

CNC USB Controller

User manual

2014-09-10

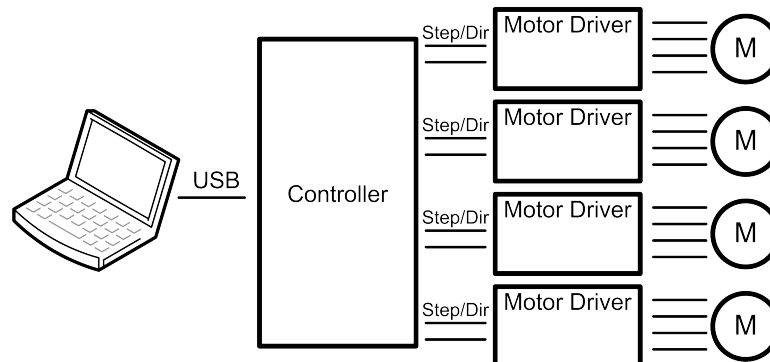
Disclaimer

CONTROLLER AND CONTROLLER SOFTWARE ARE PROVIDED TO YOU "AS IS," WITHOUT WARRANTY. THERE IS NO WARRANTY FOR THE CONTROLLER AND CONTROLLER SOFTWARE, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OF THIRD PARTY RIGHTS. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE CONTROLLER OR CONTROLLER SOFTWARE IS WITH YOU. SHOULD THE CONTROLLER OR CONTROLLER SOFTWARE PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

IN NO EVENT SHALL THE AUTHOR BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE CONTROLLER OR CONTROLLER SOFTWARE.

1 Introduction

1.1 Overview



The PlanetCNC series of USB CNC motion controllers are link between a personal computer and motor drivers supporting step/direction control. They are compatible with most drivers. The controllers use the USB port, available on all modern computers and laptops. They can serve as direct replacement or upgrade for many parallel port break-out boards.

There are currently four models available. The Mk1 controller for up to 4 axes, the Mk2 controller, using up to 9 axes, Mk2/4 controller supporting up to 4 axes and Mk3 controller supporting up to 9 axes with Ethernet support. The Mk2/4 controller is essentially a 4 axis version of the Mk2 controller, supporting the same high pulse rate and advanced software functions. It also provides DB25 and built-in screw terminal connections.

PlanetCNC controllers provide a complete, fully integrated software/hardware solution. Additional machine control software is NOT required. The USB CNC Controller software is a dedicated application, designed to fully exploit the features of the purpose-built hardware. It has many advanced features to assist day-to-day CNC machine operation.

1.2 Features and specifications:

- USB (V2.x) from PC/Laptop running Windows XP, Vista, Windows 7, 8 or 8.1 (32 bit or 64bit)
- motor driver connector pin-out is compatible with 10 pin open source interface (Linistepper, PICStep)
- controller works with most step/dir stepper and servo motor drivers available on the market
- buffered IO for maximum performance
- advanced interpolation algorithms
- start, stop, pause and resume execution of program on your machine
- standard RS274/NGC G-code (EMC2 and LinuxCNC compatible)
- advanced G-codes - G40, G41, G42 (Cutter Radius Compensation) supported
- advanced G-codes - G43, G49 (Tool Length Offsets) supported
- advanced G-codes - G54, G59.3 (Coordinate System Origins) supported
- tested with SolidCAM, MasterCAM, ArtCAM, Vectric, CamBam, MeshCAM ... generated G-code
- Profili 4-axes and 3-axes G-code supported
- import toolpath from DXF files
- import toolpath from PLT/HPGL files
- import toolpath from image files
- import toolpath from NC-Drill (Excellon) files
- import toolpath from Gerber (RS-274X) files
- toolpath simulation
- automatic homing procedure
- advanced toolchange procedures
- automatic tool length measuring
- export toolpath to G-code
- export toolpath to DXF
- SDK (software developers kit) is available
- works on MacOS with virtual machine emulating Windows

1.3 System Requirements

Minimum system requirements:

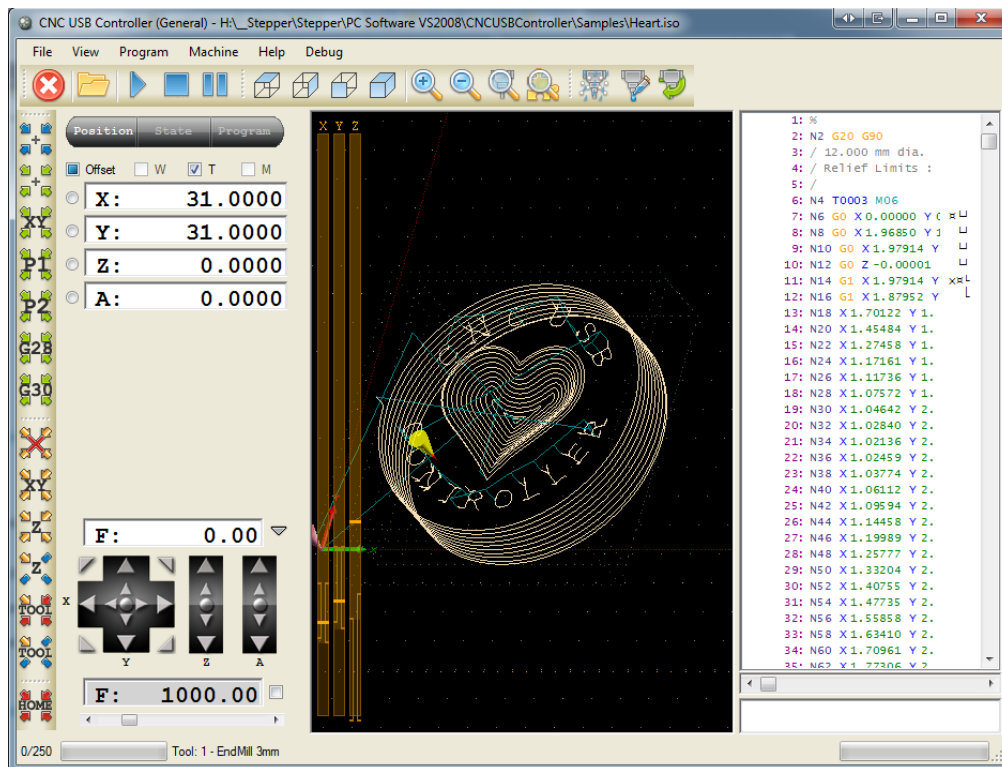
- 1 GHz or faster processor
- 512MB RAM
- 500 MB available hard disk space
- Graphics with OpenGL support
- USB 2.0 port
- .NET Framework 3.5 SP1

Recommended system requirements:

- 2 GHz or faster processor
- 2GB RAM
- 500 MB available hard disk space
- Graphics with OpenGL support
- USB 2.0 port
- .NET Framework 3.5 SP1

2 Software

2.1 Overview



CNC USB Controller software is designed to fully exploit the advanced features of controller hardware. At the same time the software remains user friendly. Even those new to CNC machining can employ advanced functions with little difficulty.

Configuration options allow for maximum flexibility, integration and customization. Automatic tool changers are supported. Tool sensor functions provide automatic tool-length detection. A 'movable' tool sensor can be used as a multifunction positioning aid.

Simulation features are designed for fast verification of NC programs. Simulation can run automatically, under keyboard or mouse control, or by selection of individual lines in an NC program. Zoom, pan or rotate of the preview does not interrupt simulation.

The software has useful G-Code manipulation and transformation functions. G-Code can be bookmarked, copied pasted and edited. It can be shifted, scaled mirrored and rotated. Code re-mapping for foam cutter applications is available.

There are many functions to assist creation of toolpaths. A wide range of content can be directly imported or converted to NC program. The 'G-Code Wizard' quickly creates pocketing and drill array toolpaths based on user specifications. .DXF layers can be managed within the software. Tool paths can export to G-Code, .DXF or .CSV.

2.2 Installation

CNC USB Controller software is compatible with Windows XP, Vista, Windows 7, Windows 8 and Windows 8.1 (32 or 64 bit). Installation is a two-part process. Driver installation is performed, after which the main application can be installed and configured. The installation process is largely 'automatic.' In most cases it's possible to accept 'default' options.

Windows 7 will display a 'publisher verification' request. This is simply a Windows 7 feature. It occurs when software from any publishers who are not part of the Microsoft 'verification' process is installed.

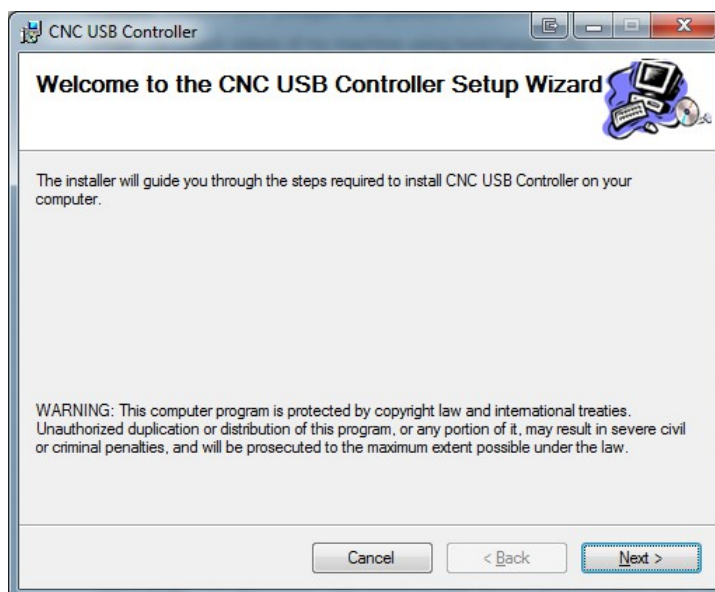
Windows 8 and Windows 8.1 driver installation will require few extra steps. Please refer to instructions below.

Some computers may require additional components to be installed. This is largely governed by the operating system and the installed software. More recent Microsoft releases contain these elements as part of the operating system. Older versions may not have them if software installed on the machine has not required their use.

Requirements:

Microsoft .NET 3.5 SP1 Framework

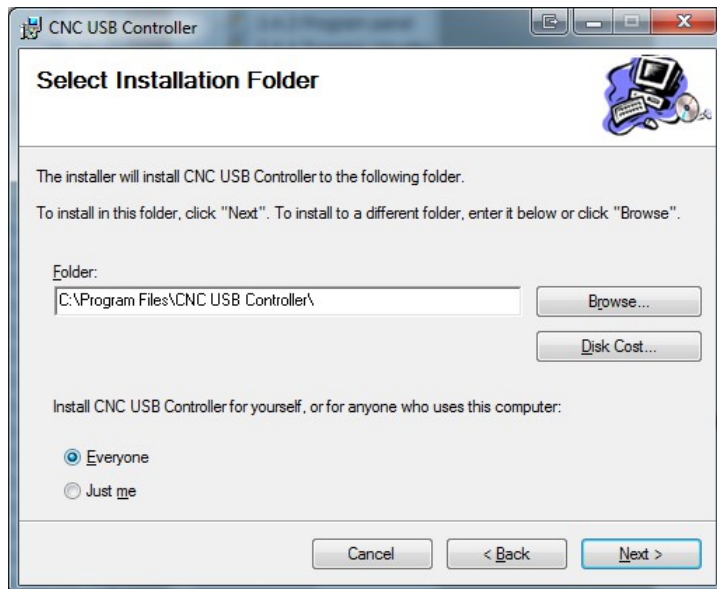
2.2.1 Software installation



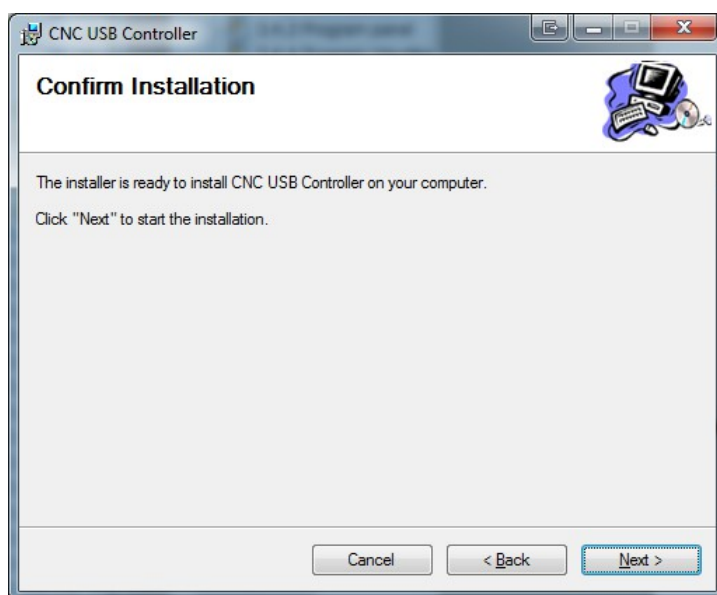
CNC USB Controller software is installed by execution of the 'CNCUSB_Setup.msi' Microsoft installer package.

The 'CNC USB Controller Setup Wizard' will launch.

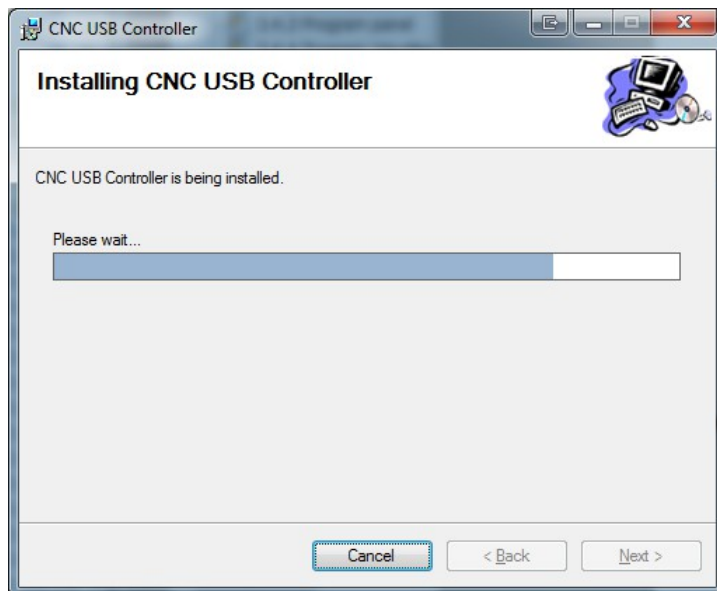
Press 'Next' to start installation or 'Cancel' to close the wizard and stop installation.



A destination folder is nominated. 'C:\Program Files\CNC USB Controller\' is the default install location. An alternate location can be selected by use of the 'Browse..' button. Press 'Next' to continue or 'Cancel' to exit the setup wizard.

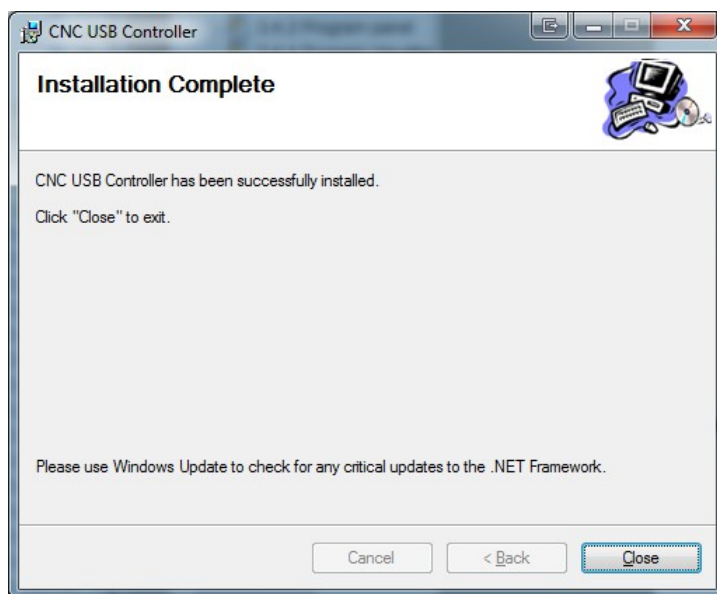


The setup wizard will request confirmation. Press 'Next' to start installation, 'Back' to adjust settings or 'Cancel' to exit the setup wizard without installation.



Wait until installation is complete.

Driver installation might start during this step. This is explained in next chapter.



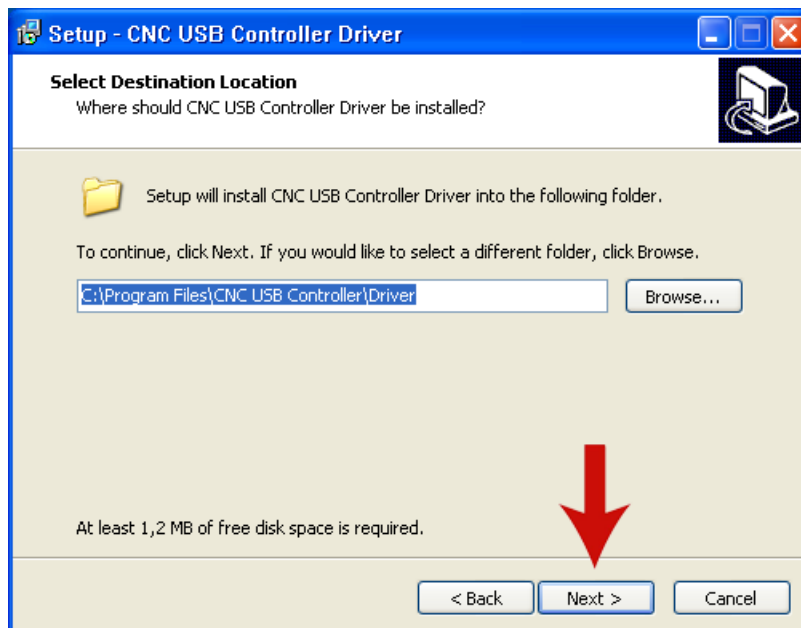
On completion the setup wizard will display a report of successful installation.

Press 'Close' to exit the wizard.

