Basic User Manual Eding CNC Software GUI

Rev.4 - Released

Document History

Rev.	Date	Author	Comment
4	11-may-2020	AB	Release, initial version.

Table of Contents

Table of Figures

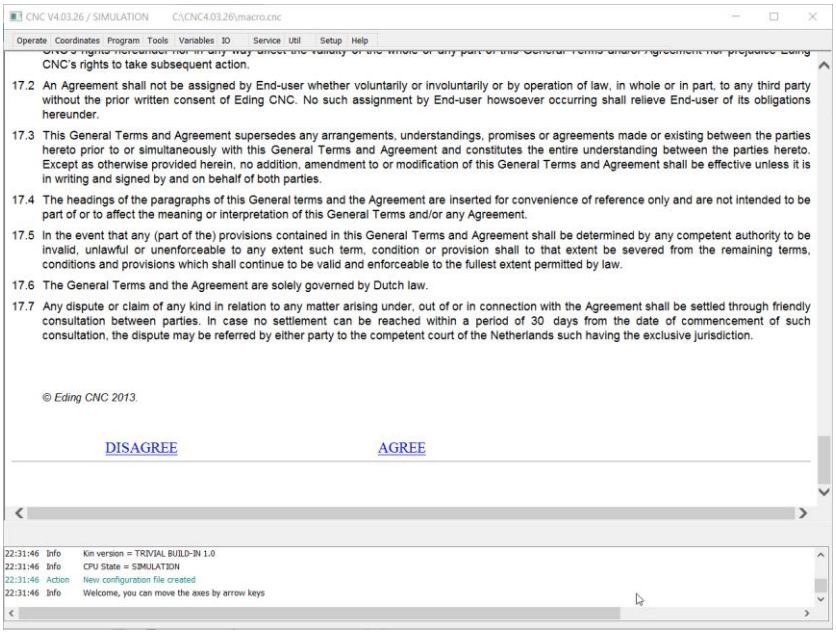
1 Scope

This manual describes how to use the EdingCNC application. The application itself is multilingual but in this manual, it is assumed that English is used as language. This document will focus on how to use the application for day-to-day operation, certain advanced features and how to setup the application will be explained in separate manuals.

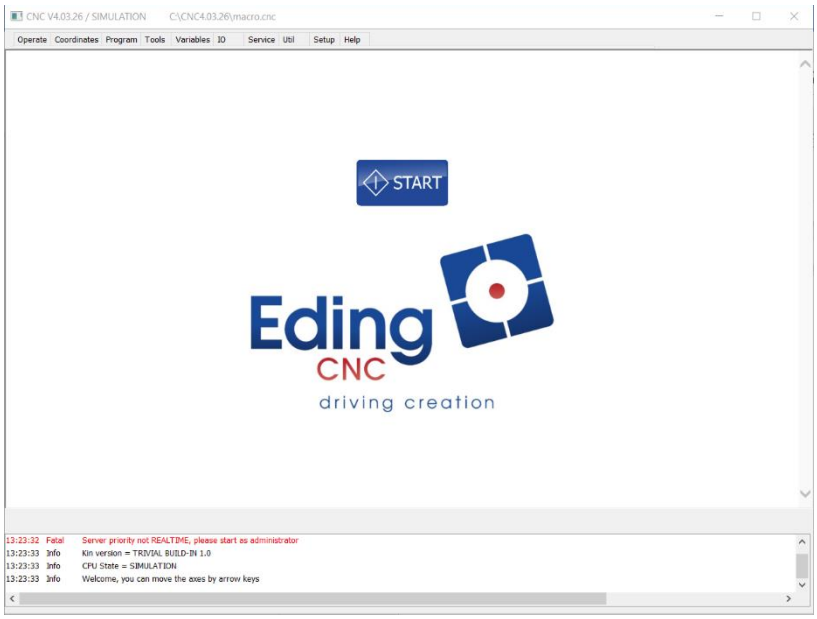
2 Getting started

2.1 First time use.

The first time the application is started the screen shows the terms for using EdingCNC.
If you agree, simple click on ‘AGREE’ at the bottom.



For your information, this screen might also appear after you have updated the application. Next, you will see the start screen:



Press  to continue to see the main operate screen.

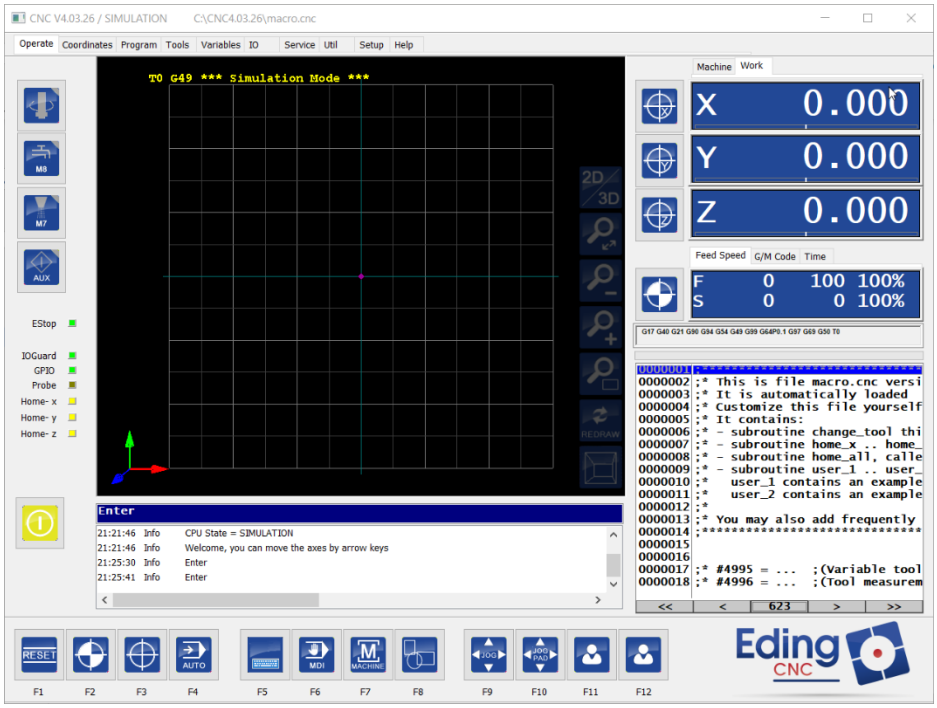
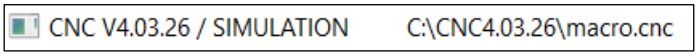


Figure 1 Main operate screen

At the top of the application information about it is displayed:

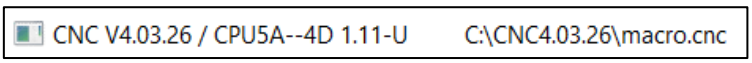
Figure 2 Software and hardware information.

This information contains information about the application version, the connected hardware, connection state and the loaded job file.



The first part “CNC V4.03.26” is the actual version of the application, the second part “Simulation” indicates that no hardware is connected, and the application is simulating connected hardware. Please note that this also means that there cannot be proper evaluation of inputs, eg. the E-STOP input, since these are actual hardware inputs of the controller.

If a controller is connected it will show this by indicating its name, for example if a CPU5A is connected:



Instead of the text ‘SIMULATION’ it is now replaced by the controller type and its firmware version (1.11).

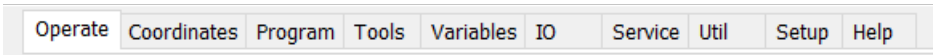
Finally, you are informed about which file is loaded and where it is located. In this case the file ‘macro.cnc’ is loaded. This file is a special file that is part of the application and contains information about the behavior of the machine eg. the homing procedure.

For now, it is only important to understand that this file will always be (re)loaded when a job is loaded. If an actual job is loaded, that filename will be shown; however, macro.cnc will also have been (re)loaded.

In other documents more information will be given about THE ‘macro.cnc’ file and how this can be used. In the next chapter(s) we will discuss the main operation screen in more detail.

3 Tabs pages overview

At the top of main operations window several tabs are visible. These tabs offer access to different screens of the application.



Some screens will only be discussed briefly because the goal is to inform you about how to use the software. The screens that are not discussed in this manual will be explained in more detail in separate manuals.

Navigation of the tabs

The different screens can be reached by clicking on each tab, but they can also be navigated through the keyboard.

Keyboard keys	Function
CTRL + TAB	Select tab to the left
CTRL + SHIFT + TAB	Select tab to the right
ALT + O	Select Operate tab

3.1 Operate screen

The ‘Operate’ screen is the main screen of the application. In this screen G-Code can be loaded and executed. Also controlling the machine is performed here.

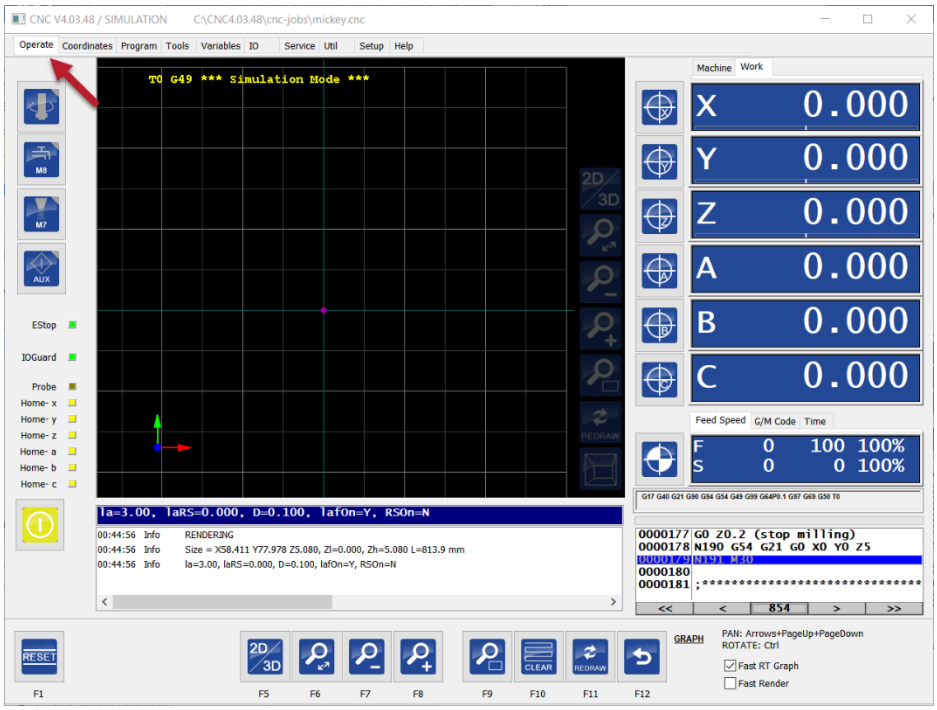


Figure 3 The 'Operate’ screen, the main screen of the application.

3.2 Coordinates screen

This screen contains buttons for controlling the coordinate systems, Z-height compensation, teach in, and more. Please note that the information that is indicated by the blue rectangle will only be present if the A-axis is selected in the '4TH MILL' mode.

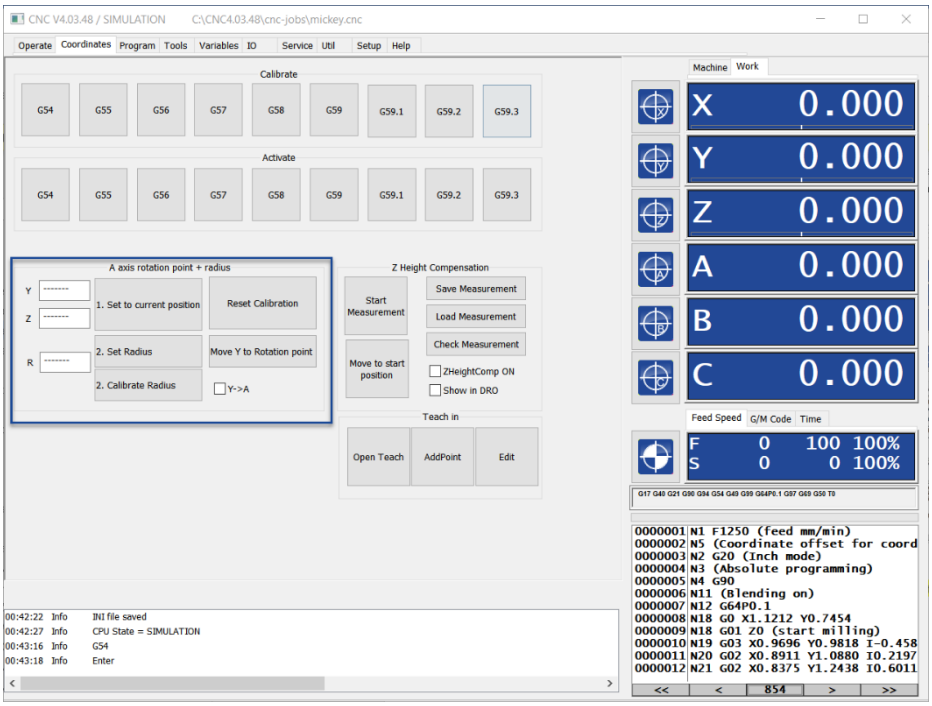


Figure 4 The coordinates screen.

The use of the coordinates screen is explained in a separate document.

3.3 Program screen

This screen can be used to load a DXF or PLT file and generate G-Code based on the operation you want to perform. This is a very basic form of CAM software; it is only intended to do simple things.

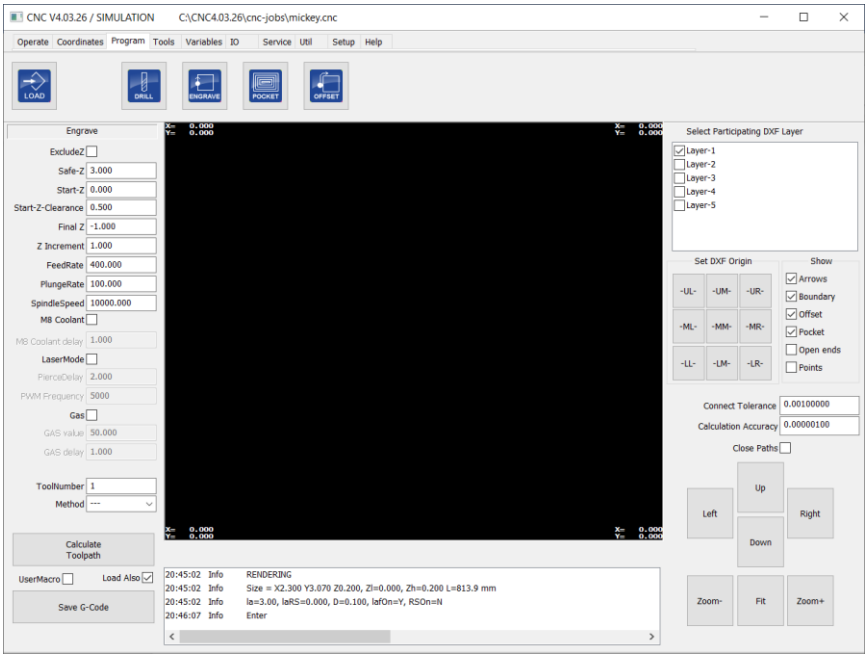


Figure 5 The program screen.

The use of this program screen is explained in a separate document.

3.4 Tools screen

This page shows a list of the tools of the machine and the offset for each tool.

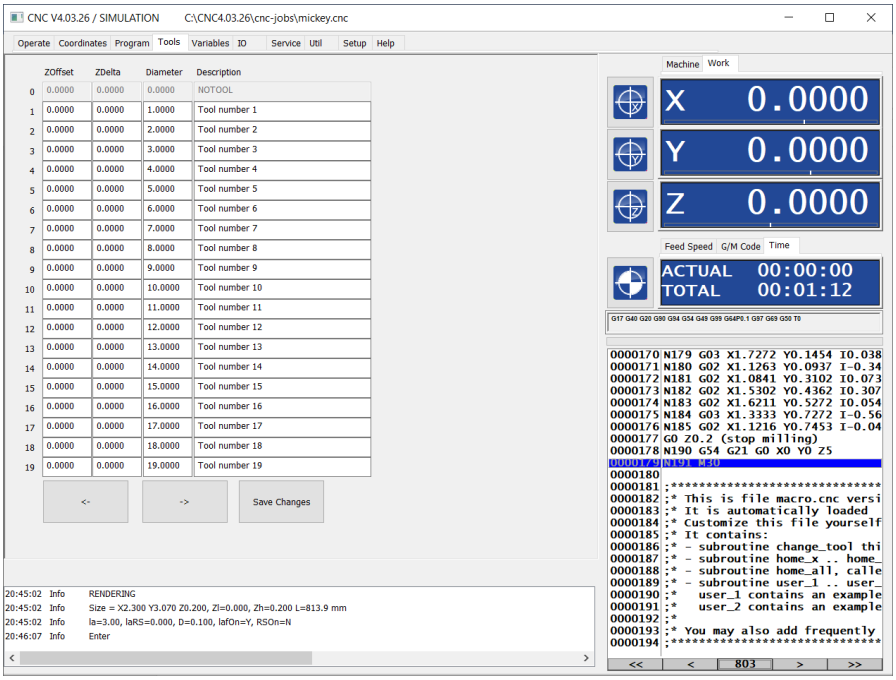


Figure 6 The tools screen.

The use of this screen is explained in a separate document.

3.5 Variables screen

This screen shows numerous variables used by the G-Code interpreter. It also contains 4 rows of ‘watches’ to show specific variables, which can be very helpful in debugging your own written macros if you are going to use the extended programming features. Furthermore, it contains numerous settings used by the system / interpreter.

