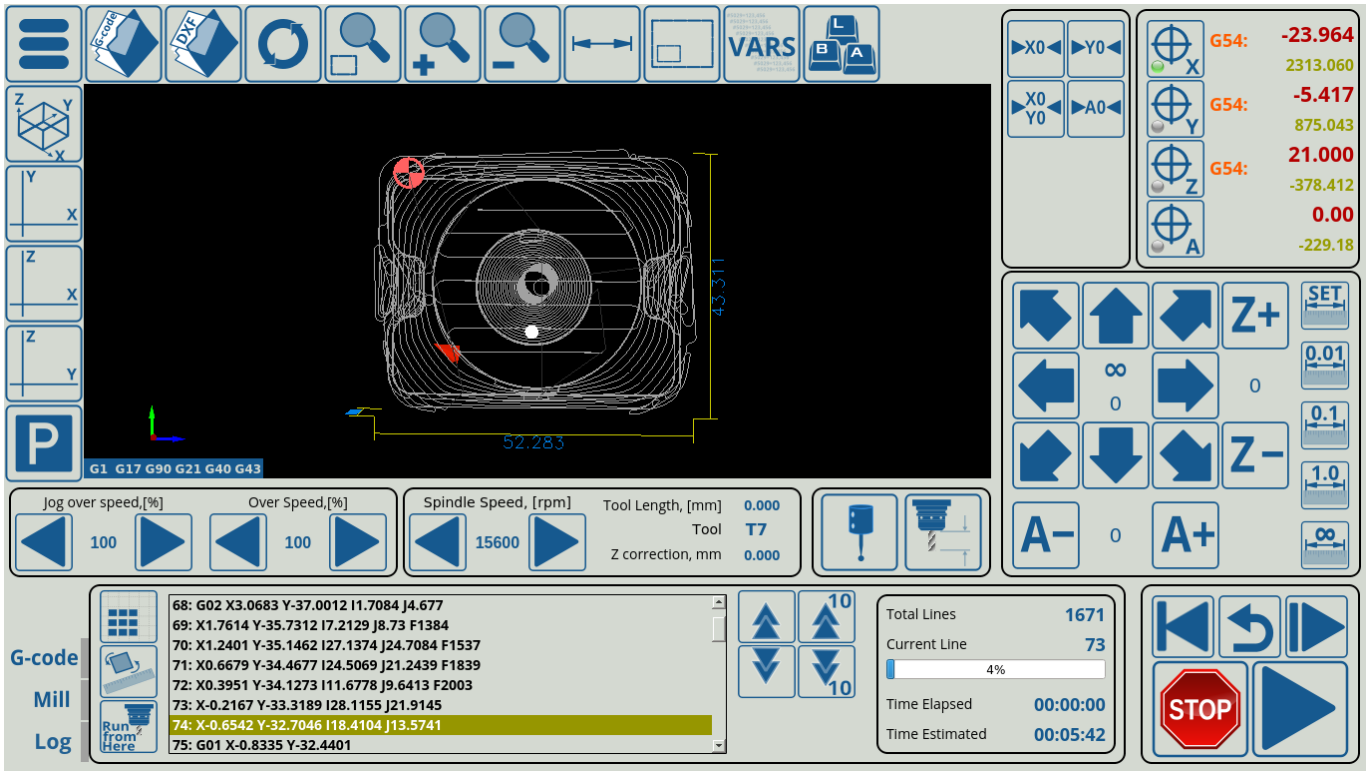
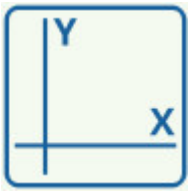
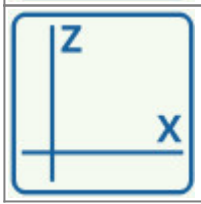
myCNC control software main screen description

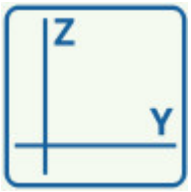
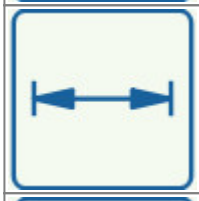
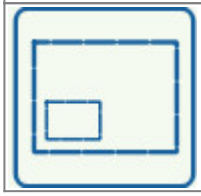
The following describes the main windows and buttons within the X1366M4 profile.

The main screen of the 4 axes Mill profile is shown below:



Description of main screen buttons and windows

	Go to the main window.
	Reorient the working area into an orthogonal view
	Reorient the file into an XY plane orientation
	Reorient the file into an XY plane orientation

	Reorient the file into an YZ plane orientation
	Open the Parking Window (see table below)
	Open a G-code file
	Open a DXF file
	Reload the program file from the hard drive
	Zoom - Fit to Window
	Zoom - Zoom In
	Zoom - Zoom Out
	Show dimensions of the nesting chart
	Show work area with the nesting chart

	Show the virtual keyboard
	Minimize the myCNC software
	Close the myCNC software
	Shut down the workstation. Note that this summons a script for the process (sudo poweroff). A similar system can be used for other actions, such as summoning applications. To have the summoned application give over control, use the “&” symbol in your custom .sh script.
	Open the myCNC settings
	Custom machine settings (see the manual for Speed Settings setup)
	A library of standard parts for cutting, milling, etc
	Open up the editor
	Open up the system diagnostics window
	Run the homing for the x-axis
	Run the homing for the y-axis