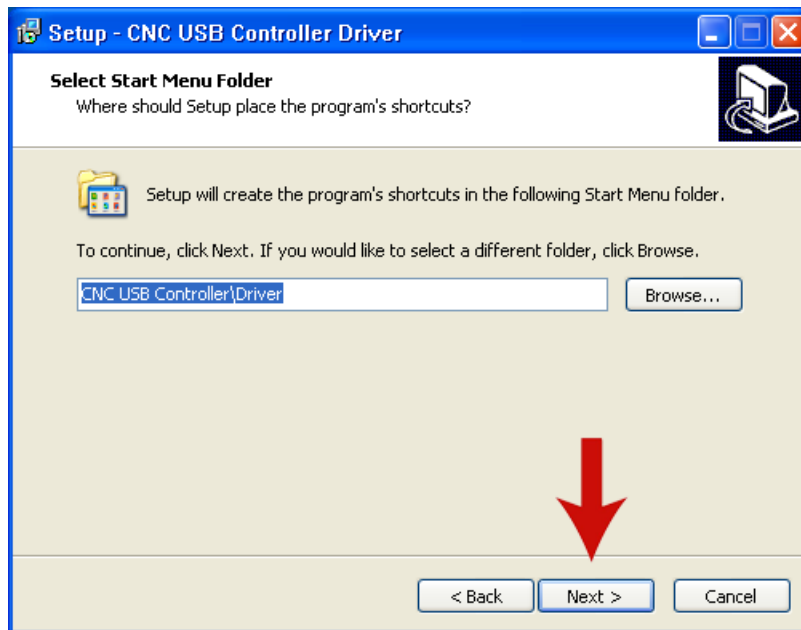
2.2.2 Driver installation



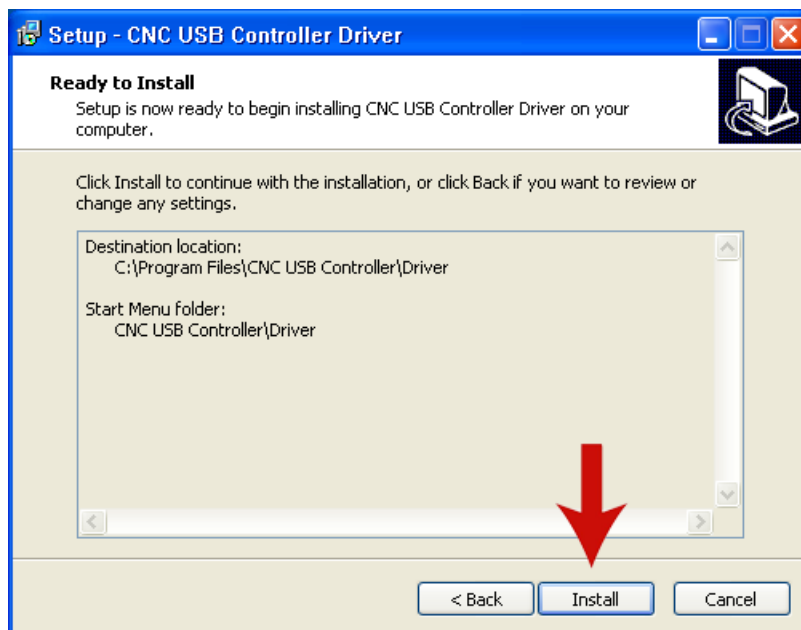
Driver installation is achieved by running “CNCUSBControllerDriver.exe”.
The ‘Driver Setup Wizard’ will launch, press ‘Next’ to continue, or ‘Cancel’ to stop installation.



The destination “C:\Program Files\CNC USB Controller\Driver” is applied by default.
An alternate location can be specified using the ‘Browse..’ button.
Press ‘Next’ to continue, or ‘Cancel’ to stop installation.



A start menu folder for shortcuts is created. “CNC USB Controller\Driver” is the default setting. An alternate folder can be specified using the ‘Browse..’ button. Press ‘Next’ to continue, or ‘Cancel’ to stop installation.

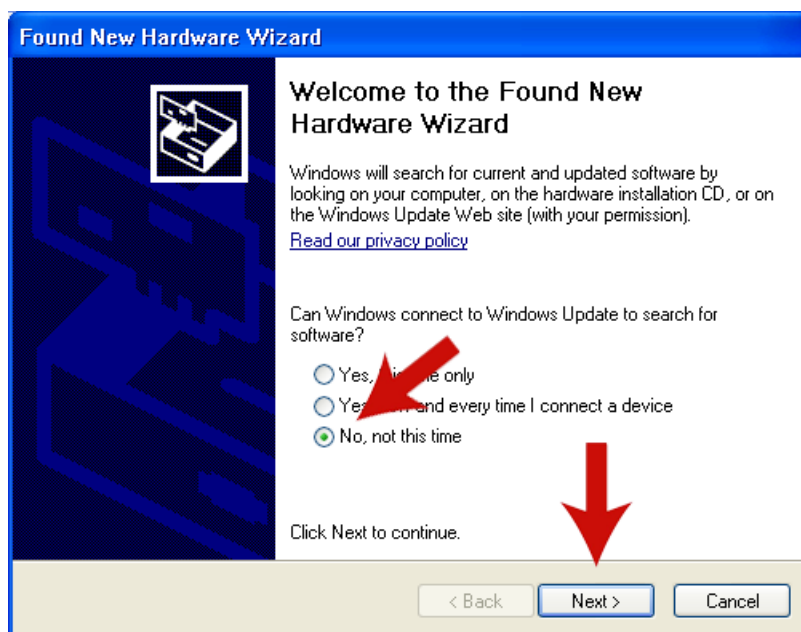


The ‘Ready to Install’ pane will appear. This provides opportunity to review, and if so desired, change setup options. Press ‘Install’ to continue, ‘Back’ to adjust settings or ‘Cancel’ to stop installation.

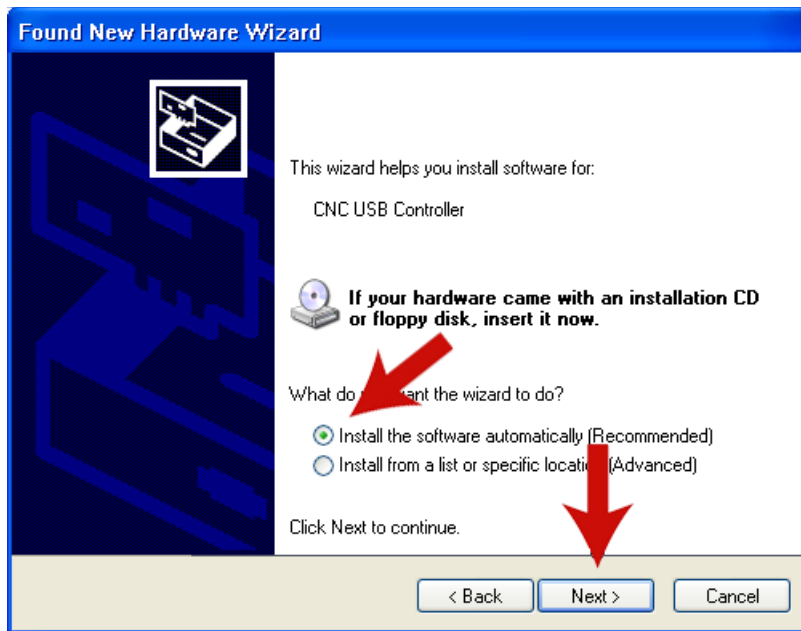


Driver is successfully installed.
Press 'Finish' to exit setup.

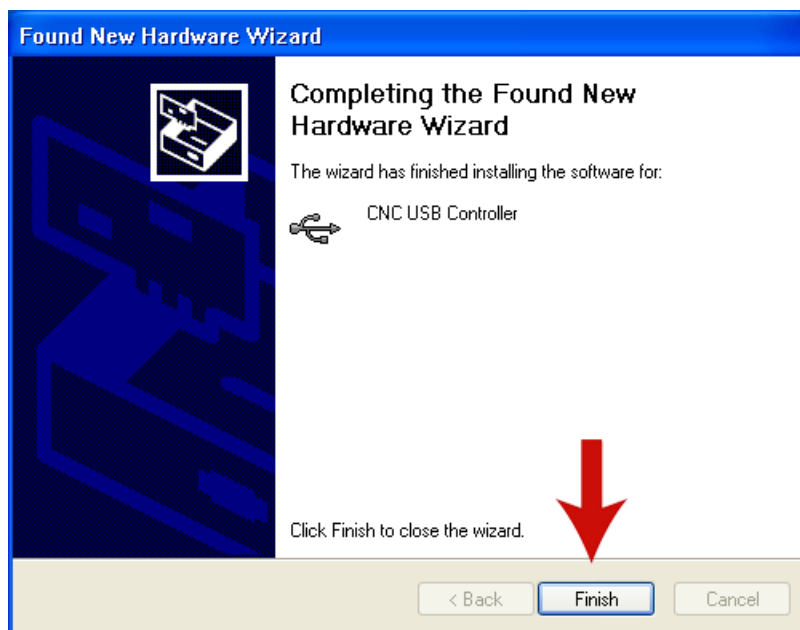
2.2.3 Completing driver installation (Windows XP)



Driver installation will complete when the CNC USB Controller is connected to the USB Port. Windows XP will detect controller hardware and the 'Found New Hardware Wizard' will launch. Use of 'Windows Update' is not required. Press 'Next' to continue, or 'Cancel' to stop installation.

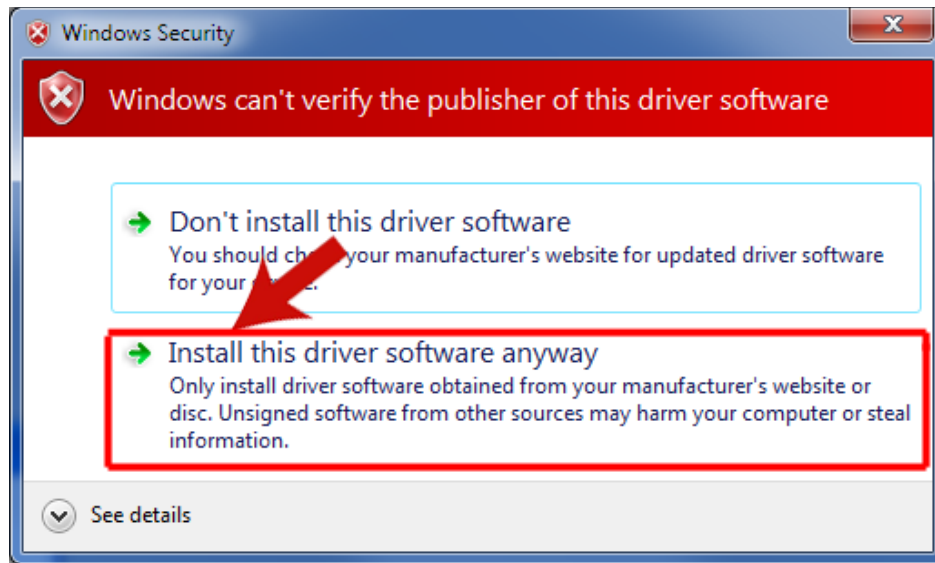


The new hardware wizard will present additional options.
'Install the software automatically' is the recommended option.
Press 'Next' to continue, or 'Cancel' to stop installation.



Windows XP will continue installation without further user input.
The 'Found New Hardware Wizard' will report success on completion.
Press 'Finish' to close the wizard.

2.2.4 Completing driver installation (Windows 7)



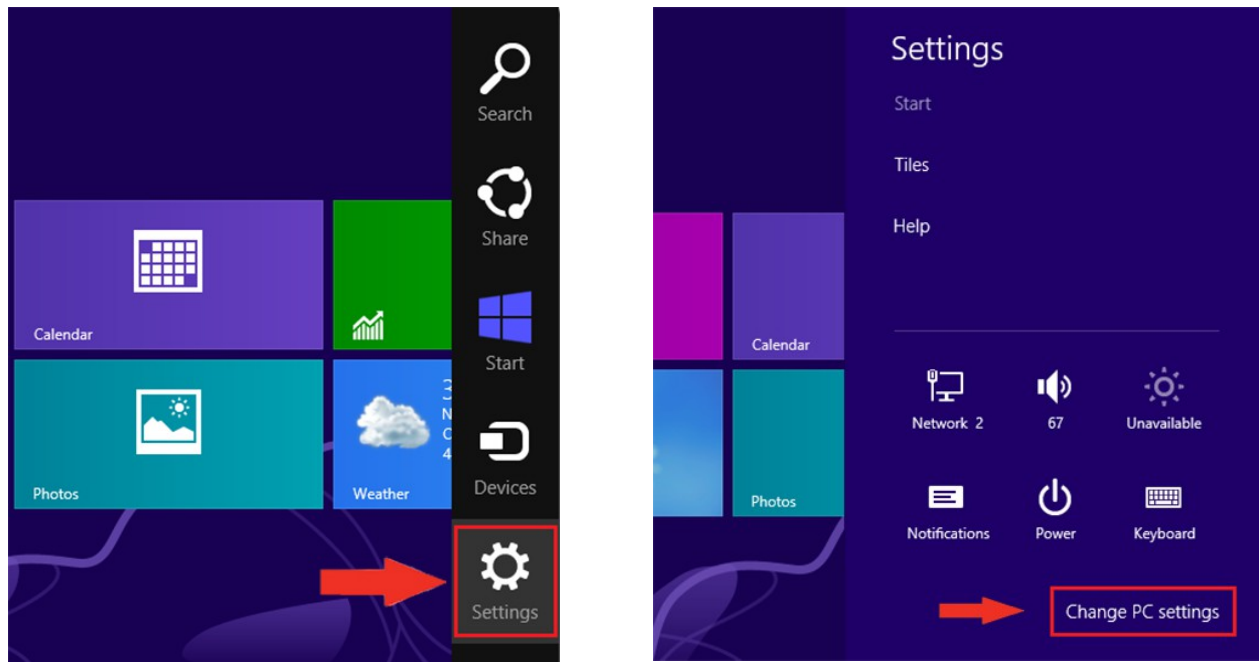
When the CNC USB Controller is connected to the USB Port, Windows 7 will detect controller hardware and begin the driver installation process. 'Windows Security' will launch a 'publisher verification' request.

Select 'Install this driver software anyway' to continue with installation, the rest of the process does not require user input. When the driver has been installed, Windows 7 will show a pop up report of success at the lower right-hand side of the 'Taskbar'.

2.2.5 Completing driver installation (Windows 8 and Windows 8.1)

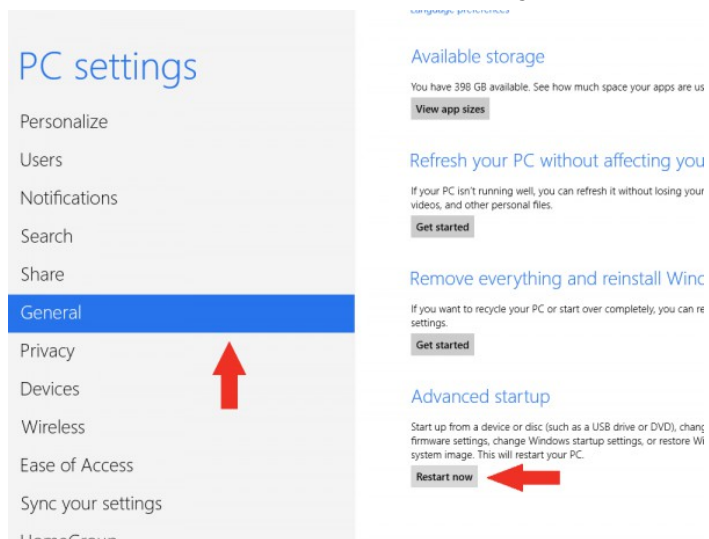
Windows 8 and 8.1 will not install drivers automatically unless they are signed.

Invoke the Charms bar and click on Settings. Open control panel by clicking on "**Change PC Settings**".



Windows 8: Select "**General**" and then "**Advanced Startup**"

Windows 8.1: Select "**Update and Recovery**" and then "**Recovery**"

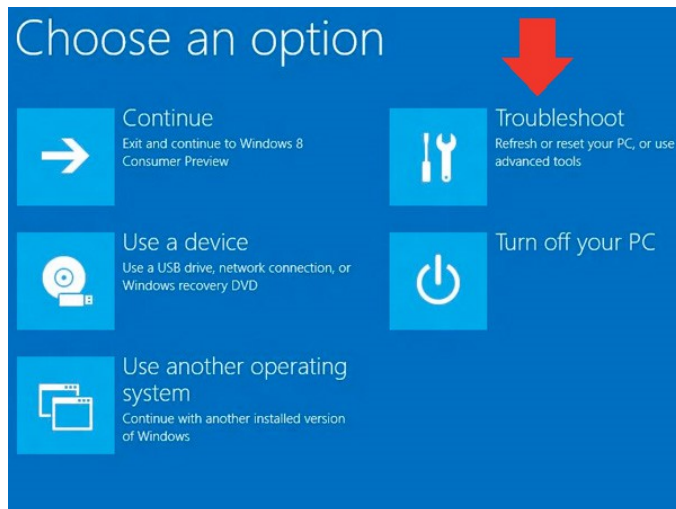


Click "**Restart now**". Now the system will restart and might take some minutes to show up the boot menu. Wait for it patiently.

After some time you will be prompted with a menu with following options:

- Continue
- Troubleshoot
- Turn off

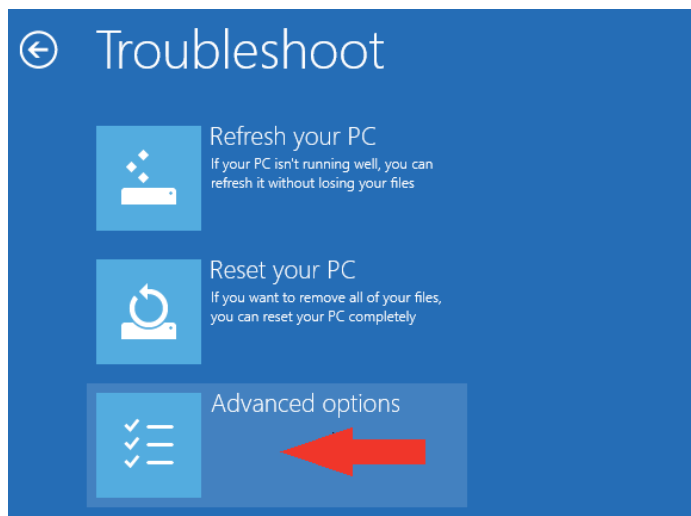
Choose "**Troubleshoot**".



Then the following menu appears:

- Refresh your PC
- Reset your PC
- Advanced Options

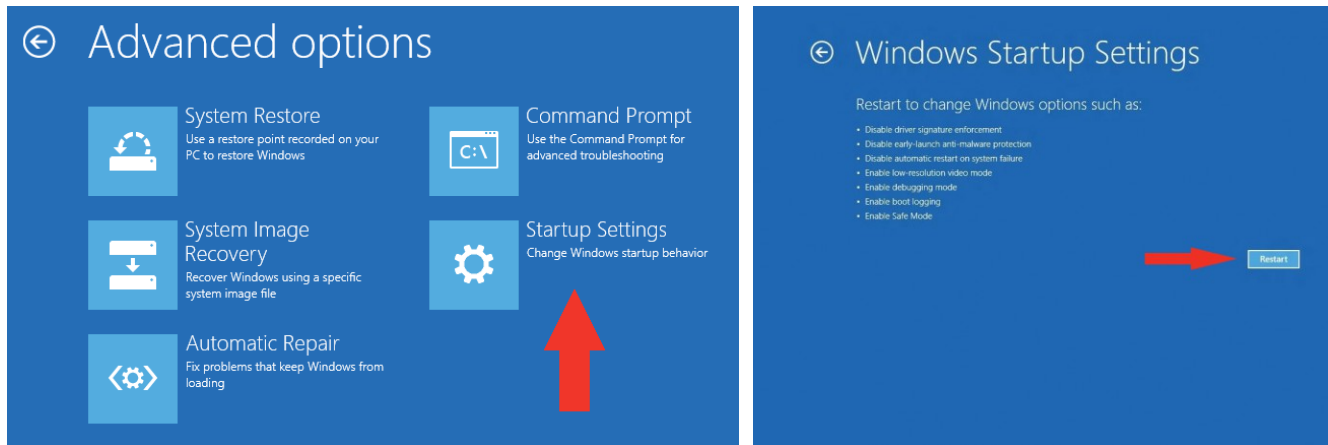
Choose "**Advanced Options**".