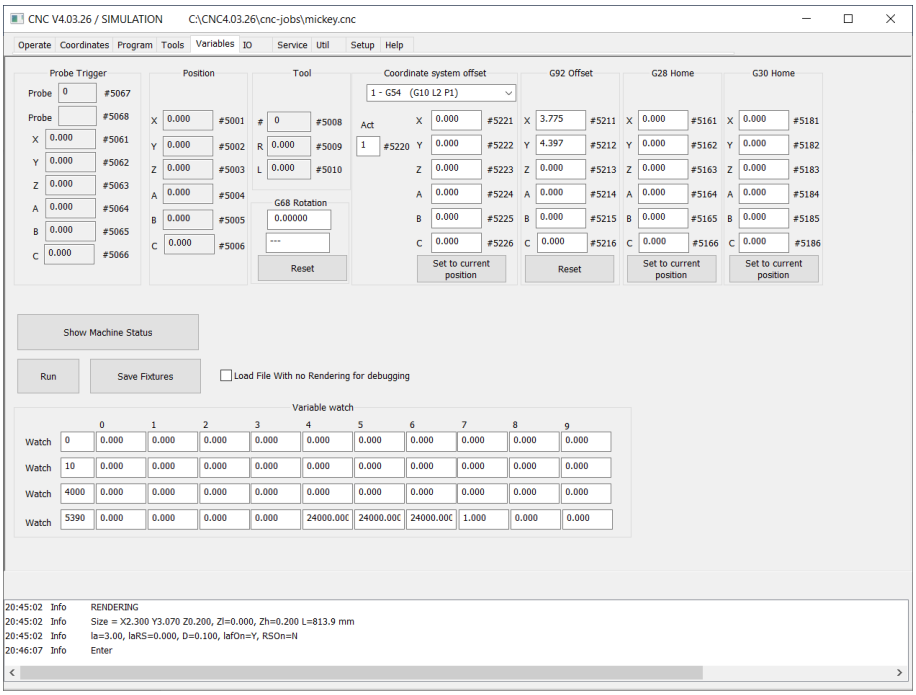


Figure 7 The variables screen.

The use of this screen is explained in a separate document.

3.6 I/O screen

The I/O screen provides an overview of the different available I/Os of the controller. The input states are shown, but you can also manually control outputs of the controller.

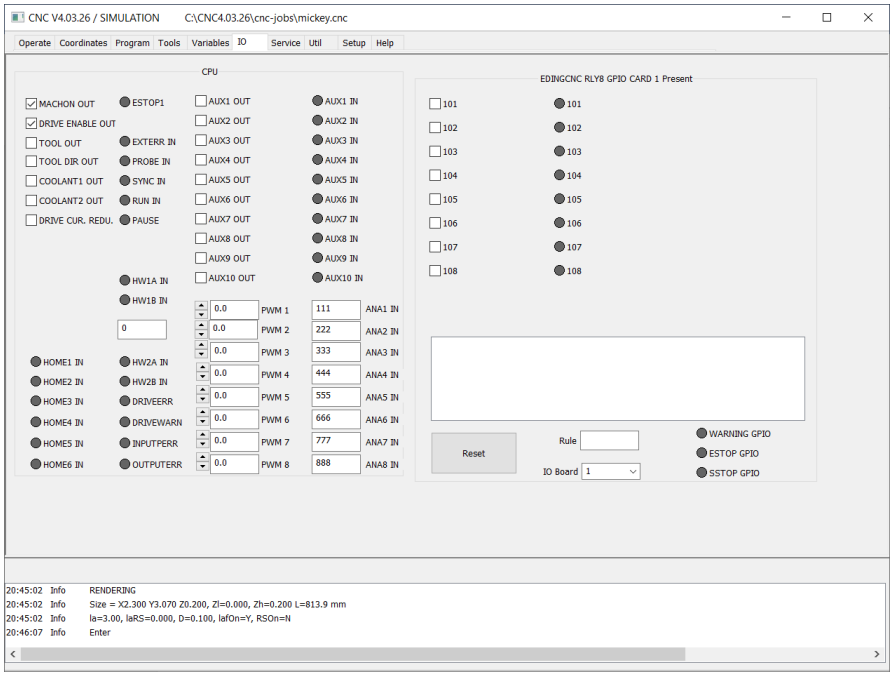


Figure 8 I/O screen.

By clicking on the checkboxes you can activate an output. Inputs are indicated by yellow indicators. In simulation mode these are not indicated. PWM outputs can be modified from 0 to 100%, and finally the value of the analogue inputs are shown.

On the right side of the tab I/O expansions are shown, its functionality is based on the I/O module that is connected.

3.7 Service screen

The service screen shows how much your machine has been operating. It can help in deciding whether the machine needs service. Also, it is possible to configure settings for automatic control of a pump for specific applications. More about this can be found in “Chapter 8 The ‘Service’ tab in more detail”.

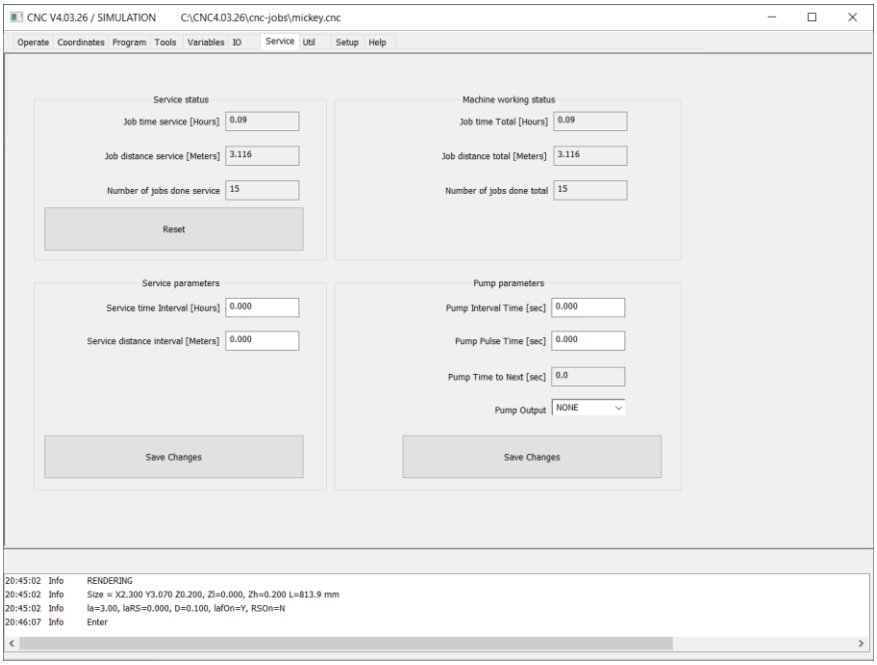


Figure 9 The service screen.

3.8 Util screen

The util screen contains helpful tools for quickly calculating the feed for a milling application.

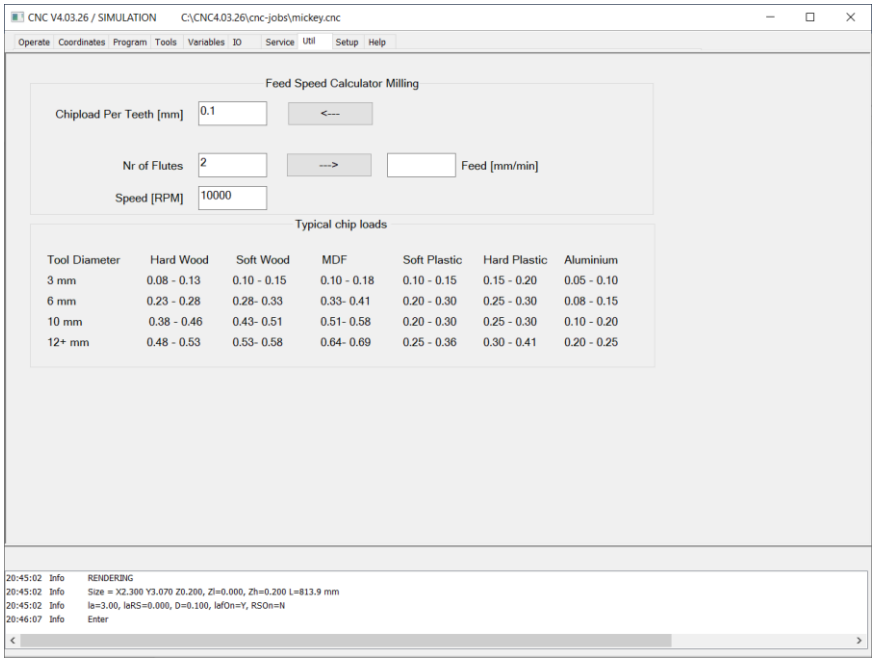


Figure 10 The util screen.

It allows to calculate the right Feed/Speed for milling. '***Chip load***' is the quantity of material that is removed by one teeth of the milling tool.

This is the most important parameter for calculating the feed given a speed. This tab is given as-is, it is important that you understand how to select the proper tool depending on the material you are working on.

3.9 Setup screen

This screen consists of multiple pages that are used for setting up the system and configure the behavior of the system.

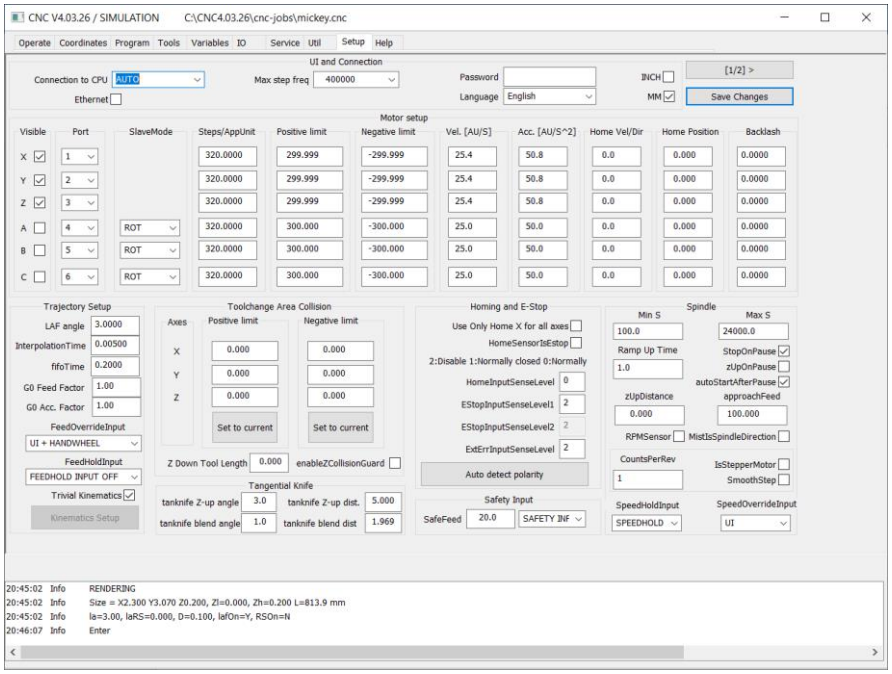


Figure 11 The setup screen (page 1 of 2).

Only change settings if you know what you are doing. Incorrect settings can lead to damage to the machine. This screen will be discussed in a separate document.

3.10 Help screen

The help screen gives a summary of the navigation buttons at the bottom of the screen and information about the support G and M codes.

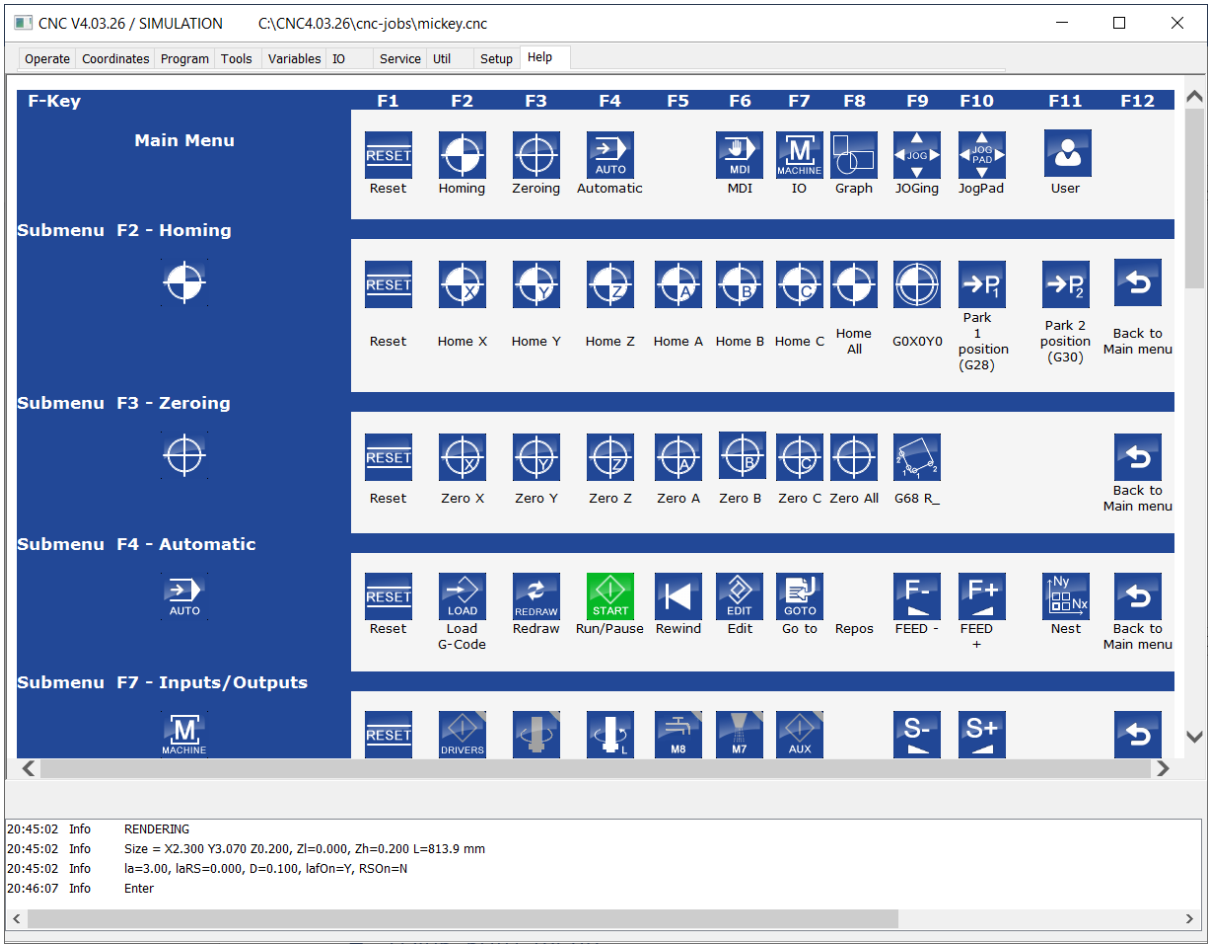


Figure 12 The help screen.

4 The ‘Operate’ screen in more detail

The ‘Operate’ screen is the screen that will used most of the time. In this chapter we will explain how to navigate and use the options that are presented. In the image below you see an overview of the operate tab screen.

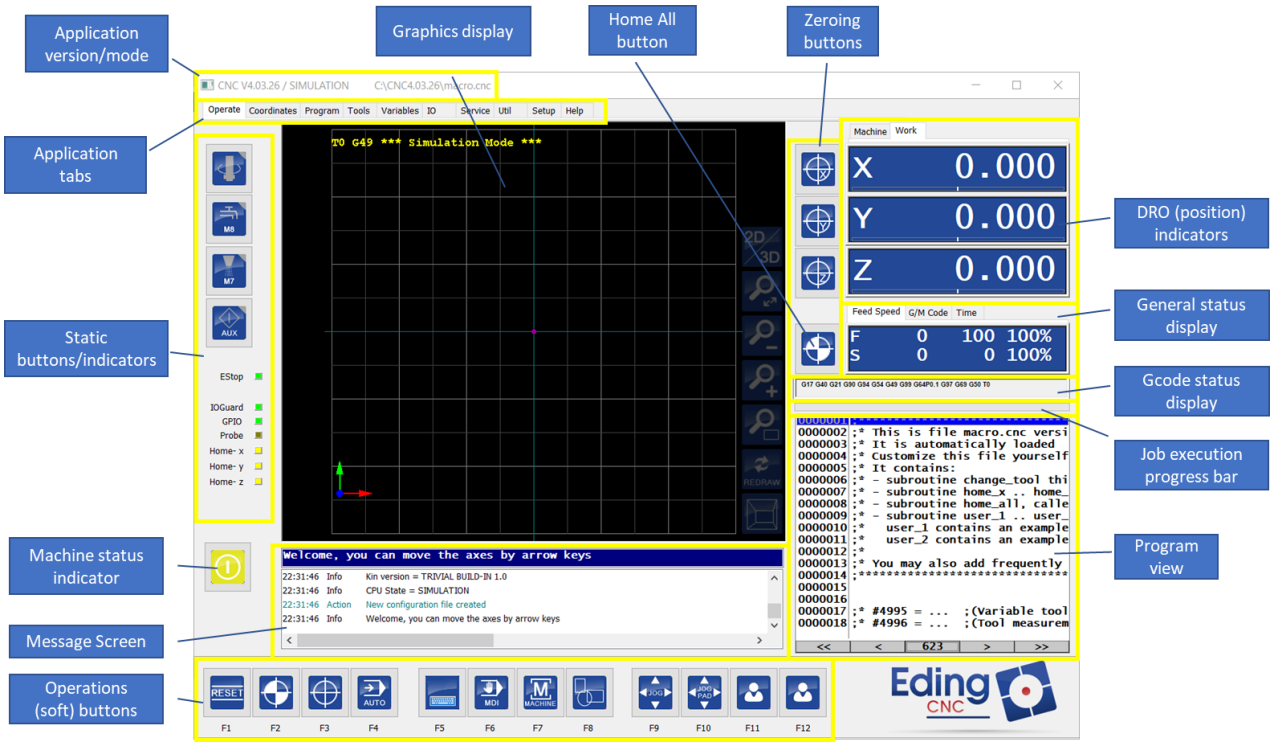


Figure 13 Operate screen.