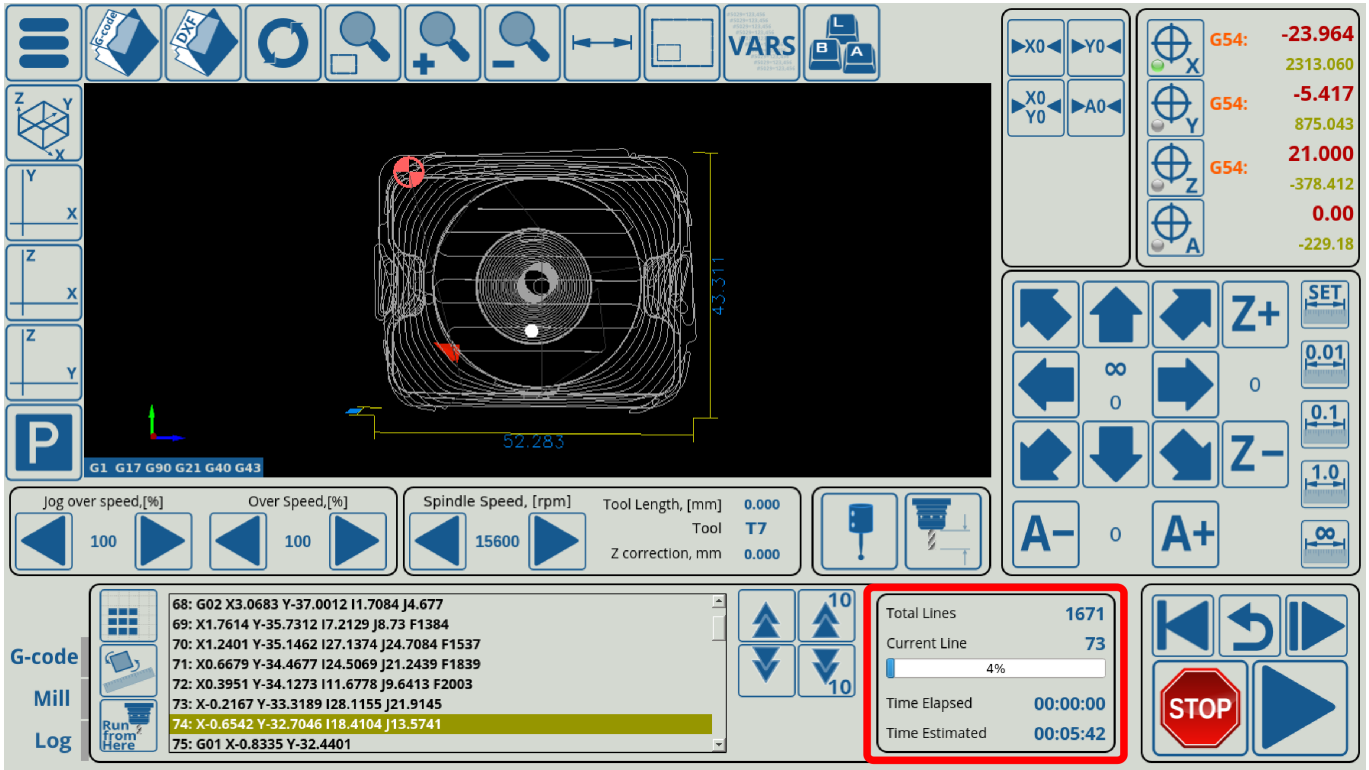
	Run the homing for the a-axis
	Run the homing for the xy plane
	Reset to zero the working x-coordinate
	Reset to zero the working y-coordinate
	Reset to zero the working z-coordinate
	Reset to zero the working a-coordinate
	Machine movement buttons (xy plane)
	Machine movement button (positive z-axis)
	Machine movement button (negative z-axis)
	Machine movement button (positive a-axis)
	Machine movement button (negative a-axis)
	Set the machine movement step size to a specified value

	Set the machine movement step size to 0.01 mm
	Set the machine movement step size to 0.1 mm
	Set the machine movement step size to 1 mm
	Set an infinite machine movement step size
	Go to the beginning of the control program
	Go forward
	Return to the working point
	Run the program
	Stop the program
	Go up one line in the program code
	Go down one line in the program code
	Go up ten lines in the program code

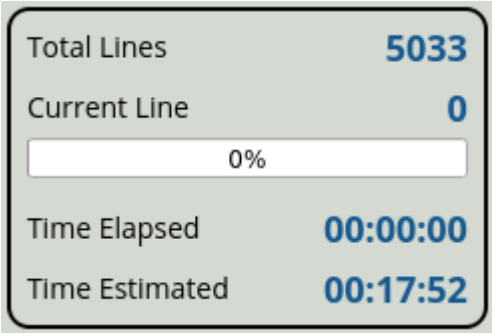
	Go down ten lines in the program code
	Open the Tool Specifications window (see table below)
	Open the Probe Sensor window (see table below)

Progress window

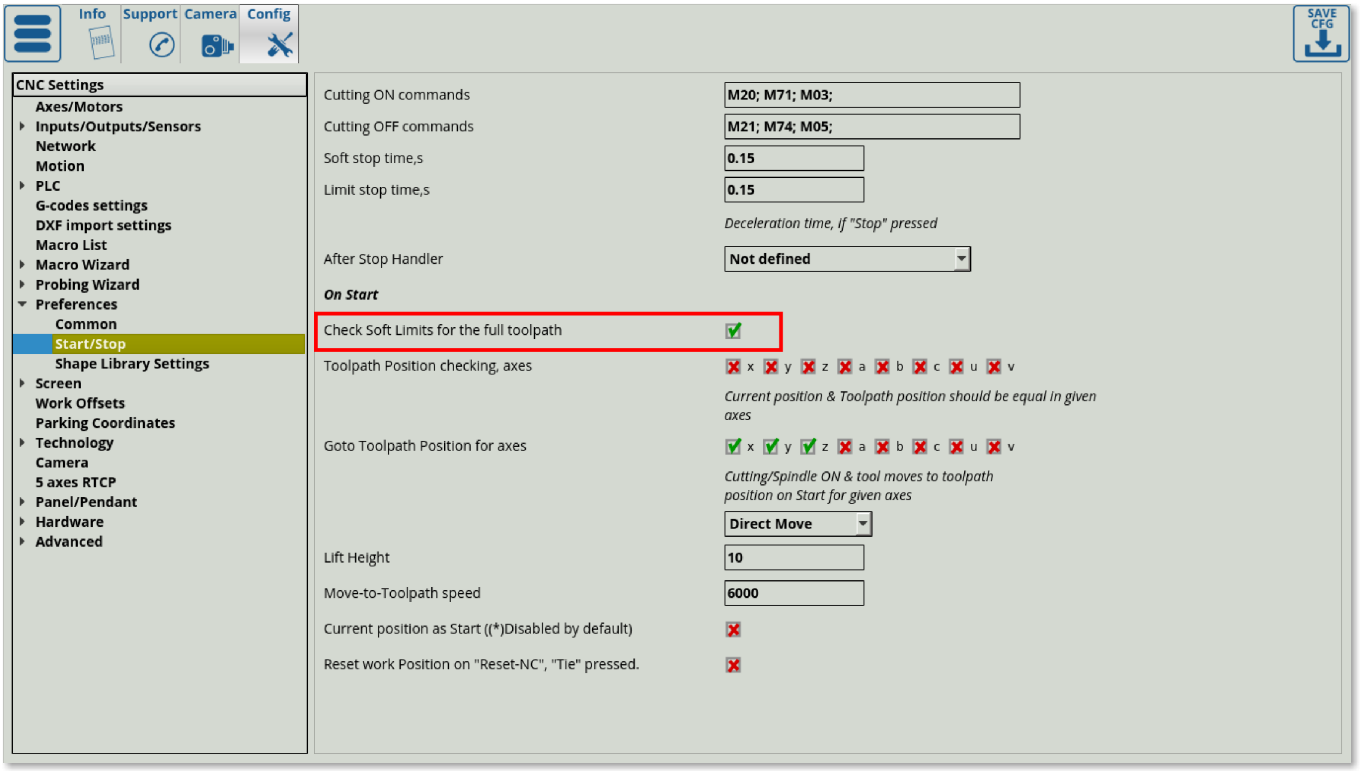
The progress window is available on the latest M4 profile versions and is located on the lower part of the main profile screen:



The progress window shows the total number of lines in the program file, the current line that the running program is on, the progress bar, the elapsed time and the estimated time remaining.