Then the following menu appears:

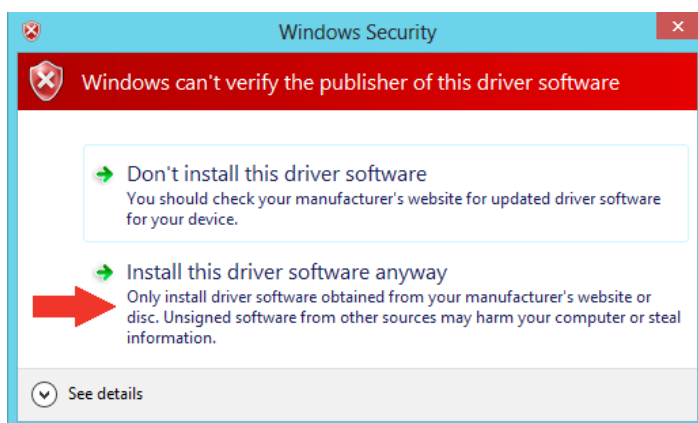
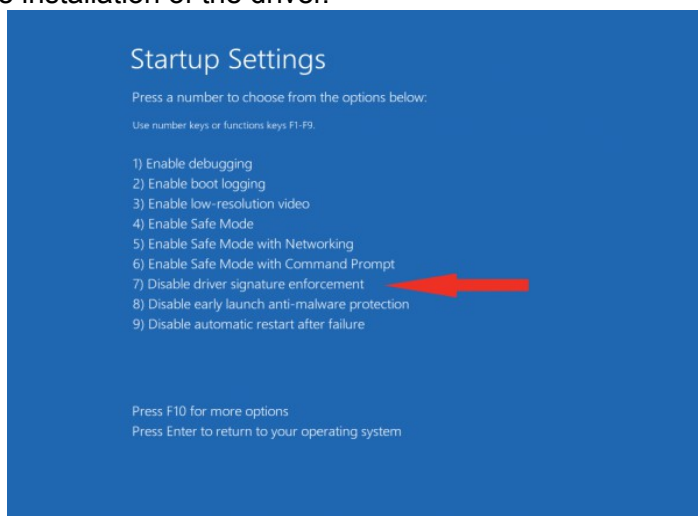
- System Restore
- System Image Recovery
- Automatic Repair
- Command Prompt
- Windows Startup settings

Choose "**Windows Startup Settings**", then Click "**Restart**".

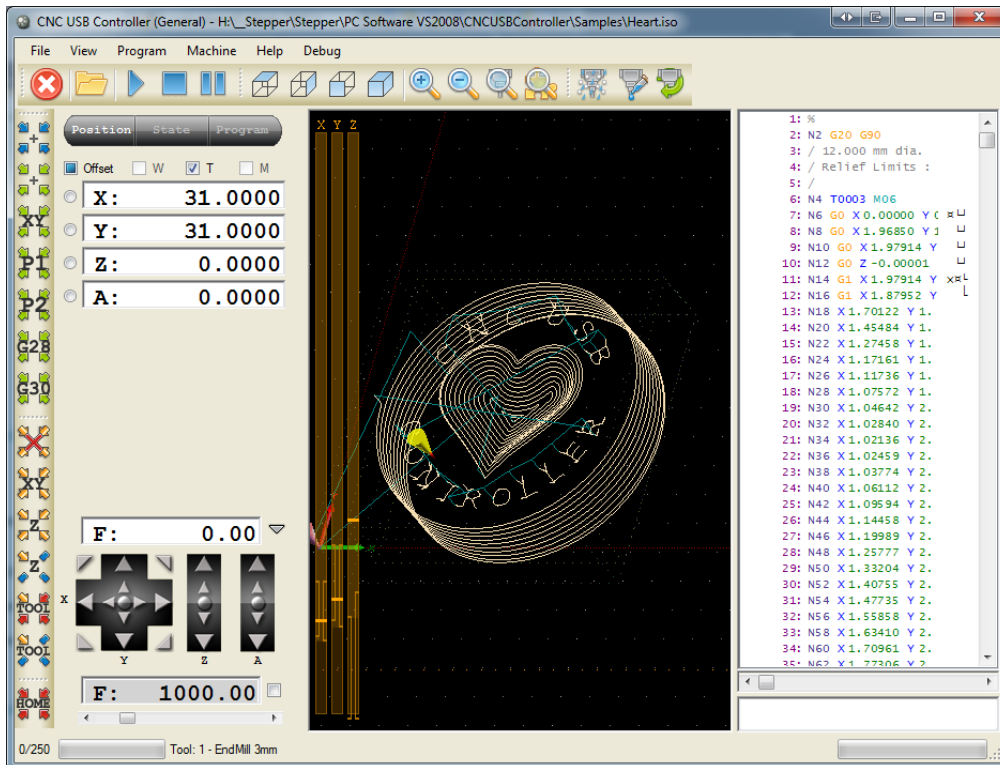


Now the computer will restart and the boot menu appears.

Choose "**Disable Driver signature Enforcement**" from the menu. Now windows will start and you can do the installation of the driver.



2.3 Main window



On launch of CNC USB Controller software the 'Main Window' will display. Data and status information for all aspects of G-Code program execution is provided on this main screen. The interface has been designed to be simple to use. Many regularly used functions are available via on-screen buttons and keyboard 'hot-keys'.

There are seven areas within the window. Each provides information or function concerning specific aspects of G-Code program execution. For the purposes of illustration, color highlights in show the various areas as described below.

- Position / State / Program panels.
- Program visualization/preview.
- G-Code panel.
- Manual data input (MDI) panel.
- Menus and toolbars.
- Status bar.
- Function shortcuts.

On screen Jog controls are provided for position adjustment if a hardware jog keyboard is not available. Alternative screen layout, with wider aspect visualization and G-Code display, is available by pressing F11.

2.3.1 Position panel

The panel displays currently active offsets, machine position and speed.

On panel are jog buttons used to move machine to desired position.

There is also speed setting with speed override slider and check box.

Offset:

W – Working offset

T – Tool offset

M – Matrix transformation

If check box is enabled then offset is set and active.

If check box is checked then offset is included in position display.

It is also used with “Set Position” and “Go To” commands.

Machine position:

Shows current machine position. If offset check box is checked then its value is included in position.

User can enter new value. It will be used to execute “Set Position” command. Each position value has small button in front which is used to “Set Position” to zero.

Executing “Set Position” will change machine absolute position! If absolute position is changed automatic tool change and fixed tool sensor position might become invalid!

The screenshot shows the 'Position' panel of a machine control interface. At the top, there are three tabs: 'Position', 'State', and 'Program'. Below the tabs, there are three checkboxes: 'Offset', 'W', 'T', and 'M'. The 'Offset' checkbox is checked. Below these are three input fields for X, Y, and Z coordinates. The X coordinate is 15.52750, the Y coordinate is 66.58750, and the Z coordinate is 12.10875. Each input field has a small circular button to its left. Below the coordinates, there are two more input fields: 'SPD' (0.00) and 'OVRD' (300.00). The 'OVRD' field has a small square checkbox to its right. Below these are two jog controls: a cross-shaped joystick for X and Y, and a vertical joystick for Z. At the bottom, there is a 'JOG' input field with the value 2500.00 and a small square checkbox to its right.

Position	State	Program
☒ Offset	☐ W	☐ T
☒ M		
X: 15.52750		
Y: 66.58750		
Z: 12.10875		
SPD: 0.00		
OVRD: 300.00	☐	
X	Y	Z
JOG: 2500.00		

Position display color description:

Red and purple: Limit switch active and machine stop enabled. (Red for negative direction and purple for positive direction)

X: 55.15625 X: 35.75250

Light red and light purple: Limit switch active and machine stop disabled.

X: -9.17250 X: -9.17250

Yellow and Orange: Outside machine soft limits with soft limits enabled. (Yellow for negative direction and orange for positive direction)

X: -179.69500 X: 30.62125

Light green: Outside machine soft limits with soft limits disabled.

X: 11.90750

Light blue: Editing enabled

Y: 751.91624

Speed display:

Shows current machine speed in units/minute.

Jog:

Click+hold on large arrows jogs until the mouse button is released. Corner buttons allow 'diagonal' or combined axis jogging. Smaller arrows, near the center of jog controls, perform a single 'step' move per click. Jog step increment is defined in 'Settings' or adjusted via a drop-down on the arrow next to the current machine speed indicator.

Speed set:

Shows set speed value. If enabled with check box then "speed override" is active. Value can be entered or adjusted with slider below.

This value also sets jogging speed.

Note: using "speed override" is not recommended method for changing speed. If you need to run NC program on machine with different speed then you should use menu command "Program/Set Speed..."

WARNING:

If absolute position is changed automatic tool change and fixed tool sensor position might become invalid!

2.3.2 State panel

The State panel displays a range of information in respect of active parameters and settings. Information has reference to current machine state and program simulation state.

Information displayed on the panel includes:

2.3.2.1 Current:

Units:	millimeters or inches
Plane:	XY, YZ or ZX
Mode:	absolute or relative
Speed:	current speed
Feed:	feed speed value
Traverse:	traverse speed value
Flood:	ON/OFF
Mist:	ON/OFF
Spindle:	direction & speed

2.3.2.2 Axes:

X, Y, Z:	Shows axis coordinate values, if line is selected from program then this is indicated with (SYM).
Work Offset X:	Working offset value of X axis
Work Offset Y:	Working offset value of Y axis
Work Offset Z:	Working offset value of Z axis
Tool Offset Z:	Tool offset value
Tool Offset X:	Tool offset value
Absolute:	True if moving in absolute machine coordinates

2.3.2.3 Origin:

Coordinate S..:	Number of used coordinate system.
Offset:	Shows all offsets of current coordinate system.

2.3.2.4 Tool:

Tool Offset Z:	Shows tool offset for Z coordinate defined with G43 g-code
Tool Offset X:	Shows tool offset for X coordinate

Tool:	tool number
Name:	tool name
Shape:	tool shape
Diameter:	tool diameter

Length: Current status of completion in percentage, number of currently executing G-code line in relation to total number of g-code lines.

Line: Number of currently executing G-code line

G-code: Corresponded G-code of currently executing G-code line

PositionStateProgram

Current

Units	Millimeters
Plane	XY
Mode	Absolute
Speed	84.32
Feed	1100.00
Traverse	1000.00
Flood	Off
Mist	Off
Spindle	Unknown (-1)

Axes

X	113.794
Y	-46.328
Z	1.459
Work Offset X	84.250
Work Offset Y	-87.565
Work Offset Z	7.914
Tool Offset Z	0.000
Tool Offset X	0.000
Absolute	False

Origin

Coordinate S...	1
Offset	X0.00000 Y0.00000 Z0.00000 ...

Tool

Tool Offset Z	0.000
Tool Offset X	0.000
Tool	0 (0)
Name	Unknown
Shape	Unknown
Diameter	0.000

Length	11.14% 81.16/728.63
Line	48
G-Code	G02 X29.55000 Y41.24000 I18...

2.3.3 Program panel

The program panel shows information about the active NC program.

Information displayed includes:

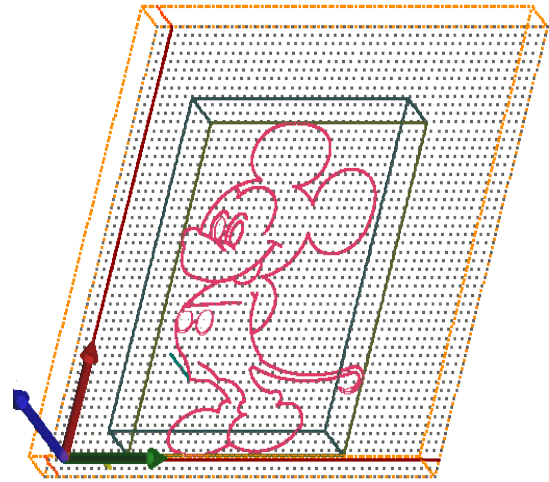
Name: name of program
Units: units used in this panel
Min/Max: minimum and maximum values used in program
Min Feed, Max Feed: minimum and maximum values used for feed moves (cutting extents)
Length: length of toolpath
Time: estimated time needed to execute program
Time (O): estimated time needed to execute program if using current speed override value.

Position		State	Program
Name			
Units		Millimeters	
Min X		16.797	
Max X		74.310	
Min Y		0.873	
Max Y		77.692	
Min Z		0.000	
Max Z		0.000	
Min Feed X		16.797	
Max Feed X		74.310	
Min Feed Y		0.873	
Max Feed Y		77.692	
Min Feed Z		0.000	
Max Feed Z		0.000	
Max Speed		1100.000	
Min Speed		1000.000	
Max Feed Speed		1100.000	
Min Feed Speed		1100.000	
Length		728.625	
Time		00:00:51	
Time (O)		00:00:51	

2.3.4 Graphical program visualization display

This display shows 3D representation of machine, tool, toolpath,...

- Orange box – machine limits
- Gray grid – machine table
- Axis arrows – displayed coordinate system origin
- Red axis lines – selected coordinate system origin
- White line – feed toolpath
- Cyan line – traverse toolpath
- Red line – selected toolpath section
- Yellow cone – current machine position
- Orange cone – simulated position
- Dark green/gray cone – G28 and G30 positions
- Dark green/gray – program extents and program cutting extents

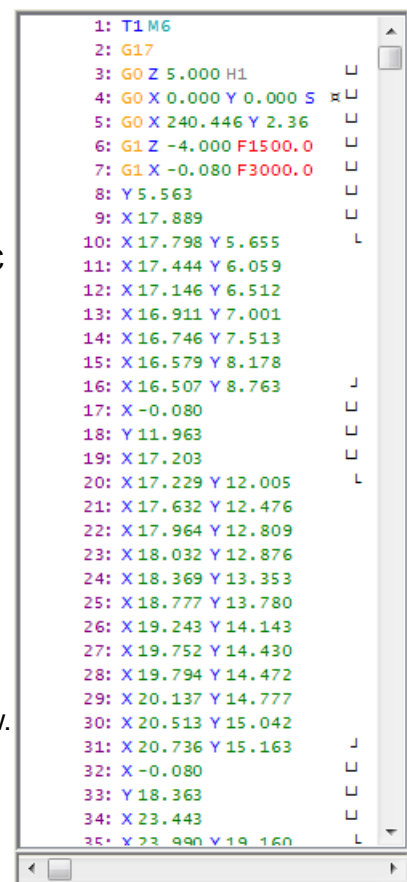


2.3.5 G-code panel

G-code panel shows current program lines.

- Red lines show a pause. The software / machine will pause NC program execution at these lines.
- Lines with dark red line numbers are bookmarked.
- Lines with $\perp$ means acceleration
- Lines with $\lrcorner$ means deceleration
- Lines with $\sqcup$ means acceleration and deceleration
- Lines with $\times$ are removed by optimizer
- Lines with $\times$ are ignored

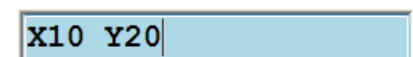
Under panel is horizontal slider which can be used for G-Code preview.



2.3.6 Manual data input (MDI) panel

MDI is text input box, which allows manual G-code input.

For example typing X10 Y20 will move machine to this position.



This input can also be used for “MDI shortcuts”. Each menu command has “MDI shortcut” assigned. For example: File/Open has shortcut “/101”. Of you type “/101” into MDI input, File/Open command will be executed.

2.3.7 Toolbar



Emergency stop:	Executes immediate Emergency Stop (E-Stop)
Open program:	Loads a G-Code program
Start execution:	Start program execution
Stop execution:	Stop program execution
Pause execution:	Pause program execution
Top View:	Display G-Code program view from the top
Side View:	Display G-Code program view from the side
Front View:	Display G-Code program view from the front
Perspective View:	Display G-Code program using 'Perspective' view
Zoom In:	Zoom display in to view details
Zoom Out:	Zoom display out to view a larger area
Zoom Tool:	Zoom display to the current tool position
Zoom Extents:	Zoom display to the G-Code program extents
Mist:	Activate / Deactivate Coolant Mist
Flood:	Activate / Deactivate Coolant Flood
Spindle:	Activate / Deactivate Spindle

Set Position – Zero:	Zero all axes. If any offset is checked, it will be used. Warning: “Set Position” will change machine absolute position! If absolute position is changed automatic tool change and fixed tool sensor position might become invalid!
Go To Position - Zero:	Moves the machine to the zero position. If any offset is checked, it will be used.
Go To Position - ZeroXY:	Moves the machine X and Y axis to the zero position.
Go To Position - Park1:	Moves the machine to the absolute Park1 position defined in ‘Settings’.
Go To Position - Park2:	Moves the machine to the absolute Park2 position defined in ‘Settings’.
Go To Position - G28:	Moves the machine to the absolute G28 position as specified in ‘Settings’.
Go To Position - G30:	Moves the machine to the absolute G30 position as specified in ‘Settings’.
Offset - Zero:	Zero’s the working offset.
Offset - Current XY:	Sets the current XY position as the XY working offset.
Offset - Current Z:	Sets the current Z position as the Z working offset.
Offset – Measure Offset Z:	Measures Z working offset at the current location, using ‘movable’ tool sensor.
Offset - Measure Tool Offset:	Measures tool length using a ‘fixed’ tool sensor.
Offset - Measure Tool Offset Here:	Measures tool length at the current location, using ‘movable’ tool sensor.
Home:	Initiate automatic homing procedure.



2.3.8 File menu

The file menu offers a group of methods for opening, closing, importing and exporting programs. Programs can be used for visualization, simulation, generating toolpath, cutting, and can be in various formats.

Most import and toolpath features display user dialogs for entry of user parameters. Description of features is provided below.

A 'G-Code Wizard' contains functions that generate pocketing or drill array toolpaths based on user parameters.

G-Code can be exported using an option best suited to user requirement. The 'Raw' option saves a binary format, used with planet-cnc.com SD card adapters.

'Settings' is where we configure and set machines main parameters.

Open ...	RCtrl+P /101
Close	/102
Recent Files	▶
Import G-Code ...	/111
Import DXF ...	/112
Import PLT/HPGL ...	/113
Import NC Drill ...	/114
Import Gerber ...	/115
Import Image 2D ...	/116
Import Image 3D ...	/117
Import Text ...	/118
Import CSV ...	/119
Import MIDI ...	
G-Code Wizard	▶
Export Toolpath to GCode ...	/141
Export Toolpath to DXF ...	/142
Export Toolpath to CSV ...	/143
Export Toolpath to Raw ...	/144
Settings ...	/151
Import Settings ...	/152
Export Settings ...	/153
Language	▶
Exit	/199

2.3.8.1 Open

Opens new program. Software will try to auto detect file format.

2.3.8.2 Close

Closes program that is currently opened.

2.3.8.3 Recent Programs

Displays a list of recently opened programs. Select the file for open from a drop-down list.