NOTE. This is the default screen, based on different configurations this can vary. For the purpose of explaining the usage and how to navigate, this standard setup is shown.

The operate screen can be operated with a mouse, keyboard only or touch screen.

4.1.1 Static buttons/indicators

This part of the application shows indicators and buttons to provide the user with direct control and information. This indicators and buttons can be changed either via customizations or depening on how the application is configured eg. milling, 3D printing or plasma cutter operation.

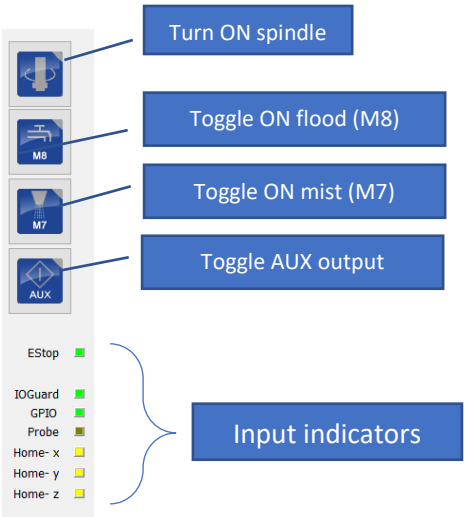


Figure 14 Static buttons/indicators.

4.1.2 Operations (soft)buttons

At the bottom of the screen ‘soft buttons’ are visible. The give the user functions that can be executed directly or will show a more selection of options. These buttons can be operated by mouse but can also directly used by pressing the function-keys that are visible below each icon.

4.1.2.1 Main menu

The main menu is the main menu, it is the top-level menu.

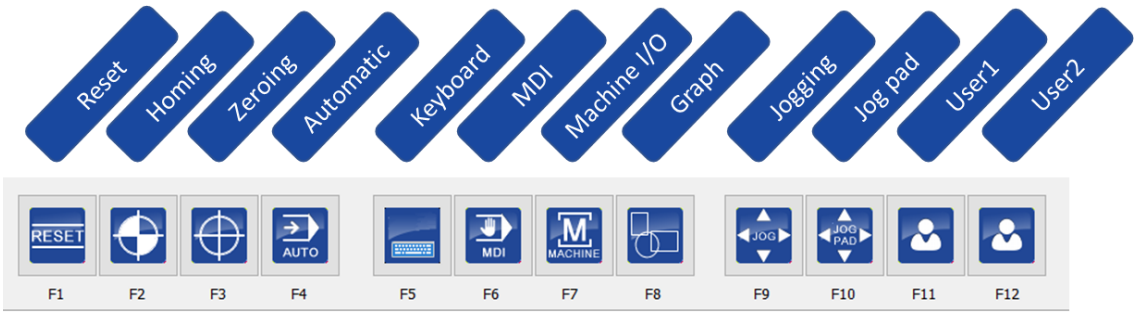


Figure 15 Main menu.

A very important button is the RESET button. It is used after starting the software to activate the system and enabling the drives.

NOTE. The amplifiers are switched ON when pressing the RESET button, you will not be able to rotate the motor shaft if the motor driver is enabled. If you still able to rotate the motor shaft by hand, this indicates that you probably need to invert the motor driver enable signal in the setup.

The RESET button is also used in other situations, e.g. recovering from an error, or stopping a running program.

4.1.2.2 Homing menu

The homing menu offers the possibility to execute a homing procedure for one or more axis.

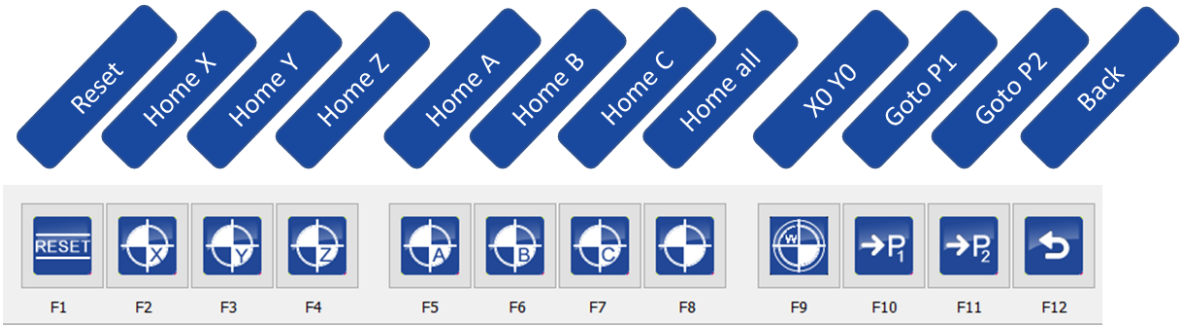
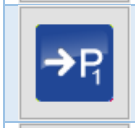
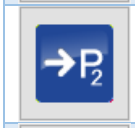


Figure 16 Homing menu.

NOTE. In the image above 6 axes are shown, if your machine has been configured with less axis only those axes will be shown.

Button	Function
	Execute a reset command.
	Start homing procedure for the indicated axis.
	Automatically start homing sequence for all axes.
	Move to position X=0 Y=0, these are the ***work*** coordinates.
	Go to park position1 (G28).
	Go to park position2 (G30).
	Return to previous menu.

NOTE. The way homing is performed, and in which sequence can be configured. The homing procedure is defined in the macro.cnc file. More about this can be found in the manual about macro programming.

IMPORTANT. Running a program and fast jogging is ***only*** possible after the machine is correctly homed, so this must be setup first. The reason is that ***collision prevention (MCA) is not active*** when the machine is not homed, so damage to the machine may occur when homing is not performed. For your information, in the setup screen the machine can be configured whether homing is required by the user.

4.1.2.3 *Zeroing menu*

The zeroing menu is intended to reset the position of the axis to the current position.

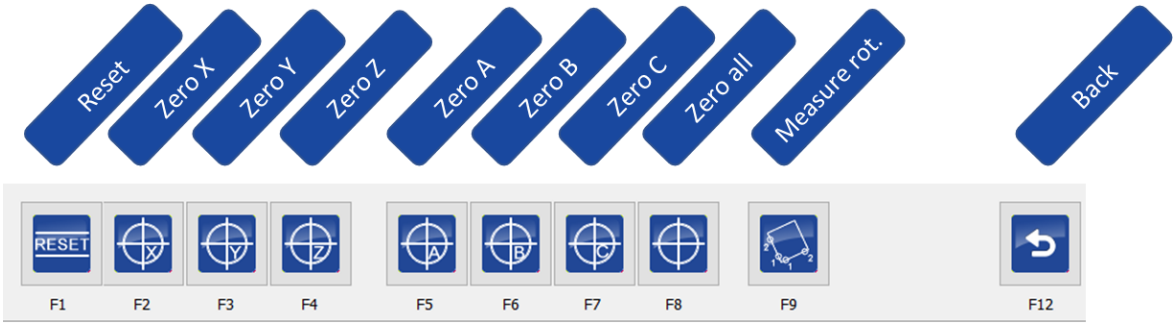
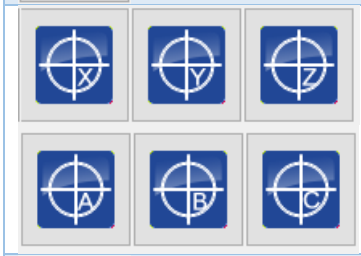


Figure 17 Zeroing menu.

NOTE. In the image above 6 axes are shown, if your machine has been configured with less axes only those axes will be shown.

Button	Function
	Execute a reset command.
	Set ***work*** coordinates for the indicated axis to zero.
	Set ***work*** coordinates for all axis to zero.
	Measure rotation (G68).
	Return to previous menu.

NOTE. If enabled, the zeroing buttons can also use a 'G10 L20' for zeroing the currently active coordinate system (G54-G59.3) instead of the global G92. More about this is explained in a separate document.

The coordinate that is set to zero refers to the **work** position, never the machine position. See also the chapter explaining the DRO. These zero axis buttons can also be found next to the DRO.

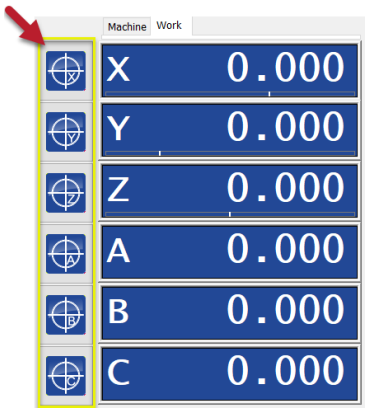


Figure 18 Zero buttons next to DRO.

Measure rotation

'Measure rotation' is a feature that is very helpful when using material that is not completely aligned in parallel to the axes. It automatically corrects the work piece/clamp for rotation. This means that it is no longer necessary to spend time to setup your clamp / material very accurately, the software will automatically correct this for you.

By selecting two points on your material the system will add a rotation offset for the loaded G-code. More about this feature will be explained in a separate document.

4.1.2.4 *Automatic menu*

The automatic menu is used to load your program and control its execution. Besides several buttons, it features several settings that can be applied on how the on how your program is executed.

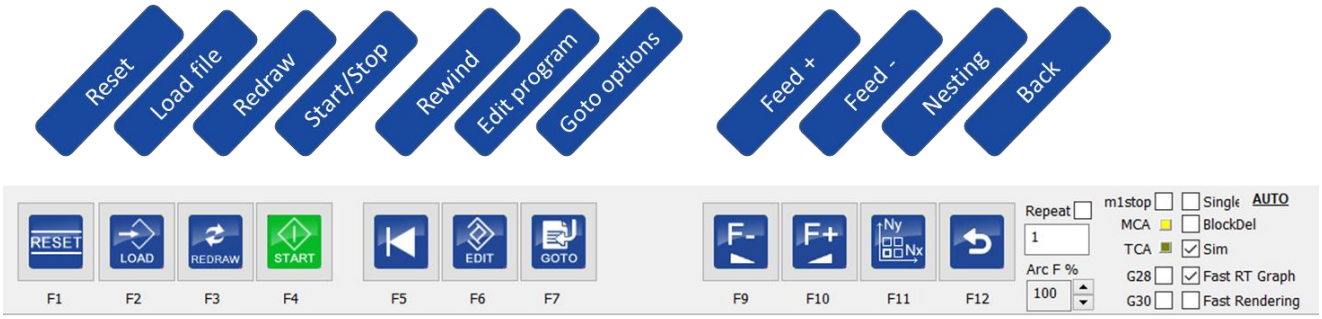
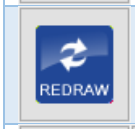


Figure 19 Automatic menu.

Button	Function
	Execute a reset command.
	Load your program.
	Redraw the output on the graphics window.
	Start/pause the execution of your program.
	Rewind the program to the start.
	Open an editor for editing you program directly.
	Shows options to jump to a position in the program. For more info, see chapter 6.
	Decrease FEED with 5%. If this button is clicked with **CTRL** button held this will increase **SPEED instead of FEED**.
	Increase FEED with 5%. If this button is clicked with **CTRL** button held this will de **SPEED instead of FEED.**
	Shows the nesting options. For more information see ”Using ‘nesting’ menu

	Return to previous menu.

Start / Pause button

When the 'Start' button is pressed it will change its color and designation to 'Stop'

Start job Pause job

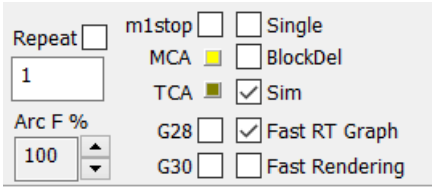
Although the text on the button says 'Stop' it will only pause the execution of the program. By pressing 'Start' again the execution will continue where it was stopped. This can be useful if you would like to simply pause the execution, move the spindle away to do something and continue. If the position has been changed during pause, the system will first return to its 'pause' position before continuing.

Pressing 'Reset' during the execution or pause will completely cancel the execution of the program and it will rewind to the start of your program.

NOTE. Pressing the ESC key during the execution of you program will also pause it. Pressing it twice will stop and rewind the execution of your program. The ESC key is ***not*** a replacement for an external E-STOP switch.

Additional settings 'AUTO' menu

On the left of the 'AUTO' menu there are more options and indicators. These options can be used for numerous situations and will give extra information about the current mode.



Button	Function
Repeat	This will repeat the same program the number of times you indicate. During the execution this number will automatically decrease to indicate the progress.
Arc F %	Reduce feed for large arcs. This is a parameter that influences the Look Ahead Feed (LAF). Only change this if you understand its usage. Example of situations a user might change this can be because of the usage of a certain tool or certain material.
m1stop	Indicate to stop program execution if an M1 code is encountered in the executed program
MCA	Indicates if Machine Collision Guard is active
TCA	Indicates if Tool Collision Guard is active

G28	Go to position G28 after program is finished
G30	Go to position G30 after program is finished
Single	Activate single step mode. Execute only one line when 'Start' or F4 is pressed.
BlockDel	When active all lines with '/' in front will not be executed.
Sim	Indicates the application is running in simulation mode.
Fast RT Graph	Enabling this option will show only a limited amount of movement line pieces in the graphics window, indicated with a yellow line (See also *Loaded program* The amount of line pieces is determined by value of ***OpenGLMaxLines***. This setting can be found in the setup tab. If this option is not selected it will show all movement line pieces, which can cause more memory to be used; or can even have the program run out of memory. Enabling this option limits the amount of memory that is used. So, use it especially when running long programs (several hours or more). This function is also automatically activated when the file size of the job is bigger than ***LongFileModeCriterion*** as defined in the setup.
Fast Rendering	This is also a very useful option for very long programs. If selected only the outlines (rectangle) of the part is drawn to save memory. This is also automatically activated when the files size is longer than **SuperLongFileModeCriterion** as defined in the setup tab.

4.1.2.5 Keyboard

This will display a keyboard on the screen. This can be useful when using a touchscreen to operate the machine.

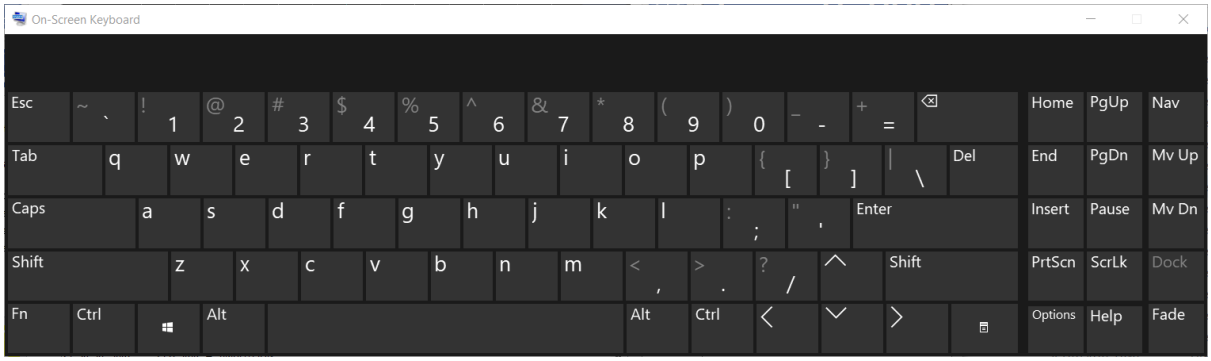


Figure 20 On Screen Keyboard.

4.1.2.6 MDI (Manual Data Input)

The Manual Data Input windows gives the operator the opportunity to enter commands directly. This can be G-code commands like ‘G1 X30 Y40’, build-in commands like ‘TCAGUARD OFF’, or call functions that are defined in ‘macro.cnc’ with the command ‘gosub <macro name>’. This last option can assist in debugging your own macro command. More about macro programming can be found in a separate document.

Figure 21 The Manual Data Input (MDI) dialog window.

This dialog will keep an history of previous used commands until the application is closed. With the cursor up or down key, previously used command can be retrieved.

NOTE. The MDI dialog can be called up at any time by pressing CTRL + F6.

4.1.2.7 Machine I/O

The machine I/O menu offers options to control outputs of the machine and the spindle speed. But also, the state of some inputs.

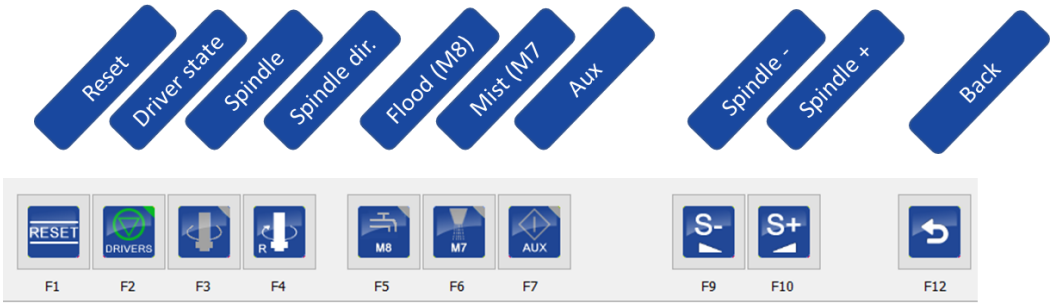
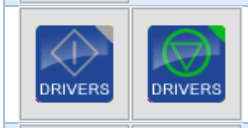
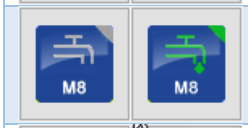
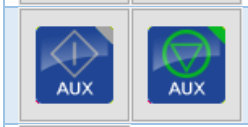
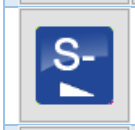
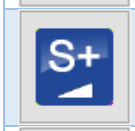


Figure 22 Machine I/O menu.