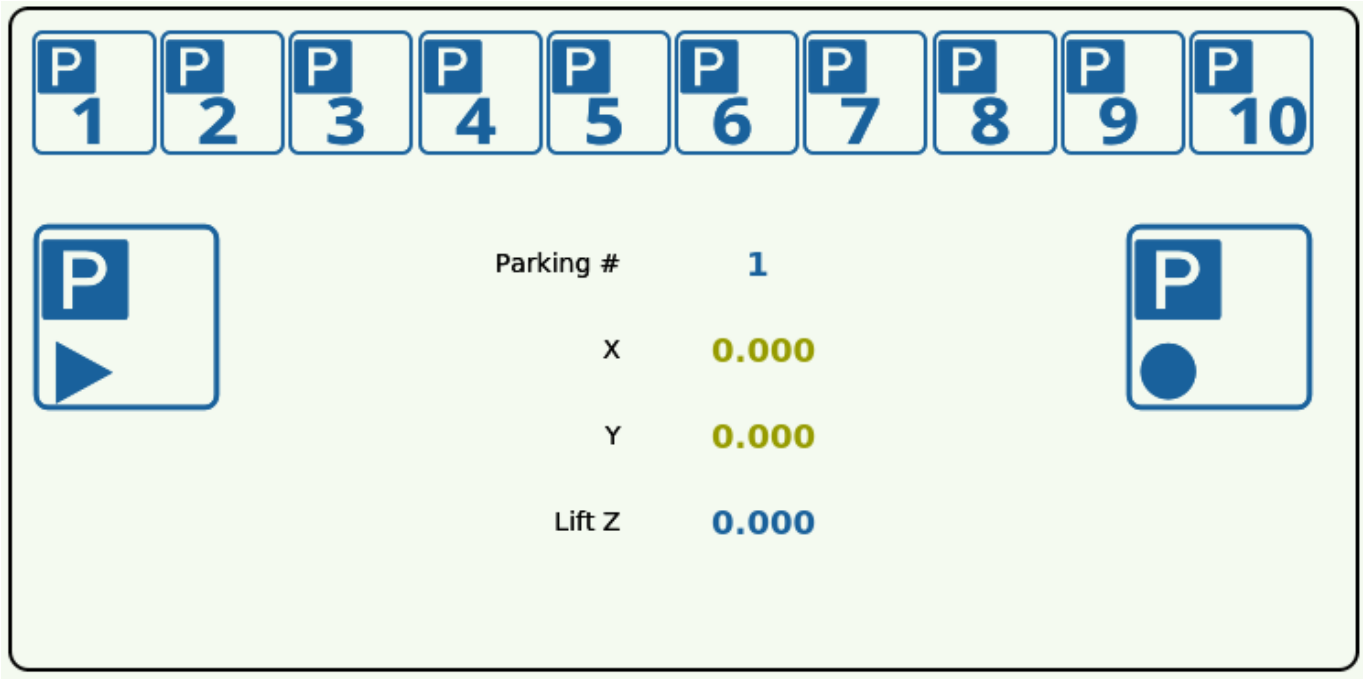


Note that the progress bar will only work properly if the following settings is turned ON: “Check Soft Limits for the full toolpath” in **Settings > Preferences > Start/Stop**.



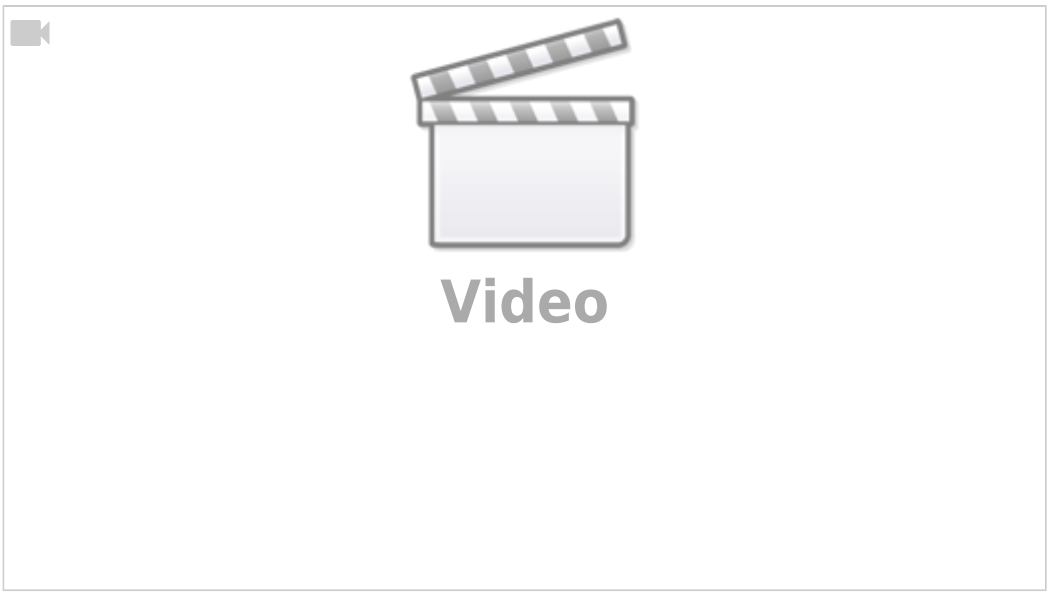
Parking Window buttons

The following Parking window opens upon pressing the Open the Parking Window button:



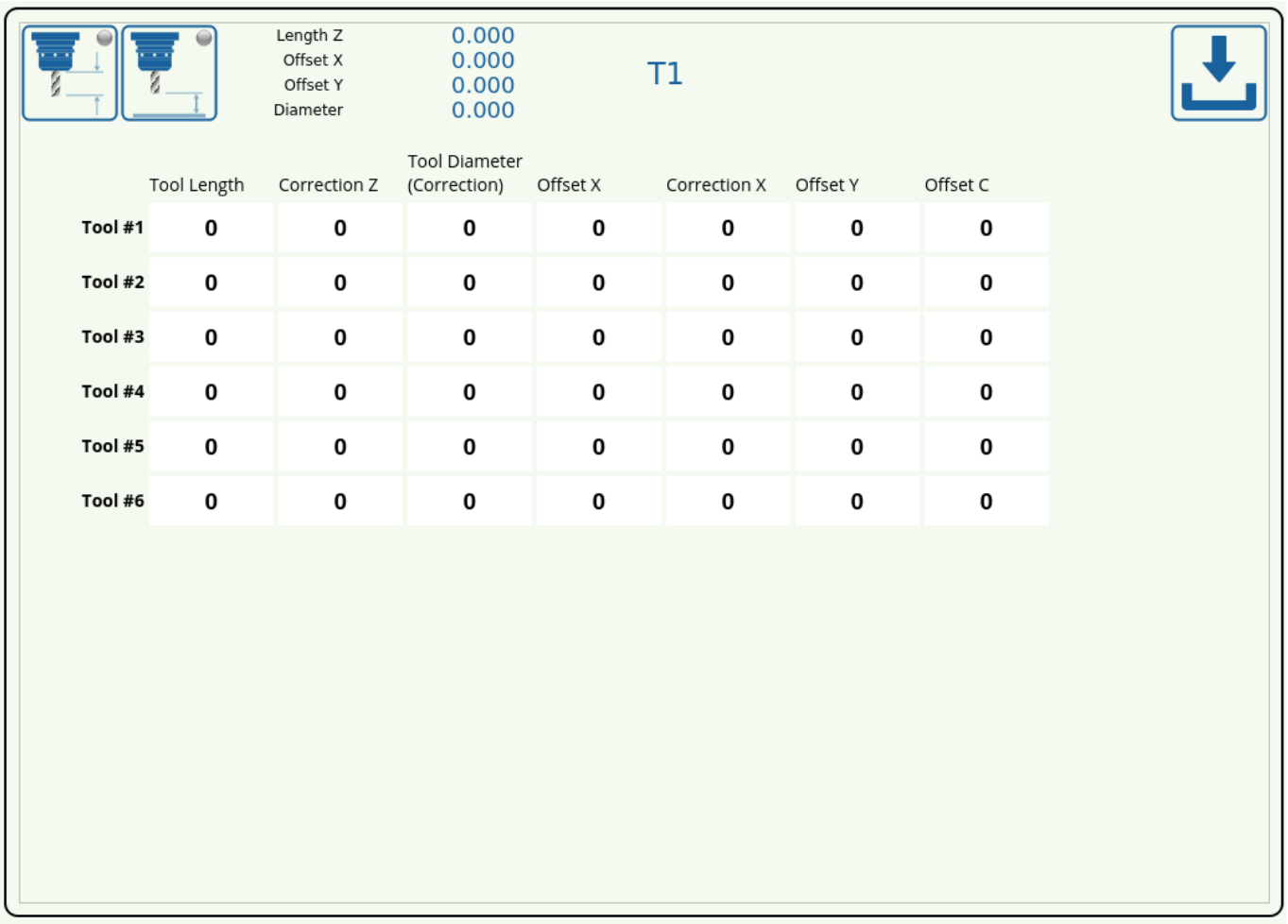
	Open the Parking Window
	Move to selected parking position
	Save current position as Selected Parking
	Choose a preset parking position