2.3.8.4 Import programs

Imported programs assist at creation of G-Code, based on a content which is supported in wide range of program formats.

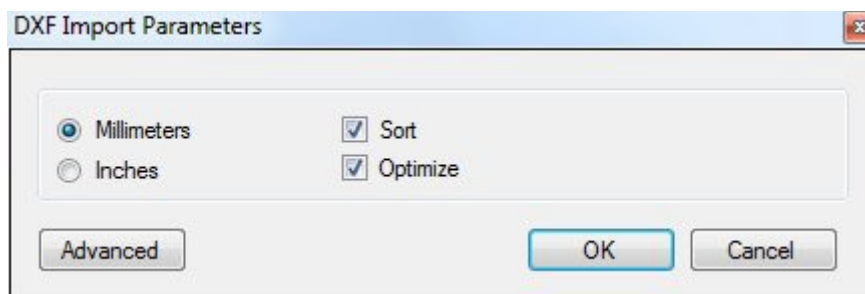
2.3.8.4.1 Import G-code

Opens program in G-code form. G-code file can use different extension. Usually extensions are: *.nc, *.tap, *.cnc, *.iso, *.gcode, *.ncf, *.txt.

2.3.8.4.2 Import DXF

2.3.8.4.3 Import PLT/HPGL

Imports program in DXF and PLT/HPGL format. This type of format usually contains vectors, that can be converted to toolpath. Most of the software for CAD drawing or vector images have option to save design in DXF format.



Sort:

Sorts elements at such particular order, that when cutting is applied, the shortest path between elements is used. This shortens cutting time.

Optimize:

This feature tries to further optimize toolpath by splitting elements entities. However, caution should be used at this feature, because there is a possibility that generated toolpath might be incorrect. This feature is very useful when Foam Cutter machine is used.

Millimeters or Inches:

DXF files have no embedded units. Millimeters or inches units must be specified. If units are not properly defined, program will be scaled incorrectly.

2.3.8.4.3.1 Advanced / General

General settings of imported DXF program can be made in General tab. You can define how layers within DXF program will be treated. You can generate 'Pause' between elements and if tool change will occur during machining process, you can enable that also. You can adjust scale factor of imported program. 'Spindle', 'Flood' and 'Mist' outputs can be turned ON at the beginning and turned OFF at the end of generated toolpath.

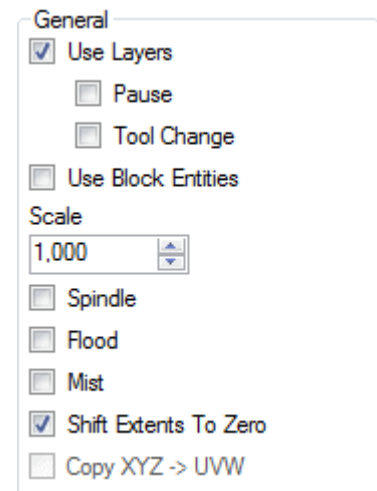
The image shows a software dialog box titled "General". It contains two main sections of settings. The left section, titled "General", includes checkboxes for "Use Layers" (checked), "Pause", "Tool Change", "Use Block Entities", "Spindle", "Flood", "Mist", "Shift Extents To Zero" (checked), and "Copy XYZ -> UVW". Below these is a "Scale" spinner set to "1,000". The right section contains "Speed" (spinner at "250,000"), "Tab Distance" (spinner at "0,000"), "Tab Size" (spinner at "3,000"), "Passes" (spinner at "1"), and "Step Down" (spinner at "1,000"). At the bottom are text input fields for "Header" and "Footer".

Section	Parameter	Value / State
General	Use Layers	Checked
	Pause	Unchecked
	Tool Change	Unchecked
	Use Block Entities	Unchecked
	Scale	1,000
	Spindle	Unchecked
	Flood	Unchecked
	Mist	Unchecked
	Shift Extents To Zero	Checked
	Copy XYZ -> UVW	Unchecked
Machining Parameters	Speed	250,000
	Tab Distance	0,000
	Tab Size	3,000
	Passes	1
	Step Down	1,000
	Header	
Footer		

2.3.8.4.3.1.1 General

Use Layers:

Elements within DXF files are usually grouped in layers. In HPGL format layer is called pan. If you choose to use layers, you will be able to see certain layers name in the generated G-code window. It also allows you to generate 'Pause' and 'Tool Change' procedure in-between layers.



E.g.: If your element is given default layer name 3, you are able to see layers name in G-code window. Layers name will appear in the beginning of layers corresponded G-code. This way you can make your generated G-code slightly more transparent.

Pause:

In between each layers G-code execution, a pause will be generated. User dialog reminds you that Pause has been generated and to resume program you must click Continue button.

Tool change:

In between each layer, a tool can be changed. Generated tool change G-code uses layers name for tool name. E.g.: If we name our layer in DXF file, '3', then generated tool change G-code will be 'T3 M6'

Use Block Entities:

Sometimes DXF elements are inside special block. With this option enabled, you choose that block entities are also used in toolpath generation.

Scale:

Apply the specified scale factor for correct dimension size.

E.g.: If millimeters are set as units, and we want to display our imported program in centimeters, we set scale factor 10.

Spindle:

When enabled, 'Spindle' ON/OFF G-code will be generated. Spindle will be turned ON in the beginning of the program, and turned OFF at the end of the program.

Flood:

When enabled, 'Flood' ON/OFF G-code will be generated. Flood will be turned ON in the beginning of the program, and turned OFF at the end of the program.

Mist:

When enabled, 'Mist' ON/OFF G-code will be generated. Mist will be turned ON in the beginning of the program, and turned OFF at the end of the program.

Shift Extents To Zero:

Generated program will Align with the zero location.

You can also align program with zero location later, if you click *Program/Shift/Extents To Zero*

Copy XYZ->UVW:

Generates program for 2nd tower when 'Hot Wire' machine type is used. With this option enabled, toolpath of UVW axes, will be symmetrical with generated toolpath of XYZ axes.

Speed:

Sets feed speed.

Tab Distance:

Distance between two tabs used for holding element in place during cut.

Tab Size:

Size of tabs.

Passes:

Number of passes that machine will do over generated toolpaths.

You can use this feature in combination with Step Down.

The image shows a settings dialog box with the following fields and values:

Speed	
250,000	
Tab Distance	Tab Size
0,000	3,000
Passes	Step Down
1	1,000
Header	
Footer	

E.g.: If you want to cut material of thickness 10mm, and you want to cut it in 5 passes, you could set: Passes: 5 and Step down: 2.000. What will happen is this: Machine will cut over generated toolpath in 5 passes total. For each pass machine will plunge for 2mm.

Note: In first pass machine will not plunge for 2mm, but it will pass at current Z offset height (e.g. surface of work material). In order that first pass will be plunge of 2mm, we must set *Safe Height/Base Height* at -2.000 value.

Step Down:

Depth increment per pass.

Header:

Inserted text will appear in G-code window at the beginning of generated G-code.

Note: If text doesn't include G-code syntax, then text must be in brackets '()'.

Footer:

Inserted text will appear in G-code window at the end of generated G-code.

Note: If text doesn't include G-code syntax, then text must be in brackets '()'.

2.3.8.4.3.2 Advanced / Safe Height

In this dialog you can set 'Safe Height' of your machine's traverse moves, you can set 'Base Height', generate 'Delay' and 'Pause' or use 'Height To Spindle' feature.

SafeHeight

Safe Height

☒ Enable

Height: 20,0000

☐ Spindle

☐ Flood

☐ Mist

☐ Outputs First

Delay: 0,0

☐ Pause

Base Height

☐ Height To Spindle

☒ Enable

Base Height: -1,0000

Traverse

☐ Spindle

☐ Flood

☐ Mist

Start Delay: 0,0

☐ Start Pause

End Delay: 0,0

☐ End Pause

2.3.8.4.3.2.1 Safe Height**Enable:**

Enable traverse moves at 'Safe Height' between elements.

Height:

'Safe Height' value.

Safe Height

☒ Enable

Height: 20,0000

☐ Spindle

☐ Flood

☐ Mist

☐ Outputs First

Delay: 0,0

☐ Pause

Spindle:

Insert 'Spindle' 'ON' G-Codes, on selected 'Safe height' before downward “plunge” move.

Insert 'Spindle' 'OFF' G-Codes, on selected 'Safe height' after downward “plunge” move.

Flood:

Insert 'Flood' 'ON' G-Codes, on selected 'Safe height' before downward “plunge” move.

Insert 'Flood' 'OFF' G-Codes, on selected 'Safe height' after downward “plunge” move.

Mist:

Insert 'Mist' 'ON' G-Codes, on selected 'Safe height' before downward “plunge” move.

Insert 'Mist' 'OFF' G-Codes, on selected 'Safe height' after downward “plunge” move.

Outputs first:

Insert 'Spindle', 'Flood', 'Mist' ON/OFF G-Codes, after downward “plunge” move from selected 'Safe height'.

Delay:

Insert 'Delay' (G04) before downward “plunge” move.

Note: If 'Outputs first' is checked, Delay will be inserted **after** downward “plunge” move and 'Spindle', 'Flood', 'Mist' 'ON' G-codes.

Pause:

Insert 'Pause' (M00) G-Code before downward ‘plunge’ move.

Note: If 'Outputs first' is checked, Delay will be inserted **after** downward “plunge” move and 'Spindle', 'Flood', 'Mist' 'ON' G-codes.

2.3.8.4.3.2.2 Base Height

Base Height feature comes useful when you are importing DXF program which includes different Z coordinate values, and you do not intend to use them in conventional way. Laser or Plotter are one of the examples where this feature can be used, since you need only ON/OFF output commands over generated Z coordinate toolpath.

Height To Spindle:

Height values of imported file are converted in 'Spindle' ON/OFF commands.

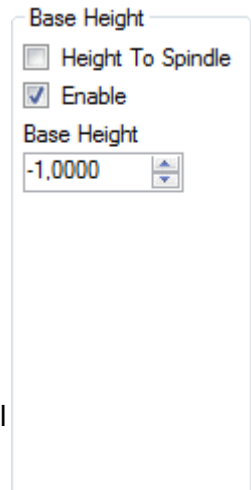
E.g.: Z+ axis movement transforms into M05 (Laser OFF) and Z- axis movement transforms into M03 (Laser ON).

Enable:

Z coordinate values of imported file will not be used. 'Base Height' value will be used.

Base Height:

Value of 'Base Height' will be depth of machines first plunge move.



The 'Base Height' dialog box contains the following elements: a title bar 'Base Height', a checkbox 'Height To Spindle' which is unchecked, a checked checkbox 'Enable', and a numeric input field 'Base Height' with the value '-1.0000' and up/down arrow buttons.

2.3.8.4.3.2.3 Traverse

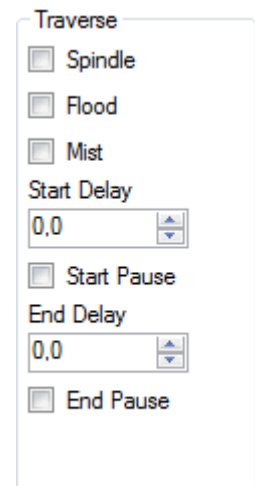
Traverse feature allows us to set machine behavior during its traverse moves. You can set outputs to be turned ON or OFF while machine is performing Traverse moves. You can generate Pause or Delay in the beginning or at the end of specified element.

Spindle:

Insert 'Spindle' ON/OFF G-Codes, after downward 'plunge' move from selected 'Safe Height'.

Flood:

Insert Flood' ON/OFF G-Codes, after downward 'plunge' move from selected 'Safe Height'.



The 'Traverse' dialog box contains the following elements: a title bar 'Traverse', three checkboxes 'Spindle', 'Flood', and 'Mist' which are all unchecked, a 'Start Delay' numeric input field with the value '0.0', a 'Start Pause' checkbox which is unchecked, an 'End Delay' numeric input field with the value '0.0', and an 'End Pause' checkbox which is unchecked.

Mist:

Insert 'Mist' ON/OFF G-Codes, after downward 'plunge' move from selected 'Safe Height'.

Start Delay:

Set Delay time after inserted 'Spindle', 'Flood', 'Mist', ON G-Code

Start Pause:

Set Pause after inserted 'Spindle', 'Flood', 'Mist', ON G-Codes

End Delay:

Set Delay time after inserted 'Spindle', 'Flood', 'Mist', OFF G-Codes

End Pause:

Set Pause after inserted 'Spindle', 'Flood', 'Mist', OFF G-Codes

2.3.8.4.3.3 Advanced / Stages

With this feature you can define which output ('Spindle', 'Flood' or 'Mist') will be turned 'ON' at specific height. You can insert 'Delay' time or 'Pause' in certain stage as also when final stage is completed. This feature comes useful in cases when you are using Plasma cutter.

The 'Stages' configuration window is divided into three panels: Stage 1, Stage 2, and End. Each panel has an 'Enable' checkbox, a 'Height' input field, and checkboxes for 'Spindle', 'Flood', and 'Mist'. Additionally, there is a 'Delay' input field and a 'Pause' checkbox for each stage.

Stage	Enable	Height	Spindle	Flood	Mist	Delay	Pause
Stage 1	☒	3.0000	☐	☐	☐	1.0	☐
Stage 2	☐	2.0000	☐	☒	☒	4.0	☐
End						0.0	☐

2.3.8.4.3.3.1 Stage 1

2.3.8.4.3.3.2 Stage 2

Enable:

Enable certain 'Stage' move.

Height:

Height of certain 'Stage' move. Height specified here, is height where 'Spindle', 'Flood' or 'Mist' will be turned ON.

Spindle:

Insert 'Spindle' ON G-Codes after stages height has been achieved.

Flood:

Insert 'Flood' ON G-Codes after stages height has been achieved.

Mist:

Insert 'Mist' ON G-Codes after stages height has been achieved.

The 'Stage 1' configuration panel shows the following settings: 'Enable' is checked, 'Height' is 3.0000, 'Spindle' is checked, 'Flood' is unchecked, 'Mist' is unchecked, 'Delay' is 1.0, and 'Pause' is unchecked.

Delay:

Set 'Delay' time after stages height has been achieved. Delay time can be used for instance when you turn 'Spindle' ON at determined stage height, and 'Spindle' needs 5 seconds to achieve its desired RPM.

Pause:

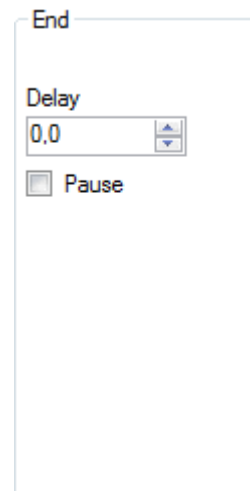
Set 'Pause' after stages height has been achieved.

2.3.8.4.3.3.3 End**Delay:**

Set Delay time at the end of final stage.

Pause:

Set pause at the end of final stage.

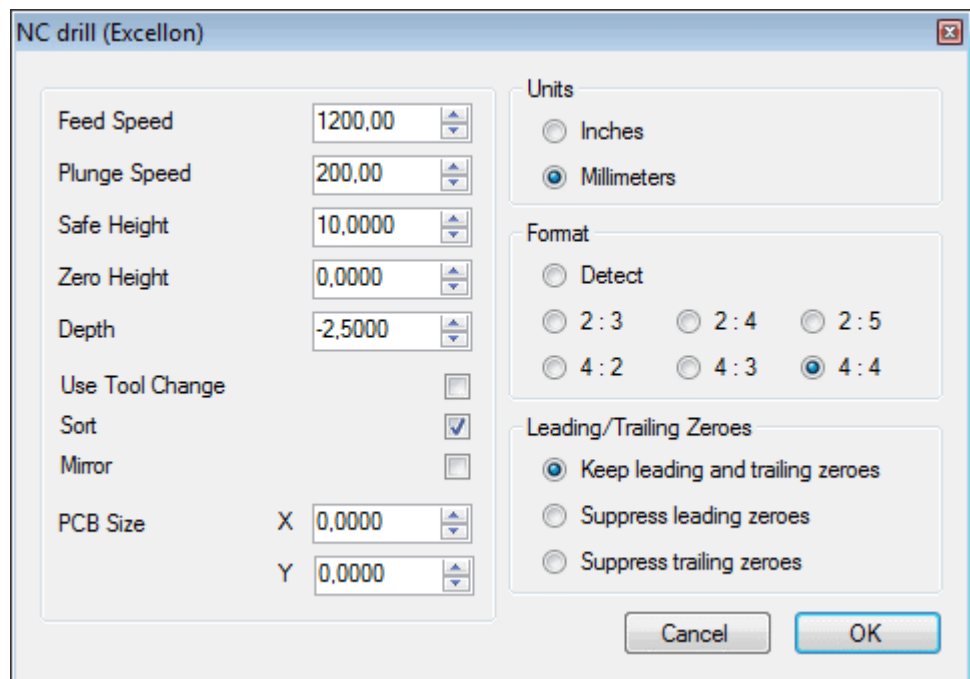


The image shows a software window titled "End". Inside the window, there is a label "Delay" above a numeric input field containing the value "0.0". To the right of the input field are small up and down arrow buttons. Below the input field is a checkbox that is currently unchecked, followed by the text "Pause".

2.3.8.4.4 Import NC Drill

Imports NC Drill (Excellon) file.

NC Drill files are generated by software for design of printed circuit boards. NC Drill files are used for drilling holes of printed circuit boards.



Feed Speed:

Sets 'Feed Speed'.

Plunge Speed:

Sets 'Plunge Speed' for plunge (down) moves.

Safe Height:

'Safe Height' value for machine traverse moves with traverse speed.

Zero Height:

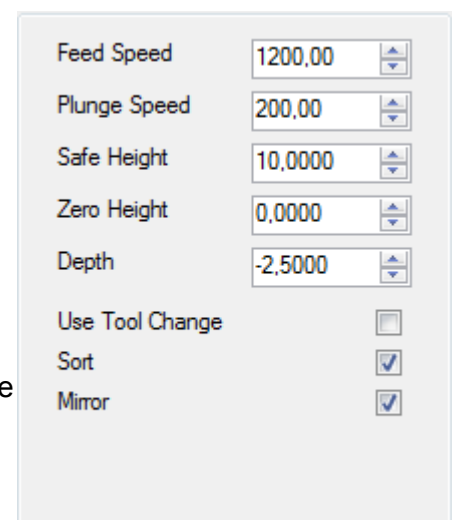
Surface height. Machine moves down to this height at 'Feed Speed'.

Depth:

Drill depth. Machine moves down to this height at 'Plunge Speed'.

Use Tool Change:

Uses tool change G-code between different drills.



Sort:

Sorts holes to optimize the toolpath to shorten working time.

Mirror:

Mirrors program in XY.

If NC Drill format definition is not included in file then you can change it. Software will try to auto-detect correct settings but if imported file looks strange then you should change these values.

