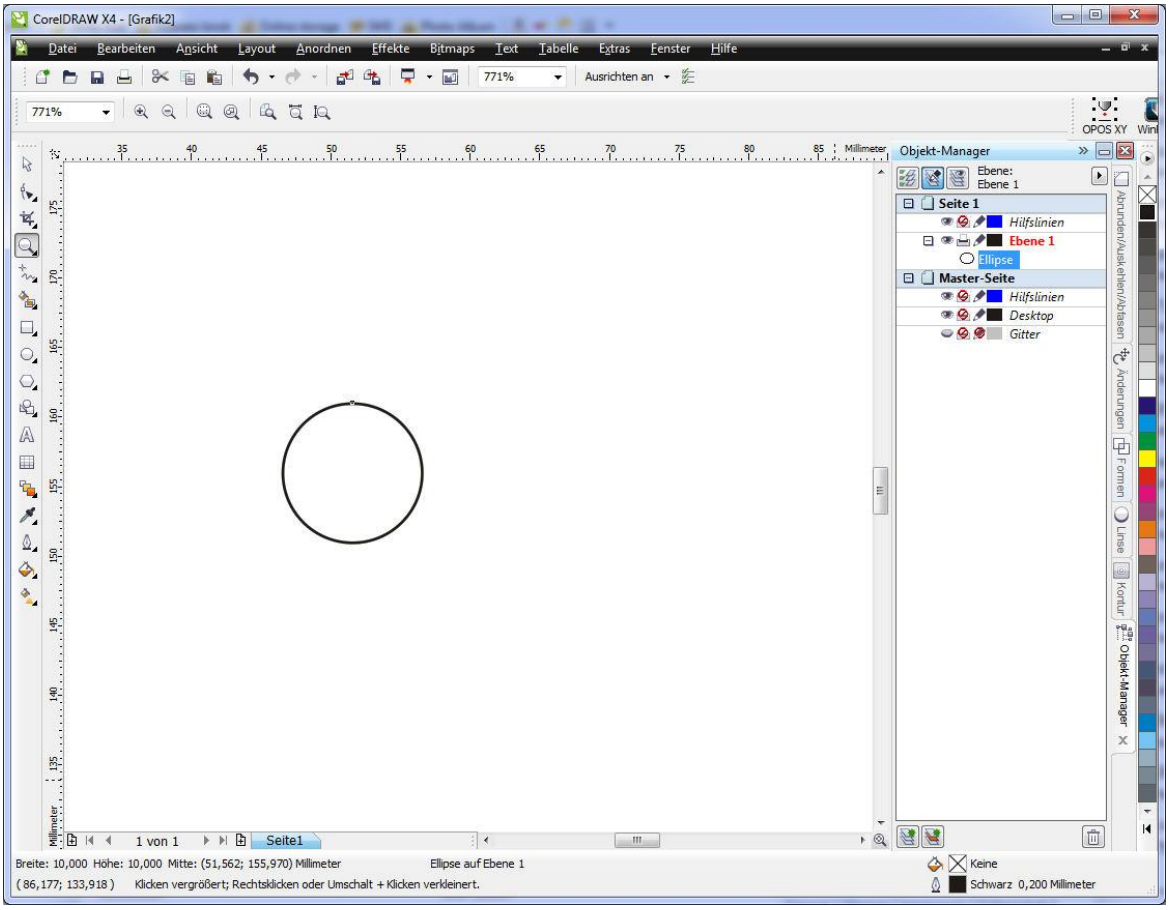
1. Creating a calibration template with Corel Draw 101
2. Calibrating the video positioning system 106
3. Creating data for printing and cutting/milling with Corel Draw 108
4. Using video positioning 110

5.14.1 Creating a Calibration Template with Corel Draw

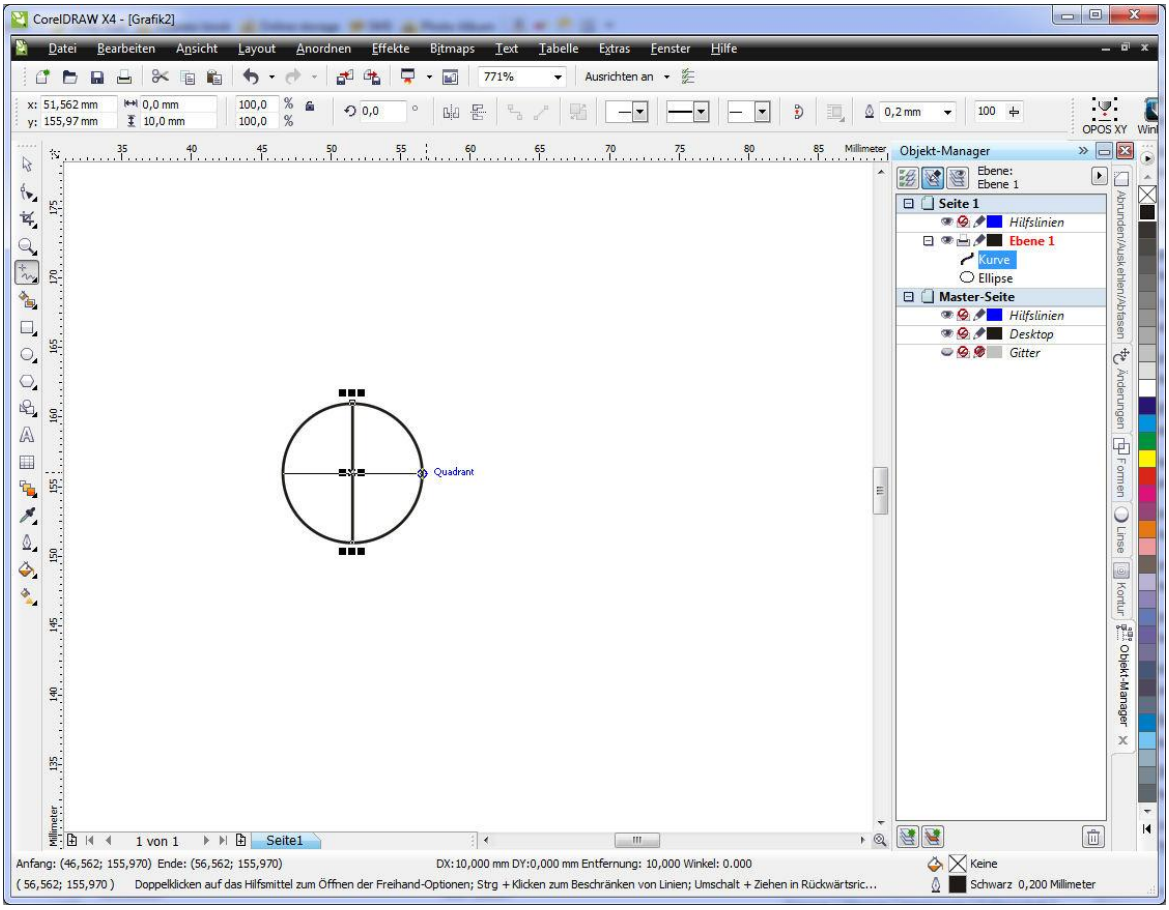
Start Corel Draw with a blank DIN A4 page.



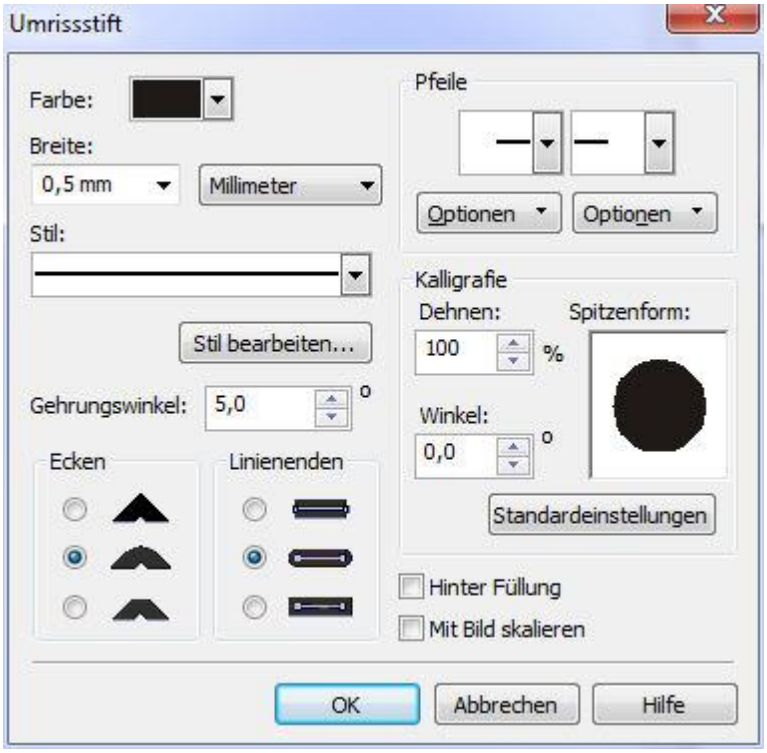
Using the **Circle/Ellipse tool**, draw a circle with a diameter of **10 mm**.

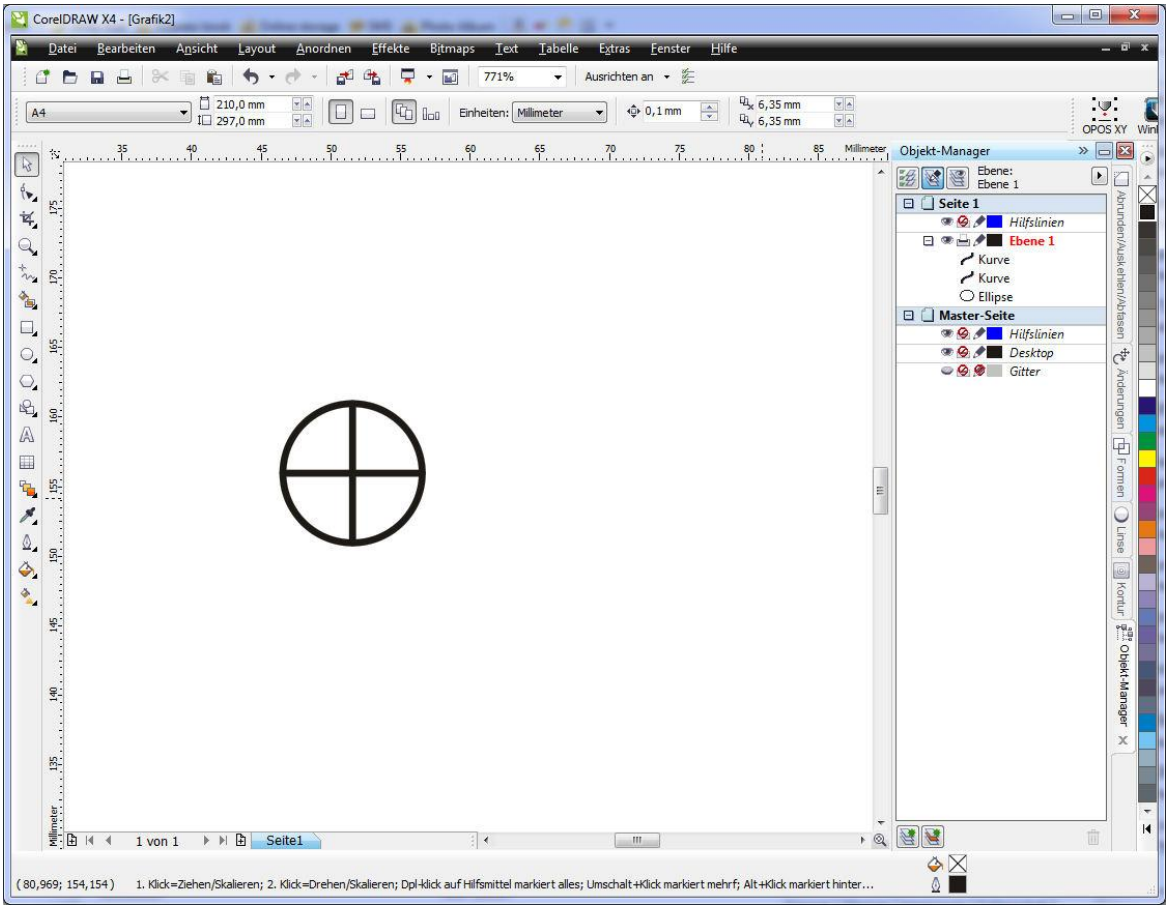


Draw two lines crossing at the center of the circle. Use the **Snap to Objects (Alt + Z)** option to align the lines from quadrant to quadrant.