Button	Function
	Execute a reset command.
	State of the motor drivers.
	Enable or disable the spindle output (CW) (G-code M3/M5).
	Direction of the spindle CCW (G-code M4/M5).
	Enable or disable the flood output (G-code M8/M9).
	Enable or disable the flood output (G-code M7/M9).
	Control of the AUX1 output.
	Decrease the speed of the spindle with 1%.
	Increase the speed of the spindle with 1%.
	Return to previous menu.

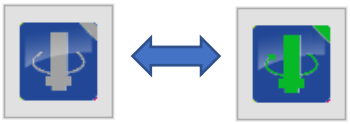
Drivers state

This button indicates whether the motor drivers are switched ON or OFF.

If the software is started the drivers will automatically be enabled. If you press on this icon or the 'Machine Status' indicator you can disable the drivers, they will then only be turned ON after the 'Reset' button has been pressed.

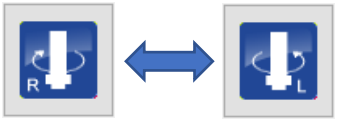
Spindle button

This button indicates whether the spindle is enabled or not. This output is also called the 'TOOL' output.



Spindle direction button

This button indicates whether the spindle is rotating clockwise (R), as seen from above or counterclockwise (L). Not all controllers have this feature.



Flood button

This button enables or disables the flood output.

Mist button

This button enables or disable the mist output. It is possible that this output is not available on all controllers. Not all controllers have this feature.

Aux button

This enables or disable the aux output.

4.1.2.8 Graph menu

The graph menu offers controls for viewing your program in the graphics display.

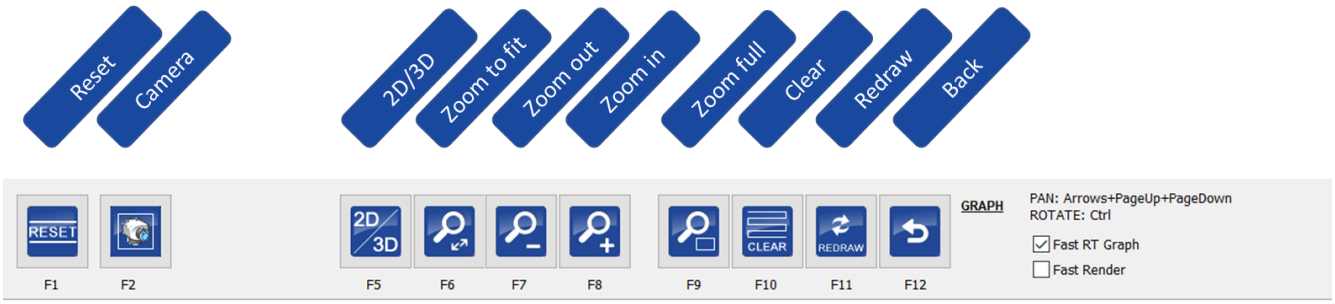
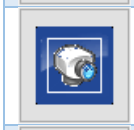
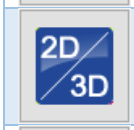
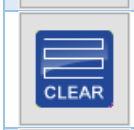
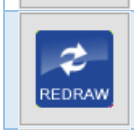
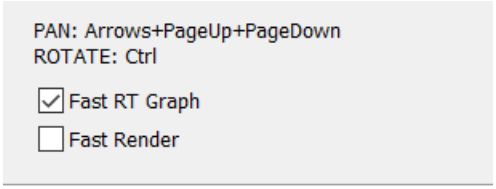


Figure 23. The graph menu.

Button	Function
	Execute a reset command.
	Show camera view (usage of the camera will be explained in a separate document).
	Switch the graphics view between 2D and 3D.
	Zoom to maximum of loaded program.
	Zoom out.
	Zoom in.
	Zoom to maximum of machine dimensions.
	Clear the graphics view.
	Redraw the output of the graphics view.
	Return to previous menu.

On the right part there are some additional parameters



More info about these options ‘Fast RT’ Graph and ‘Fast Render’ can be found here : “Fast RT Graph” and here: “Fast Rendering”.

2D/3D

This will toggle the view in the graphics viewer between 2D and 3D.

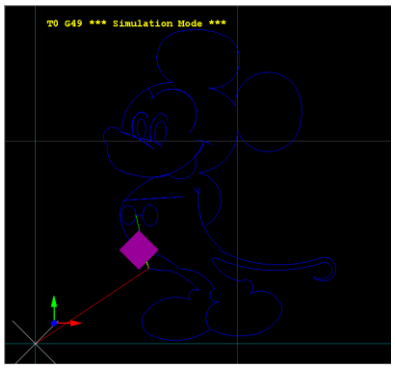


Figure 25 2D view.

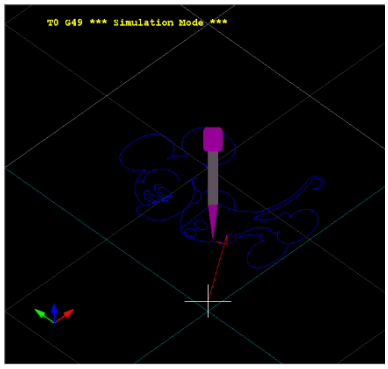


Figure 24 3D View

Zoom to fit

This will zoom till the output of your program is completely visible in your graphics view.

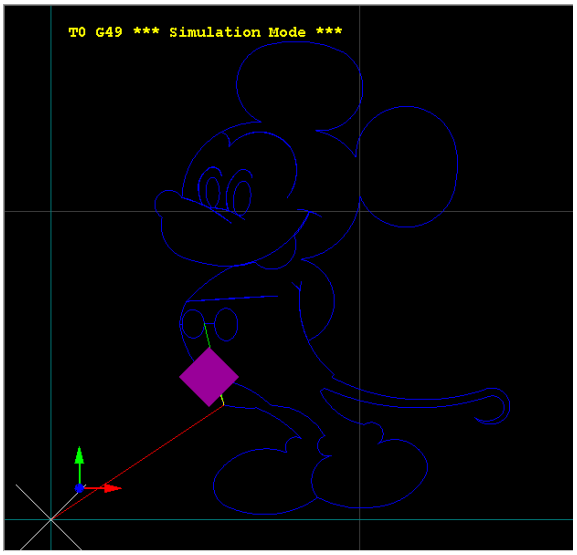


Figure 26 Zoom to fit

Zoom full

This will zoom and show the full size of the machine.

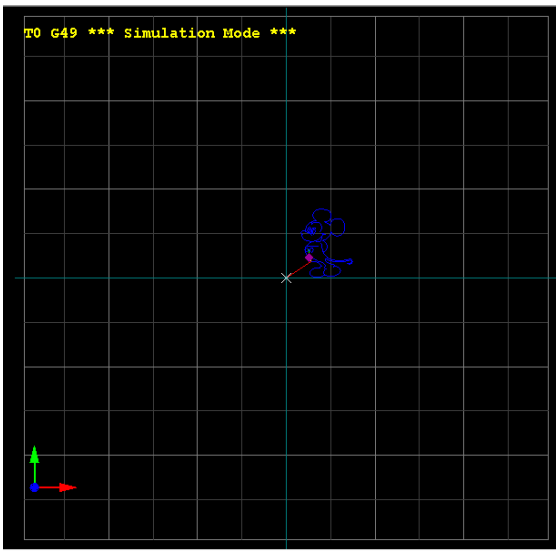


Figure 27 Zoom full

Redraw

The redraw button can be used for example, if you have moved the work home position of your machine. By pressing the redraw button the program will be shown in the graphics view on the new position. It will also always perform a 'zoom to fit'.

Cursor keys

If this menu is active, the cursor keys can be used to move around.

Key	Function
Cursor LEFT	Move view left
Cursor RIGHT	Move view right
Cursor UP	Move view up
Cursor DOWN	Move view down
CTRL + Cursor UP/DOWN	Rotate X-axis CW/CCW
CTRL + Cursor LEFT/RIGHT	Rotate Y-axis CW/CCW
CTRL + PAGE UP/PAGE DOWN	Rotate Z-axis CW/CCW

4.1.2.9 Jogging menu

Moving the axes manually is called jogging. If a keyboard is used, several keys can be used for controlling the axes. These keys are only active in the 'Operate' and 'I/O' screens.

It is also possible to use a handwheel if it is connected directly to the controller.
The jogging menu gives you the possibility to control the speed by which the axis is moved.
Based on whether the application operates in mm of inch mode either two of menus appears.

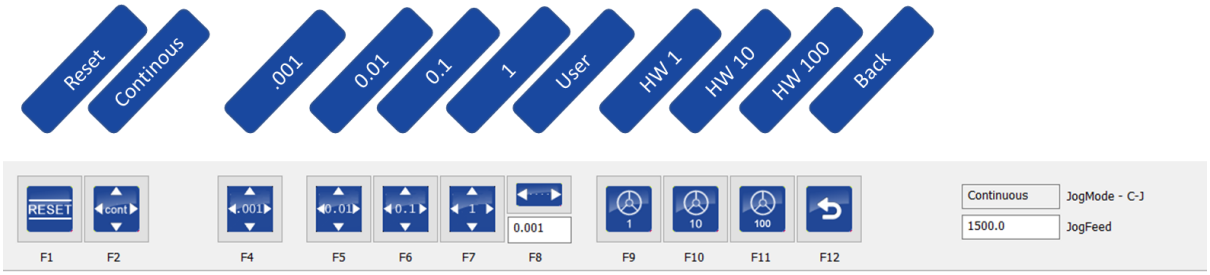


*Figure 28 Jogging menu in **mm** mode.*

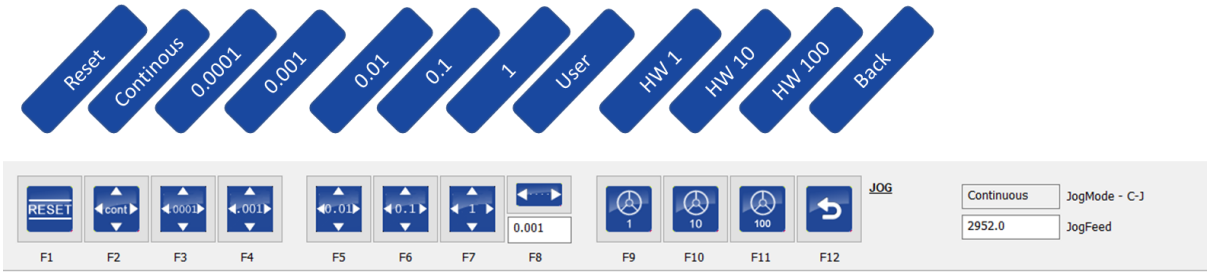
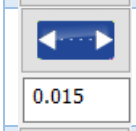
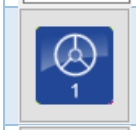
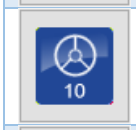
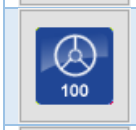


*Figure 29 Jogging menu in **inch** mode.*

Button	Function
	Execute a reset command.
	Continuous jogging, the movement of the selected axis will stop if the button is released.
	Every keypress will move selected axis 0.0001 inch.
	Every keypress will move selected axis 0.001 inch or mm.
	Every keypress will move selected axis 0.01 inch or mm.
	Every keypress will move selected axis 0.1 inch or mm.

	Every keypress will move selected axis 1 inch or mm.
	User defined step size.
	Enable hand wheel. Handwheel mode x1.
	Enable hand wheel. Handwheel mode x10.
	Enable hand wheel. Handwheel mode x100.
	Return to previous menu.

Several keys are reserved for jogging.

Figure 30 Axis jogging with keyboard keys.

The following keys are assigned to moving the axis:

Key	Function
Cursor LEFT	X-
Cursor RIGHT	X+
Cursor UP	Y+
Cursor DOWN	Y-
Page UP	Z+
Page DOWN	Z-
Home	A+
End	A-
Insert	B+
Delete	B-
Num. +	C+
Num. -	C-

The speed of the movement also depends on the fact whether the machine has already been homed. If the machine has not been homed the speed will only be a defined percentage of 'SafeFeed", this value is configured as part of the setup.

During normal jogging the speed at which the machine is moved is limited. By pressing additional keys, the used jogging speed is automatically selected.

Key	Jogging feed
No additional key	10%
+CTRL	50%
+SHIFT	100%

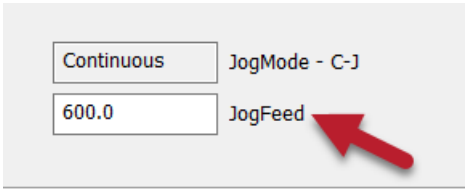


Figure 31 Jogfeed used during continuous jogging.

The currently selected jogging mode is indicated by 'JogMode' to the right of the jogging menu.

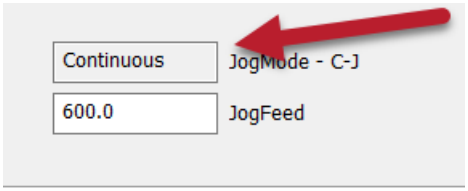


Figure 32 Currently selected jogging mode.

For your information. Since jogging causes the application to send out motion data, a delay would appear until stopping as a result due to the queue still containing motion date when the pressed key was already released. To help in this, during jogging the queue will be shortened to help responsiveness.

Hand wheel

By connecting a hand wheel directly to the controllers, it is possible to control the position very simple. The handwheel is activated by clicking on any of the three icons:

Figure 33 Hand wheel activation icons.

Activating the handwheel will show the text HW-1, HW-10 or HW-100 as JogMode.
The selected button will turn green. For example, if the '10' button is selected.

Figure 34 HW-10 selected.

There is one extra setting, by pressing **CTRL** + **ALT** + **N** or **CTRL** + **ALT** + **F9** it is possible to select a 0.1x modus. This is indicated by switching on all three icons on and the text HW-0.1.

Figure 35 HW-0.1 selected.

When the hand wheel is selected this is also indicated by changing color of the currently activate axis in the DRO. In the image below the X-axis is selected can be controlled through the handwheel.

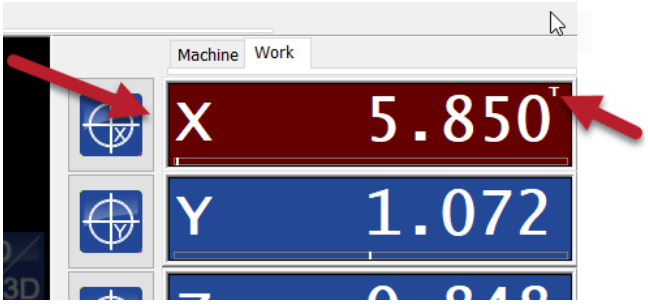


Figure 36 Selected X axis for hand wheel operation.

Also notice the 'T' that is shown at the top-right of the axis position. This 'T' means this axis is tracking an input value, in this case the handwheel. Different axis can be selected by pressing the UP or DOWN cursor keys. For more information, see also 'DRO (position) indicators

4.1.2.10 Jog Pad

The jog pad can be used for jogging by mouse or touch screen. Each arrow indicates which axis it is controlling and in what direction.

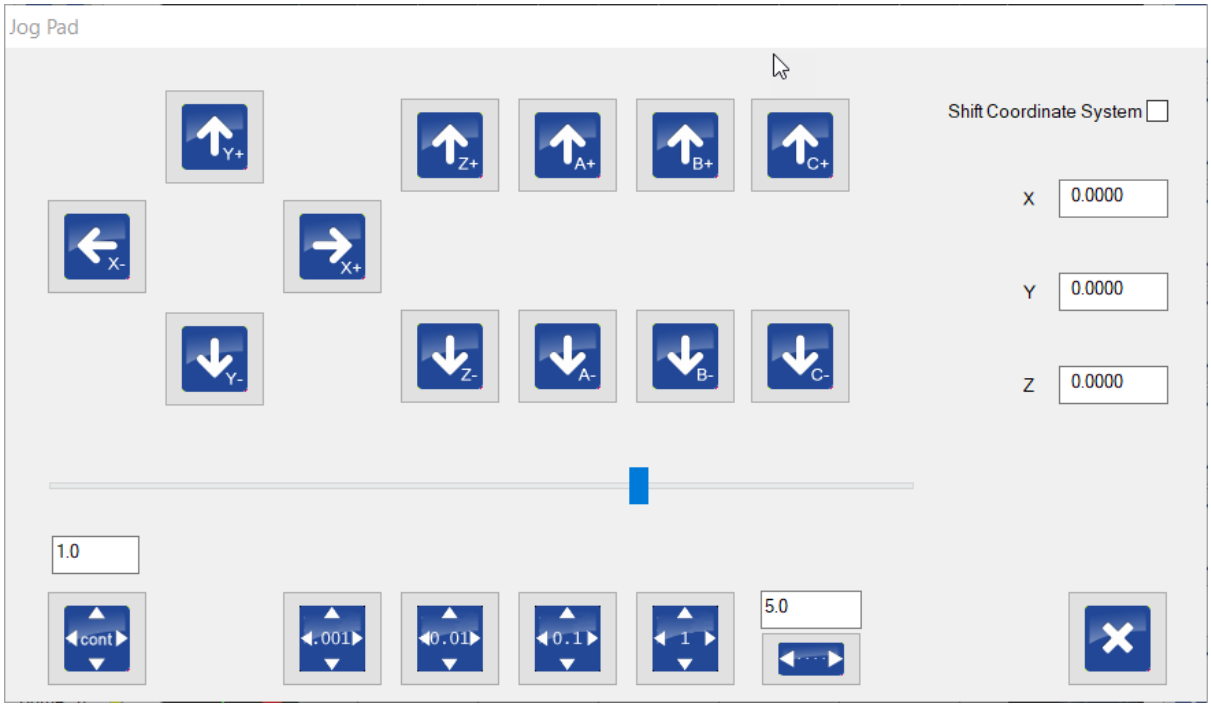


Figure 37 Jogging pad in mm mode.