2.3.8.4.4.1 Units:

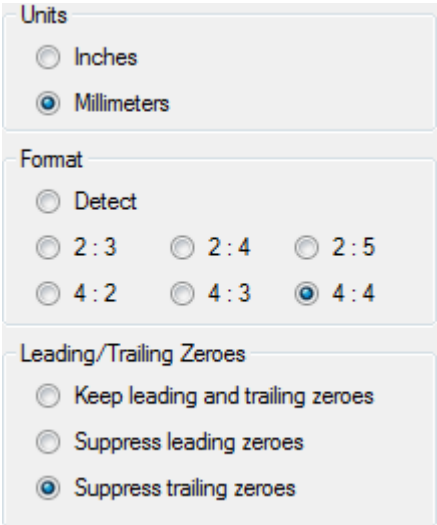
Specify millimeter or inch units.

2.3.8.4.4.2 Format:

Specify number decimal digit format.

2.3.8.4.4.3 Leading / Trailing Zeroes:

Specify leading and trailing zeroes.



The screenshot shows a settings dialog with three sections: Units, Format, and Leading/Trailing Zeroes. In the Units section, 'Millimeters' is selected. In the Format section, '4 : 4' is selected. In the Leading/Trailing Zeroes section, 'Suppress trailing zeroes' is selected.

Section	Option	Selected
Units	Inches	No
	Millimeters	Yes
Format	Detect	No
	2 : 3	No
	2 : 4	No
	2 : 5	No
	4 : 2	No
	4 : 3	No
Leading/Trailing Zeroes	4 : 4	Yes
	Keep leading and trailing zeroes	No
	Suppress leading zeroes	No
	Suppress trailing zeroes	Yes

2.3.8.4.5 Import Gerber

Gerber

Feed Speed: 990,00
Plunge Speed: 200,00
Safe Height: 5,0000
Zero Height: 1,0000
Depth: -0,1500
Tool Diameter: 0,0000
Mirror: ☒
PCB Size: X 0,0000 Y 0,0000
Use: Polygons ☐ Tracks ☒ Pads ☐
Mark Pads: Circle ☐ Oval ☐ Other ☐ Rectangle ☐ Polygon ☐ Pause ☒ Tool Number: 1
Drill Pads: Circle ☐ Oval ☐ Other ☐ Rectangle ☒ Polygon ☐ Depth: -2,5000 Pause ☒ Tool Number: 0
Mode: Isolation ☒ Engrave ☐ None ☐ Pause ☒ Tool Number: 3
Clear Copper: Enable ☐ Border: 5,0000 Pause ☒ Tool Number: 5
OK Cancel

Feed Speed:

Sets Feed speed.

Plunge Speed:

Sets Plunge Speed for plunge (down) moves.

Safe Height:

Safe Height value for machine traverse moves with traverse speed.

Zero Height:

Surface height. Machine moves down to this height at 'Feed Speed'.

Depth:

Depth of milling, when milling isolation or engraving. Machine moves down to this height at 'Plunge Speed'.

Tool diameter:

Diameter of tool used for milling.

Mirror:

Mirrors program in XY.

Feed Speed: 990,00
Plunge Speed: 200,00
Safe Height: 5,0000
Zero Height: 1,0000
Depth: -0,1500
Tool Diameter: 0,0000
Mirror: ☒
PCB Size: X 0,0000 Y 0,0000

PCB Size:

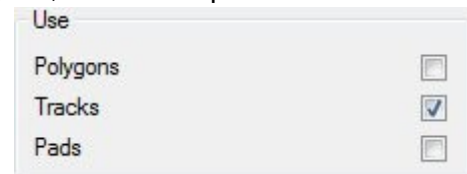
Dimensions of PCB board.

2.3.8.4.5.1 Use:

Gerber files can contain different elements such as polygons, tracks and pads.

Use Polygons:

Use 'Polygons' in toolpath calculation.

**Use Tracks:**

Use 'Tracks' in toolpath calculation. Only tracks will be visible.

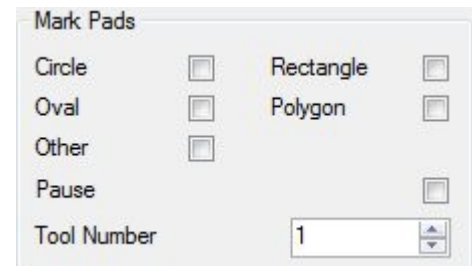
Use Pads:

Use 'Pads' in toolpath calculation. Only pads will be visible.

2.3.8.4.5.2 Mark Pads:

Pad shape can be: Circle, Oval, Rectangle, Polygon or Other.

You can mark selected pad shapes to ease manual drilling later.

**Pause:**

Insert 'Pause' (M00) G-Code at the end of marking pads.

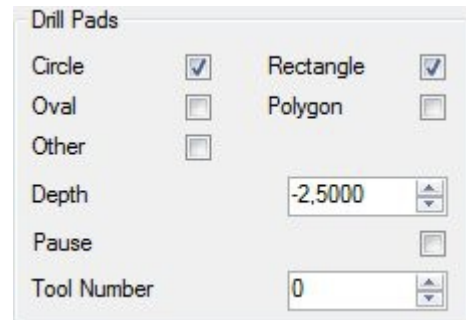
Tool Number:

Number of the tool that will be used for marking pads.

2.3.8.4.5.3 Drill Pads:

Pad shape can be: Circle, Oval, Rectangle, Polygon or Other. You can drill selected pad shapes. You might want to drill hole on circled pad for trough hole component but not for rectangle pad for surface mount component.

NOTE: Check if drill holes in generated g-code are correct.



Drill Depth:

Drill depth. Machine moves down to this height at 'Plunge Speed'.

Pause:

Insert 'Pause' (M00) G-Code at the end of drilling pads.

Tool Number:

Number of tool that will be used for drilling pads.

2.3.8.4.5.4 Mode:

Tool-path calculation mode.

Isolation:

Enable to mill electrical isolation toolpath.

Engrave:

Enable to mill center line (for example silkscreen or cutout).

None:

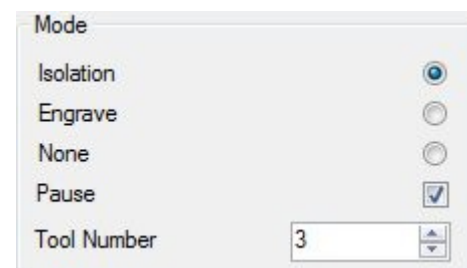
Enable if you want just mark or drill pads.

Pause:

Insert 'Pause' (M00) G-Code at the end of specified toolpath calculation mode

Tool number:

Number of the tool that will be used for specified toolpath calculation mode.



2.3.8.4.5.5 Clear copper

Enable:

Enable to generate toolpath that will clear unused copper.

Border:

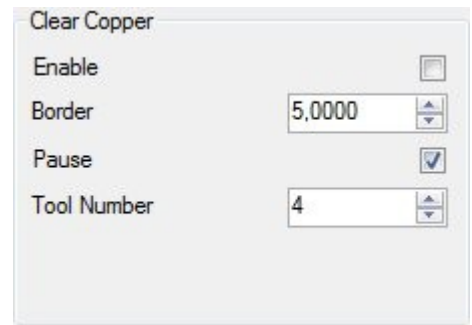
Distance from PCB outline where copper will still remain.

Pause:

Insert 'Pause' (M00) G-Code at the end of clearing copper.

Tool Number:

Number of the tool that will be used for clearing copper.



Clear Copper	
Enable	☐
Border	5,0000
Pause	☒
Tool Number	4

2.3.8.4.6 Import Image 2D

Imports image file and tries to generate 2D toolpath from image.

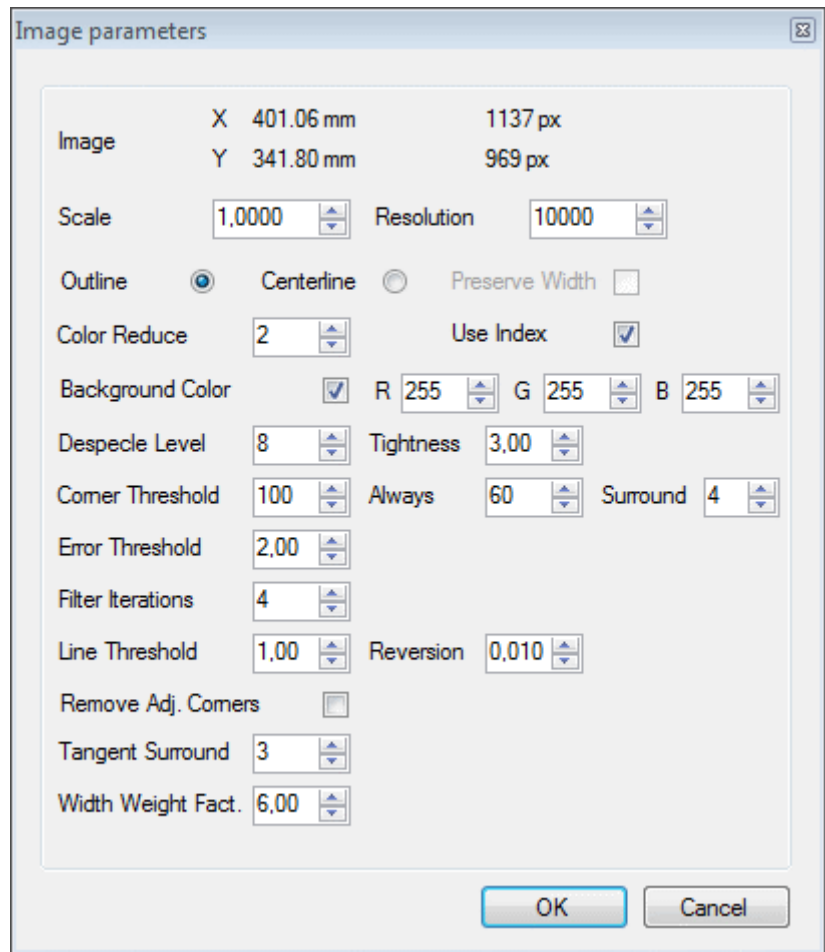


Image:

Dimensions of image in mm and in pixels

Scale:

Sets scale of imported image.

Resolution:

This is calculating algorithm for internal resolution.

Outline:

Toolpath generation is based on the outline of shape.

Centerline:

Toolpath generation is based on the centerline of shape.

Preserve width:

Centerline parameter.

Color Reduce:

Algorithm for reducing number of colors in image. This helps to generate better toolpath from image.

Use index:

When enabled, color index is used instead of color values from table.

Background color:

Sets background color that will be ignored.

Despecle Level:

Image “noise” filter removes speckles from image.

Tightness:

Despacle level parameter.

All features below are the parameters of image recognition algorithm.

Corner Threshold:**Always:****Surround:****Error Threshold:****Filter Iterations:****Line Threshold:****Reversion:****Remove Adj. Corners:****Tangent Surround:****Width Weight Fact:**

2.3.8.4.7 Import Image 3D

Imports image file and tries to generate 3D toolpath from image.

The dialog box contains the following settings:

Parameter	Value
Image X	112.50 mm
Image Y	192.50 mm
Size X	112,50
Size Y	192,50
Height Z	5,00
Diameter	2,50
Detail	1,00
Method	L (selected)
Invert	☐

Size XY:

Sets maximum XY dimensions of the generated toolpath. Initial values are determined using pixel dimensions and the image 'resolution'.

Height Z:

Maximum Z dimension of the generated toolpath.

Invert:

Invert Z values for positive or negative relief.

Diameter:

Diameter of tool that will be used. This sets Y axis step value.

Detail:

Interpolation detail level. This sets X axis step value.

Method:

Toolpath extraction can use selected color or intensity data embedded within the image. Usually luminance produces best results.

R – Red

G – Green

B – Blue

H – Hue

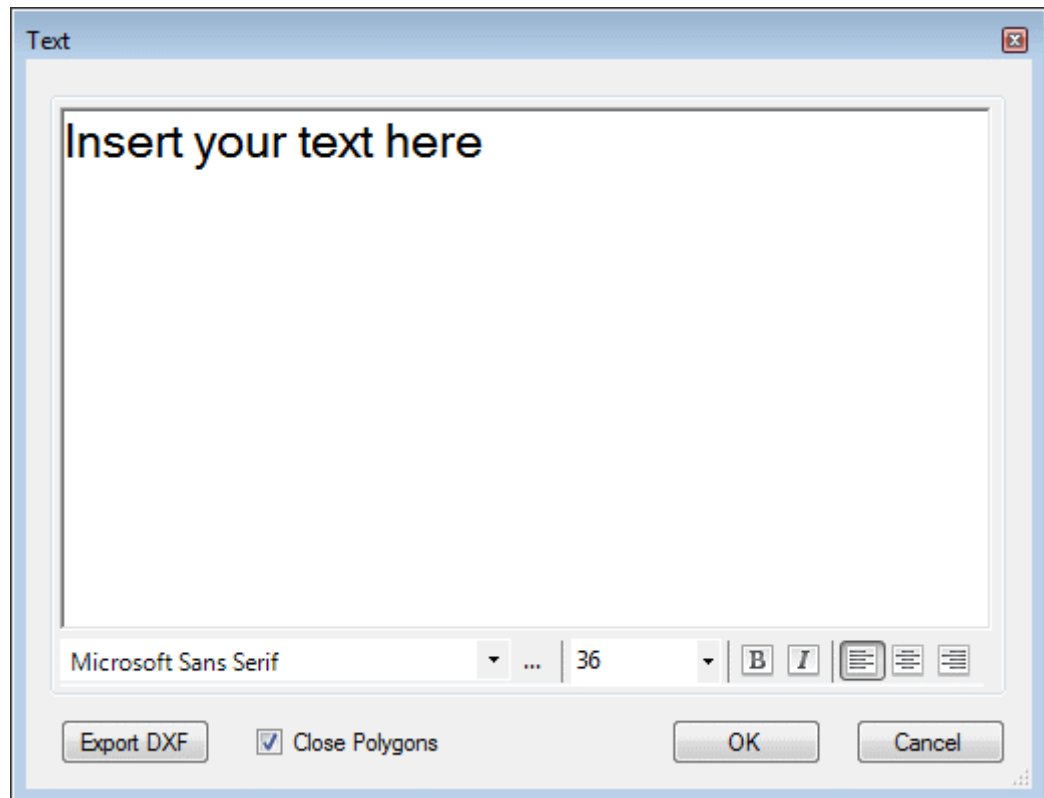
S – Saturation

L – Luminance

2.3.8.4.8 Import Text

Feature generates G-code from true-type font. Any TTF font on the system can be used, even 'graphic' fonts. Font, font size, styling and layout options can be applied.

After setting text and styling options, DXF import dialog will appear.



2.3.8.5 G-Code Wizard

Contains functions for generating programs that can be used for various tasks. You can choose between: 'Round Pocket', 'Straight Pocket', 'Drill Array', 'Rotary Roughing', 'Lathe Turning', 'Airfoil' and 'Airfoil expert'. Main purpose of G-code wizard is to simplify certain tasks, and therefore save time. Various wizards for different tasks.

2.3.8.6 Export program

Program containing tool-path can be exported to G-code, DXF, CSU or TXT file format. It can also be exported as RAW file format for use with SD card.

2.3.8.6.1 Export Tool-path To G-code

Export tool-path to G-code file.

2.3.8.6.2 Export Tool-path to DXF

Export tool-path to DXF file. Traverse moves will be skipped.

2.3.8.6.3 Export Tool-path to CSV

Export tool-path to CSV file.

2.3.8.6.4 Export Tool-path to RAW

Export tool-path to RAW file. RAW files are used with SD card.

2.3.8.7 Settings

In 'Settings' are all main configurations of parameters that are crucial for proper functionality and correct behavior of machine and attached hardware.

2.3.8.7.1 Import Settings

Imports 'Settings' file to quickly restore a known working configuration or to re-configure CNC USB Controller software to suit different machines types or applications.

2.3.8.7.2 Export Settings

Saves the active 'Settings' file as a backup of the software/machine configuration. Custom options can be stored to quickly configure machines to user requirement.

2.3.8.8 Language

Select language of application from list of supported languages.

2.3.8.9 Exit

Closes CNC USB Controller software. If the 'E-Stop' on exit feature is active, the machine performs an E-Stop. 'Settings' options allow E-Stop to deactivate spindles and interrupt all motor 'ENABLE' signals, disabling the machine if software is inactive.

2.3.9 View menu

The view menu controls various aspects of the G-Code program visualization. Only program visualization is affected by view menu options, there are no changes applied to active program G-Code.

For example, where 'Settings', options have defined user coordinate systems, 'View / Origin' options can display origin locations assigned in user coordinate systems.

If so, the 'Origin' icon will relocate in the visualization display, showing the location assigned in the selected coordinate system. This does not affect the origin in the active G-Code program. Simulation and cutting use the origin embedded in G-Code.

Origin		▶
Top View	F1 /221	
Side View	F2 /222	
Front View	F3 /223	
Perspective View	F4 /224	
Zoom In	F5 /225	
Zoom Out	F6 /226	
Zoom Tool	F7 /227	
Zoom Extents	F8 /228	
Center Tool	F9 /231	
Center View	F10 /232	
Change View	F11 /233	
Simulate		▶
Window		▶
Show		▶
Material		▶
Show IO...	/291	

2.3.9.1 Origin

The 'Origin' function displays machine origin positions established in the *Settings / Parameters* tab. Up to nine coordinate systems can be specified. Each can use a unique origin location to suit specific applications or hardware.

2.3.9.2 Top View

Display G-Code / tool-path as viewed from top.

2.3.9.3 Side View

Display G-Code / tool-path as viewed from side.

2.3.9.4 Front View

Display G-Code / tool-path as viewed from front.

2.3.9.5 Perspective View

Display G-Code / tool-path using perspective view.

2.3.9.6 Zoom In

Zoom the display in for close view of smaller details.

2.3.9.7 Zoom Out

Zoom the display out for overview of a larger area.

2.3.9.8 Zoom Tool

Zoom in for a close view of the current tool position. If G-code line is selected or program is in simulation, simulated tool position will be zoomed instead.

2.3.9.9 Zoom Extents

Center and zoom display to view whole G-Code / tool-path.

2.3.9.10 Center Tool

Centers the program visualization on the current tool position. Useful to 'find' and follow the tool when simulating large NC programs or if cut extents move the tool outside the displayed area.

2.3.9.11 Center View

Centers program visualization based on NC program extents. Useful to quickly reorient the view if 'Center Tool' has been used and it's desirable to restore a 'full' view.

2.3.9.12 Change View

Switch between horizontal and vertical position of G-code panel. Provides alternate view with wide aspect visualization and G-Code listing. This can be useful if G-Code lines or comments are wider than the 'standard' view will display or if program aspect ratio is more suited to this layout.

2.3.9.13 Simulate

Starts / Stops animated simulation of G-Code programs.

Users can pan (left mouse button), rotate (right mouse button) and zoom (mouse wheel) without interruption of simulation.

Simulation does not apply programmed cutting speed.

Shift + F12 increases simulation speed, Ctrl + F12 slows down simulation.

2.3.9.14 Window

2.3.9.14.1 Jog

Opens a jog panel maintaining on-screen jog functions. Click+hold on large arrows jogs until mouse button release. Corner buttons allow combined axis jogging. Small arrows, near the center of jog controls, perform a single 'step' move per click.

Jog	/251
Reset	/252
Large	/253

2.3.9.14.2 Reset

Resets GUI to default size and position.