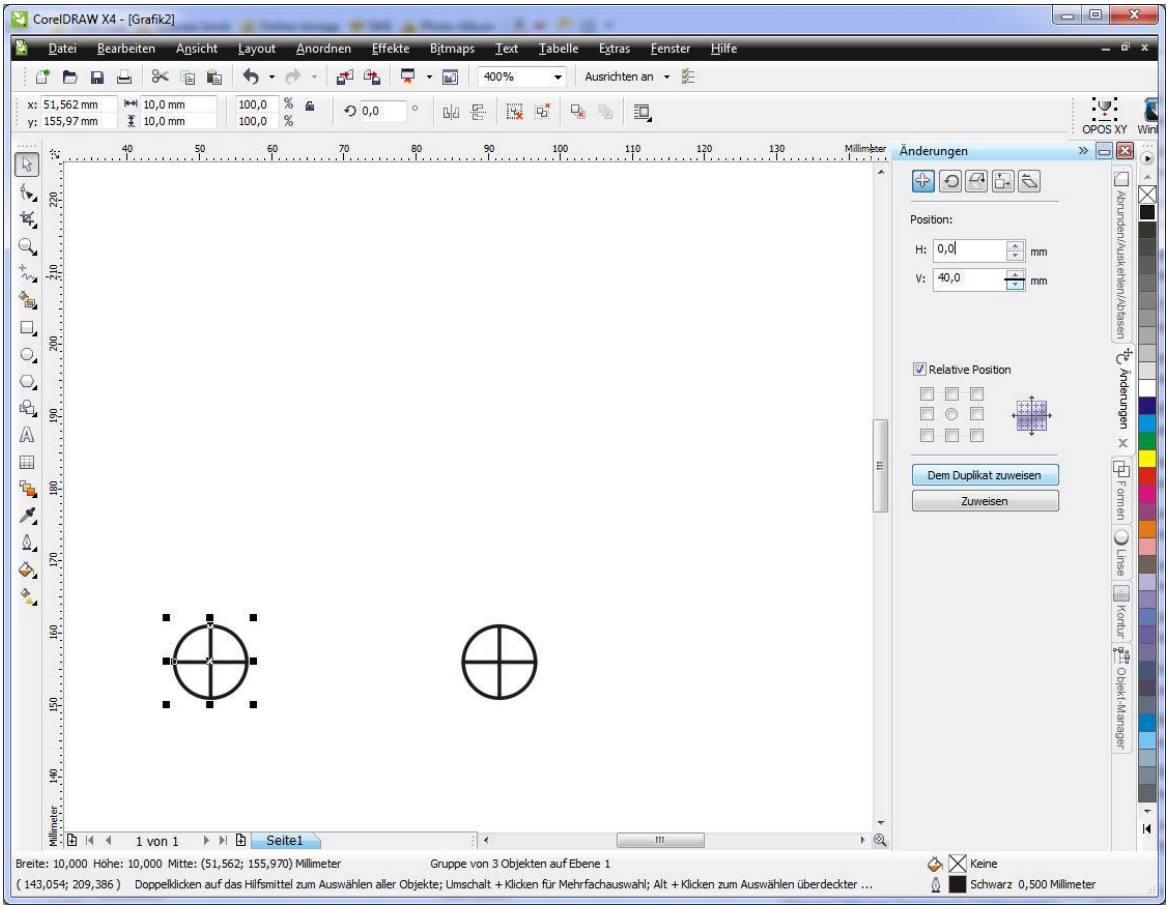


Select all three elements and **group them together**. Then assign line attributes. The line thickness should be about **0.5 mm**.

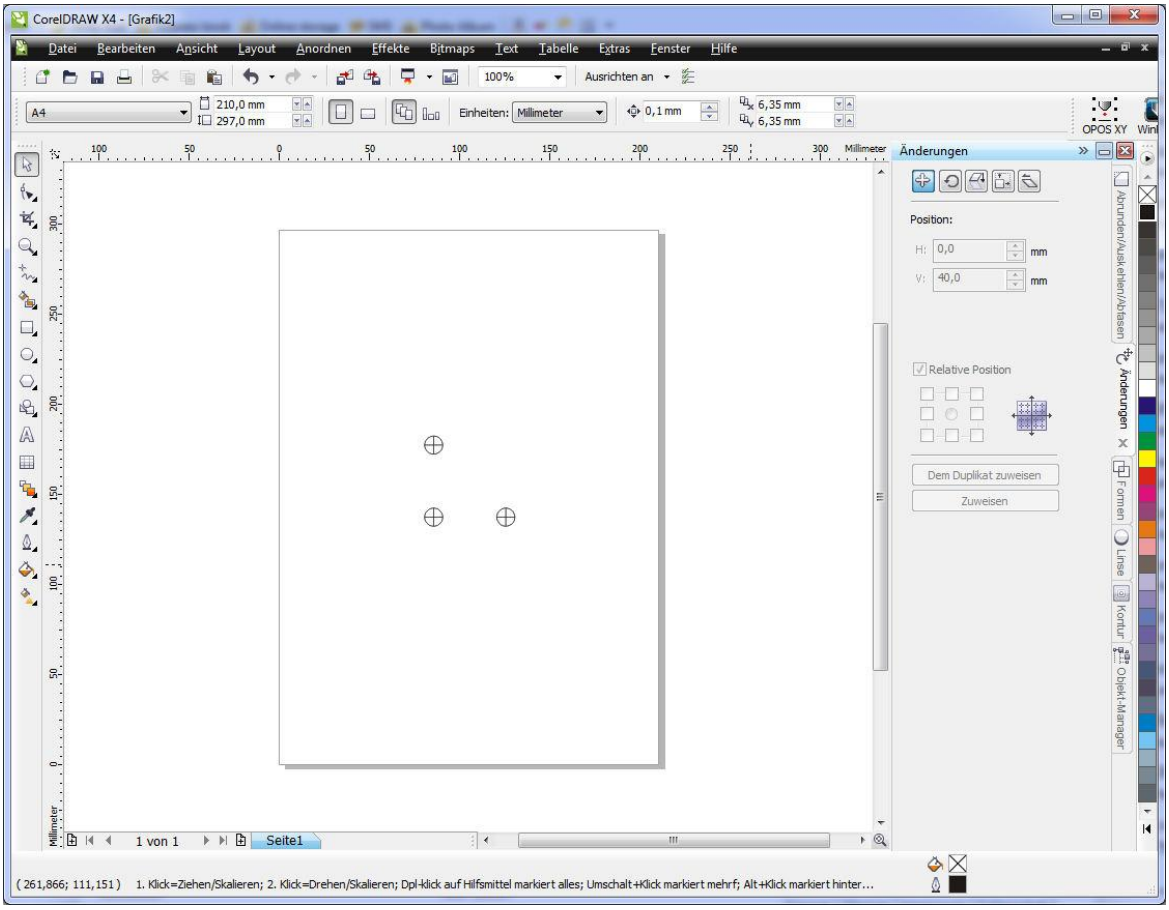




Duplicate this marker once in the X-direction and once in the Y-direction.



The finished template should look approximately like this (see example illustration).



Finally, save the file as a **calibration template**.

Note: The spacing and size of the markers can be adjusted to personal requirements. The sizes provided above are suitable for a camera distance of **120 mm** and have proven effective in practice.

5.14.2 Calibration of the Video Positioning System

Before the video positioning system can be used, it must first be calibrated. Proceed as follows:

1. Start the calibration dialog via **Settings > Calibration of the Video Positioning System**.

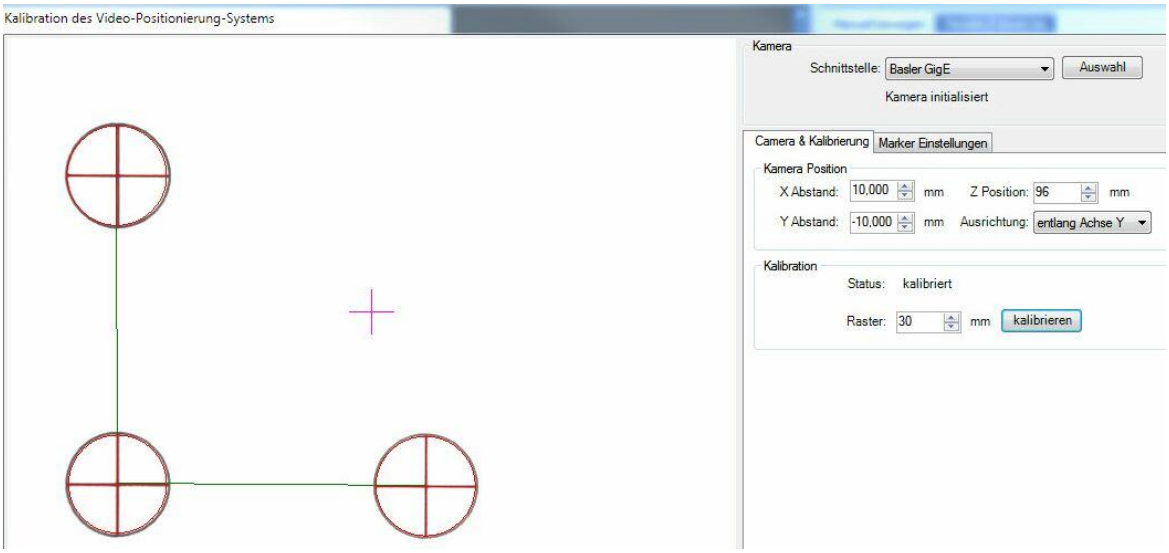


Abbildung: Kalibrierungsdialog

2. In the interface, select the **camera** to be used.

3. Print a sheet with three marker points placed at equal distances (a template is included in the program directory). Position the sheet under the camera so that all three points are visible in the center of the display window.

4. Under **Marker Type**, select the shape of the marker. In the example shown, the marker type is *Circle + Line* (more precise). The **Radius** parameter defines the circle size. The alternative option is three solid black dots.

5. Enter the **distance between the markers** into the field *Marker Distance → Object Distance*.

6. Enter the values for the **camera position** (distance between the camera and the cutter in X and Y).

7. Position the camera by manually moving the Z-axis until a sharp image is visible (the camera does not support auto-focus). Transfer the Z-distance to the material into the field *Z Position*.

Important: If the material thickness changes, the *Z Position* value must also be updated; otherwise, the camera image will be blurred and recognition will fail.

8. Fine-tune the **Marker Detection Settings** if necessary to improve recognition:
- **Threshold** – defines how visible the lines must be to be recognized. Lower values make lines easier to detect but may cause false positives; higher values detect only the clearest lines.

- **Min. Line Length** – sets the minimum detectable line length. Lines shorter than this are ignored. Smaller values include shorter lines in recognition.
- **Max. Line Gap** – defines the maximum allowed gap along a line to still treat it as a single line. Gaps larger than this are interpreted as two separate lines.

9. Start the calibration by pressing the **Calibrate** button. Three red crosshairs should appear aligned with your marker points and connected by two green lines (forming an "L" shape). If so, the calibration is complete.

You can now close the calibration dialog.

Tips:
- Ensure sufficient lighting and print quality for reliable detection of the markers.
- Poor lighting or low print quality may cause recognition errors.

Measuring the Camera Offset

1. Place a thin piece of material on the machine.
2. Depending on the tool in use (knife or milling cutter), briefly dip into the material to create a reference mark.
3. Set the **relative coordinates** to zero.
4. Move the Z-axis to the camera's operating height.
5. Use X and Y movement to align the crosshair precisely with the mark.
6. The displayed RX and RY values now correspond to the **camera offset**.

5.14.3 Creating Data for Printing and Cutting/Milling with Corel Draw

When preparing data for printing and cutting/milling, a few important points must be considered. The following example demonstrates the general procedure. While Corel Draw is used here (as in the creation of calibration markers), other programs such as Adobe Illustrator can also be used.

Step 1: Create Layers

1. Start Corel Draw with a blank document.
2. Create three new layers and rename them:
- **Print**
- **Cut**
- **Marker**
3. Set **Print** as the active layer and import the object to be printed (in this example, a sign that will be printed on a flatbed printer and then milled along its contour).

If the imported data is on the wrong layer, move it to the **Print** layer.

Step 2: Prepare the Cut Path

1. The file already contains a contour (a light outline around the object).
2. Select this line, assign it a distinctive color, and move it to the **Cut** layer.
3. Create a rectangle that completely encloses the graphic.
4. Assign the rectangle a different, clearly visible line color and place it on the **Cut** layer as well.

Step 3: Add Registration Marks

1. Open the file containing the calibration markers.
2. Copy one marker into the current document and duplicate it three times, so that four markers are available.
3. Use the **Align to Objects (Alt+Z)** option to position the markers at the corners of the rectangle so that their centers lie exactly on the rectangle corners.
4. If necessary, move the markers to the **Marker** layer.

The finished document should now contain:

- The **Print** layer with the design.
- The **Cut** layer with the contour and rectangle.
- The **Marker** layer with the registration marks.

Step 4: Export Files

Two separate files must be exported from the document:

- **For printing**:
  - Disable the **Cut** layer.
  - Export the file as PDF or EPS for printing.
- **For cutting/milling**:
  - Disable the **Print** and **Marker** layers.
  - Export the file (e.g., via Adobe Illustrator).

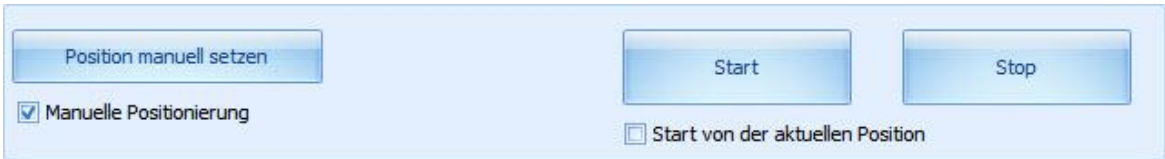
 **Tip:** Adobe Illustrator is advantageous because it preserves different line colors, which are recognized in cncGraF 8.

Step 5: Workflow in cncGraF 8

- The rectangle surrounding the contour defines the page size but is **not cut/milled**.
- In cncGraF 8, this rectangle can be deactivated via the tool library.
- After printing, place the printed sheet on the CNC machine bed.
- Start the **marker recognition** process to align the cutting/milling path with the print.

5.14.4 Using Video Positioning

Once the camera has been selected and calibration completed, the **Video Positioning** function can be used. In the **Camera** tab, the live video image is displayed along with a toolbar.



Toolbar Options

1. Start

- Activates the Video Positioning function.
- The registration marks are automatically approached and detected.

For automatic detection, the printed sheet must be positioned at the machine's **origin point** (X=0, Y=0). The drawing must also be loaded at its original- position 79.

- **Start from current position**: If the first (bottom-left) registration mark is already visible in the camera's field of view, it will be taken as the initial reference mark.