More info on the Parking widget and how to add it to profiles which don't display it by default:



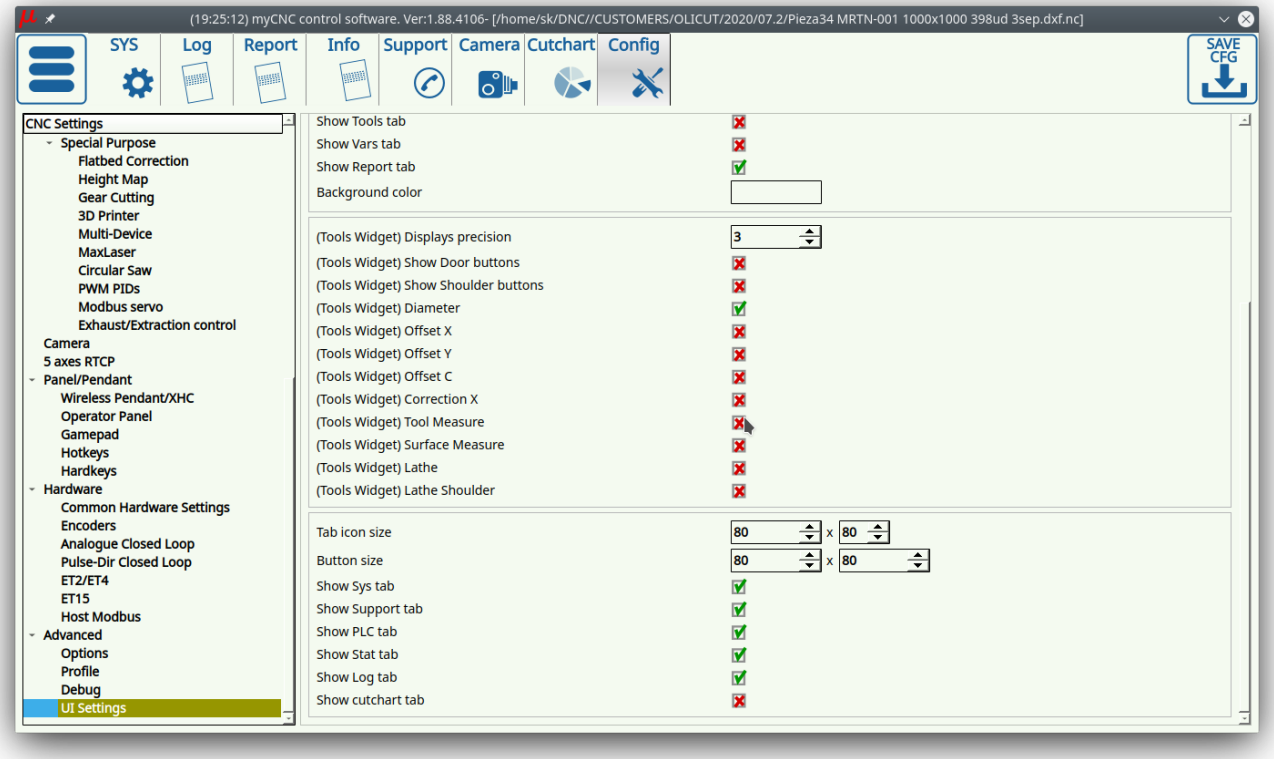
Tool Specifications Window buttons

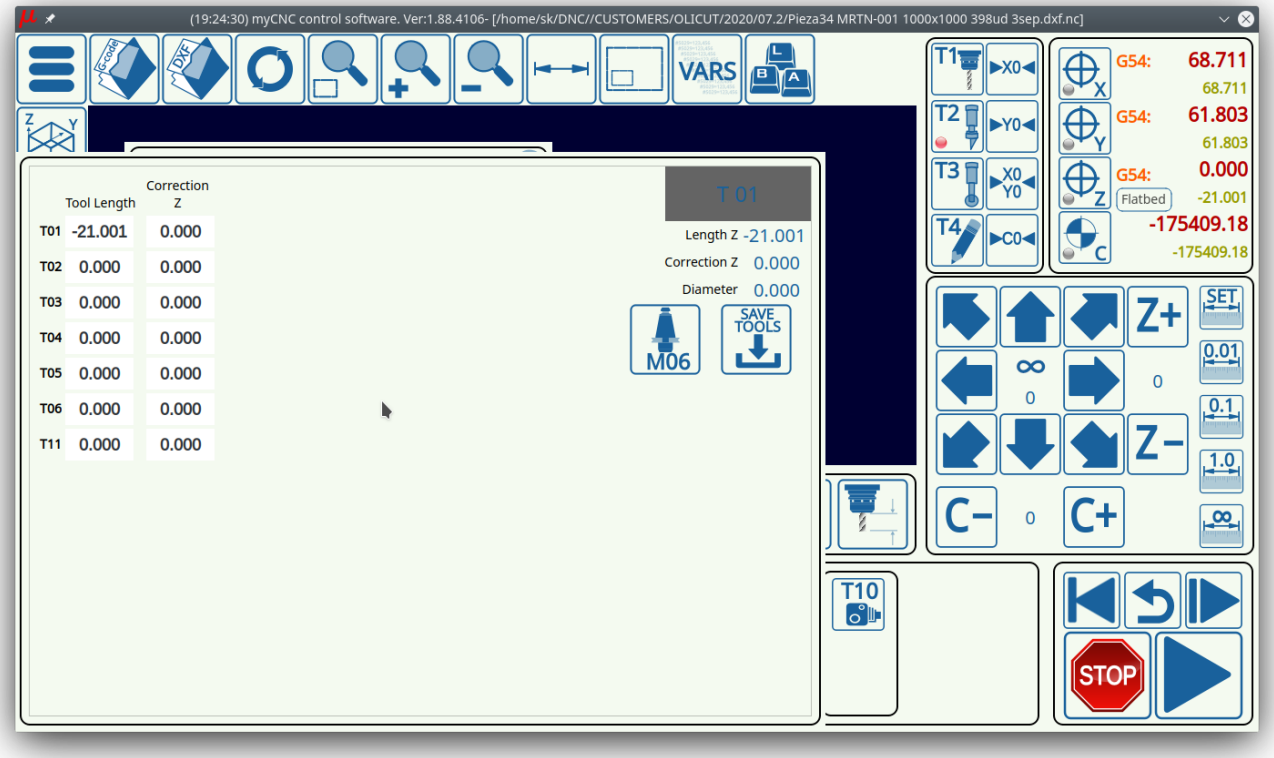
Upon pressing the Tool Specifications button, you are presented with the following window:



	Tool Length	Correction Z	Tool Diameter (Correction)	Offset X	Correction X	Offset Y	Offset C
Tool #1	0	0	0	0	0	0	0
Tool #2	0	0	0	0	0	0	0
Tool #3	0	0	0	0	0	0	0
Tool #4	0	0	0	0	0	0	0
Tool #5	0	0	0	0	0	0	0
Tool #6	0	0	0	0	0	0	0

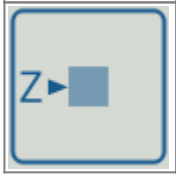
On newer profile and software versions, you are able to add additional buttons to the Tool window, by going into Settings > Advanced > UI settings:

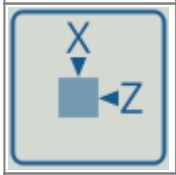
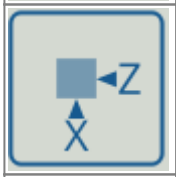




The following buttons are available:

	Run the tool length procedure (M421 macro)

	Run the surface measure procedure (M400 macro)
	Run the M06 macro
	Save the current state of the tools table
	Run the M425 command
	Set the tool setter offset
	Run the M421 command
	Close door
	Open door
	Console probe down
	Console probe up
	Left tool measure (M465)

	M464 X- Tool Measure
	M463 X+ Tool Measure
	M461 ZX- Tool Measure
	M460 ZX+ Tool Measure
	Right tool measure (M462)
	Offset X
	+Tune Z

Note that button for the M421 macro (run tool length procedure) can work with either a confirmation dialog for the Rapid Speed Length (parameter #20), or without it. The code for the two buttons is shown below:

1) Option with confirmation

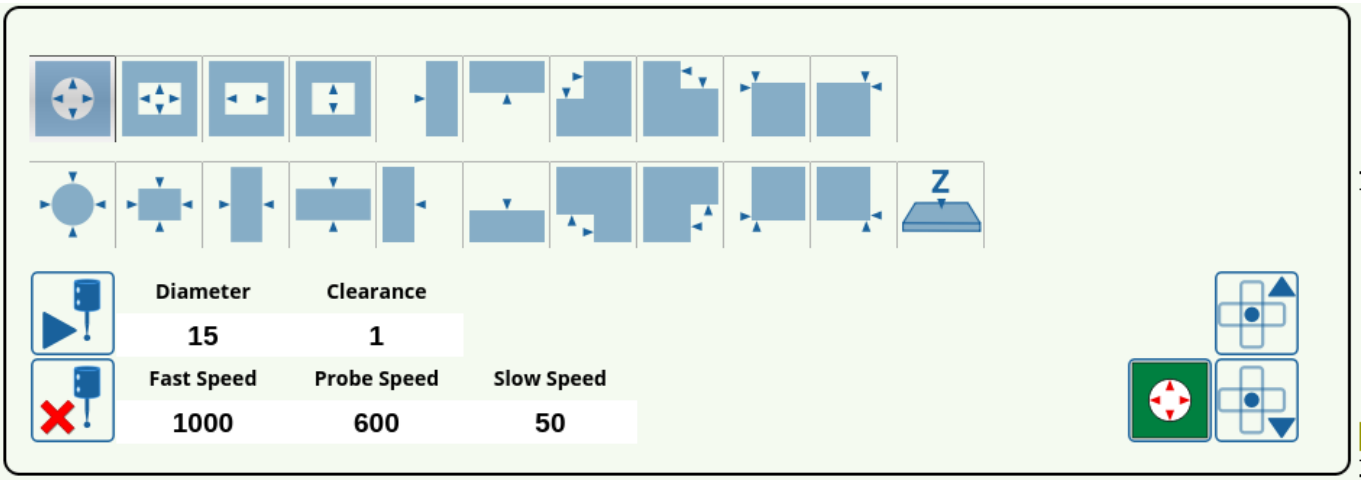
```
<gitem where="x-toolsview" position="10;10"
 width="860" height="605" name="toolsview" type="toolsview"
 fontStyle="normal" fontSize="12" noclose="1"
 tool-measure="confirm"
 bgColor="##b-main"  ></gitem>
```

2) Option without confirmation

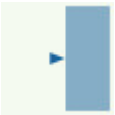
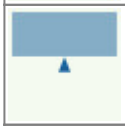
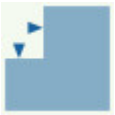
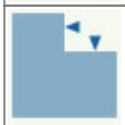
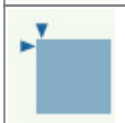
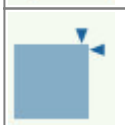
```
<gitem where="x-toolsview" position="10;10"
 width="860" height="605" name="toolsview" type="toolsview"
 fontStyle="normal" fontSize="12" noclose="1"
 tool-measure="direct"
 bgColor="##b-main"  ></gitem>
```

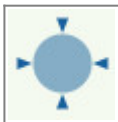
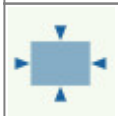
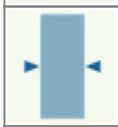
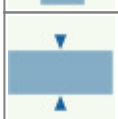
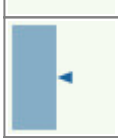
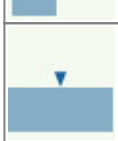
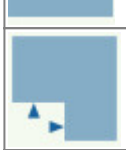
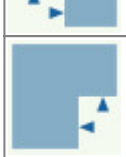
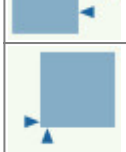
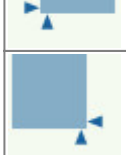
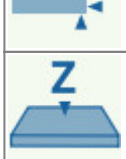
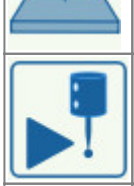
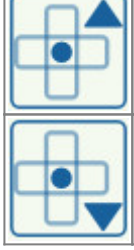
Probe Sensor Window buttons

Upon pressing the Probe Sensor button on the main screen, you are presented with the following window:



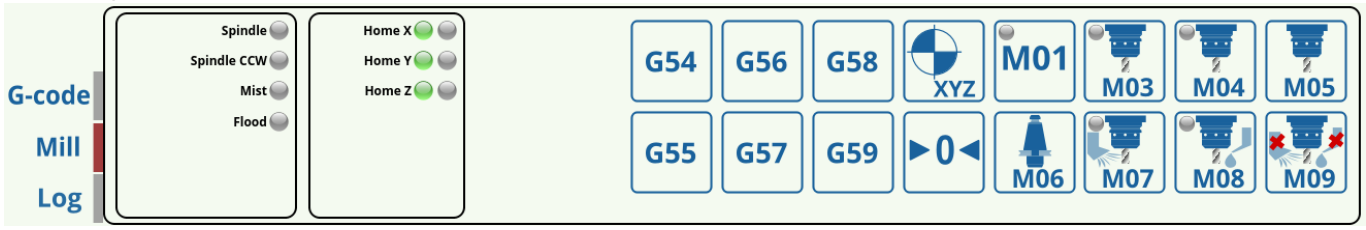
The probe sensor allows to find the edges and the corners of simple shapes and register those in the myCNC software. The following buttons can be utilized from within the software:

	Run the centering procedure for a round hole
	Run the centering procedure for a rectangular hole
	Run the centering procedure for the internal x-axis
	Run the centering procedure for the internal y-axis
	Find the edge for the positive x-axis
	Find the edge for the positive y-axis
	Find the internal corner for the negative x- and positive y-axis
	Find the internal corner for the positive x- and positive y-axis
	Find external corner for negative x- and positive y-axis
	Find external corner for positive x- and positive y-axis

	Run centering for an external round shape
	Run centering for an external rectangular shape
	Run centering for an external x-axis
	Run centering for an external y-axis
	Find the edge for the negative x-axis
	Find the edge for the negative y-axis
	Find internal corner for positive x- and positive y-axis
	Find internal corner for negative x- and positive y-axis
	Find external corner for positive x- and positive y-axis
	Find external corner for negative x- and positive y-axis
	Find the surface in the z-axis
	Run the probing procedure
	Close the probing window
	Move to the hole center
	Move the probe console up
	Move the probe console down

Mill Window buttons

Upon clicking the **Mill** button in the lower left corner of the main screen, you are presented with the following window:



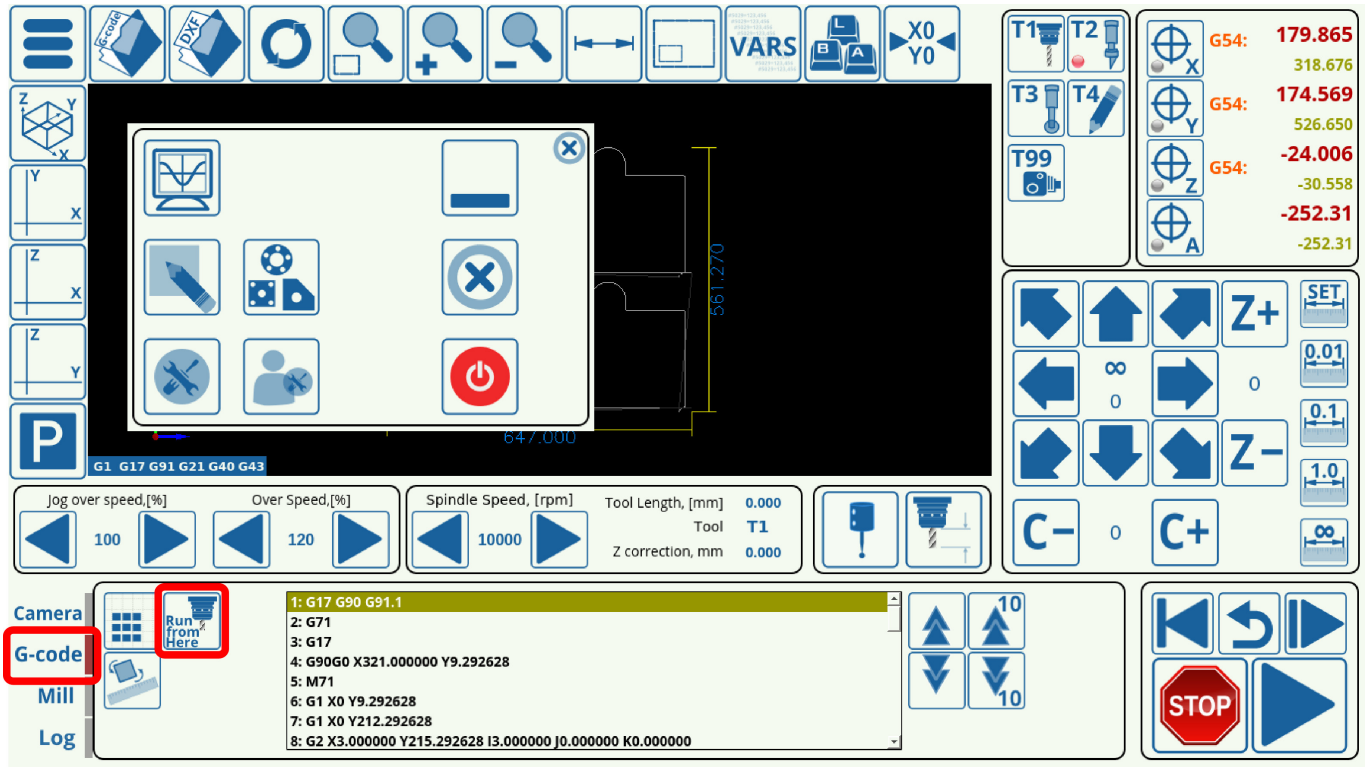
	Select the preferred coordinate system
	Run the homing procedure for all axes
	Move to work position
	Turn spindle on (clockwise rotation)
	Turn spindle on (counter-clockwise rotation)
	Turn spindle off
	Switch the working tool
	Turn the mist coolant on

	Turn the lubrication on
	Turn the coolant and lubrication off
	Conditional machine stop