2.3.9.14.3 Large

Displays larger toolbar screen buttons, for use with a touchscreen display.

2.3.9.15 Show

'Show' item opens a sub-menu of elements that can be displayed as part of G-Code visualizations. There are many helpful aids to provide useful feedback. Checked items are visible in visualizations, unchecked items are not.

Show Grid	/2601
Show Axes	/2602
Show Working Area	/2603
Show Tool	/2604
Show Tool Handle	/2605
Show G28, G30	/2606
Show Park	/2607
Show Tool Length	/2608
Show Tool Change	/2609
Show Stored Positions	/2610
Show Labels	/2611
Show Extents	/2612
Show Direction	/2613
Show Completed	/2614
Show Path History	/2615
Show Gauge	▶
Large Tool Handle	/271
Tool Radius	/272
Follow Tool	/273
Coordinates	/280
3D	/281
G-Code	/282
Perspective	/289

2.3.9.15.1 Show Grid

Display of grid on screen.

2.3.9.15.2 Show Axes

Display of coordinate system axes.

2.3.9.15.3 Show Working Area

Display of working area defined in *Settings/Axes/Homing*.

2.3.9.15.4 Show Tool

Display of tool.

2.3.9.15.5 Show Tool Handle

Display of tool handle.

2.3.9.15.6 Show G28, G30

Display of 'G28' and 'G30' positions on working area.

2.3.9.15.7 Show Park

Display of 'Park' positions on working area.

2.3.9.15.8 Show Tool Length

Display of tool length.

2.3.9.15.9 Show Tool Change

Display of tool positions for tool change set in *Settings/Tool Table*.

2.3.9.15.10 Show Stored Positions

Display of positions stored in *Settings/Stored Position 1-9*

2.3.9.15.11 Show Labels

Display of labels.

2.3.9.15.12 Show Extents

Display of toolpath extents. Toolpath is outlined from minimum to maximum edges for all axes.

2.3.9.15.13 Show Direction

Display of direction.

2.3.9.15.14 Show Completed

Display of completed toolpath. Color of completed toolpath by default is green.

2.3.9.15.15 Show Path History

Display of toolpath history. Tool leaves a footprint displayed as line for every move it makes.

2.3.9.15.16 Show Gauge

Display of gauges for selected axis. Gauge is displayed in left side of 3D display. Gauge helps with the sense of orientation within of axis minimum and maximum limit values.

2.3.9.15.17 Large Tool Handle

Display of large tool handle. Large tool handle helps to locate the tool over 3D display.

2.3.9.15.18 Tool Radius

Display of tool radius. If tool diameter is defined in *Settings/Tool Table/General tab/Diameter*, tool diameter is displayed on 3D screen.

2.3.9.15.19 Follow Tool

3D display follows tool over toolpath.

2.3.9.15.20 Coordinates

Enables/disables Coordinate display panel in main user window.

2.3.9.15.21 3D

Enables/disables 3D display

2.3.9.15.22 G-Code

Enables/disables G-code display panel in main user window.

2.3.9.15.23 Perspective

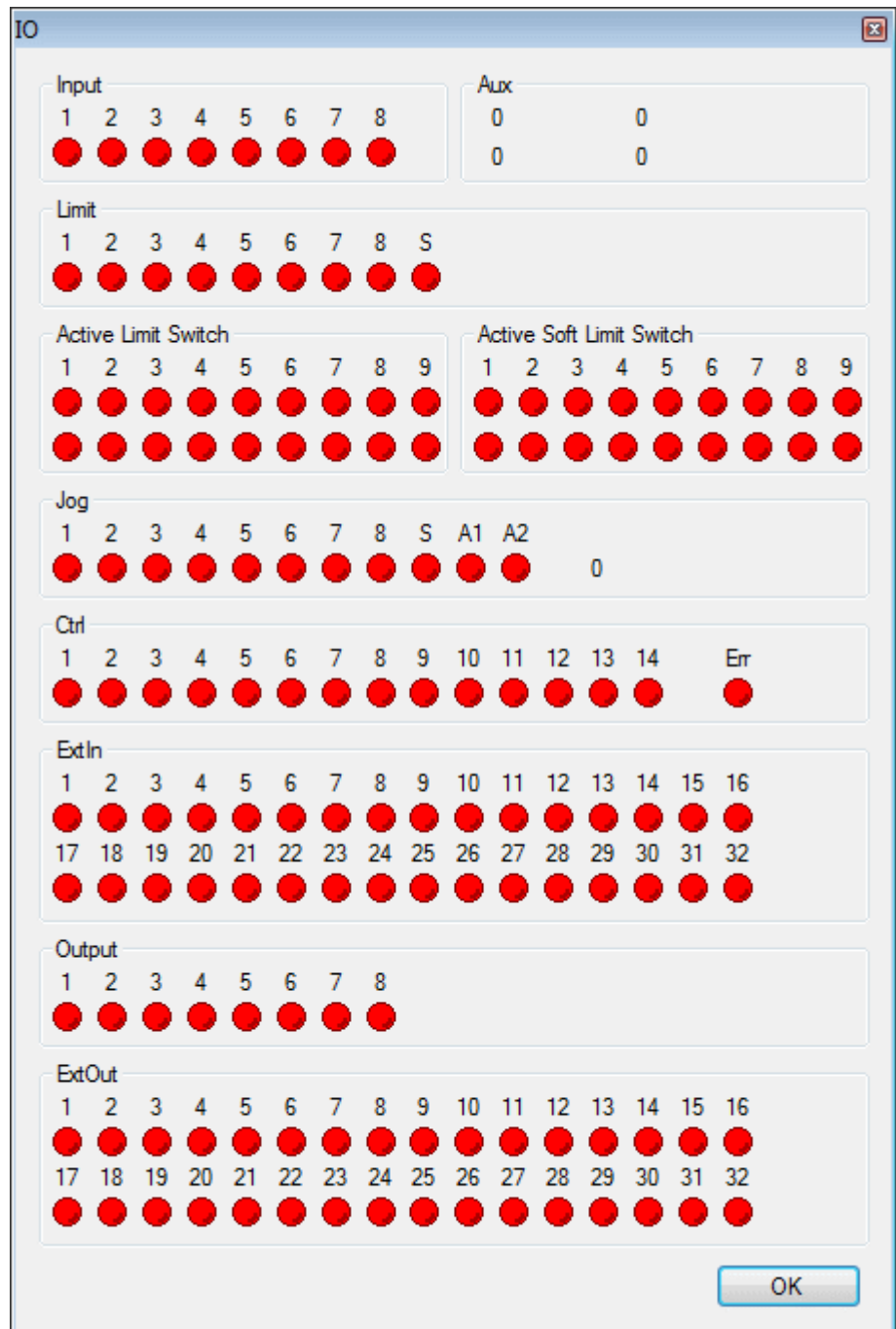
Enables/disables perspective view

2.3.9.16 Material

Opens a list of materials defined in *Settings/Materials*. Selected materials can be displayed as solid or complete with textured appearance.

2.3.9.17 Show IO

'Show IO' allows us to observe input and output pins status. It comes very useful when we need to check if certain input/output is properly activated. ON / OFF indication depends on connected hardware and use of 'Invert' options in other 'Settings' tabs.



2.3.9.17.1 Input

Indication panel of 'Input' connector pins.

2.3.9.17.2 Aux

Indication panel of 'Aux' connector pins.

2.3.9.17.3 Limit

Indication panel of 'Limit' connector pins.

2.3.9.17.4 Active Limit Switch

Indication panel of active 'Limit Switch' connector pins.

2.3.9.17.5 Active soft Limit Switch

Indication panel of active 'Soft Limits'

2.3.9.17.6 Jog

Indication panel of 'Jog' connector pins.

2.3.9.17.7 Ctrl

Indication panel of 'Ctrl' connector pins.

2.3.9.17.8 ExtIn

Indication panel of 'ExtIn' connector pins.

2.3.9.17.9 Output

Indication panel of 'Output' connector pins.

2.3.9.17.10 ExtOut

Indication panel of 'ExtOut' connector pins.

2.3.10 Program menu

The program menu provides G-Code processing functions to conform NC programs to user requirements. In most cases only parameter entry is required to make desired adjustments.

Functions to navigate NC programs and 'bookmark' blocks of code are also available. NC programs can be 'sectioned' by use of toolchange or pause codes.

Shift, Scale and Rotate can be applied to entire programs. These functions can assist with repeat usage of the same NC program or defined program block, for corners, patterns and so forth.

Lines		►
Bookmarks		►
Shift		►
Scale ...	/341	
Mirror		►
Rotate XY ...	/345	
Advanced		►
Set Speed ...	/351	
Calculate Speed ...	/352	
Copy XYZ -> UVW	/361	
Swap XYZ <-> UVW	/362	
Convert Arcs To Lines	/371	
Convert All To Lines	/372	
Edit G-Code ...	/381	
Edit DXF ...	/382	
Copy To Clipboard	LCtrl+C /391	
Paste From Clipboard	LCtrl+V /392	

2.3.10.1 Lines

'Lines' functions allow easy navigation through G-Code.

2.3.10.1.1 Select Next:

Next G-code line in G-code window is selected.

2.3.10.1.2 Select Next Page:

Next page in G-code window is displayed.

2.3.10.1.3 Select Previous:

Previous G-code line in G-code window is selected.

2.3.10.1.4 Select Previous Page:

Previous page in G-code window is displayed.

Select Next	/311
Select Next Page	/3110
Select Previous	/312
Select Previous Page	/3120
Select Next Pause	/313
Select Previous Pause	/314
Select Next Tool Change	/315
Select Previous Tool Change	/316
Select Next Comment	/317
Select Previous Comment	/318

2.3.10.1.5 Select Next Pause:

Next G-code line containing 'Pause' is selected.

2.3.10.1.6 Select Previous Pause:

Previous G-code line containing 'Pause' is selected.

2.3.10.1.7 Select Next Tool change:

Next G-code line containing 'M6' G-code is selected.

2.3.10.1.8 Select Previous Tool Change:

Previous G-code line containing 'M6' G-code is selected.

2.3.10.1.9 Select Next Comment:

Next line containing comment is selected.

2.3.10.1.10 Select Previous Comment:

Previous line containing comment is selected.

2.3.10.2 Bookmarks

G-code lines can be bookmarked. Bookmarked G-code lines can be further used with 'Machine/Bookmarks' features.

'Bookmarks' are set by selection of the G-Code line where the bookmark is required and use of the 'Toggle' item or press of the Tab key. Multiple lines can be bookmarked. When a line has been bookmarked it highlights.

Selection of a 'Bookmark' and press of the Tab key will clear it. Alternatively all 'Bookmarks' can be cleared at once using 'Clear All'. When bookmarks are cleared the highlight is removed.

2.3.10.2.1 Select Next:

Next bookmarked line in G-code window will be selected.

2.3.10.2.2 Select Previous:

Previous bookmarked line in G-code will be selected.

Select Next	/321
Select Previous	/322
Select All	/323
Toggle	Tab /324
Clear All	/325

2.3.10.2.3 Select All:

Selects all bookmarked G-code line.

2.3.10.2.4 Toggle:

Toggle bookmark on selected G-code line.

2.3.10.2.5 Clear All:

Clears all bookmarks from G-code lines.

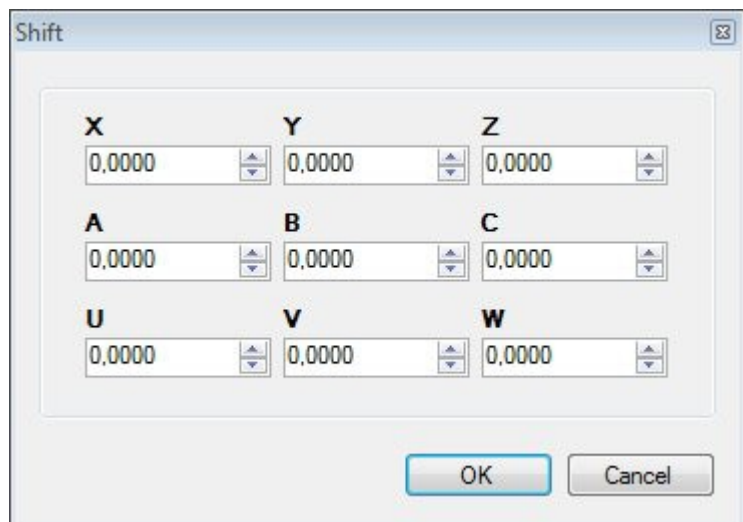
2.3.10.3 Shift

'Shift' translates or moves the tool-path in 3D space using one of many options.

User ...	/331
Extents To Zero	/332
Extents To Position	/333
Center To Zero	/334
Center To Position	/335
Extents To Zero XY	/336
Extents To Position XY	/337
Center To Zero XY	/338
Center To Position XY	/339

2.3.10.3.1 User:

Displayed dialog allows user to specify value of the distance in each axis, for which tool-path will be shifted.



The 'Shift' dialog box is a standard Windows-style window with a title bar and a close button. It contains a grid of nine input fields for specifying shift distances. The first row contains fields for X, Y, and Z axes. The second row contains fields for A, B, and C axes. The third row contains fields for U, V, and W axes. Each field is a text box with a value of '0,0000' and a small up/down arrow icon to its right. At the bottom right of the dialog are 'OK' and 'Cancel' buttons.

X	Y	Z
0,0000	0,0000	0,0000
A	B	C
0,0000	0,0000	0,0000
U	V	W
0,0000	0,0000	0,0000

OK Cancel

2.3.10.3.2 Extents to Zero:

Aligns the 'minimum' edges of the tool-path extents with the machine zero position.

2.3.10.3.3 Extents to Position:

Aligns the 'minimum' edges of the tool-path extents with the current machine position.

2.3.10.3.4 Center to Zero:

Aligns the center of the tool-path with the machine zero position.

2.3.10.3.5 Center to Position:

Aligns the center of the tool-path with the current machine position.

2.3.10.3.6 Extents to Zero XY:

Aligns the 'minimum' edges of the tool-path extents with the machine zero position for X and Y axes only.

2.3.10.3.7 Extents to Position XY:

Aligns the 'minimum' edges of the tool-path extents with the current machine position for X and Y axes only.

2.3.10.3.8 Center to Zero XY:

Aligns the center of the tool-path with the machine zero position for X and Y axes only.

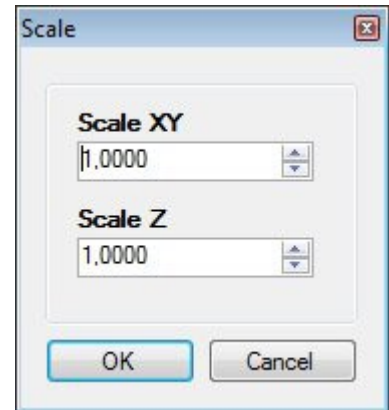
2.3.10.3.9 Center to Position XY:

Aligns the center of the tool-path with the current machine position for X and Y axes only.

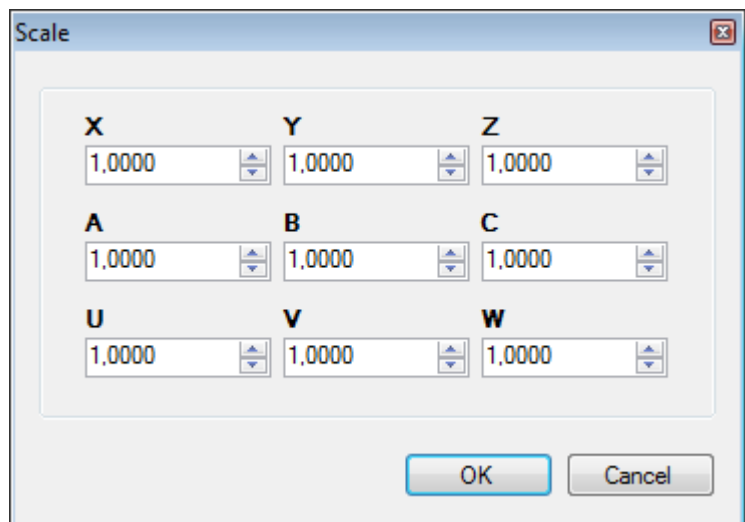
2.3.10.4 Scale

'Scale' adjusts dimensions of the tool-path using 'factor' based scaling. Default value of 1.00 is equivalent to 100% scale or actual size. A scale factor setting of 2.00 increases tool-path dimensions to 200% of the original size in the relevant axis.

When tool-path includes arcs, scale dialog offers only values of scaling for XY and Z axes.



When tool-path does not include any arcs, scale dialog offers values for all axes. You can use 'Convert Arcs To Lines' to convert G-code with arcs to G-code without arcs.



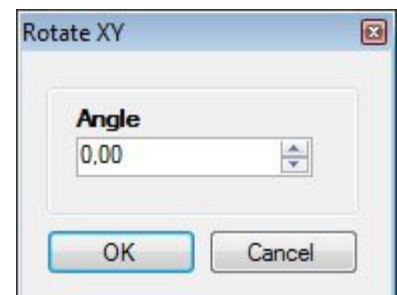
2.3.10.5 Mirror

'Mirror' function mirrors the tool-path over X, Y or Z axis. Axis of reflection is selected in the sub-menu.

Mirror X	/342
Mirror Y	/343
Mirror Z	/344

2.3.10.6 Rotate XY

'Rotate XY' rotates the tool-path XY plane with the origin as the center of rotation. Rotation angle is specified in dialog. Positive values rotate the tool-path in a clockwise direction. Negative values cause anti-clockwise rotation.



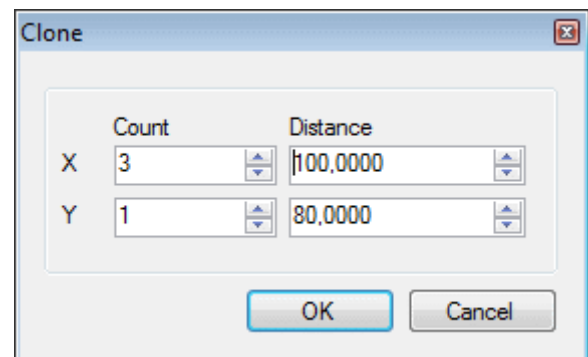
2.3.10.7 Advanced:

Advanced menu includes features that help user to modify tool-paths in multiple useful ways.

Clone ...	/3401
Wrap ...	/3402
Transform ...	/3411
Transform From Points ...	/3412
Warp ...	/3421

2.3.10.7.1 Clone:

When user needs to mill more pieces of the same tool-path, “Clone” feature can be used.



Count XY:

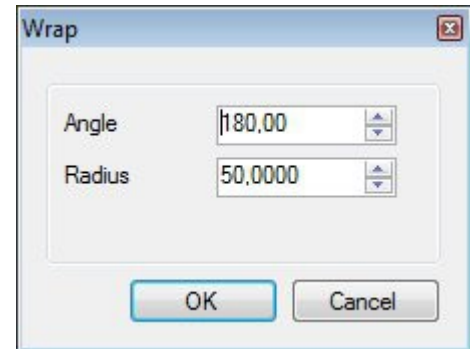
You can define how many clones of original tool-path will be copied in X and Y direction.