2. Manual Positioning

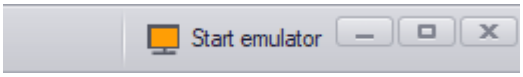
- When this option is enabled, marks can be positioned manually.
- The machine can be moved either through the **Manual Move** menu or by clicking directly on the video image.
- Pressing the **Set Position Manually** button assigns the current position (the cross in the camera image) as the marker.

6 Move

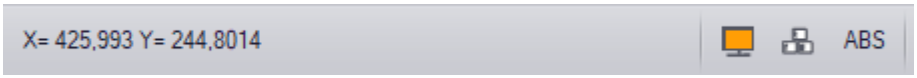
The pull-down main menu **"Move"** contains all machine-specific functions for controlling the movement of the CNC machine. This chapter is structured as follows:

6.1 Emulator

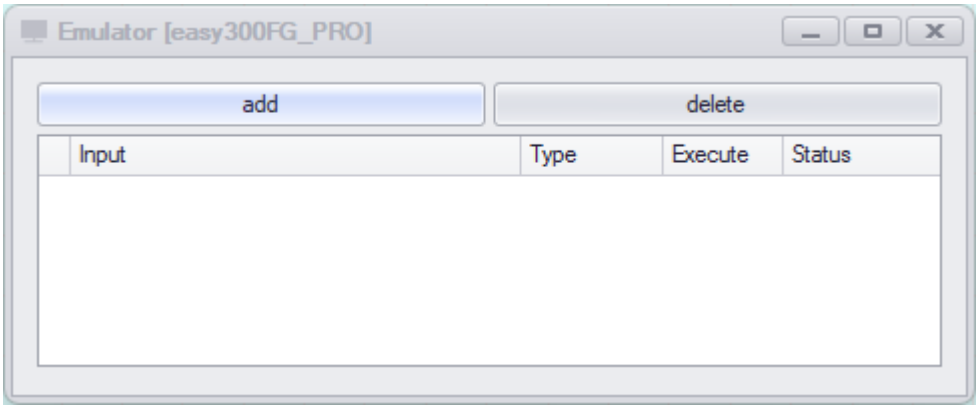
The **Emulator** simulates the controller smc5d-m4 and allows the operation of a **virtual CNC machine** without requiring any hardware. It can be started via the **emulator icon** located in the top-right corner of the main menu.



Once the emulation is running, the **orange emulator icon** will appear in the status bar at the bottom of the window.



In addition, the **Emulator dialog window** opens in the Windows taskbar. Here, inputs can be defined and then manually toggled using the mouse. For example, this allows the tool length sensor input switch to be activated virtually via mouse click.



To check the execution of individual vectors, you can alternatively use the **basic simulation** via the menu: **Window → Simulation**.

6.2 Milling / Drilling

The milling process can be started either by clicking the milling/drilling icon  in the horizontal toolbar or by pressing F9 (Menu: Operation → Milling/Drilling). Before the process begins, the Job Parameters dialog opens, where the final settings for the machining job can be defined.

There are two variants of the Job Parameters dialog:

- Job Parameters 114 **for 2D files** (e.g., HPGL, DXF, …)
- Job Parameters 118 **for G-Code files** (DIN66025)

6.2.1 Milling / Drilling 2D Data

2D file formats such as **HPGL, DXF, EPS, PostScript** do not contain any information required for CNC machine control. Therefore, essential parameters like processing order, clearance height, Z-depths, feed rates, etc., must be defined either in the Tool Library 76 dialog or in the **Job Parameters** dialog.

Job Parameters for 2D Files

In the *Job Parameters* dialog, the following settings can be defined:

Selecting Data and Processing Order

As an example, consider PCB milling. A PCB layout typically consists of an HPGL file and the corresponding drill file. cncGraF 8 can open, edit, and display both files within a single document. In the first step, the *Job Parameters* dialog specifies which dataset should be processed. You can choose between **vectors** and **drill points** (see Figure 2). If the document contains only drill data (Sieb & Maier) or only vector data (HPGL, DIN 66025), then only the available data type is shown in the dialog.

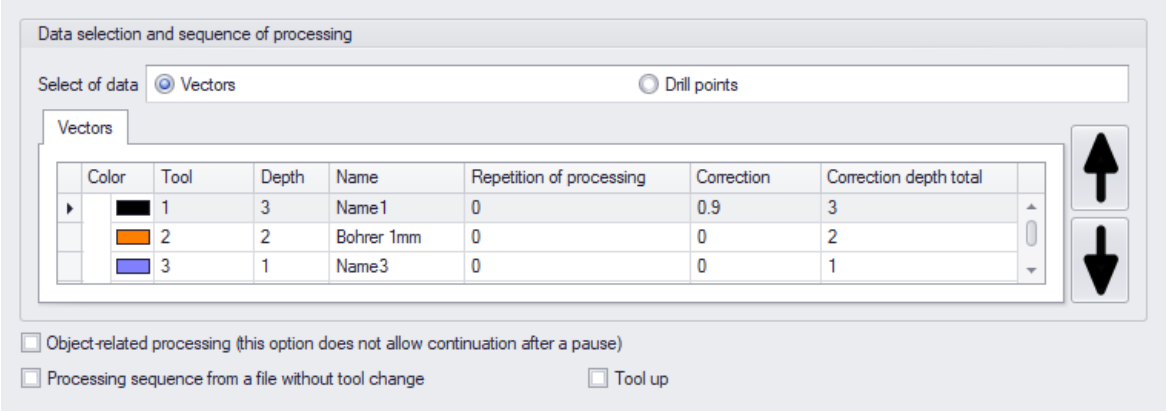


Figure 1: If both vectors and drill points are present, a data selection is required.

The tool lists under **“Vectors”** and **“Drill Points”** display all tools used in the order in which they will be processed (from top to bottom) (see Figure 2).

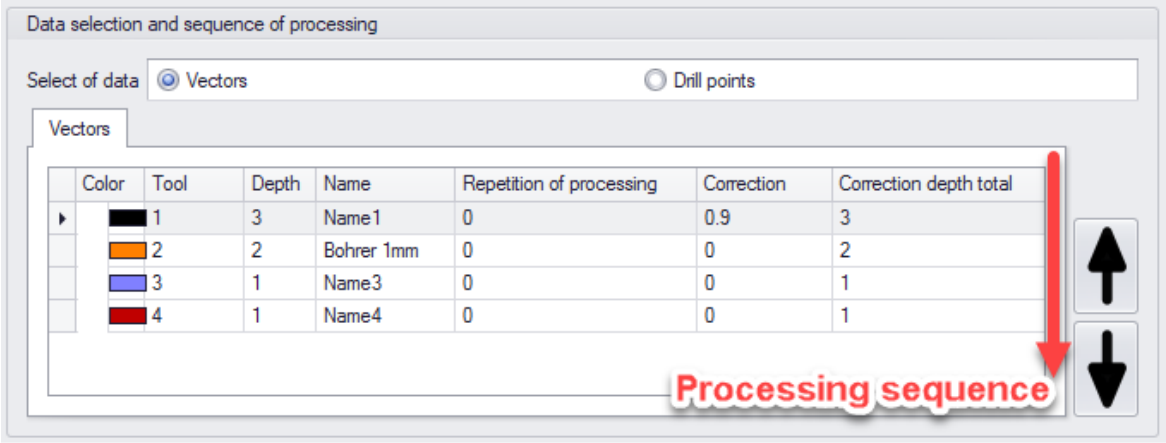


Figure 2: The option **"Tool Change"** is active. The processing order can be changed by selecting a tool and using the arrow buttons.

Note: For **DIN66025 (G-Code) data**, the processing order **cannot** be changed.

Repeat Machining and Step-Down Correction

The values for **"Repeat Machining"** and **"Step-Down Correction in millimeters"** are entered directly by clicking on the corresponding row and column in the tool table.

- A value of **0** means the machining is carried out once.
- A value of **1** means the operation is repeated once.

In combination with Z-axis step-down correction, this function is useful whenever the desired material removal cannot be achieved in a single pass (e.g., milling grooves in metal, surface grinding, etc.).
cncGraF 8 offers the following options for **Repeat Machining**.

1. **"Object-based machining"** can only be used for 2D data. In this mode, one object is fully processed before the machine moves on to the next object (see example below).

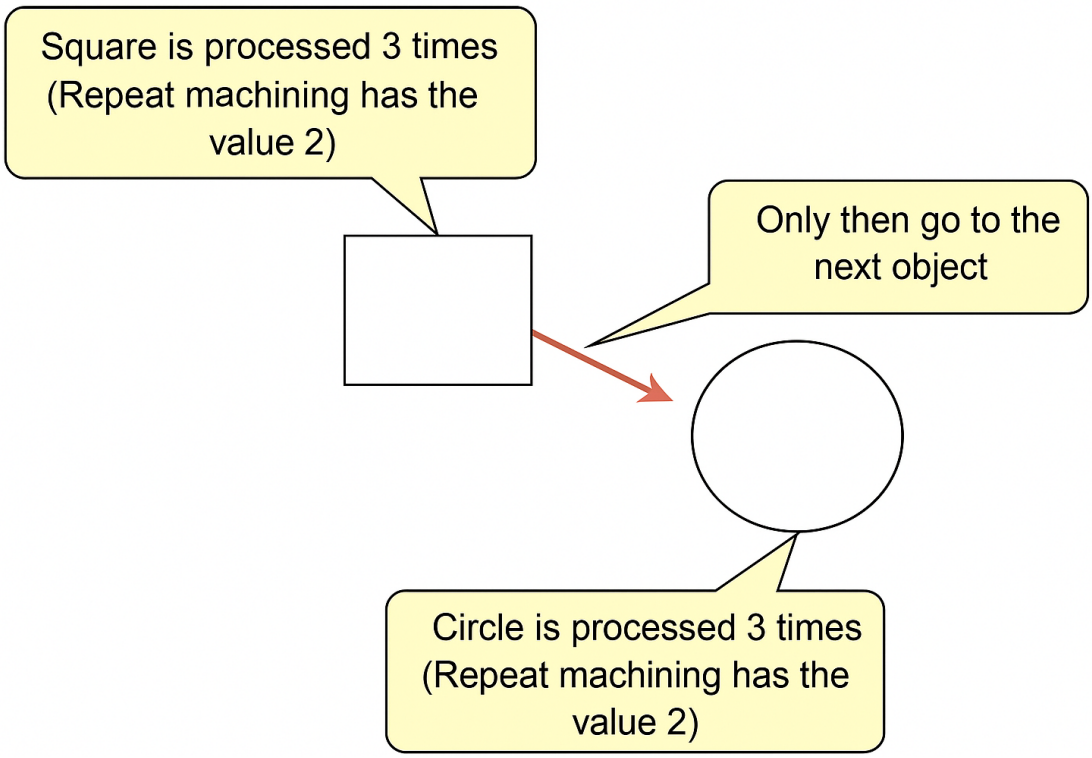


Figure 3: Object-based machining

Tool lift – After each object is processed, the tool is lifted.

Warning: When "Object-related processing" is enabled, 'Continue' (no pause) is not possible!

2. **"Repeat processing without changing the file order"** executes the milling process in the same sequence as the tools are stored in the file. In this case, the tools are taken into account. **Example**: In the HPGL file, the following tool sequence is defined: SP1, SP3, SP1, SP2, SP1. Here, all elements for tool SP1, SP3, and finally SP2 are processed first. After that, the process is repeated.

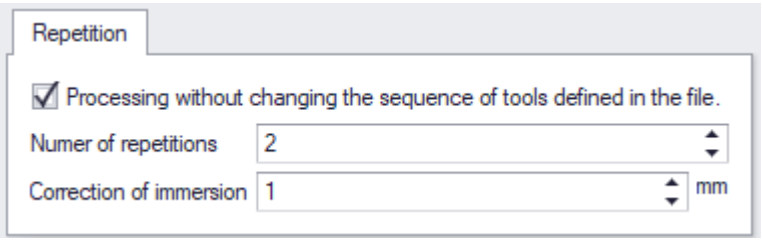


Figure 4: *"Repeat processing without changing the file order"* is active. The processing of the entire file is repeated twice with a depth correction of 1 mm.

3. **"Processing order from file without tool change"** executes the milling process exactly as it is stored in the file. In this case, the tools are ignored. Example: In the HPGL file, the following tool sequence is defined: SP1, SP3, SP1, SP2. Here, all elements for tool SP1, SP3, SP1, and finally SP2 are processed.

4. If the above-mentioned options are disabled, the value **"Repeat processing"** from the tool list will be executed in the corresponding tool order.

Properties
Under *Properties* you will find the following functions:

- **Tool change:** If this option is active, the option *Measure tools* can also be enabled. After a tool change, the tool length sensor is approached and the new tool is measured in order to determine and compensate for the length difference. A simple microswitch (probe) can serve as the sensor. After the measurement, machining continues with the new tool. To save time, the tool length can also be taken from the tool library. For this to work, some settings must be configured. Further information on *Tool length from the tool library* can be found in the chapter Tool Library 76 and in the chapter Tool Measurement 131.


More details about tool measurement can be found in the chapter Tool Measurement 131.