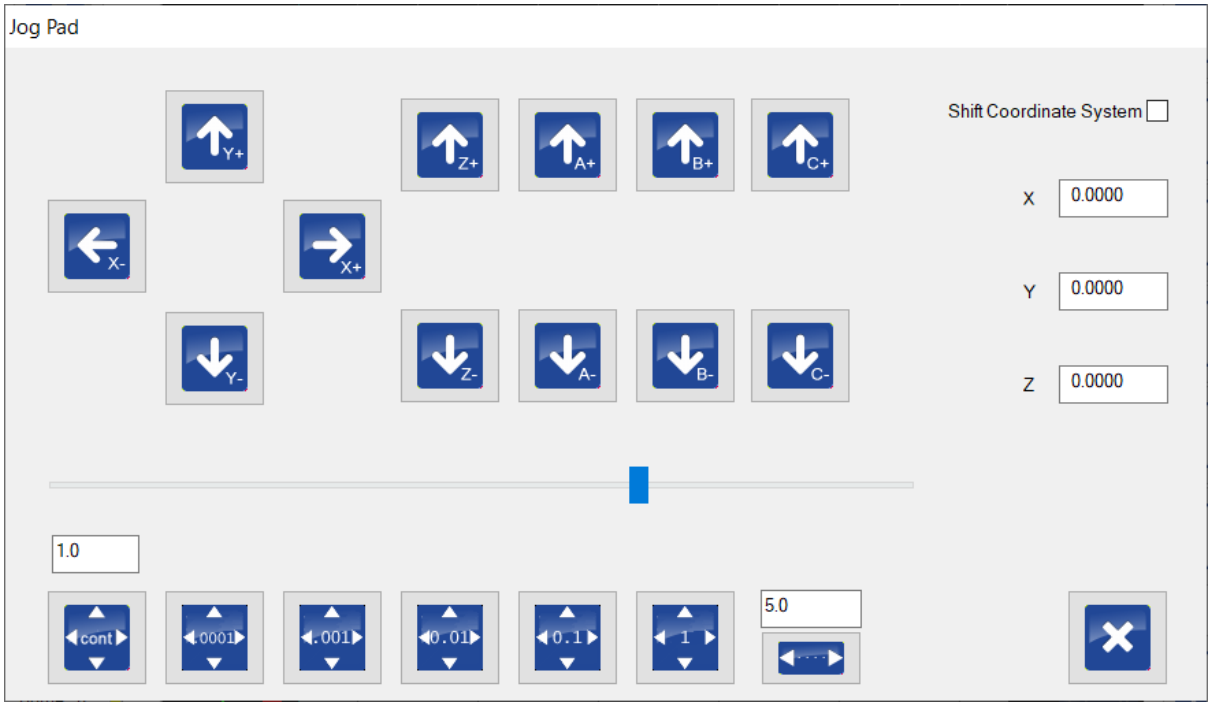
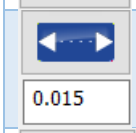


Figure 38 Jogging pad in inch mode.

In the image above all six axes are shown because all six axes are active. However, if less axes are active only these will be shown.

Button	Function
	Continuous jogging, the movement of the selected axis will stop if the button is released.
	Every keypress will move selected axis 0.001 inch.
	Every keypress will move selected axis 0.001 inch or mm.
	Every keypress will move selected axis 0.01 inch or mm.
	Every keypress will move selected axis 0.1 inch or mm.
	Every keypress will move selected axis 0.1 inch or mm.
	User defined step size.
	Close the Jog Pad

The current selected mode and step size is indicated by the textbox above the ‘CONT’ button.

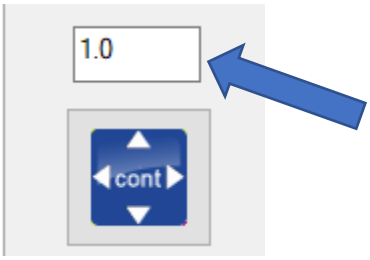


Figure 39 Current selected Jog Pad mode and step size.

It is also possible to set a custom step. The value can be filled in above the ‘user selected’ button, and then click on the button below to activate its value.

Figure 40 User selected step size.

Shift Coordinate System

If a non-continues mode is active there will also be an extra option available called ‘Shift Coordinate System’.

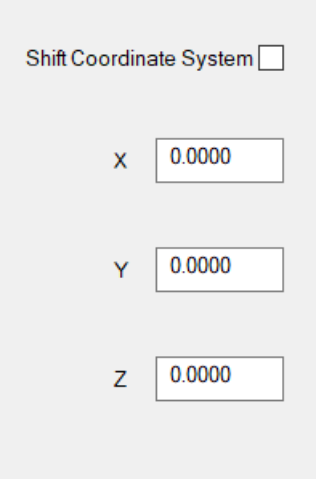


Figure 41 Shift Coordinate System fields.

When ‘Shift Coordinate System’ is checked, jog-step functions will move the physical axes one step at a time. However, the work position on-screen remains the same. This is accomplished by modifying the active G92 offset.

The amount of shift is shown at the right side for each axis. To reset the value to 0, which has no influence on the active offset nor machine position, uncheck, and then check “shift coordinate system”.

This feature is useful when e.g. during engraving you want to run the G-Code program again, but a little deeper in Z. Eg. you want to run the program 0.1 mm deeper, select jog step 0.1 and check ‘Shift Coordinate System”. Now press de arrow down button to move Z 0.1mm down. Notice that the axis moves down but that the position remains the same. When you run your engraving program again the engraving will be 0.1 mm deeper into the material.

This option is also very handy during turning. When your program has run and you measure the work piece and see its diameter is still a bit too big. So now use the –X button to compensate the diameter. Run the program again and your work-piece diameter will be correct.

4.1.2.11 User1 & User2 menu

It is also possible to add new custom functionality to the application. These functionalities can be associated to a so called 'User Button'. There are 20 buttons available for your own use. Buttons 1-10 and buttons 11-20.

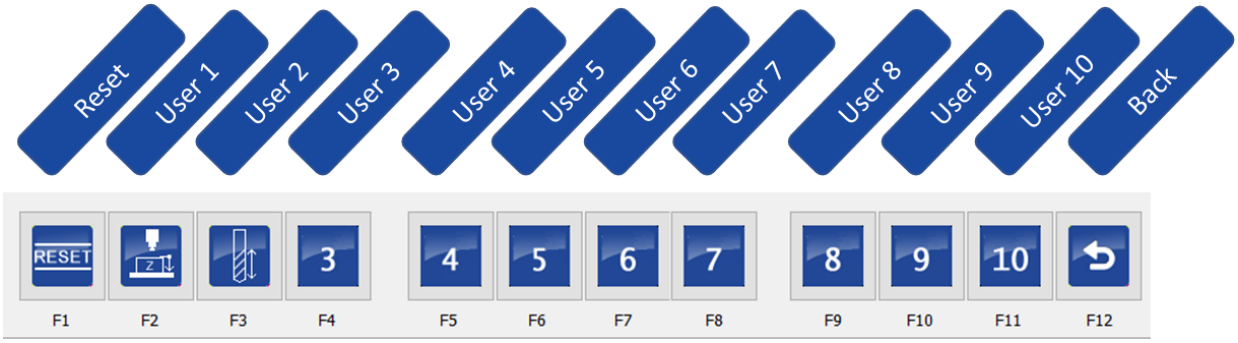


Figure 42 User buttons 1-10 menu.

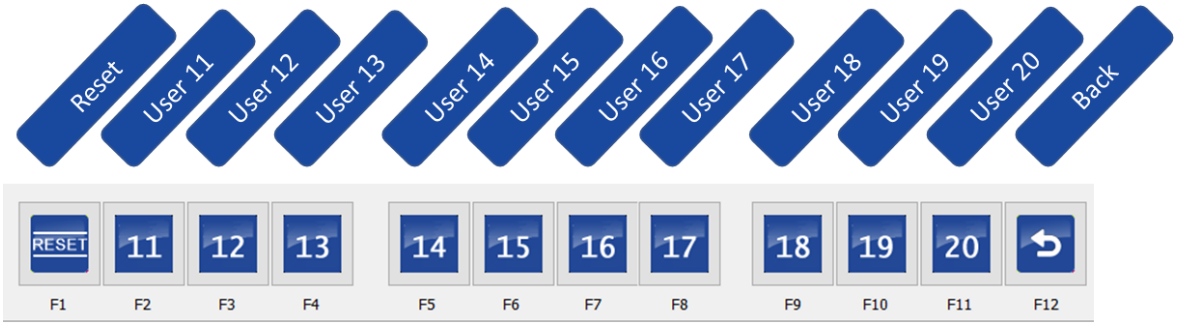


Figure 43 User buttons 11-20 menu.

Each button can have its own icon to indicate its functionality. Details about how to use your own icons are explained in a separate document.

User button 1-3 functions

Default user button 1-3 will have some functionality. The offered functionality is given as-is, and probably needs to be customized to your machine. Details about this are explained in a separate document.

Button	Function
	Zero the Z coordinate using a flexible tool setter positioned on top of the material
	Measure the tool length and put the length in the tool-table using a fixed tool setter
	User button 3. Example of a custom dialog screen.

4.1.2.12 Machine ON button

This button is also an indicator. Its color indicates the status of the machine as some sort o traffic light.

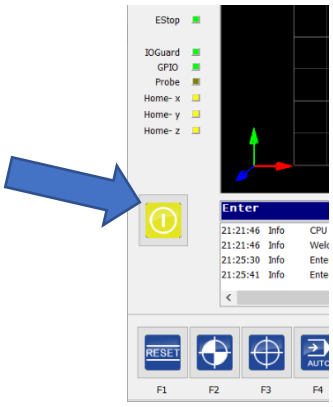


Figure 44 The Machine ON button.

It can indicate several things:

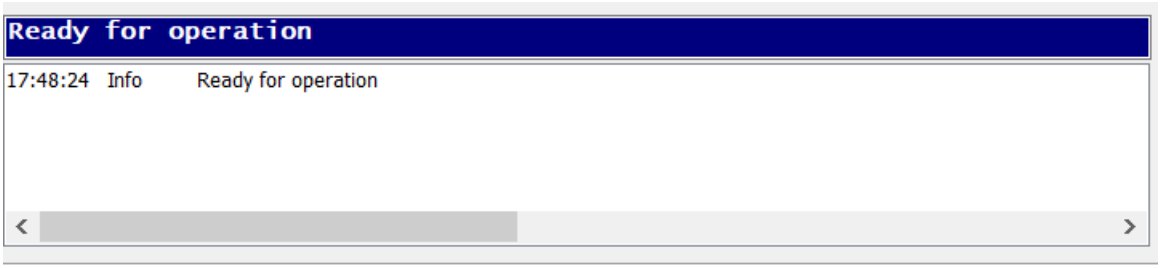
	Machine is off, drivers are switched off
	Drivers are switched on, but machine is not homed yet.
	Machine is on, waiting for operator action.
	Machine is running
	Error or E-stop condition.
	Error, E-stop is still active

If the system is configured to use a safety relay, this button can also be used to activate the safety relay.

When you start the application, you must press reset F1. This will enable the drives, the machine ON button left will be yellow flashing, this means the machine is ready but must be homed first (if this has been indicated as mandatory in the setup screen). In simulation mode it will immediately turn yellow, since it is not possible to evaluate the value of hardware inputs since no controller is connected.

4.1.3 Message screen

The message screen will display the status of the machine. But it will also display warnings and errors or other kind of information.



4.1.4 Homing/zeroing buttons

The buttons beside the axes's positions are for zeroing the work position, on the background a G92 command is executed to perform this. The zero buttons can also be found in the zero submenu, especially for people who do not like using the mouse at the machine.

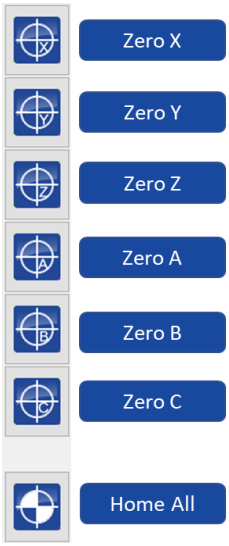


Figure 45 Zeroing/homing button next to DRO.

NOTE. In the image above 6 axes are shown, if your machine has been configured with less axis only those axes will be shown.

4.1.5 DRO (position) indicators

The Digital Read Out shows the actual position of each axis. Based on whether it operates in MMs of inches the DRO will show the digits in the DRO.

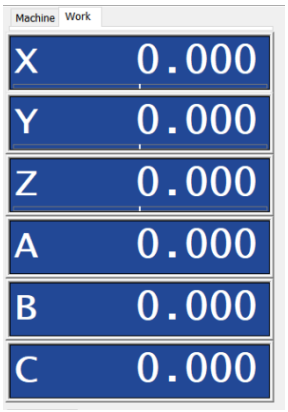


Figure 46 Digital Read Out (DRO) in mm mode.

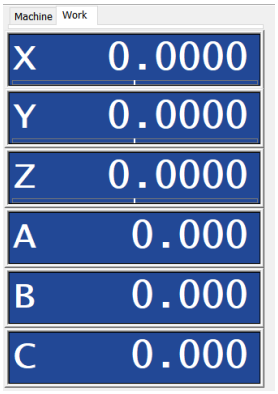


Figure 47 Digital Read Out (DRO) in inch mode.

Axis A-C are rotational axes, this will not change whether the system operates in millimeters or inches. In both case the position is in degrees.

It is important to understand there are two type of positions. Machine coordinates and work coordinates. The machine position is absolute, this is the real position of the machine. The other type of position is called work coordinates. Work coordinates can be anywhere on the machine, and its zero position can be defined manually by the operator of the machine. This position can be set using the ‘zeroing’ buttons as explained in the previous chapter.

Based on the type of position you are looking at, the background color of the DRO will change to show this.

A. Machine coordinates, homed. (green background):

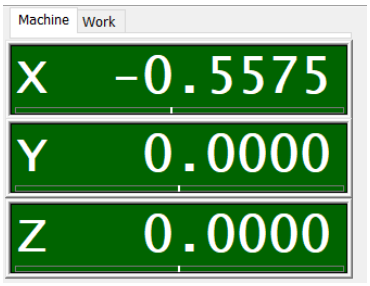


Figure 48 Machine coordinates DRO (A).

B. Work coordinates, homed. (blue background):

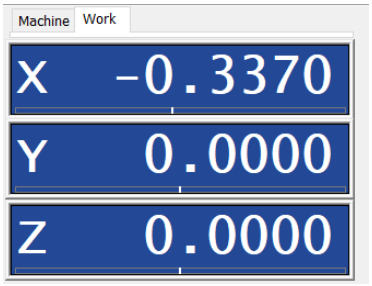


Figure 49 Work coordinates DRO (B)

If the machine is not homed, the white text will be black to indicate this. This will not occur in simulation mode because homing needs hardware input. Therefor the machine is assumed to be homed.

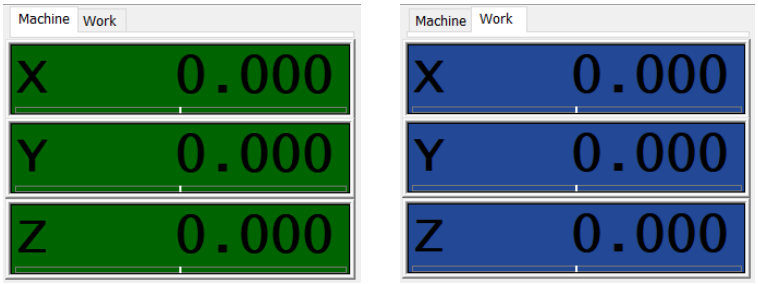


Figure 50 DRO of machine and work coordinates with not-homed position.

The machine coordinate system will never change. To be able to have defined starting put to start our operation we use the work coordinates.

For example, suppose your G-code file contains a work piece that is created with an origin of X=0, Y=0, Z=0. This is because you have drawn your part in a CAD program beginning from these coordinates and then converted to G-Code. Now you have put your raw material somewhere on the machine, probably not at coordinates X=0, Y=0, Z=0. You can know jog to the position you want and tell the machine this new position is by pressing the zeroing buttons next to the DRO or in the 'Zero Menu' and the new work position will be X=0, Y=0, Z=0 position, so now you have changed the work coordinates. Remember, the machine coordinates will remain identical. By the way, it can be a good

idea to define the upper surface of the material as Z=0, such that a negative Z value goes into the material.

If the work position has been changed after the G-code file has been loaded graphics windows will not yet reflect the actual new position. To update this press 'Redraw' in either the 'graphics menu' or the button in the 'graphics windows', you will now see exactly where the part is going to be milled on your machine bed. This home position is also indicated by cyan colored cross in the x-y plane. See also 'Graphics display in more detail'. This is also performed at the start of the execution of a loaded file.

Setting position

The DRO can be used in several ways for controlling the current position of the axis. Some methods of doing this:

1. **Clicking on an axis**

   This presents a window which offers the user to preset the actual **work** coordinate through a 'set position' G92 code (or G10L20 code depending on your setup).