Run from here

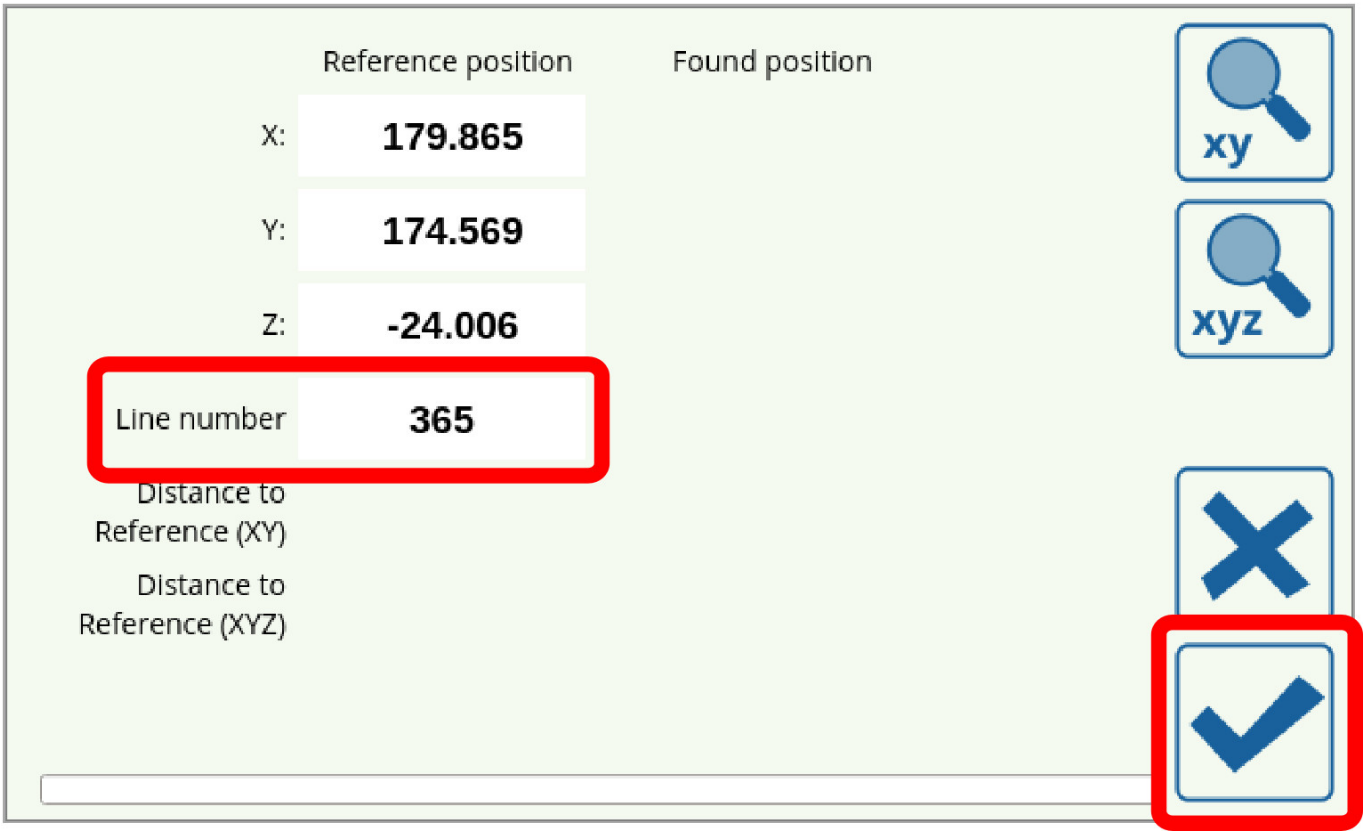
In an event of an emergency stop or failure of a mill machine running a program file, it is often useful to be able to open the file back up afterwards and be able to continue from the middle of the program without having to go through all the steps again. This is often complicated by the fact that a large program file consists of many lines of code, making it difficult to easily navigate to the place where the cutting process stopped initially. In order to facilitate finding this spot, the **Run from Here** process is put in place in myCNC software.

Upon opening the **G-Code** tab and pressing the **Run from here** button, you are presented with the following screen:



Upon clicking the **Run from Here** button, the following screen is brought up:

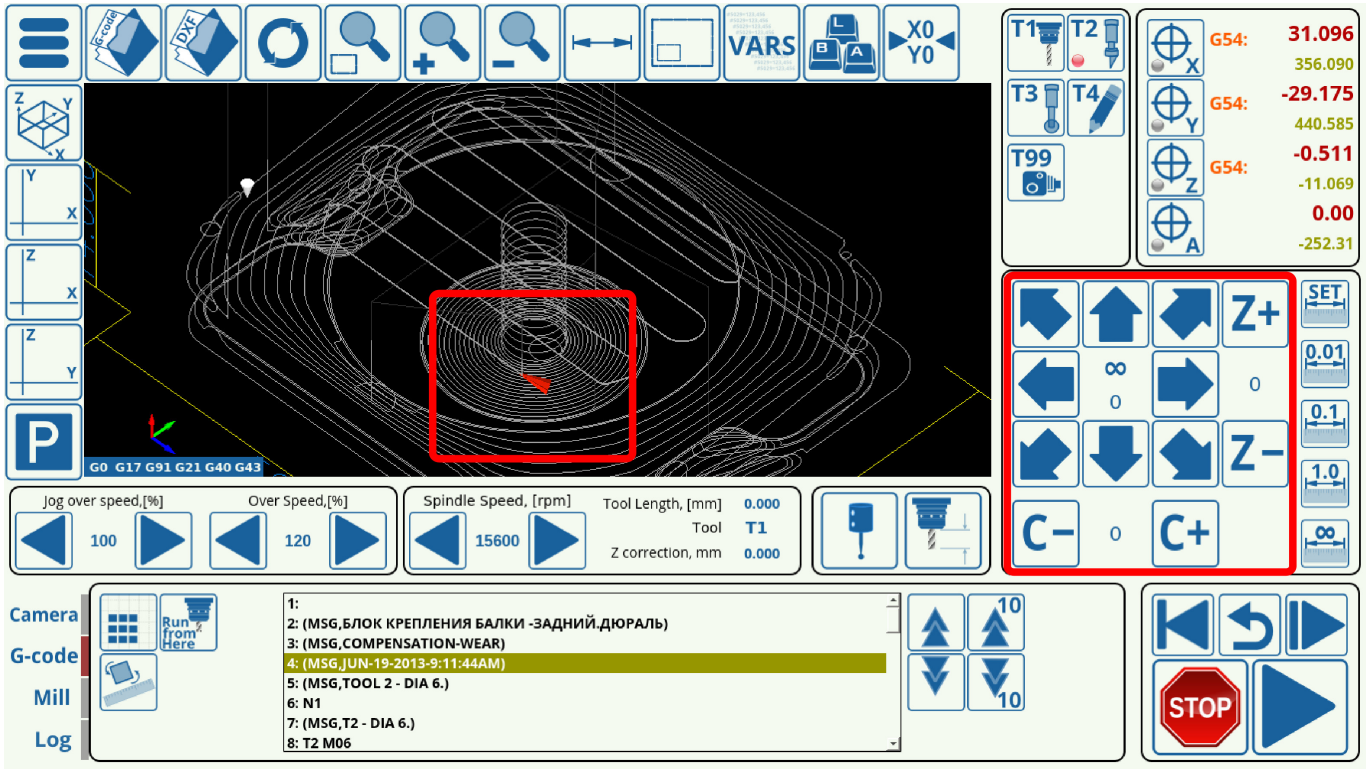
If you know the particular line number from which you want to resume the cutting process, you can enter it in the **Line Number** field, and then press OK.



If, however, you do not know the particular line number at which the program was stopped, you can use the **Found Position** function in order to approximately locate the correct line.