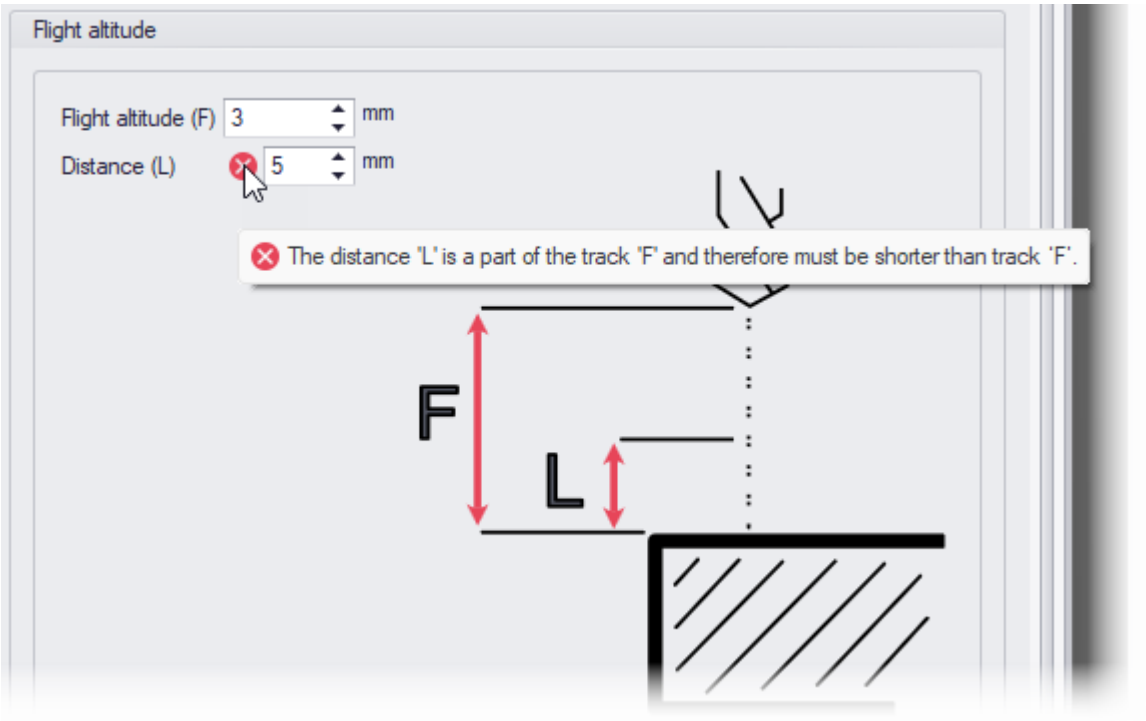
- **At the end** of the milling or drilling process, tasks such as raising the tool, moving to the zero point, moving to the park position, performing a homing move, or depositing the tool (only for automatic tool changers) can be carried out automatically.

- After a pause, tasks such as raising the tool, moving to the zero point, or moving to the park position can be executed automatically. More information about *Pause* can be found in the chapter 'Pause' 126.

- If the surface of the workpiece has been probed beforehand, height compensation can be activated. More about height compensation can be found in the chapter Workpiece Probing 127.

Clearance Height 
The clearance height *F* defines the distance between the tool tip and the material during rapid moves. The distance *L* is a subsection of the clearance height and is executed at the feed rate of the Z-axis. If the distance *L* is set to 0, then the entire descent from clearance height is executed at rapid speed.

The clearance height can be set automatically using the button . In this case, the current Z-position of the CNC machine is taken as the new clearance height.



6.2.2 Milling/Drilling from G-Code (DIN66025) Data

Unlike 2D files such as HPGL or DXF, G-Code (DIN66025) files already contain all the information required for execution on the CNC machine. Therefore, only a few additional settings are necessary.

Processing Order
The tool sequence is defined in the G-Code (DIN66025) file and cannot be changed. For better clarity, it is displayed in the *Job Parameters* dialog (see figure).

Figure: In the following example, tool T2 is used first, followed by tool T1.

Properties
The following functions are available under *Properties*:

- **Tool Change**: When this option is enabled, the *Measure Tools* option can also be activated. After a tool change, the tool length sensor is triggered to measure the new tool and calculate the length difference for compensation. A simple microswitch (probe) can serve as the sensor. Once the measurement is complete, the machining process continues with the new tool. To save time, tool lengths can also be retrieved from the tool library. For this to work, certain

settings must be configured. For more details on this, see the chapters Tool Library 76 and Tool Measurement 131.

For more information on tool measurement, see the chapter Tool Measurement 131.

- The clearance height is the distance between the tool tip and the material during rapid traverses.
- At the end of the milling or drilling process, actions such as raising the tool, moving to the zero point, moving to the park position, performing a reference run, or depositing the tool (only for an automatic tool changer) can be executed automatically.
- After a pause, actions such as raising the tool, moving to the zero point, or moving to the park position can also be executed automatically. For more information on pauses, see the chapter Pause 126.
- If the surface of the workpiece has been probed beforehand, height correction can be activated. For more details on height correction, see the chapter Probing the Workpiece 127.

Warning: The functions *Height Correction* and *Add Clearance Height to Output Data* modify the toolpaths of the G-code file.

6.2.3 Speed

With the smc5d controller, the speed of the CNC machine during milling/drilling can be adjusted in real time by modifying the controller's clock rate. The adjustment is made using the slider in the status bar at the bottom of the monitor. If a handwheel is connected, a handwheel icon appears (see figure).

Figure: Handwheel active – the speed can be adjusted via the handwheel or with the mouse using the slider.

The following options are available:

- Using the slider or the handwheel dial, you can set the starting speed at which the machining process begins. If it is unclear whether the file to be processed is error-free, it is recommended to start with a reduced speed. The speed defined in the file is scaled proportionally (0–100%). This starting speed applies to milling/drilling operations as well as moves to zero, park, or measurement points.
- During active machining, the speed can be continuously adjusted from 0 to 100% using the slider, the handwheel dial, or the +/- keys on the keyboard.

In the menu *Settings* > *Options* > *General* → *Speed* or via the toolbar icon, you can configure when the starting speed should automatically reset to 100%.

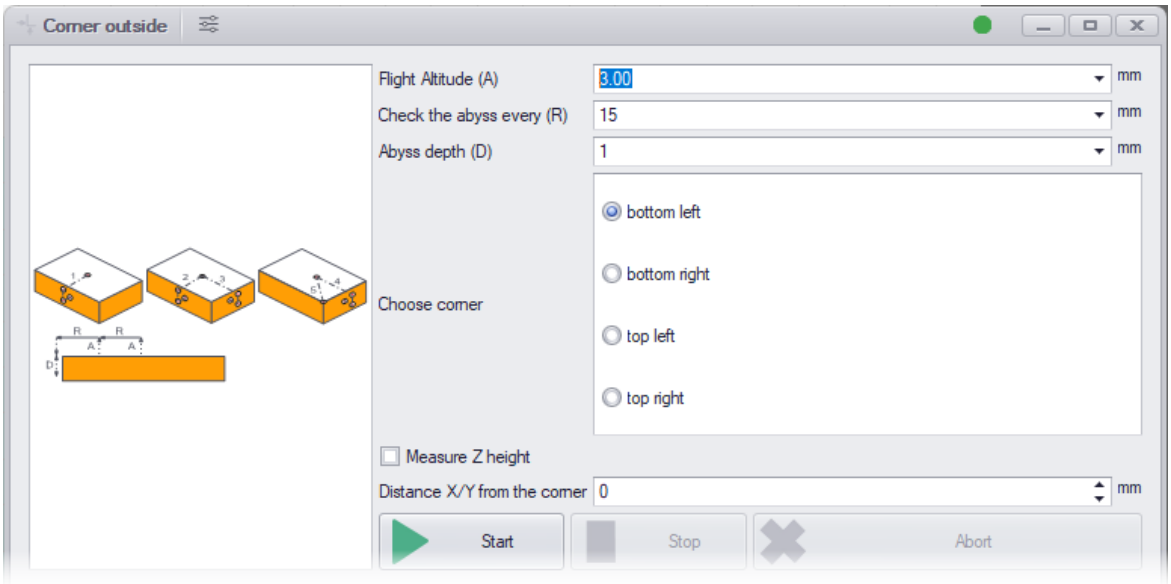
The available options are:
- Reset at job end or job abort
- Reset when loading a new file
- Reset when restarting the program

6.3 Probe Functions

In the *Probe* section (menu: *Operate* > *Probe* or via the toolbar), various functions for the 3D probe are available. These functions can be used to determine, for example, edges or midpoints, with the results applied for zero point determination.

Each function, such as *Outside Corner*, opens its own dialog window with parameters and an explanatory graphic. Every dialog window includes, in addition to its title, an option switch and a round green symbol indicating the input status. In the options, you can define whether the dialog window should automatically close after measurement and whether the zero point should be updated. Additionally, an information message can be displayed.

The green input status symbol is used to check the probe's functionality. When the 3D probe is triggered, the symbol toggles between on and off.

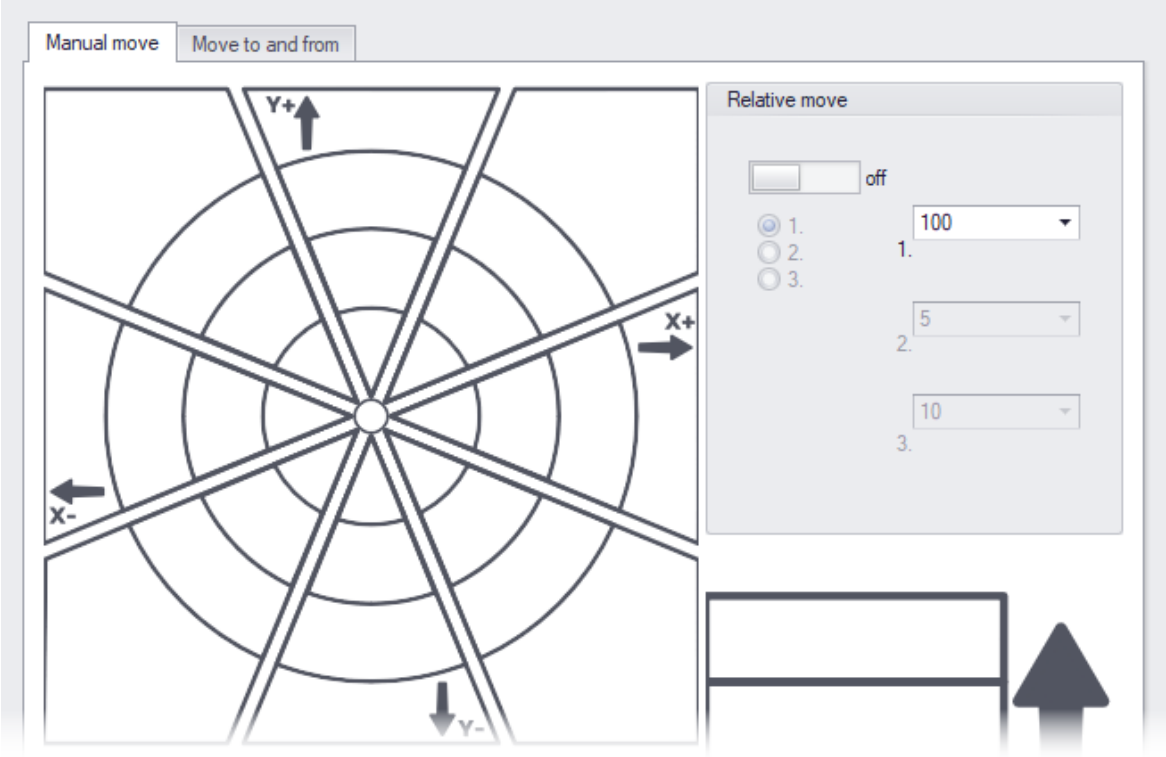


In the *Positions* dialog (menu: *Settings* > *Positions* > *Measuring Point*), multiple measurements can be enabled. There you can define how many times the measurement should be

repeated and which maximum deviation is allowed. When this option is enabled, the probing process is carried out multiple times, the average value is calculated, and the measurement accuracy is improved.

6.4 Manual Movement

The *Manual Movement* dialog provides all functions needed to move the machine manually. The window is docked to the right side of the main window. Using the tack icon, you can define whether the window should auto-hide or remain permanently visible.

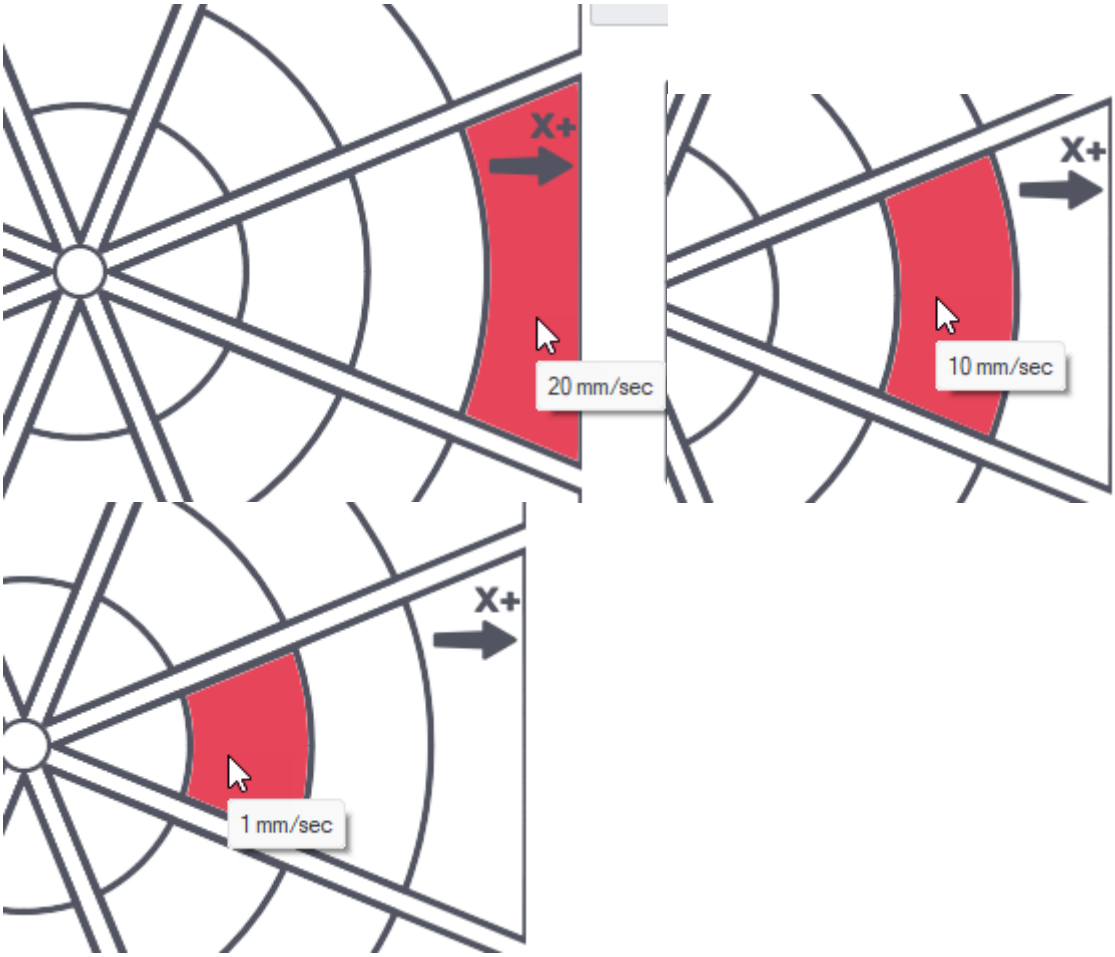


The *Manual Movement* dialog includes the following functions:

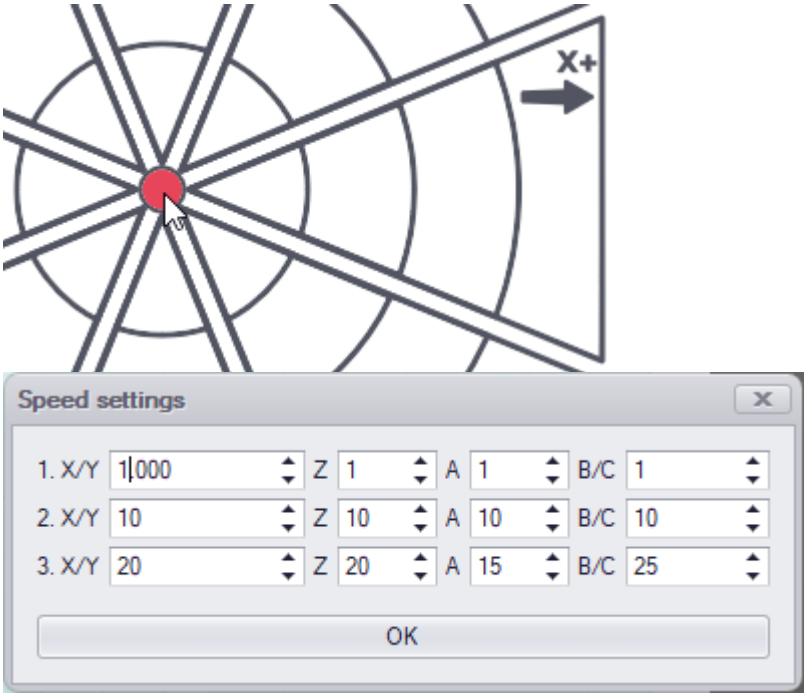
- Manual axis movement
- Setting travel speeds
- Relative movement
- Jogging / Move to position
- Switching pump, spindle, and other outputs
- Executing macros

Manual Axis Movement

The *Manual Movement* window consists of a vector graphic divided into axes and zones. By clicking the individual fields, the desired axis and its associated speed are selected. Each field has its own speed assigned (see figure series).



Setting Travel Speeds
In the middle of the graphic, there is a circle. By clicking this circle, the *Set Speeds* dialog opens, where an individual speed can be defined for each field.



Relative Movement

The *Relative Movement* function allows moving a defined relative distance. After entering the desired distance and clicking the corresponding axis button — which simultaneously determines the travel direction — the relative move is executed.

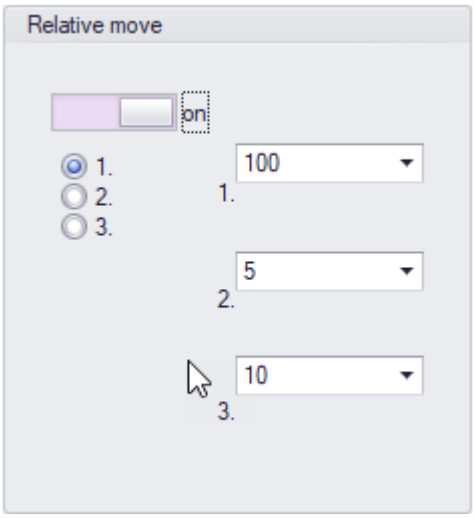


Figure: Distance 1 with 100 mm is selected. When clicking X+, the machine moves 100 mm in the positive X direction.

Jogging / Move-to

The *Move-to* and *Jogging* functions are used to approach a specific position. By entering X, Y, and Z coordinates, the position can be reached either directly or by jogging. If the option *Set X/Y*

position with mouse is enabled, the coordinates can be transferred directly into the X and Y input fields with a left mouse click.

Switching Pump, Spindle, and Other Outputs

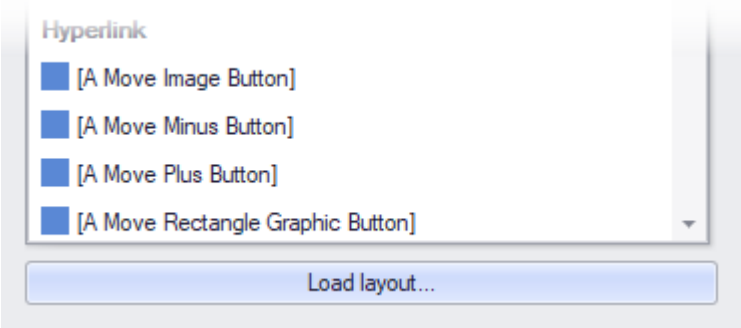
Pump and spindle can be switched on or off by clicking the respective buttons. Using the *Customize Layout* context menu, additional buttons such as collet or spindle speed can be added. Furthermore, free buttons can be configured to execute macro commands. This allows the *Manual Movement* window to be individually customized. More information can be found in the chapter Customize Manual Movement 125.

6.4.1 Customize Manual Movement

All dialogs, including the *Manual Movement* window, can be customized via plug-and-play. To do this, select the *Customize Layout* option from the context menu. The context menu opens when you right-click on a free space in the *Manual Movement* window between the elements.

After selecting this option, the *Layout Customization* dialog opens.
At the bottom area, elements are displayed that can be inserted into the *Manual Movement* window via drag & drop (select the element, hold down the left mouse button, and move it). Existing elements can be removed in the same way. In addition, all elements can be moved and adjusted.

By right-clicking again inside the *Manual Movement* window, an extended context menu opens with additional functions for editing and customizing the window.
Several predefined layouts of the *Manual Movement* window are available. A layout can be loaded via the *Load Layout...* button in the lower area of the *Layout Customization* dialog.



If something goes wrong during layout editing, the original layout can be restored using *Load Layout...*.

6.5 Performing Homing and Moving to Positions

Various functions are available under the *Drive* menu to move to specific positions, such as homing to the zero point or moving to the park position. One of the most important functions is *Homing*, which is required for correct machine positioning. After program start, the axis positions in cncGraF 8 are unknown. Therefore, a homing procedure must first be performed. For safety reasons, this typically begins with the Z-axis moving upward, followed by the X, Y, and other axes.

In the *Homing* dialog, you can select which axes should be referenced. If the check mark for an axis is removed, that axis will be excluded from the homing process. For CNC machines without reference switches, or during initial setup, the reference point can also be defined manually via *View* > *Set Reference Point*.... Once homing is completed, the absolute coordinates in the program's main menu are reset to their initial values.

By default, homing is executed at reduced speed, which can take considerable time on large CNC machines. The *Fast Homing* function therefore first performs a rapid move to a predefined position near the reference point (X, Y, and Z) (see chapter *Machine Setup* > *Axes* > *Homing*). The actual homing process then starts from this point.


The *Fast Homing* function cannot be used immediately after program start. It is only available if:

- the CNC machine's position is already known, and
- all axes have already been referenced.

To move to a zero point, park position, or measurement position, it must first be defined in the *Edit Positions* dialog (menu: *Settings* > *Positions* or via the toolbar icon).

During homing, the travel direction is indicated by arrows, which serve as a control for the settings:
<-- = move toward the reference switch
-> = move away from the switch

6.6 Pause and Abort

While the CNC machine is running, all functions are disabled—except for *Pause* and *Abort*. Both functions stop the machining process but differ in their behavior:

- **Pause**: The machine is stopped using a deceleration ramp. Machining can then be resumed without requiring a new homing procedure.
- **Abort (ESC key)**: The machine is stopped immediately without a deceleration ramp. This may cause step losses, which makes it mandatory to perform a homing procedure afterward to reestablish the correct position. ist.

6.7 Check Position

The *Check Position* function (Menu: **Run > Check Position**) performs a homing cycle. In contrast to a normal homing cycle, the current position is verified, and at the end, the step losses of each axis are displayed.

Important: After configuring all machine parameters, this function must be executed to verify both the settings and the hardware.

For detailed instructions on performing a position check, see the chapter Step Losses 46.

If a CNC machine shows step losses, several factors may be responsible. Common causes include:

- Excessive speeds → verify in the dialog *Machine Parameters > Check Speeds*.
- Incorrect motor power supply (too little or too much current) → check hardware settings (motor drivers).
- Improper motor current reduction → verify in *Machine Parameters > Pin Assignment*.
- Incorrect step signal direction → adjust the step signal for the stepper motors in *Machine Parameters > Pin Assignment > Invert Step Signal*.
- Hardware issues → axes move sluggishly, jerk, or get stuck.

6.8 Probe Workpiece

The *Probe Workpiece* function (Menu: **Run → Probe → Probe Workpiece**) enables milling on uneven surfaces. Before the machining process begins, the surface is scanned in the Z-direction using a predefined grid. The collected data is then used to correct the Z-axis so that the milling tool maintains a constant cutting depth – within the specified tolerance – during machining.

Functions of the *Probe Device* dialog

Edge Clearance
Prevents errors when probing near the edges of the workpiece by ensuring the probing area does not reach the very edge.

Probing Area
Defines the size of the probing area in the X and Y directions (in millimeters).

Abort on Tolerance
Safety function: Compares the current measurement with the previously recorded value. If the difference exceeds the set tolerance, the machine stops immediately.

Warning: The *Abort on Tolerance* safety function should always be enabled.

Grid Division
The probing grid in the X and Y directions is determined by dividing the length and width of the probing field.
Example: With a side length of 100 mm and a division of 5, a total of six probing points are created, each spaced 20 mm apart. For relatively flat surfaces, only a few grid points are sufficient to achieve the desired precision of height correction.
For verification, the probing area can be displayed by enabling *Show Probing Area* or by selecting **Main Menu > View > Probing Area** after the probing process.

The dimensions of the final engraving or milling pattern must always be smaller than the defined probing area.

Lift Probe
Defines the clearance distance of the tool tip above the material during rapid traverses.

Probe Deviation
At the end of the probing process, the program calculates ideal values for each grid field (rectangle) based on the measured data. The deviation is the difference between the ideal value and the measured value. The largest deviation is always displayed.
If the deviation exceeds 0.1 mm, the grid field may contain a bump or dent. In such cases, a finer grid should be chosen and the surface re-probed.

The deviation can also be displayed later via **Main Menu > View > Probe Deviation...**

Export / Import Probing Data
The probing results can be exported or imported (**Menu: File → Probing Data**). Data can be saved as DXF (lines or points), DIN66025, text files, or as a cncGraF 8 SCAN file. SCAN files can later be re-imported into cncGraF 8 using the import function.

6.9 Calibrate Tool Length Sensor

In order to measure tool length, a one-time calibration of the tool length sensor is required. This is carried out using a wizard. Before starting the wizard, a measuring point must be defined under **Settings > Positions > Measuring Point**. Instructions and notes for this can be found in the chapter Measuring Points 73.

Important: The correct input for the tool length sensor must be selected in the machine parameters and should be verified. To do this, press the tool length sensor input and check in the

status bar (via the USB/Ethernet symbol). The input must change, and the input number must match the setting under **Machine Parameters > Pin Assignment > Measure Tool Length**.

The tool length sensor calibration wizard guides you step by step through the following tasks:

1. **Reference Run**
   For accurate position detection, a reference run must first be executed. If it has already been carried out, this step can be skipped by selecting *Next*.

2. **Measure Machine Table Height**

- Mount any tool.
- Lower the Z-axis until the tool tip touches the machine table.
- Save the table height by selecting *Click here to save the table height* in the dialog.

From the saved table height and the material thickness, the zero point height (Z0) is calculated. The material thickness starts at the height of the machine table and ends at the top surface of the material (see figure).