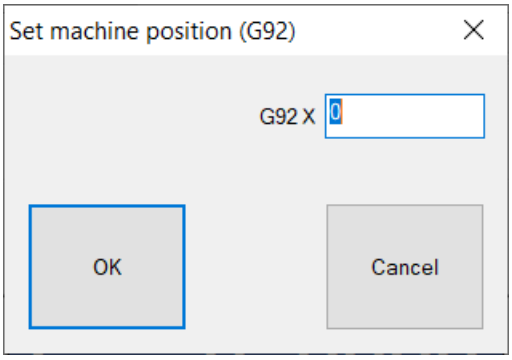


Figure 51 Set machine position (G92) dialog.

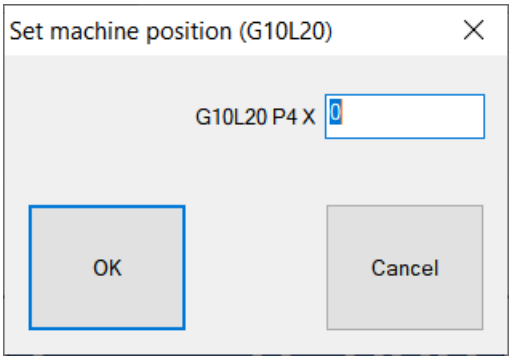


Figure 52 Set machine position (G10 L20) dialog

   Simply enter the value that you want the current position to have as work coordinate. Please note, that the machine coordinates will not have changed. This will not cause any motion of the machine.

2. ***Holding CTRL + clicking on axis (in work coordinate window)***

This present again a window, but this time it offers a 'linear movement' G0 of that axis.

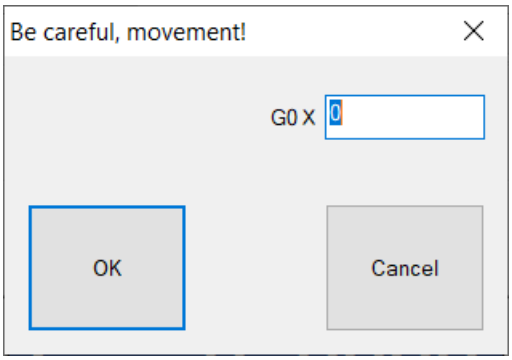


Figure 53 Move to work coordinate position dialog.

By entering a value, the axis will ***move*** to that ***work*** coordinate.

3. **Holding CTRL + clicking on axis (in *machine* coordinate window)**

This present a window, but this time it offers a 'linear movement' G53 G0 of that axis.

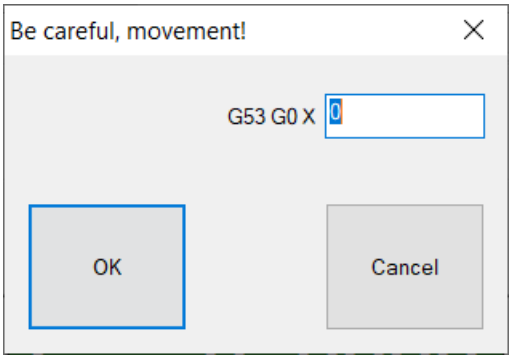


Figure 54 Move to machine coordinate position dialog.

By entering a value, the axis will travel to that ***machine*** coordinate.

Machine limits

The DRO will also show the limits of each axis. This is indicated by a small mark below the actual position indicator. See the red arrow in the image below.

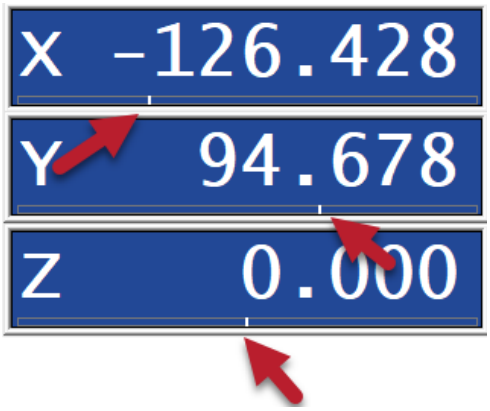


Figure 55 DRO showing limit of each axis.

4.1.6 General status display

The general status display show information about the machine. This display consists out of three tabs:

A. Feed Speed
B. G/M Code
C. Time

Each tab can be selected by clicking on the tab, but it can also be switched by pressing ‘CTRL + V’.

Feed Speed

The ‘Feed Speed’ tab shows for the feed and spindle speed the actual values, set values and percentage. If you do a G1 in this example, the feed will be 100.

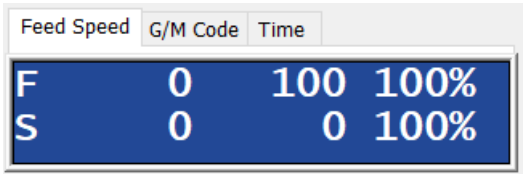


Figure 56 Feed and Spindle speed display.

G/M Code

The ‘G/M Code’ tab shows the actual G code and M code status as well as the actual tool number and the machine state, READY, RUNNING, P-JOB, HW-P etc.

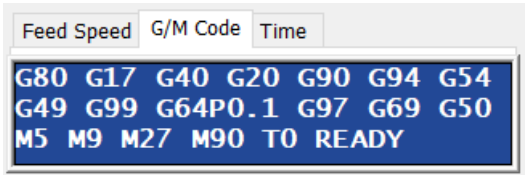


Figure 57 G/M code display.

Time

The ‘Time’ tab shows the actual running time of a job and the estimated total time.

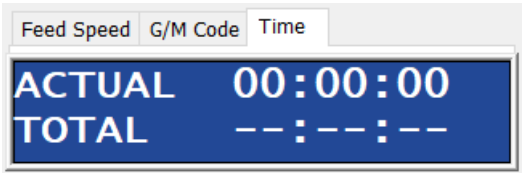


Figure 58 Time display.

4.1.7 G-Code status display

This display continually shows an overview of active G-codes.

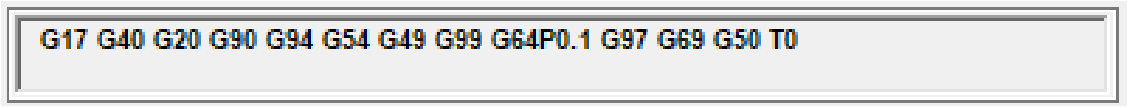
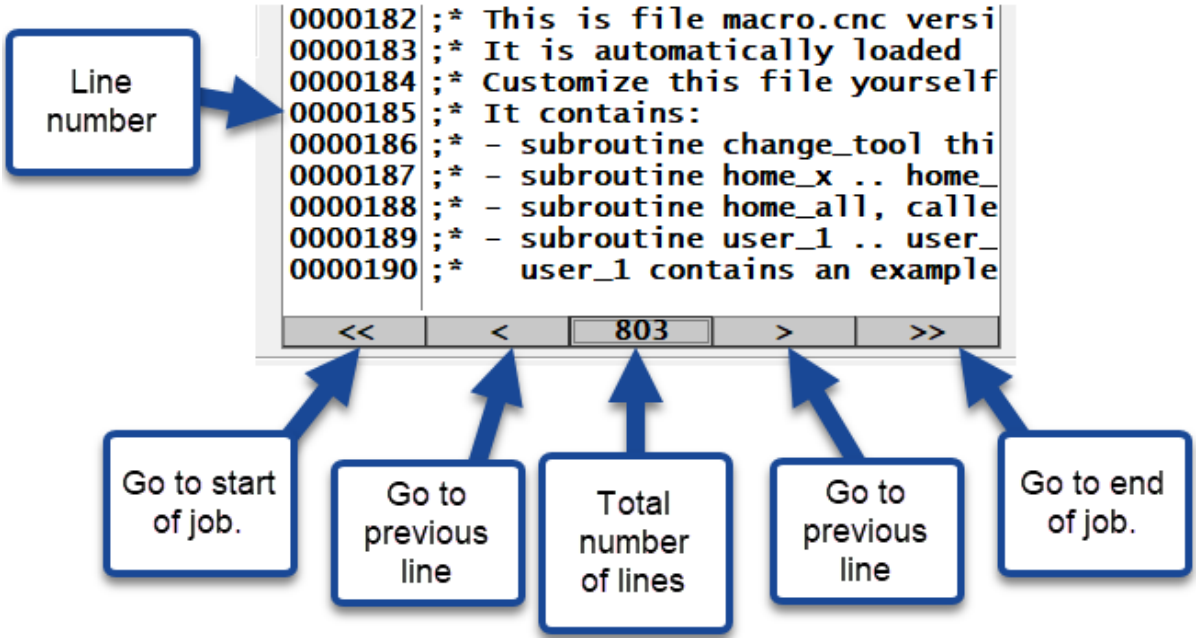


Figure 59 G-code status display.

4.1.8 Job view

The loaded G-code file is listed here. Because it is possible to load files with up to millions of lines there is not a standard scrollbar, but instead buttons. The image below shows how these buttons work.



Clicking on the single arrow will normally advance one line, however this can be increased.

Pressed key	Jumped number lines
+CTRL	25 Lines
+SHIFT	100 Lines
+CTRL+SHIFT	100 Lines

Furthermore, clicking on the 'total number of lines will display a dialog box

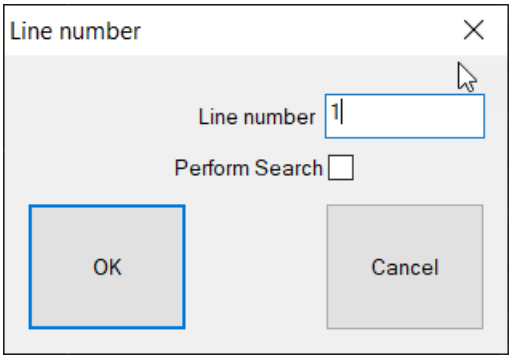


Figure 60 Goto line number dialog box.

In this dialog box you can enter the line number you would like to jump to. Simply fill in the line jumper and press OK. It will now show the requested line.

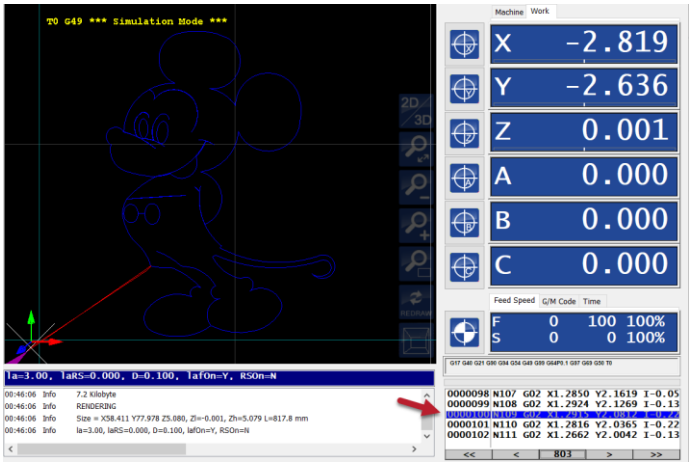


Figure 61 Goto line without 'Perform Search' checked.

Optionally, the 'Perform Search' can be checked. In that case the graphics display shows the progress up to this line. So, all lines up to the searched line will run through the interpreter.

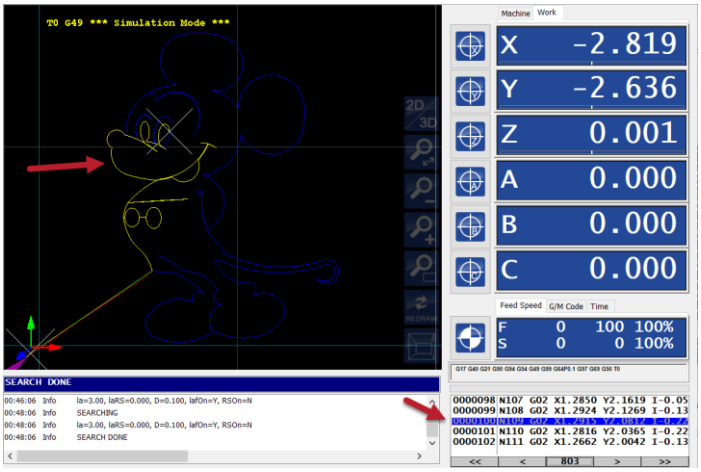


Figure 62 Goto line with 'Perform Search' checked.

4.2 Graphics display in more detail.

When the application has started the graphics display will show default view below. On the screen several things are shown:

> NOTE. The buttons on the right side are only visible if OpenGL is supported by the driver of your graphics card. Navigation in the graphics screen will somewhat be limited. Updating the driver of you graphics card might solve that issue.

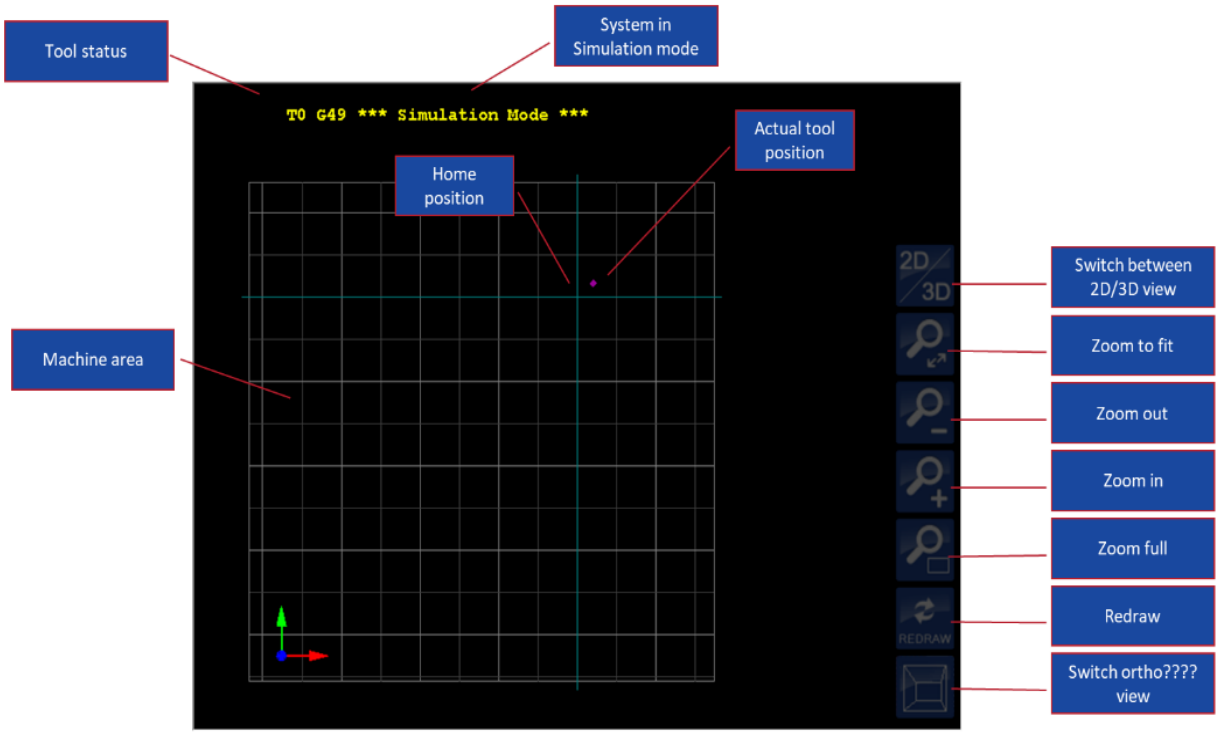
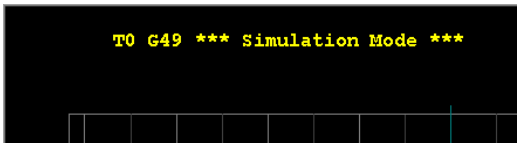
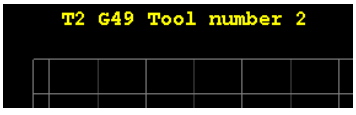


Figure 63 Overview graphics window.

At the top, the actual tool is shown, in this case it is tool 0 (T0). Also, it shows that G49 is active, meaning that 'tool length offset is canceled'.



If a controller is connected the text '*** Simulation Mode ***' will of course not be visible, instead it will show the name of the active tool, for example:



The next thing you see in the graphics view is the actual size of your machine, but also the actual current home-position on your machine. The grid that is shown is 50mm in mm mode or 2 inch in inch mode, projected on the machine bed (X-Y surface). This grid is drawn starting at the zero position of X and Y. This means when the zero position is moved, this grid also moves. Please note that the actual size of machine does *not* change.

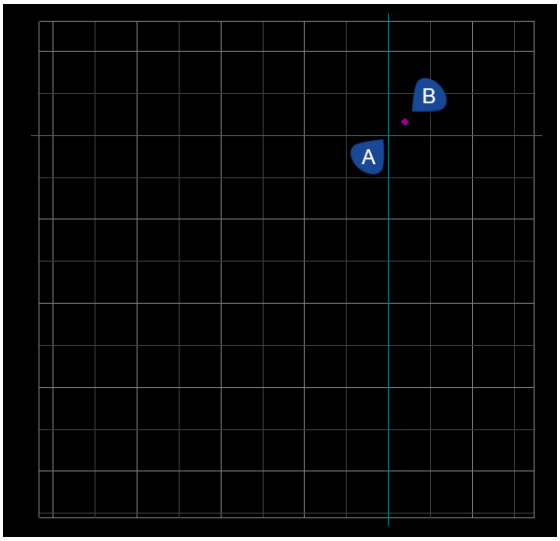


Figure 64. View of machine with actual position and work coordinate system origin.

Furthermore, the current work coordinate system origin is shown as a cyan colored cross in the x-y plane; indicated here by a '**A**'. The current position of your machine is show by the magenta point; indicated here by '**B**'.

At the bottom right, three arrows are shown. These arrows show the positive direction of the X, Y and Z. These help in understanding in what direction the machine moves.