Distance XY:

You can set the distance in X and Y direction between two neighboring clones. Default XY distance is set to G-code extents.

2.3.10.7.2 Wrap:

This feature “wraps” g-code around X axis to be used on machines with 4th axis. If g-code contains arcs (G2/G3) then you should use 'Convert Arcs To Lines' first. To have correct visualization on display, in *File/Settings/General/Machine type*, Rotary ABC feature must be used.



Angle:

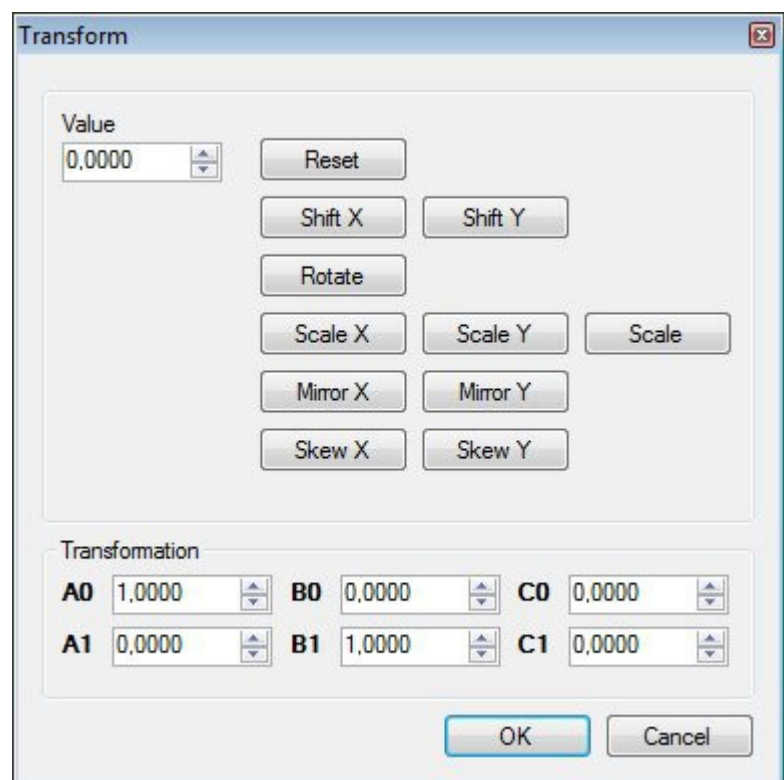
Value represents span in degrees over cylinders surface, over which G-code will be wrapped.

Radius:

Value represents cylinder radius that is mounted on A axis.

2.3.10.7.3 Transform:

Tool-path can be modified with the help of Transform feature. User enters desired value and clicks the desired transformation button. Transformation matrix combines and shows all used transformations.



Reset:

Resets transformation matrix to default value.

Shift X:

Shifts tool-path for entered value in X direction.

Shift Y:

Shifts tool-path for entered value in Y direction.

Rotate:

Rotates the tool-path XY plane for entered value.

Scale X:

'Scale X' adjusts dimensions of the tool-path using 'factor' based scaling. A scale factor of entered value 2.00, increases tool-path dimensions to 200% of the original size in the X axis.

Scale Y:

'Scale Y' adjusts dimensions of the tool-path using 'factor' based scaling. A scale factor of entered value 2.00, increases tool-path dimensions to 200% of the original size in the Y axis.

Scale:

'Scale' adjusts dimensions of the tool-path using 'factor' based scaling. A scale factor of entered value 2.00 increases tool-path XY dimensions to 200% of the original size.

Mirror X:

Mirrors the tool-path over X axis.

Mirror Y:

Mirrors the tool-path over Y axis.

Skew X:

Skews the tool-path in X coordinate.

Skew Y:

Skews the tool-path in Y coordinate.

Transformation:

Transformation matrix combines and shows all used transformations.

2.3.10.7.4 Transform from points:

“Transform from points” calculates transformation of first group of points to second group of points.

The dialog box titled "Transform From Points" contains the following fields and controls:

- From Point 1:** X: 0,000, Y: 0,000. Buttons: <, -
- To Point 1:** X: 0,000, Y: 0,000. Buttons: <, -
- From Point 2:** X: 0,000, Y: 0,000. Buttons: <, -
- To Point 2:** X: 0,000, Y: 0,000. Buttons: <, -
- From Point 3:** X: 0,000, Y: 0,000. Buttons: <, -
- To Point 3:** X: 0,000, Y: 0,000. Buttons: <, -
- Buttons:** OK, Cancel

From Point 1:

X: X coordinate of Point 1, from the first group of points

Y: Y coordinate of Point 1, from the first group of points

From Point 2:

X: X coordinate of Point 2, from the first group of points

Y: Y coordinate of Point 2, from the first group of points

From Point 3:

X: X coordinate of Point 3, from the first group of points

Y: Y coordinate of Point 3, from the first group of points

If **From** position coordinates are coordinates from G-code, you use  button to insert values from selected G-code line.

If **From** position coordinates are machine position coordinates, you use  button to insert values from current machine position.

To Point 1:

X: X coordinate of Point 1, from the second group of points

Y: Y coordinate of Point 1, from the second group of points

To Point 2:

X: X coordinate of Point 1, from the second group of points

Y: Y coordinate of Point 1, from the second group of points

To Point 3:

X: X coordinate of Point 1, from the second group of points

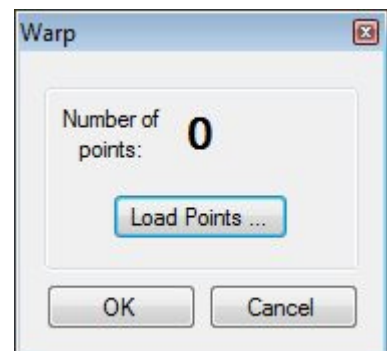
Y: Y coordinate of Point 1, from the second group of points

If **To** position coordinates are coordinates from G-code, you use  button to insert values from selected G-code line.

If **To** position coordinates are machine position coordinates, you use  button to insert values from current machine position.

2.3.10.7.5 Warp:

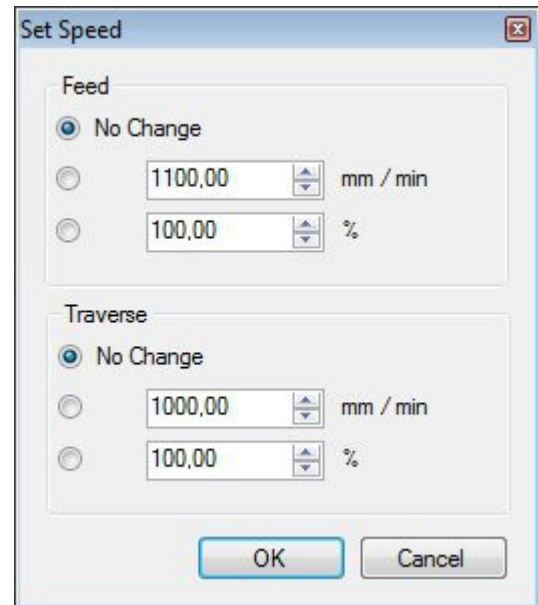
Applies “Warp” feature to current tool-path, based on measured points. If measured points are saved into file, you can load them with “Load points” button.



2.3.10.8 Set Speed

'Set Speed' provides alternate means to specify feed and traverse motion rates applied in machining the G-Code program. Current G-Code 'F' word values are overridden.

This is preferred method for changing speed. Use this and not "speed override" on main screen.

The image shows a 'Set Speed' dialog box with a title bar and a close button. It is divided into two sections: 'Feed' and 'Traverse'. Each section has three radio button options: 'No Change' (selected), a numerical input field with up/down arrows, and a percentage input field with up/down arrows. In the 'Feed' section, the numerical value is 1100,00 and the percentage value is 100,00. In the 'Traverse' section, the numerical value is 1000,00 and the percentage value is 100,00. At the bottom right are 'OK' and 'Cancel' buttons.

2.3.10.8.1 Feed

No Change:

There will be no changes in speed value for Feed Rate.

Second Option:

New speed value for Feed Rate will be set.

Third Option:

New speed value for Feed Rate can be in percentage of the existing value for Feed Rate.

2.3.10.8.2 Traverse

No Change:

There will be no changes in speed value for Traverse Rate.

Second Option:

New speed value for Traverse Rate will be set.

Third Option:

New speed value for Traverse Rate can be in percentage of the existing value for Feed Rate.

2.3.10.9 Calculate Speed

This speed calculator widget which helps user to calculate 'Feed Rate' value based on the material, tool and spindle RPM used.

The 'Calculate Speed' dialog box is a window with a title bar and a close button. It contains several input fields and buttons. On the left, there is a 'Material' dropdown menu set to 'Aluminium'. Below it is a 'Cutting Speed' input field with the value '300.00' and a range of '(250.00 to 350.00)'. There are two radio buttons for units: 'meters/min' and 'feet/min', with 'feet/min' selected. Below this is a 'Tool' section with 'Flutes' set to '2' and 'Diameter' set to '6.0000'. There are also radio buttons for 'mm' and 'in', with 'mm' selected. In the center, there is a 'Spindle Speed' section with 'RPM' set to '0.00' and a calculated value of '(4851.04)'. Below this is a 'Chip Load' input field with the value '0.0075' and a range of '(0.0050 to 0.0100)'. There are radio buttons for 'mm' and 'in', with 'in' selected. On the right, there is a 'Feed Rate' input field with the value '0.000' and two radio buttons for 'mm/min' and 'in/min', with 'mm/min' selected. There are two '>>' buttons between the sections. At the bottom right, there are 'OK' and 'Cancel' buttons.

2.3.10.9.1 Material

Various materials be selected from drop down menu.

Cutting Speed:

Based on the selected material, Cutting Speed is advised.

- Aluminium
- Brass
- Bronze
- Cast Iron
- Copper
- Magnesium
- Plastic
- Mild Steel
- Cast Steel
- Tough Steel
- Stainless Steel
- MDF
- Wood

2.3.10.9.2 Tool

User enters number of flutes and diameter of used tool and specifies measurement units.

2.3.10.9.3 Spindle Speed

Spindle Speed is calculated based on the 'Material' and 'Tool' parameter values.

2.3.10.9.4 Chip Load

User enters 'Chip load' value and specifies measurement units.

2.3.10.9.5 Feed Rate

'Feed Rate' value is calculated based on the 'Spindle Speed' and 'Chip Load' parameter values.

Please note: This calculator only roughly estimates 'Feed Rate'.

2.3.10.10 Copy XYZ → UVW

G-Code applied to the XYZ or 1st tower is duplicated / modified to create code for the UVW or 2nd tower. Usually used with foam-cutters.

2.3.10.11 Swap XYZ ↔ UVW

G-Code applied to the 1st and 2nd towers is exchanged / modified so that XYZ code becomes UVW code and vice versa. Usually for use with foam-cutters.

Example: Program is made to create left wing of plane. By use of swap command existing program creates right wing of plane.

2.3.10.12 Convert Arcs To Lines

Converts Arc entities into segmented lines. The value specified at *Settings/Options/Interpolation* determines line segment length.

2.3.10.13 Convert All To Lines

Converts all entities into segmented lines. The value specified at *Settings/Options/Interpolation* determines line segment length.

2.3.10.14 Edit G-Code

Opens a copy of the active NC program using external editor. After editing, you need to save file under same file name, return to CNCUSB software without closing the editor. Changes in g-code will be reloaded.

2.3.10.15 Edit DXF

The dialog will show, identifying layer content in the imported DXF file. Entities within the DXF can be re-ordered or removed. The G-Code line where the entity appears in the file is identified. G-code describing the entity is selected.

When an entity is selected in the panel, it is highlighted in the visualization display. The code associated with the entity is selected in the G-Code panel listing.

A DXF file must first be imported or this function is unavailable.

2.3.10.16 Copy To Clipboard

Copies entire contents of the G-Code panel to the windows clipboard.

2.3.10.17 Paste from Clipboard

Pastes windows clipboard content to the G-Code panel.

2.3.11 Machine menu

The machine menu provides means to directly interact with controlled machine and attached hardware.

Emergency Stop	Escape /400
Start	/401
Start Loop ...	/402
Start Raw File ...	/403
Start From Selected Line	/404
Start From SD Card	/405
Stop	Backspace /406
Pause	Pause /407
Bookmarks	▶
Mist	LCtrl+M /421
Flood	LCtrl+F /422
Spindle	▶
Outputs	▶
Go To	▶
Set Position	▶
Offset	▶
Tools	▶
Home	/470
Camera	▶
Transformation	▶
Capture & Measure Points	▶
Soft Limits	/490
Firmware	▶

2.3.11.1 Emergency Stop

Sends an Emergency stop or E-Stop to the machine causing immediate 'hard' stop. Controller stops sending control signals to motor driver.

Note: Using only E-Stop is not sufficient security solution when using servomotors. E-Stop doesn't prevent machine from crashing if encoder starts to send faulty data to motor driver. Some sort of double prevention system should be used for full-proof security solution, where e-Stop signal would trigger relays that would disconnect the power supply to the motors and limit switches would only serve as reference switches.

2.3.11.2 Start

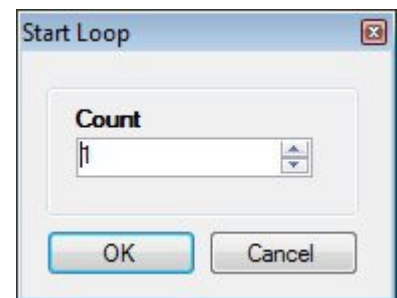
Starts execution of the current G-Code program.

2.3.11.3 Start Loop

Starts looped execution of the current G-Code program.

Count:

Value represents number of loops of the G-code program.



2.3.11.4 Start Raw File

Opens program in RAW format and starts execution.

2.3.11.5 Stop

Stops the machine using a normal stop. Deceleration is used if this is set in *Settings/Options/Stop/Decelerate*.

2.3.11.6 Pause

Pause / Resume execution of the current G-Code program. Deceleration is used if this is set in *Settings/Options/Pause/Decelerate*.

2.3.11.7 Bookmarks

Executes the active G-Code program based on bookmarks.

Start - From Bookmark To Bookmark	/411
Start - From Beginning To Bookmark	/412
Start - From Bookmark To End	/413

2.3.11.7.1 Start – From Bookmark to Bookmark :

Execution of program from first bookmarked line to last bookmarked line. First and last bookmarked lines are included in execution.

2.3.11.7.2 Start - From Beginning To Bookmark:

Execution of program from first G-code line to first bookmarked line. Bookmarked line is included in execution.

2.3.11.7.3 Start - From Bookmark To End:

Execution of program from first bookmark to last G-code line. Bookmarked line is included in execution.

2.3.11.8 Mist

Toggle coolant 'Mist' output ON / OFF. Mist output is set in *Settings/Output/Pins*.

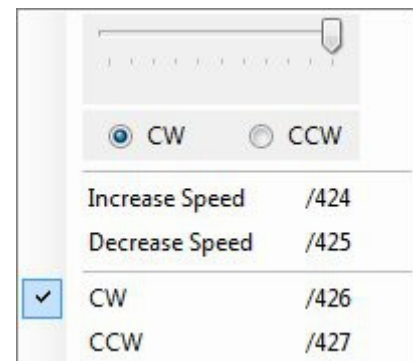
2.3.11.9 Flood

Toggle coolant 'Flood' output ON / OFF. Flood output is set in *Settings/Output/Pins*.

2.3.11.10 Spindle

Toggle 'Spindle' output ON / OFF. A sub-menu provides slider based speed control and direction options.

Spindle output is set in *Settings/Output/Pins*.



2.3.11.11 Outputs

Toggle controller outputs ON / OFF using a sub-menu.

Mk1 controller has three outputs, configured as outputs 1, 2 and 3.

Mk2 9 axis controller has seven outputs, configured as outputs 1-7.

Mk2/4 controller has three outputs, configured as outputs 1, 2 and 7.

Mk3 controller has eight outputs, configured as outputs 1-7.

Mk3/4 controller has three outputs, configured as outputs 1, 2 and 6.

2.3.11.11.1 Output Ext:

Toggle controller external outputs ON / OFF using a sub-menu. Mk3 has an option for additional 32 outputs.

2.3.11.12 Go To

Options of this item move the machine to a specified location.

Use Homing Sequence	/430
Zero	/431
Zero XY	/432
Park 1	/433
Park 2	/434
G28	/435
G30	/436
User ...	/437
Stored Position	▶
Selected Line	/439
Selected Line XY	/43901
Tool To Camera XY	/43902
Camera To Tool XY	/43903

2.3.11.12.1 Use Homing Sequence:

If this option is enabled, 'Go To' features moves will obey homing sequence rules as configured in *Settings/Axes/Homing/Sequence*.

2.3.11.12.2 Zero:

If currently active coordinate display shows absolute coordinates, then all axes move to their zero absolute position (G53 G0 X0 Y0...).

If currently active coordinate display shows relative coordinates, then all axes move to their zero relative position (G0 X0 Y0...).

2.3.11.12.3 Zero XY:

Only X and Y axes move to their zero position.

If currently active coordinate display shows absolute coordinates, then X and Y axes move to their zero absolute position (G53 G0 X0 Y0...).

If currently active coordinate display shows relative coordinates, then X and Y axes move to their zero relative position (G0 X0 Y0...).

2.3.11.12.4 Park1:

Machine moves to 'Park1' position defined in *Settings/Parameters/Park 1*. Park 1 position is absolute position.

2.3.11.12.5 Park2:

Machine moves to 'Park2' position defined in *Settings/Parameters/Park 2*. Park 2 position is absolute position.

2.3.11.12.6 G28:

Machine moves to 'G28' position defined in *Settings/Parameters/G28*. G28 position is absolute position.

2.3.11.12.7 G30:

Machine moves to 'G30' position defined in *Settings/Parameters/G30*. G30 position is absolute position.

2.3.11.12.8 User:

Opens user data entry dialog. The desired position for up to 9 axes can be entered. Machine moves to entered relative position.

2.3.11.12.9 Stored Position:

Machine can move to one of nine positions, previously stored in *Machine/Set Position/Store Current Position To/Stored Position 1-9*. Stored positions are absolute positions.

2.3.11.12.10 Selected Line:

Machine moves to position of selected G-code line.

2.3.11.12.11 Selected Line XY:

Machine moves to XY position of selected G-code line.

2.3.11.12.12 Tool To Camera XY:

Tool moves to XY position of camera view.

2.3.11.12.13 Camera To Tool XY:

Camera view moves to position of tool.

2.3.11.13 Set Position

Applies the current machine position to a named location selected from the sub-menu. Executing “Set Position” will change machine absolute position! If absolute position is changed automatic tool change and fixed tool sensor position will become invalid!

2.3.11.13.1 Enable:

For safety precautions we must check Enable option before we apply any new machines position to certain named location selected from the sub-menu.

☒	Enable	/440
☐	Zero	/441
☐	Zero Axis	▶
☐	Park 1	/442
☐	Park 2	/443
☐	G28	/444
☐	G30	/445
☐	User ...	/446
☐	Stored Position	▶
☐	Selected Line	/448
☐	Selected Line XY	/44801
☐	Camera XY	/44802
☐	Store Current Position To	▶
☐	Store Camera Position To	▶

2.3.11.13.2 Zero:

If currently active coordinate display shows absolute coordinates, then absolute coordinates will be changed to zero.