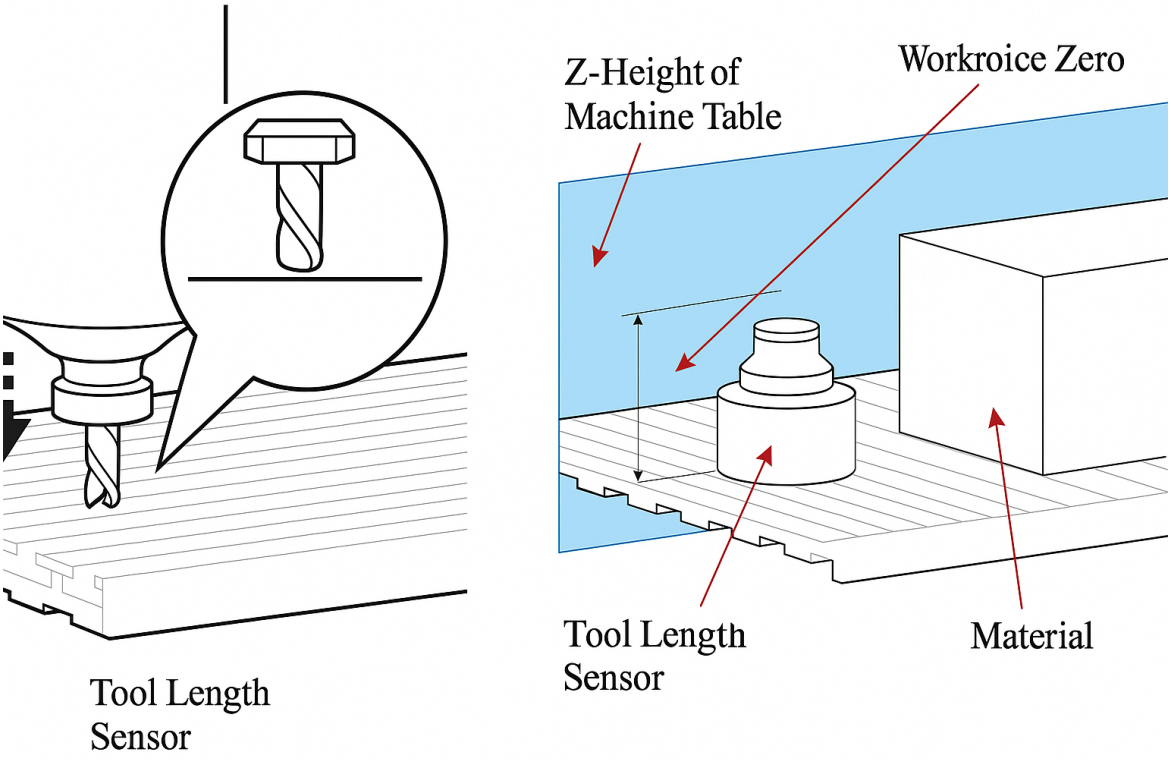


Figure: Table height and material thickness together define the zero point height.

In the **Positions** 70 dialog, the zero point height can be defined by entering the material thickness.

3. Summary
On the final page of the wizard, the table height and the switching height of the tool length sensor are displayed. Clicking *Finish* saves the values and closes the wizard.

4. Measure Tool Length Sensor
The tool length sensor is approached to determine the switching height.

 If the tool length sensor is no longer used, the calibration can be disabled (**Run > Tool Length Sensor > Enable/Disable Calibration**).

5. Tool Length Sensor – Properties
The determined values *Table Height* and *Switching Height* can be manually adjusted in the **Tool Length Sensor Properties** dialog. This dialog can be accessed via **Run > Tool Length Sensor > Tool Length Sensor Properties** in the main menu.

 **Caution:** Editing these values should only be done in exceptional cases and requires in-depth system knowledge.

6.10 Measure Tool

To measure the tool length (**Menu: Run → Tool → Measure Tool**), a tool length sensor is required. This sensor must first be set up through a one-time calibration.

 For more information on measuring points, refer to the chapter **Positions >** Measuring Points 73.

6.11 Tool Change

The **Tool Change** function (**Menu: Run → Tool →** Tool Change 40) can only be used if an automatic tool changer and a park position 70 are available. Two types of tool changes are supported:

Manual Tool Change
The change is performed manually by the machine operator at the defined park position.

Automatic Tool Change
The change is carried out by the automatic tool changer. If the selected tool number exceeds the number of available slots in the tool changer, the change must be performed manually at the defined park position.

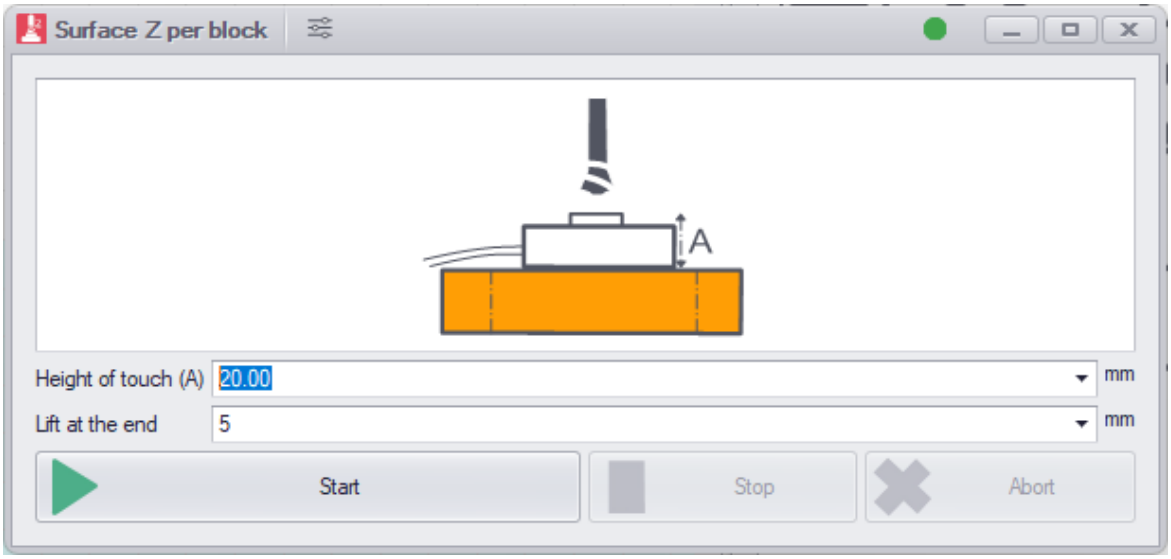

To execute the tool change, the option **Tool Change** must be activated in the **Job Parameters** dialog (F9).

6.12 Automatic Measurement of the Zero Point with Probe Block

cncGraF 8 can automatically measure the Z-height of the zero point. This requires a probe block or a 3D probe. With a 3D probe, the zero point for the X and Y axes can also be automatically determined. For more details on the 3D probe, see the chapter Probes 120.

Requirements for Probe Block

- In the menu Settings > Machine Parameters > Pin Assignment, the input for automatic measurement must be defined.
- The assigned pin can be identified via View > Interface. Press the sensor – the input will change its status, confirming the correct pin.
- The measurement speed for the Z-axis must be set very low (Machine Parameters > Speeds > Measurement Speed for Zero Point X, Y, and Z).
- The exact probe block height in millimeters must be specified (see figure).



7 Edit

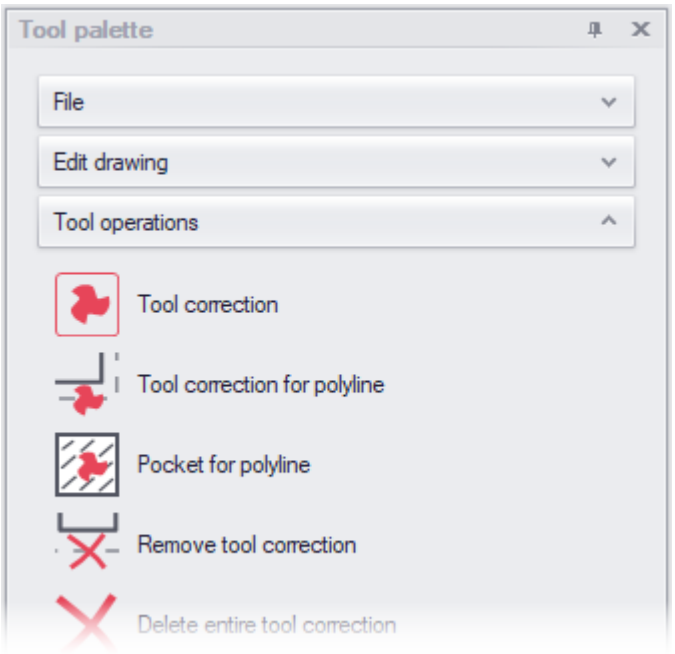
cncGraF 8 can process two types of data:

2D files (e.g., DXF)
These files do not contain machine-specific parameters such as feed rates, tool definitions, machining sequences, or tool compensation. For milling, this information must be added within the program. This approach is particularly suitable for beginners, as it allows simple 2.5D tasks, such as milling or drilling front panels, to be implemented quickly.

G-code files (DIN66025)
These files are complete machine files and contain all the required information for CNC operation. Handling them is more complex but provides full functionality and flexibility.

Since G-code files (DIN66025) are already fully prepared for the milling process, they are not edited. The functions described in this chapter apply exclusively to the editing of 2D files (e.g., DXF).

The editing functions for 2D files can be found in the **Tool Palette** window (Menu: *Window* > *Tool Palette*).



The **Tool Palette** can be customized in the menu *Settings* > *Options* > *General & Tool Palette* by hiding functions that are not required.

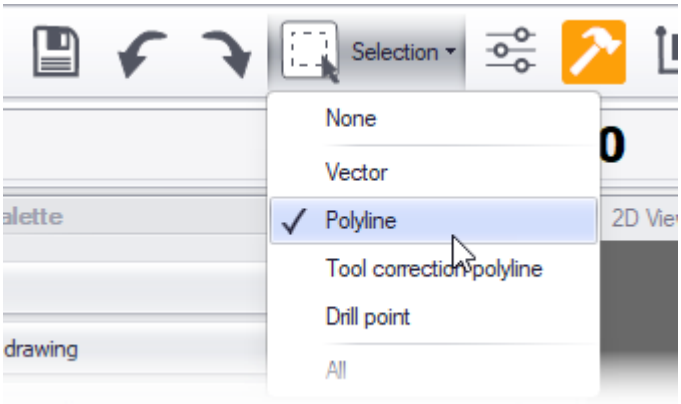
Edit Drawing

The *Edit Drawing* section provides basic functions for positioning, rotating, or mirroring a drawing on the machine surface or a defined workpiece.
Additionally, it is possible to stretch a drawing (mainly useful for testing the CNC machine) or round corners.

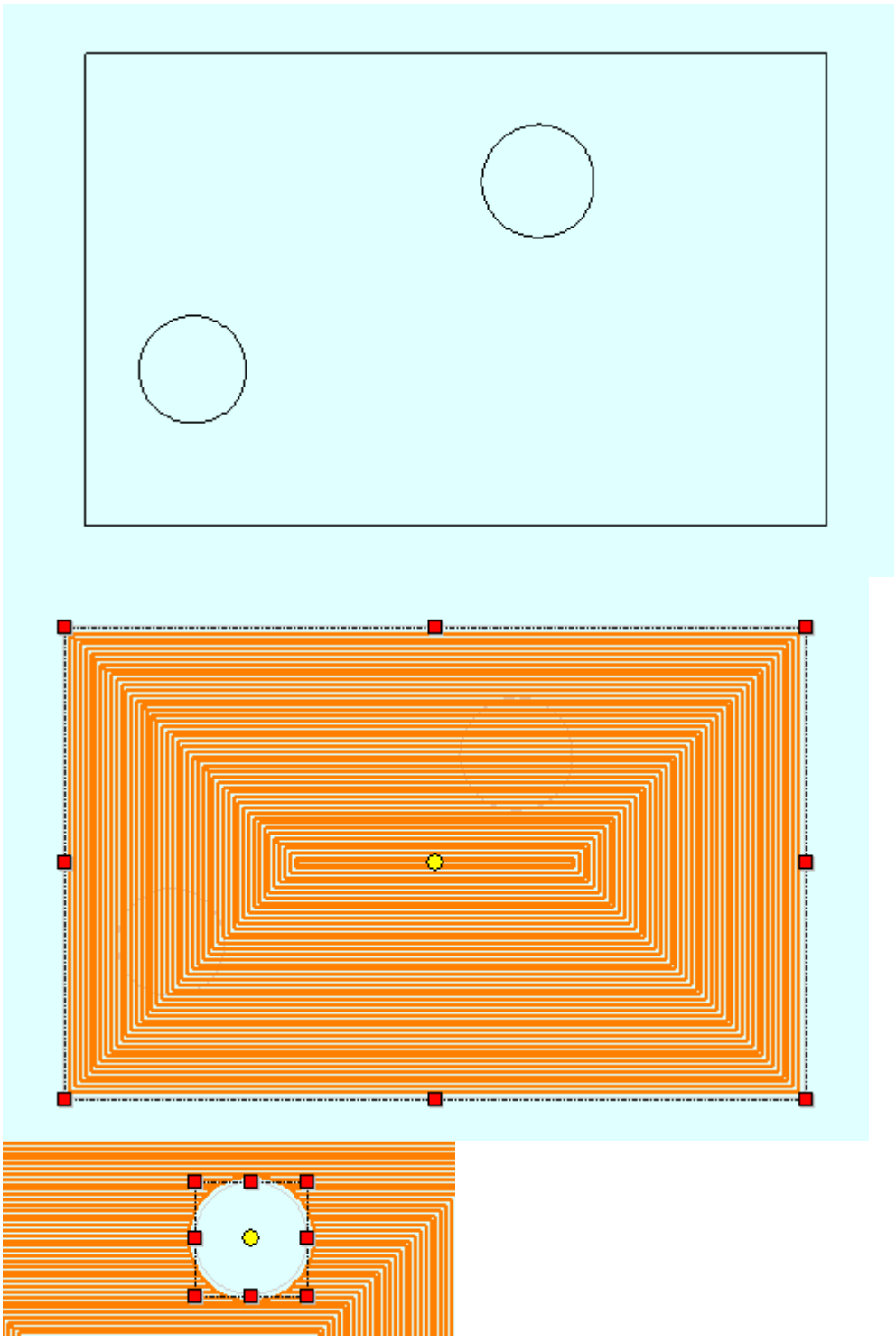
The most important function in this area is **positioning on the workpiece**, as this is usually the first step after opening a drawing.