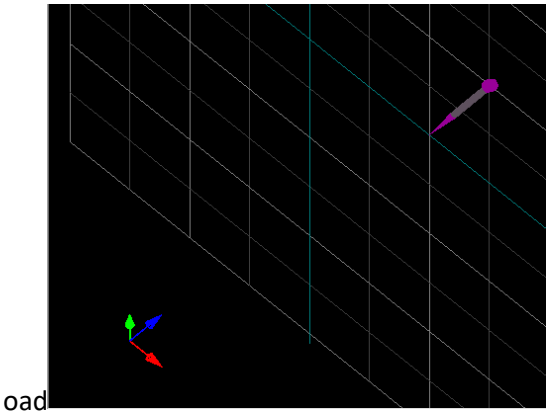


Figure 65 The colored arrow indicates positive direction of axis.

Arrow	Direction
Red	X+
Green	Y+
Blue	Z+

On the right side you see several buttons that might look familiar. They are the same as seen in the graph menu that was discussed previously. For a detailed description please have a look at that chapter.

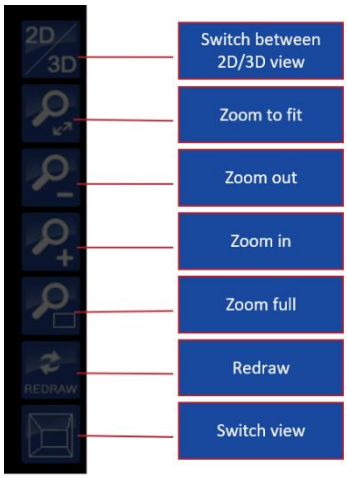


Figure 66 Overview graphics screen buttons.

Button	Function
	Switch the graphics view between 2D and 3D
	Zoom to maximum of loaded program
	Zoom out
	Zoom in
	Zoom to maximum of machine dimensions
	Redraw the screen
	Switch the view between Isometric and orthogonal

With the 2D/3D button it the view can be toggled between 2-dimensional or 3-dimensional. The images below show the difference.

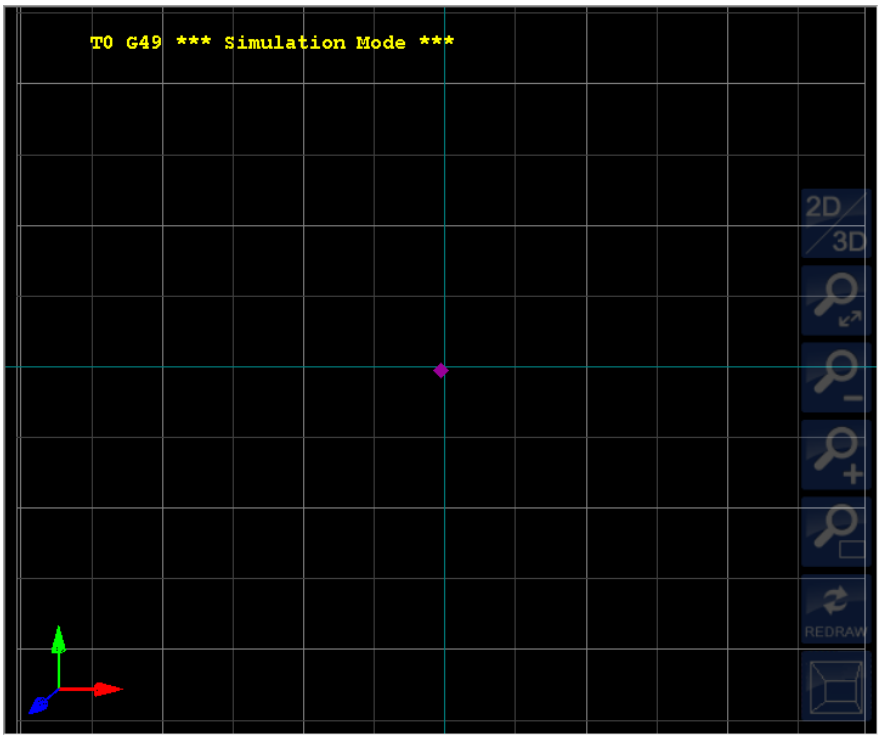


Figure 67 2D view (default).

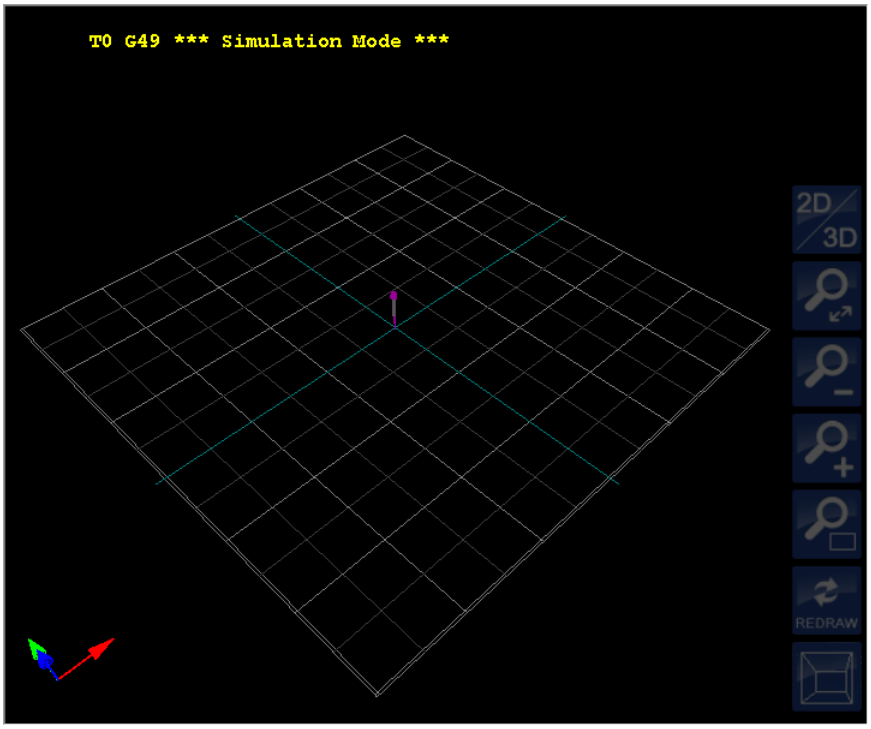


Figure 68 3D view.

Please note that in a 3D view you will see that the magenta point that is visible in 2D is now visible as a magenta pointer which tip at the bottom indicates the actual position.

Finally, with the bottom button you can toggle how the 3D view is shown either isometric or orthogonal.

Navigation

In the graphics view you can navigate with your mouse if OpenGL is active. In that case you can use the mouse like this:

Mouse movement	Navigation
Left mouse button + mouse	Pan the screen
Right mouse button + mouser	Zoom in / out
Mouse wheel	Zoom in / out
CTRL + left button + mouse	Rotate

Loaded program

When a program is loaded it will completely parse through the g-code interpreter and the tool path is shown in the window. For a representative view it is important that the axes limits are correctly filled in and that the machine is homed manually or automatic. The current work coordinate system origin is shown as a cyan colored cross in the x-y plane. When you press the preview update button, a preview is shown of the loaded g-code program. The preview is created by running the entire g-code file through the interpreter. So, when interpreter encounters an error, it is shown in the log window and in the operate view the program list box shows the line in a red color. Note that there can be inaccuracy in what the display shows, this is there because of performance and memory usage limitation reasons.

If the job is loaded a white cross will indicate the last position of the job.

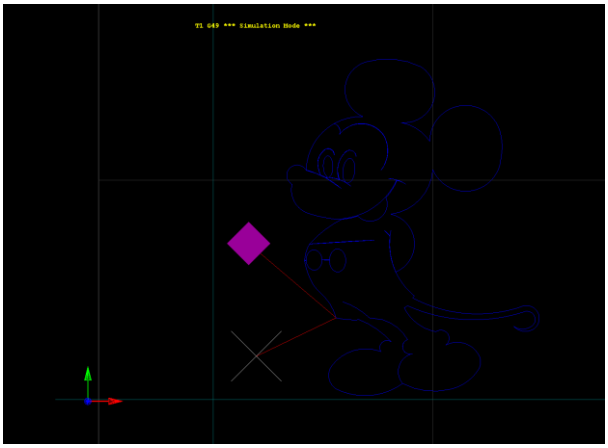


Figure 69 White cross indicating last position of loaded job.

Through a number of line colors the user can observe what the machine is actually doing.

Line color	Meaning
Blue	Toolpath from loaded program
Red	G0 movement from loaded program
Yellow	Executed movement
Green	Executed G0 movement

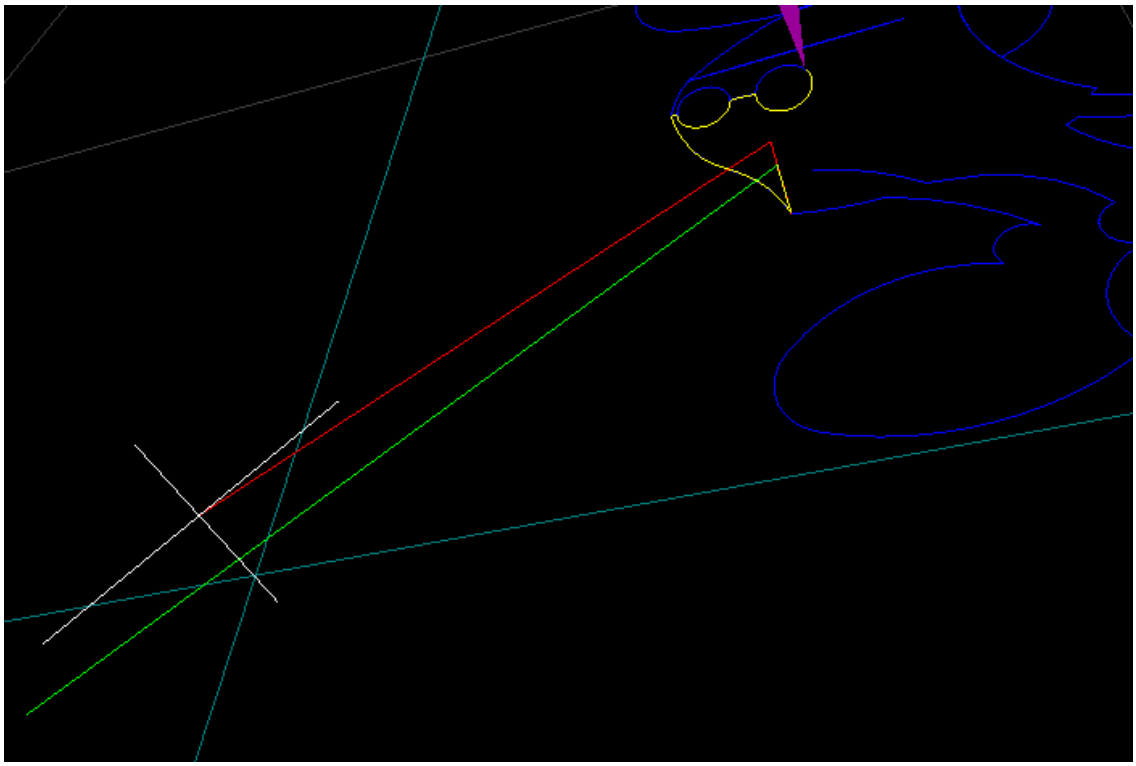


Figure 70 Overview line colors in graphics screen.

The yellow line will not constantly be visible but will disappear after a certain distance if this has been configured to save memory space. This length is defined in the setup of the machine.

5 Keyboard shortcuts

Numerous functions can be accessed through keyboard short cuts.

Key / Shortcut	Function
Escape	
Control Q	Quit program
Control TAB, Control shift TAB	Mode selection operate/coordinates/Program ..etc
Alt+O	Activate operate tab
Control V, Control shift V	Status tab next, previous
CTRL + W	Toggle Work/Machine coordinates
CTRL + F6	toggle MDI, will always work.
Alt F1	Main menu
Alt F2	Home menu
Alt F3	Zero menu
Alt F4	Auto menu
Alt F7	Machine IO menu
Alt F8	Graphics menu
Alt F9	Jog menu
Alt F10	Jog Pad
Alt F11	User 1 menu
Alt F12	User 2 menu
User macro execution	
Alt 1,2,3 .. 0, Ctrl+Alt 1,2,3 .. 0	Execute User macro 1 – User Macro 20
Homing	
Control R	Reset
Control H,	Home all
Control 1	Zero x
Control 2	Zero y
Control 3	Zero z
Control 4	Zero a
Control 5	Zero b
Control 6	Zero c
HANDWHEEL/JOG CONTROL	
Control + shift + X	Handwheel on X
Control + shift + Y	Handwheel on Y
CTRL + SHIFT + Z	Handwheel on Z
CTRL + SHIFT + A	Handwheel on A
CTRL + SHIFT + B	Handwheel on B
CTRL + SHIFT + C	Handwheel on C
CTRL + ALT + N / CTRL + ALT + F9	Handwheel X0.1
CTRL + N	Handwheel X1
CTRL + O	Handwheel X10
CTRL + P	Handwheel X100
CTRL + SHIFT + N	Jog continue (Handwheel mode off)
CTRL + J	Jog mode up
CTRL + SHIFT + J	Jog mode down
Alt + shift + A	Select JOG A
Alt + shift + B	Select JOG B
Alt + shift + C	Select JOG C
Alt + shift + X	Select JOG X
Alt + shift + Y	Select JOG Y
Alt + shift + Z	Select JOG Z
Control+Alt+Shift+O	Select JOG Speed Low
Control+Alt+Shift+P	Select JOG Speed Med
Control+Alt+Shift+Q	Select JOG Speed High

Control+Alt+Shift+R	Select JOG Step 0.01
Control+Alt+Shift+S	Select JOG Step 0.1
Control+Alt+Shift+T	Select JOG Step 1
Alt+Shift+P	Start Jog+ selected axis
Alt+Shift+N	Start Jog- selected axis
Alt+Shift+S	Stop Jog
Program execution	
Control I	Load g-code file
Control G, +shift	Run, Pause
Control T	Toggle Single line
Control + B	Toggle Blockdelete
Control F, +shift	+Feed, -Feed
Control S, Control + shift + S	+Speed, -Speed
Control+Alt+S	Speed override 100%
Control+Alt+F	Feed override 100%
Teaching	
Alt+Shift+O	Open Teach
Control+Alt+Shift+A	Add Point
Control+Alt+Shift+E	Edit
I/O Control	
Control D, Control + shift D	Spindle On right, Spindle Off
Control E, Control + shift D	Spindle On left, Spindle Off
Control K	Toggle Flood
Control L	Toggle Mist
Control M	Toggle Aux1
Control+Alt+Shift+1	Toggle Aux1 out
Control+Alt+Shift+2	Toggle Aux2 out
Control+Alt+Shift+3	Toggle Aux3 out
Control+Alt+Shift+4	Toggle Aux4 out
Control+Alt+Shift+5	Toggle Aux5 out
Control+Alt+Shift+6	Toggle Aux6 out
Control+Alt+Shift+7	Toggle Aux7 out
Control+Alt+Shift+8	Toggle Aux8 out
Control+Alt+Shift+9	Toggle Aux9 out
Control+Alt+Shift+0	Toggle Aux10 out

6 Using ‘goto’ menu

Sometimes you might want to start at a certain place in your program instead of starting from the beginning. In that case using the go-to functionality can be very useful. If will enable you to jump to any line or jump to a tool change inside your program. Use this functionality to start at given line number or tool change instead of starting from the beginning.

If the ‘Automatic menu’ is active, pressing the ‘Goto’ button or pressing the F7. It will show a new dialog on the left of the screen.

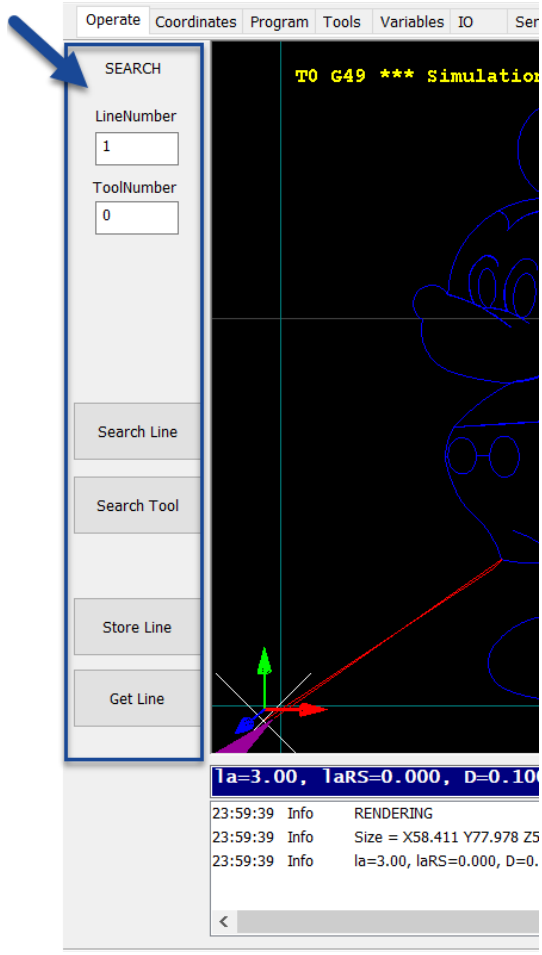
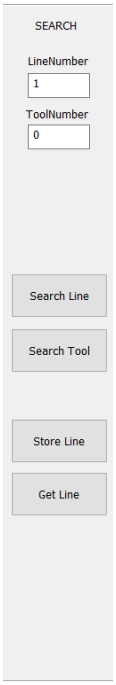


Figure 71 GOTO dialog.



If you have not entered a line number, but have paused the execution, the line number will show the current line of the job. This happens also when you press reset when paused. During Pause mode only jog movements are allowed.