In order to do so:

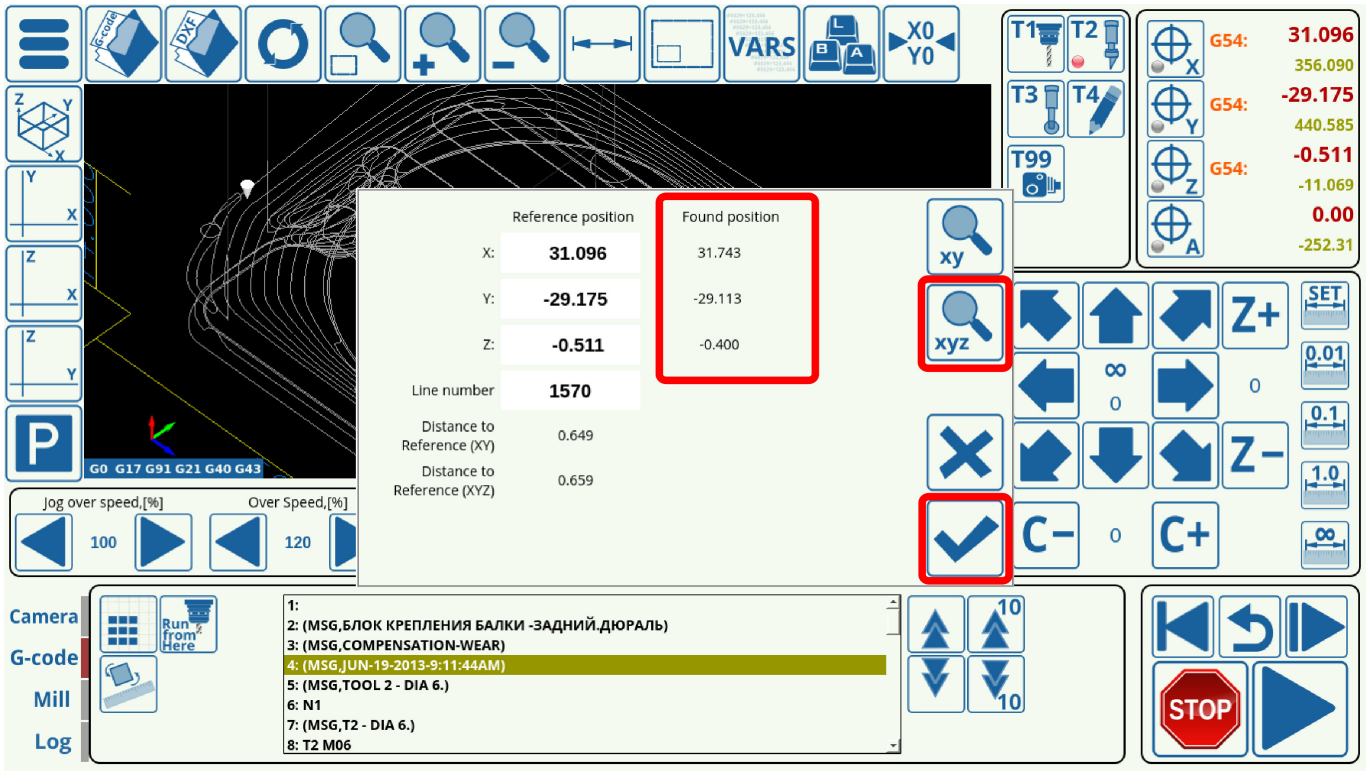
1. Navigate to the spot as close as possible to the one where the working tool has initially stopped using the onscreen buttons.



2. Click the **Run from Here** button.

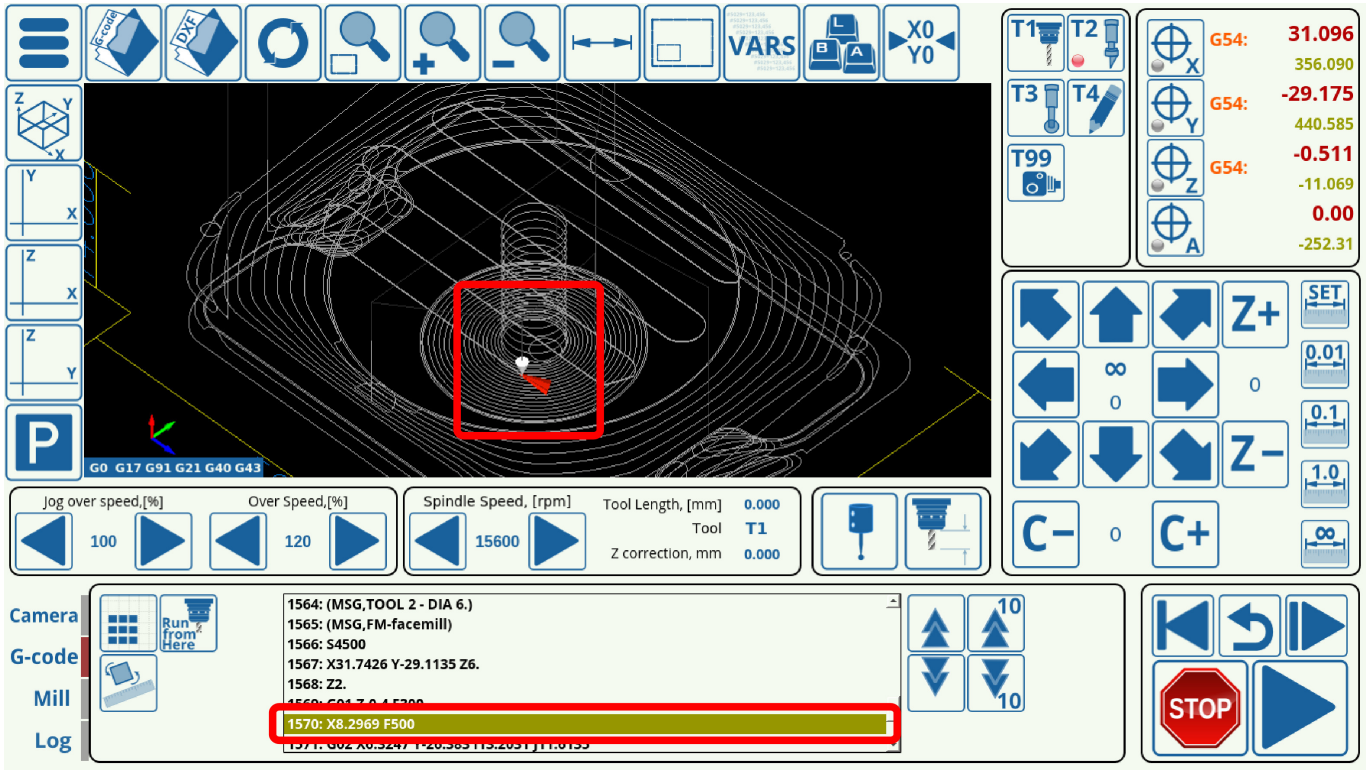
3. Depending on the specifics of your part, you can choose to either use the search in the xy or the xyz planes. The xy search is mostly used for parts that are largely flat, without a significant z-axis component to them, while the xyz will try to locate the closest point in the program file for all three axis and is primarily used for 3D components.

For this example, xyz search will be chosen, as the part used is not a simple 2D projection.



After pressing the xyz button, the Found Position coordinates appear in the popup screen. Press OK.

4. After returning to the main screen, it can be seen that the white pointer has jumped to the new Found Position, and that the selected line is now different (1570 in this example).



The program can now be run from this position without having to manually search for the correct line.

From:
http://cnc42.com/ - **myCNC Online Documentation**

Permanent link:
http://cnc42.com/mycnc/mycnc_setup_examples/four_axes_mill

Last update: **2022/09/14 12:50**