Tools
The *Tools* section contains functions for tool operations:

Automatic tool compensation (radius compensation) can be applied to the entire file.
To apply tool compensation to polylines, select the desired polyline with the mouse using the *selection rectangle* (see Figure) and then calculate the radius compensation with the *Set* button. More information about the selection rectangle can be found in the respective chapter.



A **pocket with an island** is created by first calculating the pocket. Then, the inner polyline is selected with the mouse and confirmed with the *Clear* button (see Figures).



The tool palette also provides the **Tool Properties** function. This allows the tool number and/or milling direction to be adjusted for a selected element (polyline).

Convert points into drill points:

cncGraF 8 supports importing Sieb & Maier files. Since this format is only used by a few programs (mainly PCB layout software), cncGraF 8 provides a tool to generate drilling data directly from 2D files. For this, drill points in the 2D file (HPGL or DXF) must be drawn as points • or – recommended – as crosses +.

Simple tabs can be set by clicking the desired vector.

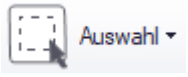
Lead-in and lead-out can be added as a line or arc. In this case, the machine approaches the contour laterally in the X and Y directions (no plunge in Z).

Machining Sequence

This section defines the processing order within a tool. Polylines can be moved to the beginning or end of the machining sequence or assigned a starting point.


Important: The *Path Optimization* function can correct errors in DXF files – such as duplicate vectors or open polylines – and optimize toolpaths.

7.1 Selection Rectangle

The *Selection Rectangle* function allows you to edit parts of a drawing (move, rotate, copy, or delete) using the mouse. First, you need to select the data type via the symbol 

in the horizontal toolbar (options: vectors, polylines, radius-compensated polylines, drill points, or all).

Afterwards, a cross-shaped mouse pointer appears on the graphical surface. By holding down the left mouse button, you can draw a rectangle that encloses the elements to be edited. Once released, the selection rectangle appears with the chosen drawing elements highlighted (see Figure).

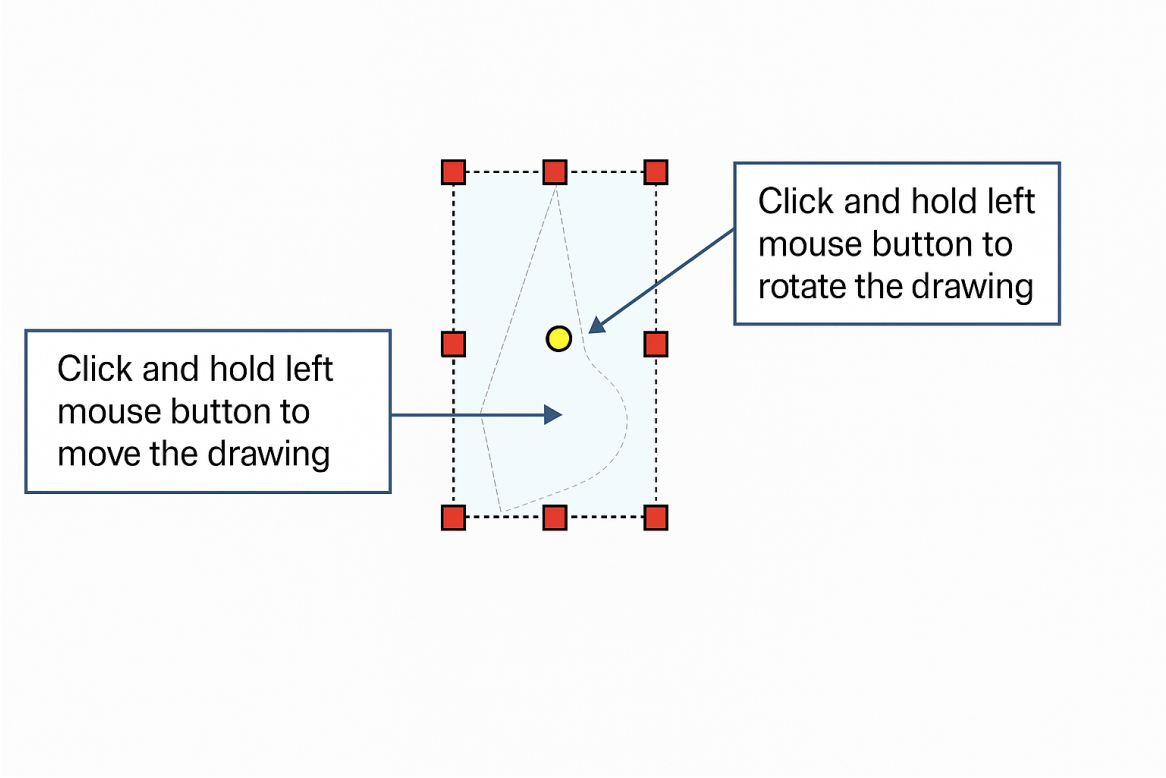


Figure: Content of the selection rectangle can be rotated, copied, or moved.

After making a selection with the mouse, you can now perform the desired action by choosing the corresponding function key and clicking the left mouse button on the workspace.

Description	*Key*	*Menu Command*
Delete	[Del]	Edit > Selection Rectangle Content > Delete

Description	*Key*	*Menu Command*
Copy	[Ctrl + C] and left mouse button	Edit > Selection Rectangle Content > Copy
Move	[Ctrl + X] and left mouse button	Edit > Selection Rectangle Content > Move
Rotate	Selection rectangle center + left mouse button	none
Change milling direction and tool number	Right mouse button > Properties...	none

7.2 Drag Knife Compensation


The drag knife compensation can only be applied to **HPGL files**.

Before a cutting operation, the orientation of the swivel knife is unknown. Therefore, it must first be aligned in the **Y-direction** (Figure 1).

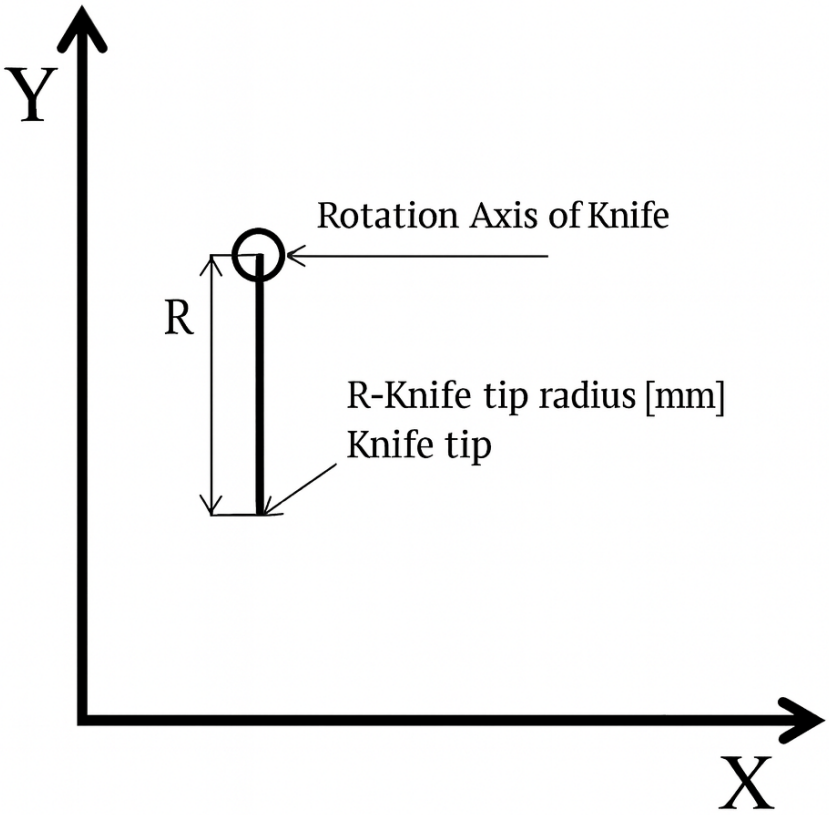


Figure 1: Starting position of the swivel knife

Immediately after the start, the knife is precisely aligned using an alignment cut in the Y-direction (Figure 2).

Figure 2: Alignment cut

Drag Knife Compensation
In the dialog window *Drag Knife Compensation*, the swivel knife radius is corrected. This window can be opened via the main menu: **Edit > Drag Knife Compensation…**

Drag Knife Parameters

- **Knife Offset:** This is the distance between the knife's pivot axis and the knife tip (knife radius).
- **Start Angle:** The start angle defines the angular difference at which the compensation path for the drag knife is calculated. The larger the start angle, the rounder the cut corners will be. A value of **8 degrees** is recommended.
- **Scaling:** With scaling, the drawing can be enlarged as desired. After the calculations have been performed, the drawing must not be scaled any further, since this would also change the knife radius.

Drag Knife Alignment

- **Offset Distance:** Defines the distance of the alignment cut from the actual contour.
- **Length:** Specifies the length of the alignment path.
- **Align:** If this option is checked, the machine will perform the alignment cut.usrichtungsstrecke.

 The *Offset Distance* value should be chosen so that the alignment cut lies **outside of the contours**, otherwise the actual contour could be damaged!

Once all parameters have been set, you can confirm them with **OK**, and cncGraF 8 will calculate the cutting paths.

 When aligning the Z-zero point, be sure to consider the cutting depth of the foil (or cardboard).

7.3 Nesting

The **Nesting** function (Window: *Tool Palette* > *Extras*) arranges 2D elements in a space-saving way on a workpiece. Multiple files can also be added simultaneously for nesting.

7.4 TeachIn

TeachIn (Window: *Tool Palette* > *Extras*) is a tool used to move the machine to specific positions and then save these positions in an HPGL file or a G-Code file (DIN66025).

After opening the TeachIn function, the desired positions can be reached using Manual Move 121 and added to the output list via *Add*.
Once all points are entered, the output list can be saved as an HPGL or G-Code file.

8 TCP Server/Client and Modbus TCP Client Communication

To send or receive data via TCP or Modbus TCP, data channels must (Menü: *Extras -> Admin of data channels*) first be defined. These channels are divided into receive channels and send channels. An example of a send channel is *"Send CNC machine X position"*, which transmits the current X position. A receive channel could be *"Execute 'Start Job' once the value in column 'P' is received."*

Note: For send channels, it is recommended to use the transmission type *"On value change"* to avoid unnecessary traffic caused by multiple transmissions.

Configuration of TCP and Modbus Clients/Servers
In the corresponding dialog window, TCP servers or TCP/Modbus clients can be created. Filters allow you to link previously defined data channels with the connections.

Important: The transaction ID is unique for each request (usually incremented sequentially). It assigns incoming responses to the correct data channels and is therefore particularly necessary when receiving commands.

TCP/IP Server/Client
- Packets (strings) of variable length are transmitted.
- The end of a packet is marked by a delimiter (e.g., <EOF> or \r\n). The delimiter can be chosen freely.
- Identifiers can be added at the beginning or end of a packet, e.g., X= for the X position.
- Example: X=147,141<EOF>

Modbus TCP (Client only)
- Modbus packets such as Holding Registers or Coils are used.
- Each packet contains a transaction ID (ushort), which is required for uniquely assigning the response.

9 Integrated Text Editor

The internal text editor is a fully functional editor that can load G-Code (DIN66025) and HPGL files, offering the following features:

- **Syntax highlighting** for G-Code (DIN66025), with commands marked in blue.
- In the text editor, DIN66025 files can be extended with additional commands using **autocomplete**. Press **CTRL + Space** to open the autocomplete menu (see Figure 1).

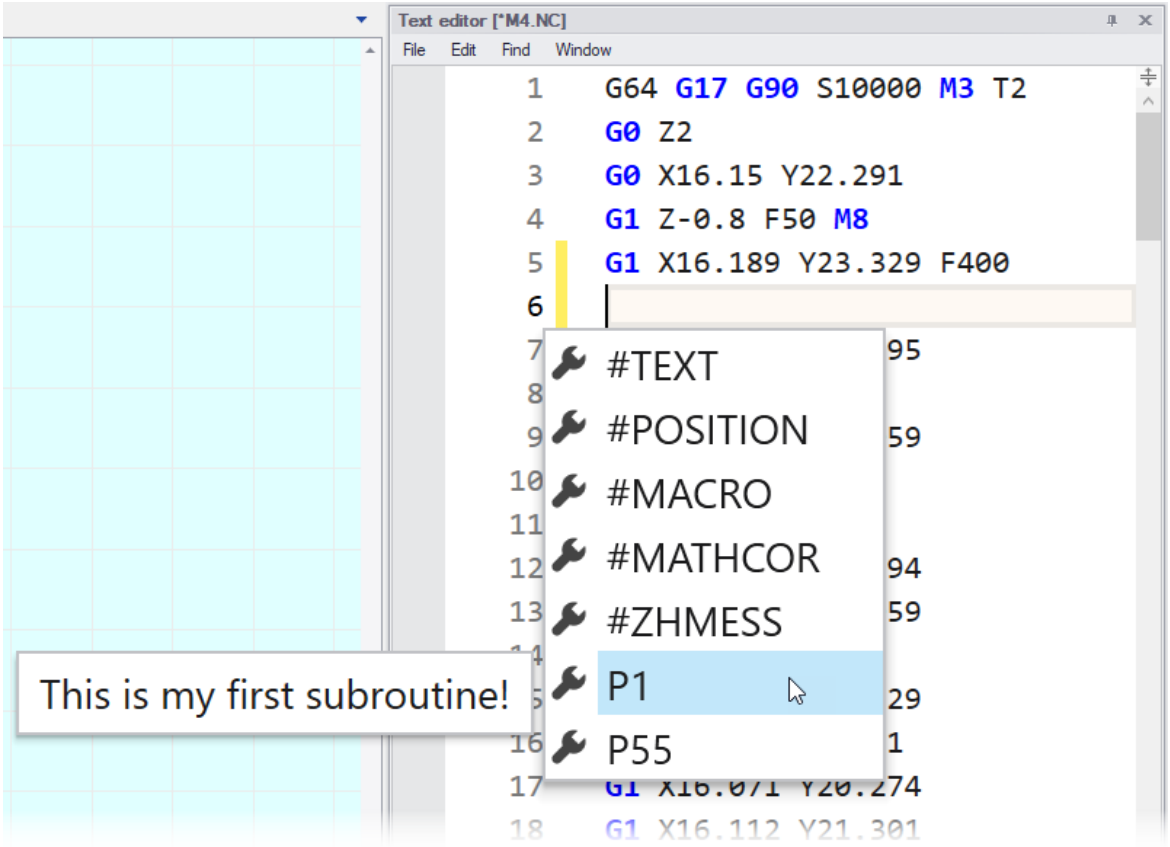


Figure 1: Autocomplete was triggered in line 8 using **CTRL + Space**, and subroutine P1 (*this is my first subroutine!*) was selected.