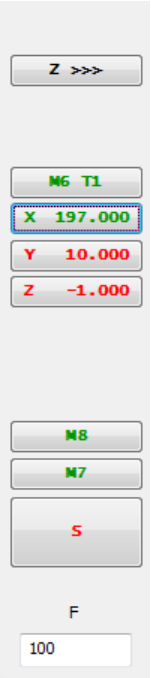
Note that you need to do a reset while being paused if you need to perform e.g. a tool change.

Press '***Search Line'*** to run the interpreter in Search mode up to the given line number.

Press '***Search Tool***' to find a tool change for the given tool number.

You can store a line number and retrieve the stored line number using the Store/Get Stored buttons. The stored line will also be stored if you close the application and be available if the application is loaded again so you could continue where you have stopped.

If 0 is used as 'Tool Number' the interpreter will stop at every tool change when 'Search Tool' is pressed.



When you press the RUN button (F4) after a search or pause, the application will try to synchronize the required state with the current state(for example, the position). That might result in the following popup dialog. If any axis is not at the correct position or the spindle or coolants are not correct, it allows you to synchronize the actual situation with the required situation manually. A green color means that the indication action is synchronized, and red color indicates that your machine is currently not yet synchronized with the appropriate position or settings at the selected line.

The '**Z >>>**' button will start moving the Z axis completely up.

The, in this case, '***M6 T1***' button shows the tool according to the interpreter, this button is not visible at a start after pause, only at start after search.

If the color is green the current tool matches the tool from the search status.
If the color is red, the tool does not match, and you can start a tool change by pressing the button.

The axis buttons show the position according to the interpreter on the searched line. Green is a match, red is not a match, pressing the button to move the axis to the correct position.

> If any axis is not synchronized, it will be done automatically when the '***Start***' button is pressed.

The "***M8***" or "***M7***" buttons allow to switch on the coolants.

The '**S**' button switches the spindle ON with correct S value from the Search status.

The value below the "F" is the plunge rate, the feed rate for the movement towards the work piece. The feed rate will only be active in this case. After the application has started it will use the feed rate as indicated by the program.

Finally, press the '***Run***' button. This will start a G1 motion with the designated plunge rate towards the search positions, then restore the Feed to the search feed and start machining from there.

7 Using ‘nesting’ menu

It is sometimes nice to have a job being done several times. There are two ways of doing this. Either you reposition your machine each time manually, set the new start position and start the job again. Or you can use ‘nesting’. This is a feature that allows a product to be produced multiple times automatically. This nesting option is provided through the ‘***Automatic menu***’

Please note, nesting can only be performed if the machine is in READY state.

If you press on the ‘***Nesting’*** button a new number of options are shown in a dialog on the left of the screen.

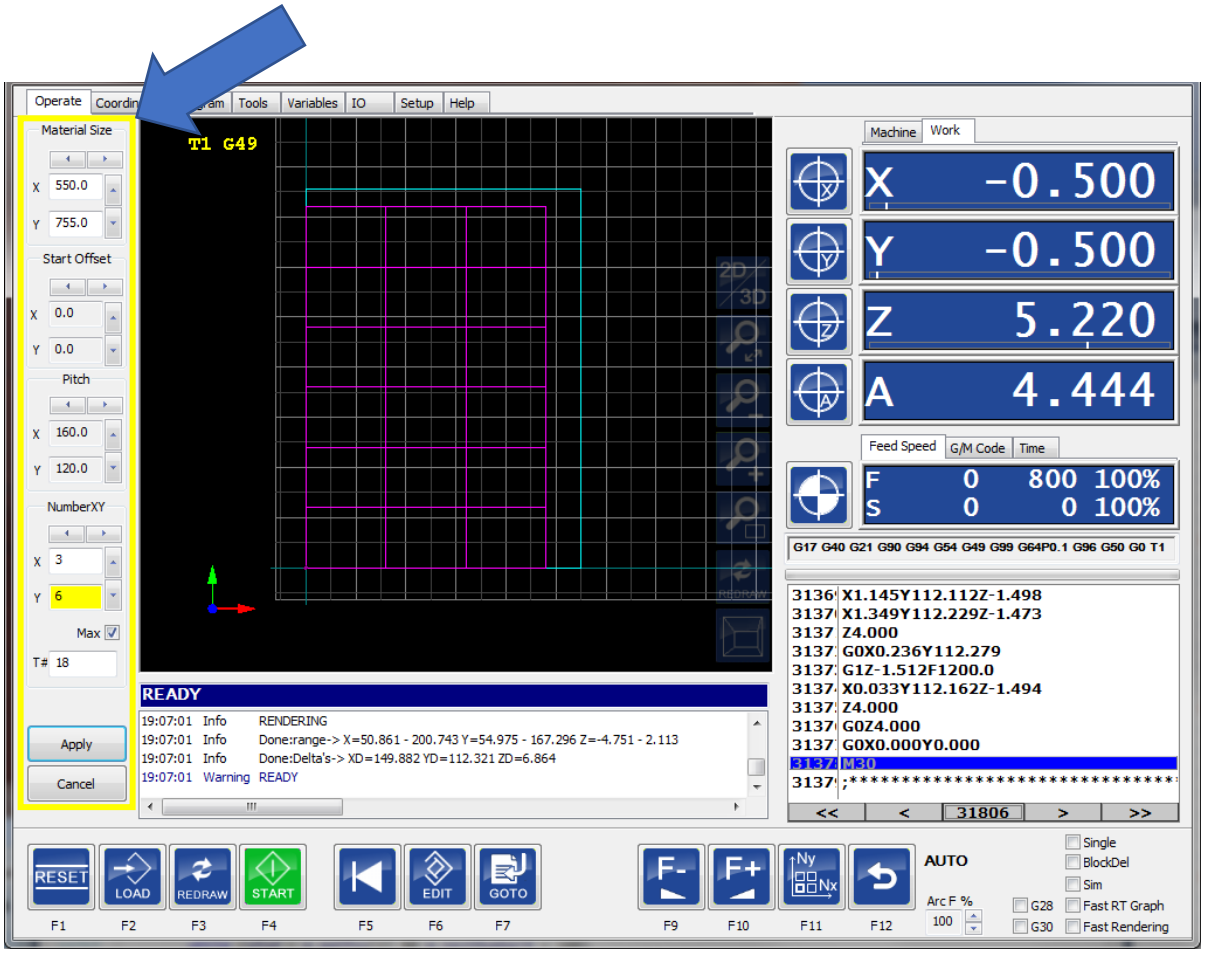


Figure 72 Nesting dialog.

The software will automatically draw a purple square around the loaded object. Through the dialog you can setup the nesting of the loaded object.

This presents several options:

Material size	The material size in X and Y.
Start offset	Change the start position for the nested result.
Pitch	Change the distance between the nested objects.
Number	Change the number of nested objects in X and Y direction.
Max	If this is clicked the software will automatically determine the maximum nesting possibilities.
T #	Show how many times the object has been nested.
Apply	Apply the current settings to the program.
Cancel	Cancel nesting, back to only one product.

The '**Nesting**' button (F11), can be pressed to show or hide the nesting dialog.

To use nesting several conditions need to be met:

- Nesting internally uses coordinate system offset *G59.3*, *the coordinate system offsets should not be used in the program*, otherwise nesting will not work, so no G54 .. G59.3 allowed in the program.
- The G54 offset should be 0.
- G92 is allowed, but if changed during the program it must be set back to the original value at the end of the program.
- The program must end with M30 otherwise nesting will not work.

Tip, use M60 instead of M30 when the spindle should not stop between the work pieces.

The values that are set in the dialog box can also be set in the G-Code:

%mx=200 Material size X

%my=200 Material size Y

%dx=200 the pitch X

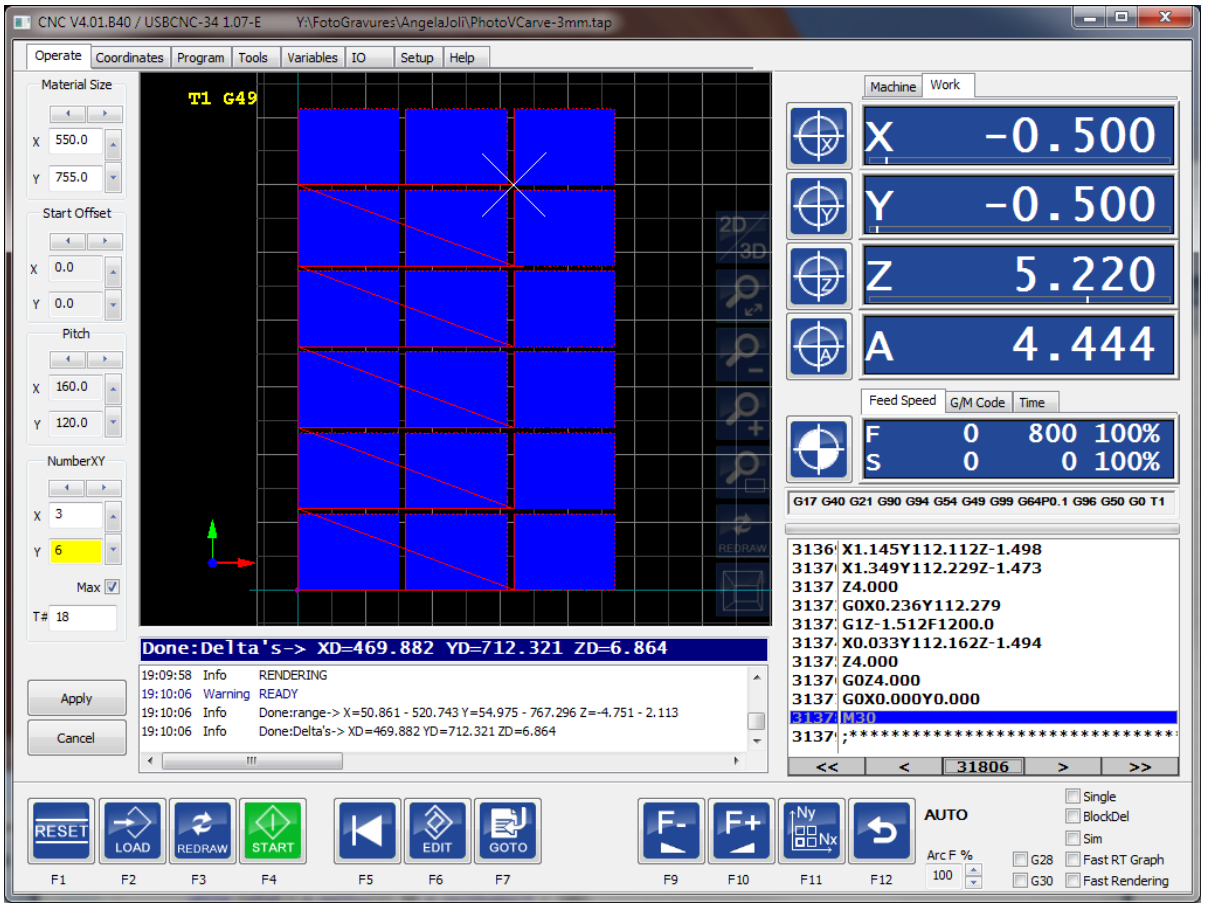
%dy=200 the pitch Y

%ox=200 Offset X

%oy=200 Offset Y

The 'offset' is the distance from the edge of the material. 'Pitch' is the distance between each part.

After pressing the Apply button, the nesting is applied to the program and the result is shown.



It is recommended to create the G-code file for the product such that X0 Y0 is at the lower left side.

If you prefer not to start at the beginning, use the ‘Goto’ function and apply the ‘Nest X’ and ‘Nest Y’ values, as shown in the image below.

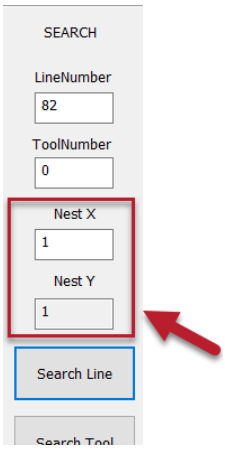
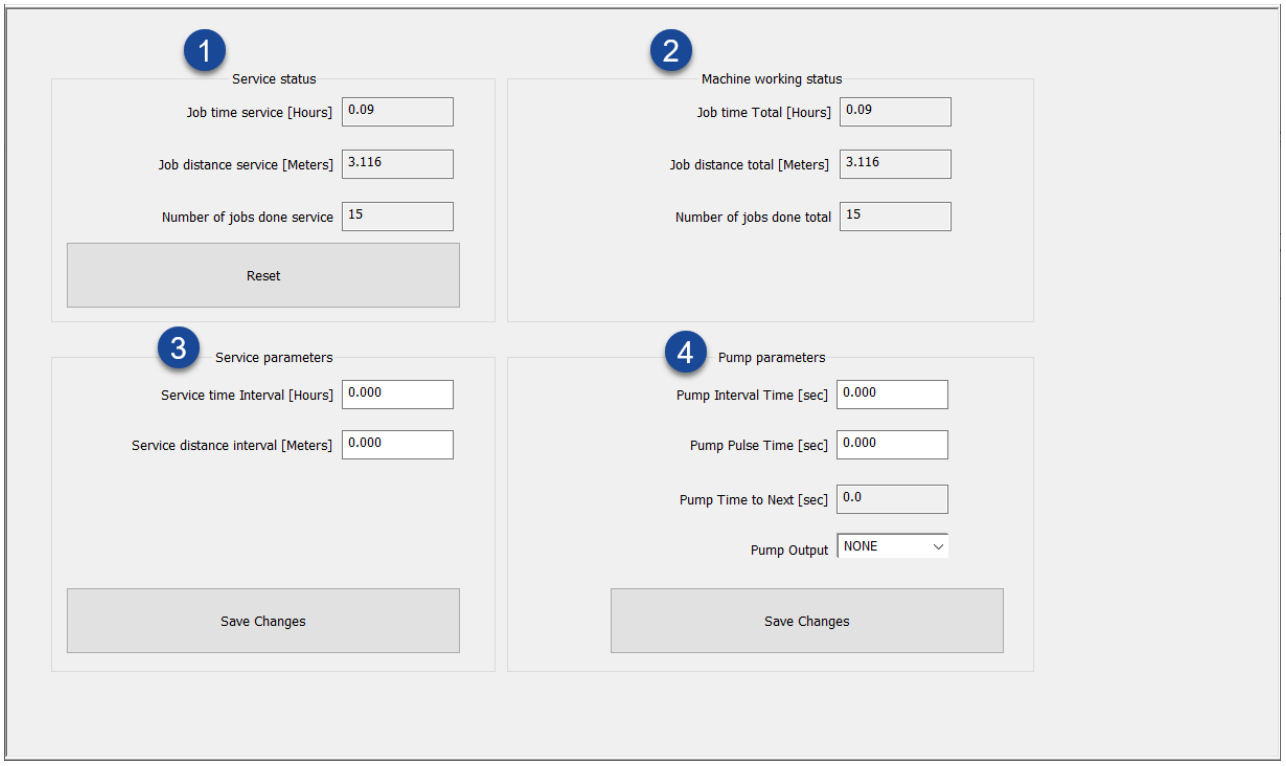


Figure 73 Nesting options during Search.

8 The ‘Service’ tab in more detail

This page shows how much your machine is used and if it needs service. Also, it can be used to set an ‘pump’ output that is activated at certain configurable times. The service status fields can be reset, for example after service has taken place. The machine working status cannot be reset.



You can see:

1. **Service status**

Job time service	Number hours since the last service.
Job distance	The distance the machine has milled in meters since last service.
Number of jobs done service	Total number of jobs ***completed*** since last service.
Reset	Reset these values

2. **Machine working status**

Note, these values are accumulated during the lifetime of the machine and ***cannot*** be reset.

Job time total	Total of job time during the lifetime of the machine
Job distance	The distance the machine has milled in meters.
Number of jobs done service	Total number of jobs done since last service

3. Service parameters

Here the conditions are set when the next service interval is required. If either of these conditions are met, it will be indicated to the user. If service is required a message will be shown. This message will be “Machine service time interval expired, please apply service”.

Service Time interval	The time interval for the next service is required, the software will show a message when this passed at the end of a job.
Service distance interval	The traveled distance till the next service interval. The software will show a message to indicate the machine needs service.

4. Pump parameters

If your system is, for example, fitted with an automatic machine oiling system; the parameters shown are used to define the intervals for this system.

Pump Interval Time	Time in seconds between pump actions.
Pump Pulse Time	Duration of one pump action
PumpTimeToNext	Time to next pump action (***read only***).
Pump Output	Selected output used for PUMP, set to NONE for no pump action
Save changes	Set the ‘Pump Time To Next’ to 5 seconds once for testing. After that every Pump Interval Time’.

This pump control will start as soon as you press ‘Save Changes’. Once started the field ‘PumpTimeToNext’ indicates when the next pump action will be performed. The first time that ‘Save changes’ is pressed the PumpTimeToNext will be set to 5 seconds so you can test the system.

ADVANCED GUI stuff

Pre-condition:

- **CNC.INI**
- **Macro.CNC**
- **Setup configuring**
-

❖ Calibrate/Activate coordinate systems

~~Nesting~~

~~Search line~~

- Measuring the rotation
- Linear Pitch compensation
- Z Height compensation
- Teach in
- Mapping X/Y G-code to cylinder on A-Axis
- XY Non rectangular Cross compensation
- Speed-PWM compensation
- Automatic vacuum sections

Losse manuals:

- Program Page, DXF and HPGL import
- ATC
- Using a toolsetter
- Using Probe(???)
- Setting up Eding CNC
- Homing >>>
- LAF
- G-code interpreter (seperate manual)
- Debuggin G- code
- 3D printing
- Plasma application
- MDI commands