- In the text editor's main menu (*Edit -> Subroutines*), subroutines can be managed. Subroutines can be inserted into DIN66025 files with the help of autocomplete.

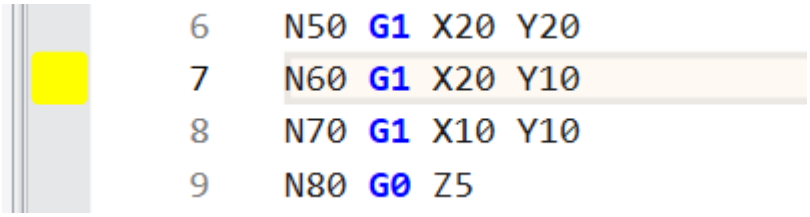


Figure 2: A pause was set at line 7

- In the text editor's main menu (*Edit -> Subroutines*), subroutines can be managed. Subroutines can be inserted into DIN66025 files with the help of autocomplete.

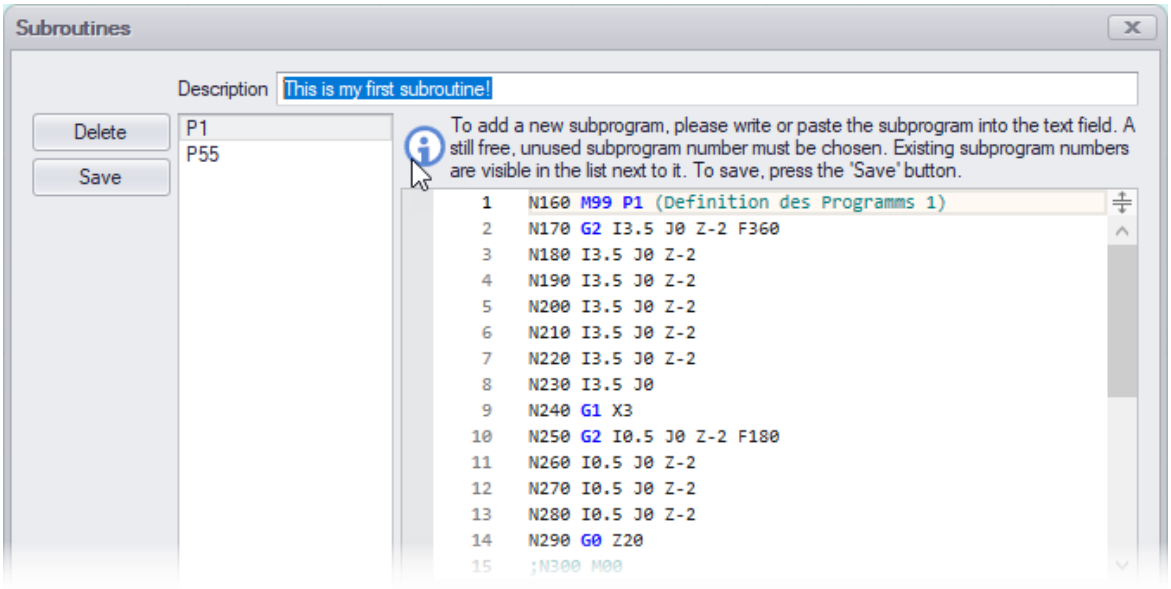
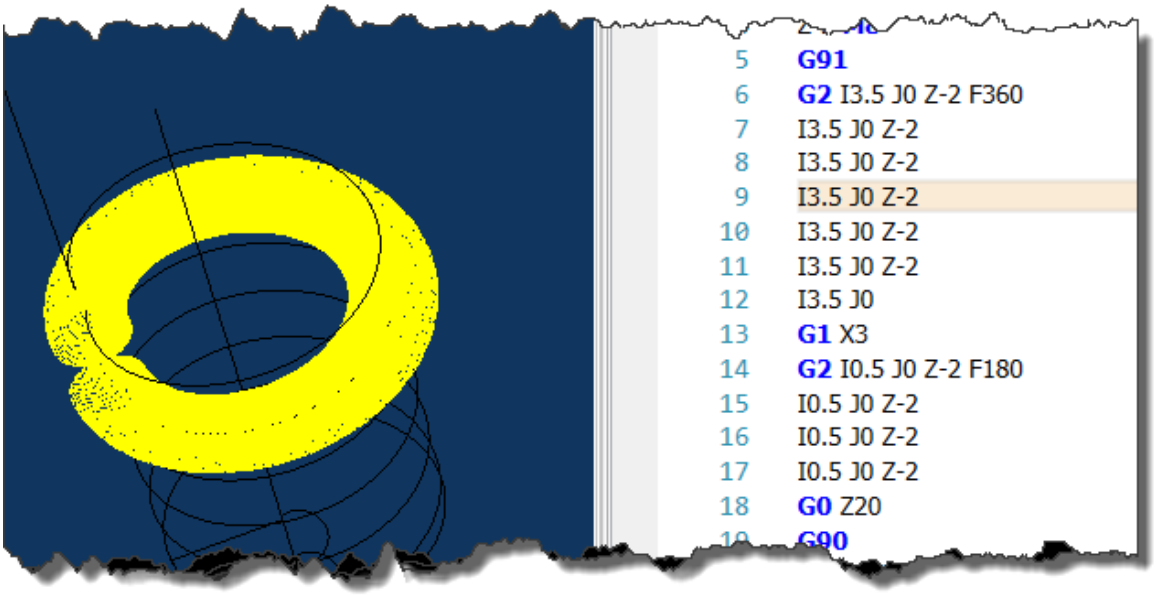


Figure 3: The description of the subroutine appears as a tooltip in the autocomplete menu

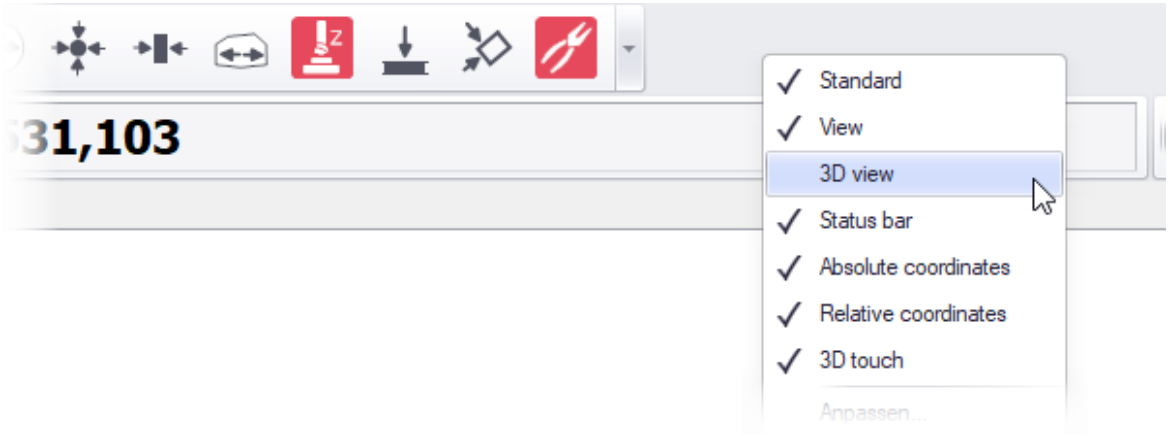
- The text editor also provides **Find & Replace**, **Go to...**, and a **Print function** with print preview (*Menu: Search and File -> Print*).
- In the 3D view, DIN66025 files can be checked. The selected line in the text editor is highlighted in the 3D view with a yellow cylinder.



The text editor is a **dockable window** and can be positioned on any side of the main window.

10 3D View

The **3D View** function makes it possible to generate a 3D model and display G-code lines in the 3D window. The function is accessed via *Window > 3D View*. Additionally, a 3D toolbar is available, which can be displayed in the menu by right-clicking.

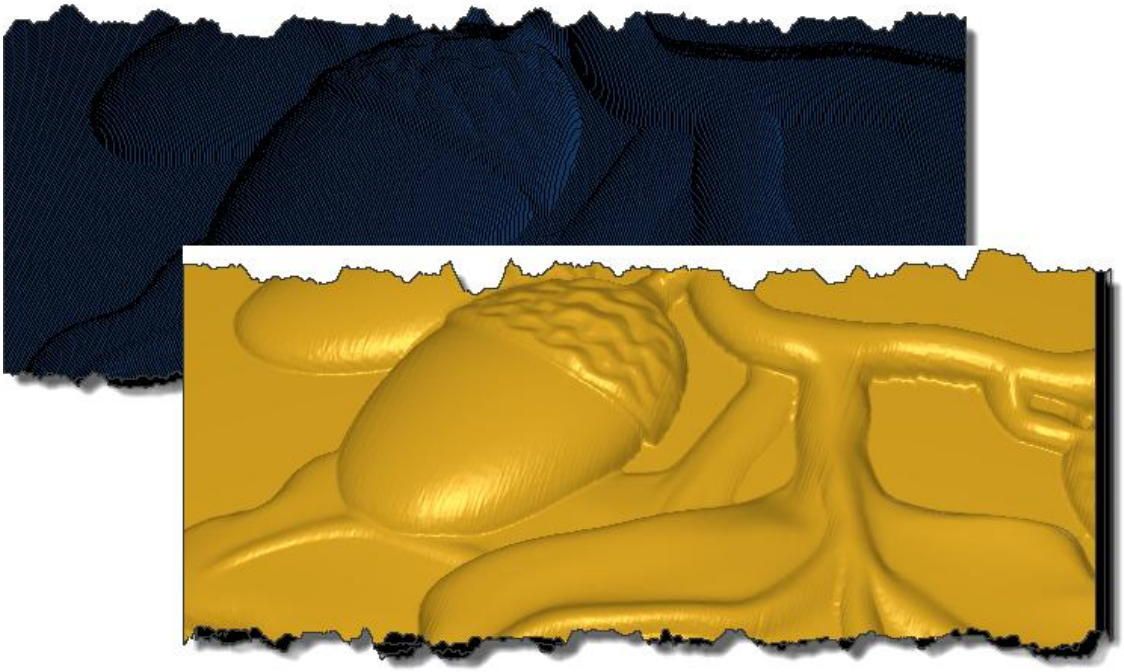


The 3D View requires a **powerful graphics card** with **OpenGL support**!

3D Model

In the 3D View, a 3D model can be generated. The following conditions must be met:

- Suitable tools (tool type, diameter) must be defined in the tool library for the loaded data.
- A workpiece must be defined for the 3D model generation. In the *Workpiece Parameters* dialog, a sufficiently large workpiece in X, Y, and Z must be specified.



DIN66025 Commands
DIN66025 commands can be verified in the 3D View. In the text editor 142, the currently selected line is highlighted in the 3D window (yellow cylinder). By pressing **CTRL + left mouse button**, a vector can be selected directly in the 3D View.

11 System Settings

All settings, except for the tool library, are stored in the configuration file **config.xml**. The configuration file is an XML file and can be opened in any text editor. In the main menu, the configuration file can be loaded via *Settings → Settings → Load...*.

The **config.xml** file is located in the directory: ...\Users\[BENUTZERNAME]\AppData\Roaming\cncGraF8\.

The configuration file contains several parameters that are not available through dialogs. To change these parameters, open a previously saved **config.xml** file in a text editor.

Save your cncGraF 8 settings by selecting *Settings → System Settings → Save*. Keep the generated system file in a secure location (not on the same PC, but for example on a USB stick or CD-ROM).

12 Fehlerbehebung

Error	Possible Solution
Error: No connection to the control electronics **smc5d**	- Disconnect and reconnect the USB cable. - Try using a different USB cable. - Increase the parameter *Interrupt after* in the menu *Machine Parameters → Connection*. - Switch the entire electronics off and on again. - Exit the program and restart it. - Check the Windows Device Manager to verify whether the control electronics **smc5d** are correctly recognized by the Windows operating system. In the *USB Controllers* section, **smc5d** must be listed.
During the homing process, at least one axis moves slowly away from or onto the switch	The homing direction is set incorrectly and must be changed. For correct adjustment of the homing switches, use the *Homing Switch Assistant*. For more details, see the chapter Homing Switches 31.
Machine makes noise but does not move	The stepper motors are receiving insufficient current. The motor current must be inverted. For further details, see the chapters Pulse Settings for the Drivers 45 and Motor Check 45.
Machine runs fine at first, but after a few minutes the connection drops or error messages occur	The PC may be shutting down after a few minutes. Disable the power management in the BIOS. Also check the power options in the Windows Control Panel.

Important: To prevent the PC from causing interruptions, power management in the BIOS must be completely disabled. Also review the power options in the Windows Control Panel.

Index

- A -

- C -

- D -

- E -

- G -

- H -

- I -

- J -

- K -

- L -

- M -

- N -

- P -

- R -

- S -

- T -

- U -

- V -

- W -

- Z -