If currently active coordinate display shows relative coordinates, then absolute coordinates will be changed so that relative coordinates will be zero.

2.3.11.13.3 Zero Axis:

Sets the current machine position as zero position of selected axis.

If currently active coordinate display shows absolute coordinates, then absolute coordinates of selected axis will be changed to zero.

If currently active coordinate display shows relative coordinates, then absolute coordinates of selected axis will change so that relative coordinates will be zero.

X	/44101
Y	/44102
Z	/44103
A	/44104
B	/44105
C	/44106
U	/44107
V	/44108
W	/44109

2.3.11.13.4 Park1:

Sets the current machine position as 'Park1' position.

2.3.11.13.5 Park2:

Sets the current machine position as 'Park2' position.

2.3.11.13.6 G28:

Sets the current machine position as 'G28' position.

2.3.11.13.7 G30:

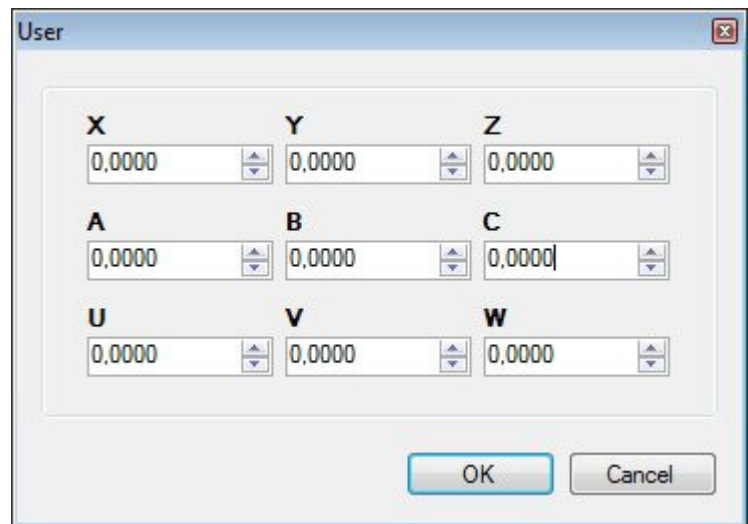
Sets the current machine position as 'G30' position.

2.3.11.13.8 User:

Opens the data entry dialog. Position can be defined for up to 9 axes.

If currently active coordinate display shows absolute coordinates, then absolute coordinates are set.

If currently active coordinate display shows relative coordinates, then absolute coordinates of selected axis will change so that relative coordinates have entered values.



The 'User' dialog box is a standard Windows-style window with a title bar and a close button. It contains a grid of nine input fields for setting machine coordinates. The fields are arranged in three rows and three columns, labeled X, Y, Z in the first row; A, B, C in the second row; and U, V, W in the third row. Each field contains the value '0,0000' and has small up and down arrow buttons to its right. At the bottom right of the dialog are 'OK' and 'Cancel' buttons.

X	Y	Z
0,0000	0,0000	0,0000
A	B	C
0,0000	0,0000	0,0000
U	V	W
0,0000	0,0000	0,0000

2.3.11.13.9 Stored Position:

Sets the current machine position as one of the nine stored positions.

2.3.11.13.10 Selected Line:

Sets the current machine position to selected G-code line.

2.3.11.13.11 Selected Line XY:

Sets the current X and Y axes position to selected G-code line.

2.3.11.13.12 Camera XY:

Sets the current machine position to origin of Camera XY offset.

2.3.11.13.13 Store Current Position To:

Stored Position 1-9:

Stores current machine position to one of nine Stored Position options.

Park 1:

Stores current machine position as 'Park 1' position.

Park 2:

Stores current machine position as 'Park 2' position.

G28:

Stores current machine position as 'G28' position.

G30:

Stores current machine position as 'G30' position.

Stored Position 1	/449001
Stored Position 2	/449002
Stored Position 3	/449003
Stored Position 4	/449004
Stored Position 5	/449005
Stored Position 6	/449006
Stored Position 7	/449007
Stored Position 8	/449008
Stored Position 9	/449009
Park 1	/449011
Park 2	/449012
G28	/449021
G30	/449022

2.3.11.13.14 Store Camera Position To:

Stored Position 1-9:

Stores current camera position to one of nine Stored Position options.

Park 1:

Stores current camera position as Park 1 position.

Park 2:

Stores current camera positions as Park 2 position.

G28:

Stores current camera position as G28 position.

G30:

Stores current camera position as G30 position.

Stored Position 1	/449001
Stored Position 2	/449002
Stored Position 3	/449003
Stored Position 4	/449004
Stored Position 5	/449005
Stored Position 6	/449006
Stored Position 7	/449007
Stored Position 8	/449008
Stored Position 9	/449009
Park 1	/449011
Park 2	/449012
G28	/449021
G30	/449022

2.3.11.14 Offset

Sets the working offset based on current machine position or user defined parameters.

2.3.11.14.1 Zero:

Sets working offsets to zero

Zero	/451
Current	▶
Current XY	/45251
Current Z	/45252
User ...	/453
Selected Line	/454
Selected Line XY	/45401
Camera XY	/45402
From Tool Sensor	▶
From Program	▶
From Material	▶
Surface Distance ...	/457
Measure Offset XY	▶
Measure Offset Z	/459
Sensor Setup ...	/45901

2.3.11.14.2 Current:

Sets working offset to current machine position for selected axes or axis.

2.3.11.14.3 Current XY:

Sets working offset to current machine position for X and Y axes only.

2.3.11.14.4 Current Z:

Set working offset to current machine position for Z axis only.

XYZ ABC UVW	/45201
X	/45211
Y	/45212
Z	/45213
XY	/45221
UV	/45222
ABC	/45231
A	/45232
B	/45233
C	/45234
U	/45241
V	/45242
W	/45243

2.3.11.14.5 User:

Opens the data entry dialog. Entered value is set as working offset for all 9 axes.

2.3.11.14.6 Selected Line:

Sets the working offset to selected G-code line.

2.3.11.14.7 Selected Line XY:

Sets the working offset to selected G-code line for X and Y axes only.

2.3.11.14.8 Camera XY:

Sets working offset to current Camera view position for X and Y axes only.

2.3.11.14.9 From Tool Sensor

Sets working offset based on positions detected using the tool sensor.

Current X+:

Current X-:

Sets working offset to current position for the X axis. Uses sensor size to compensate for positive or negative sensor placement.

Current Y+:

Current Y-:

Sets working offset to current position for the Y axis. Uses sensor size to compensate for positive or negative sensor placement.

Current Z:

Sets working offset to current position for the Z axis.

Current X-	/4541
Current X+	/4542
Current Y-	/4543
Current Y+	/4544
Current Z	/4545

2.3.11.14.10 From Program

Sets the working offset based on positions from G-Code program.

Corner:

Sets working offset to 'minimum' corner of loaded G-Code program extents.

Corner XY:

Sets working offset to 'minimum' corner of loaded G-Code program extents for X and Y axes only.

Corner	/4551
Corner XY	/4552
Center	/4553
Center XY	/4554
Simulated Position	/4555

Center:

Sets working offset to center of the loaded G-Code program extents.

Center XY:

Sets working offset to center of the loaded G-Code program extents for X and Y axes only.

Simulated Position:

Set working offset to position of simulated tool of loaded G-code program.

2.3.11.14.11 From Material

Sets the Working offset based on positions from 'Materials' saved in *Settings/Materials*.

Corner Top:

Sets working offset to materials bottom left corner on the top surface.

Corner Bottom:

Sets working offset to materials bottom left corner on the bottom surface.

Center Top:

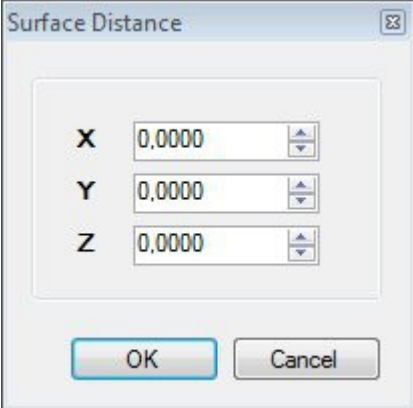
Sets working offset to materials center on top surface.

Center bottom:

Sets working offset to materials center on bottom surface.

2.3.11.14.12 Surface Distance

These XYZ values are offset values that represent the distance from the center of rotation to the surface of the piece mounted on the rotational axes. If you use 4th rotational A axis then only Z value needs to be applied.

A screenshot of a software dialog box titled "Surface Distance". It contains three input fields for X, Y, and Z coordinates, each with a value of "0,0000" and a small up/down arrow icon to its right. At the bottom of the dialog are two buttons: "OK" and "Cancel".

Surface Distance	
X	0,0000
Y	0,0000
Z	0,0000
OK Cancel	

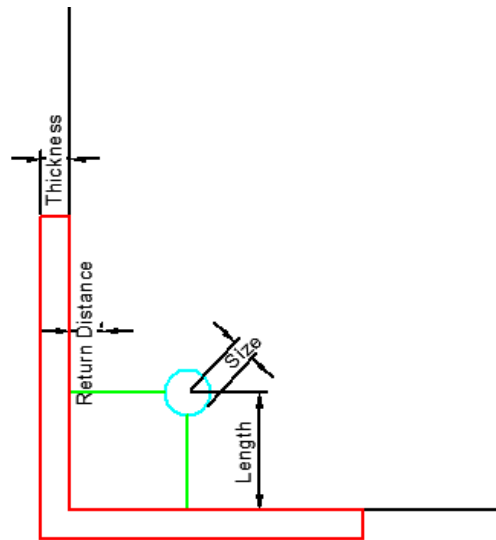
2.3.11.14.13 Measure Offset XY:

This feature obtains working offset coordinate values of corner and center of a circle. Corner can be measured from the inside and also from the outside. Center of circle can be obtained by measuring from the inside as also from the outside of the circle.

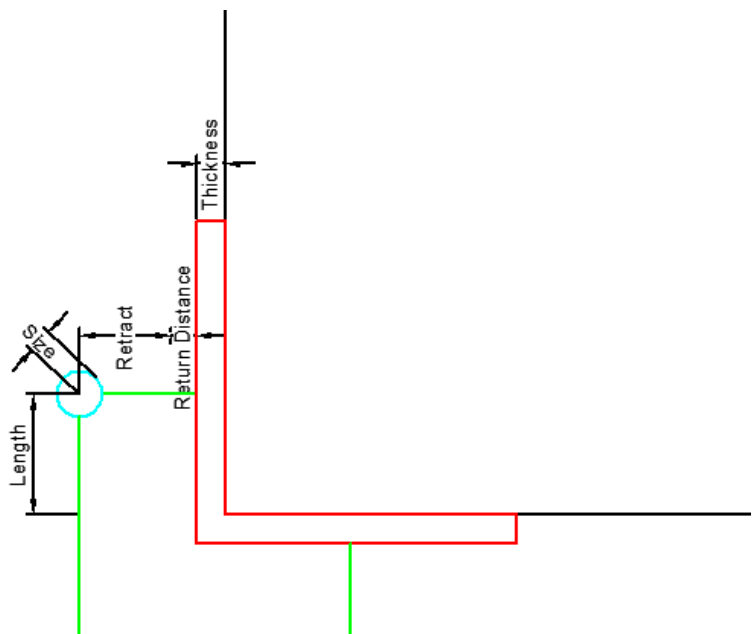
Corner Inside	/45801
Corner Outside	/45802
Sensor Setup ...	/45803
Center Inside	/45811
Center Outside	/45812
Sensor Setup ...	/45813

Corner Inside:

Machine will measure inside corner offset with two moves. Green lines represent the tool-path of the machine.

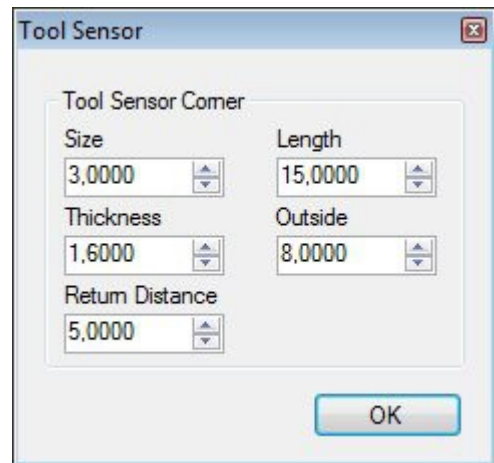
**Corner Outside:**

Machine will measure outside corner offset with two moves. Green lines represent the tool-path of the machine.



Sensor Setup - Tool Sensor Corner:

For better understanding and description of parameters refer to pictures above.



Size:

If we use sensor or touching probe to measure X and Y coordinates then this value represent radius of stylus ball at the point of touching probe.

Length:

The distance which machine travels to another measuring point.

Thickness:

Thickness of sensor used. If you are using PCB clad as sensor, then this value is the thickness of the PCB clad.

Return Distance:

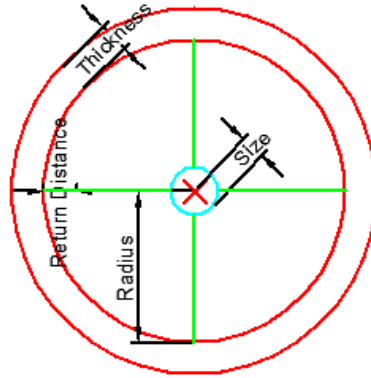
Value represents distance for which tool returns once it has touched the surface of sensor.

Retract:

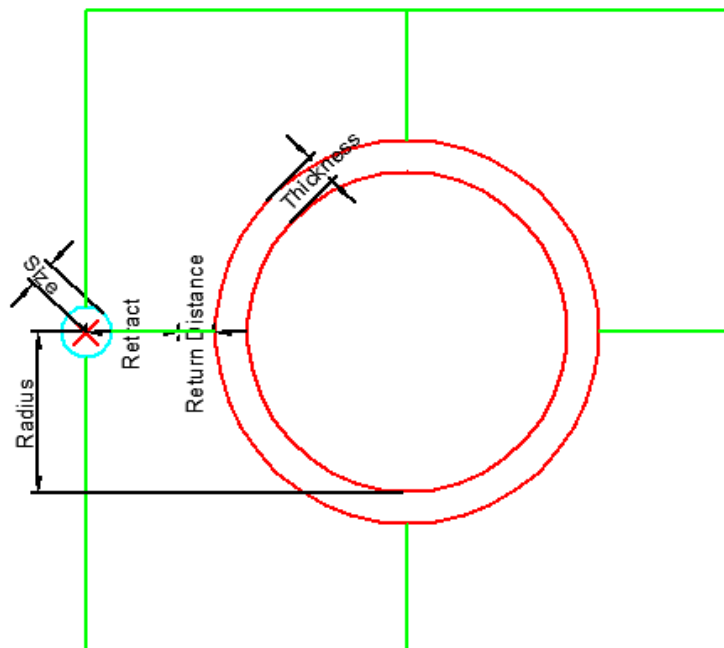
Value represents distance for which machine ascends (from position to which previously returned to).

Center Inside:

Machine will measure inner center offset with four automated moves. Green lines represent the toolpath of the machine.

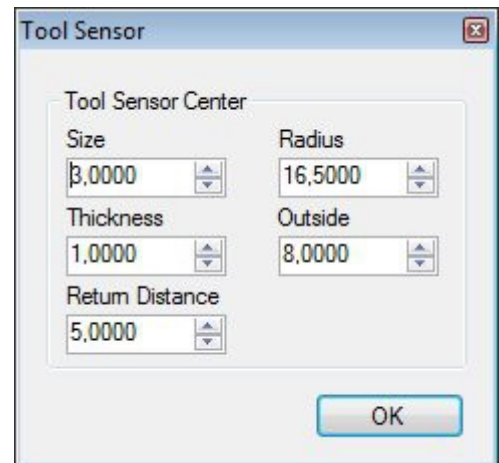
**Center Outside:**

Machine will measure outer center offset with four automated moves. Green lines represent the toolpath of the machine.



Sensor Setup – Tool Sensor Center:

For better understanding and description of parameters refer to pictures above.



Size:

If we use sensor or touching probe to measure X and Y coordinates then this value represent radius of stylus ball at the point of touching probe.

Length:

The distance which machine travels to another measuring point.

Thickness:

Thickness of sensor used. If you are using PCB clad as sensor, then this value is the thickness of the PCB clad.

Return Distance:

Value represents distance for which tool returns once it has touched the surface of sensor.

Retract:

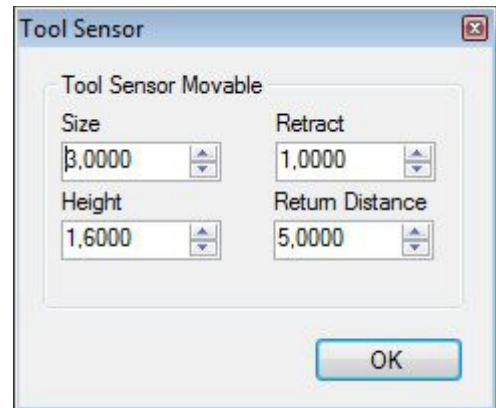
Value represents distance for which machine ascends (from position to which previously returned to).

2.3.11.14.14 Measure Offset Z

Measures the Z axis working offset using movable tool sensor.

2.3.11.14.15 Sensor Setup

Provides convenient means to adjust sensor parameters without recourse to the main settings panel. This can be useful if it's necessary to adjust the 'Size' parameter when tools are changed.



Size:

If we use sensor or touching probe to measure X and Y coordinates then this value represent radius of stylus ball at the point of touching probe. This is the same as Height parameter for measuring Z coordinates.

Return Distance:

Value represents distance for which tool returns once it has touched the surface of sensor.

Height:

Thickness of material that is used for sensor.

Retract:

Value represents distance for which machine ascends (from height to which previously returned to).

2.3.11.15 Tools

Active:	Empty
Change	►
Select	►
Zero Tool Offset	/461
Measure Tool Offset	/462
Measure Tool Offset Here	/463
Set Tool Offset ...	/464
Set Tool Offset From Active Tool	/465
Measure Tool Length	/466
Measure Tool Length Here	/467
Sensor Setup ...	/468

2.3.11.15.1 Active

Displays the active tool or 'Empty' if no tool has been mounted. Active tool is also indicated in bottom left corner of CNC USB software.

2.3.11.15.2 Change

Initiates the tool-change procedure for manual or auto tool-change.

In the case of manual tool change the machine will move to the tool change position for the operator to perform the change. When appropriately configured in 'Settings', tool changers perform the tool-change automatically (Tn M6).

2.3.11.15.3 Select

Software cannot detect if a tool is mounted or which tool is mounted. This option informs software of the actual tool status to initialize tool change procedures (M61 Qn).

2.3.11.15.4 Zero Tool Offset

Sets tool offset to zero.

2.3.11.15.5 Measure Tool Offset

Measure Tool offset using a fixed tool sensor. The machine moves to the tool sensor and measures offset. It auto returns to the initial position if 'Return' is enabled in *Settings/Tool Sensor/Tool Sensor Fixed/Return*.

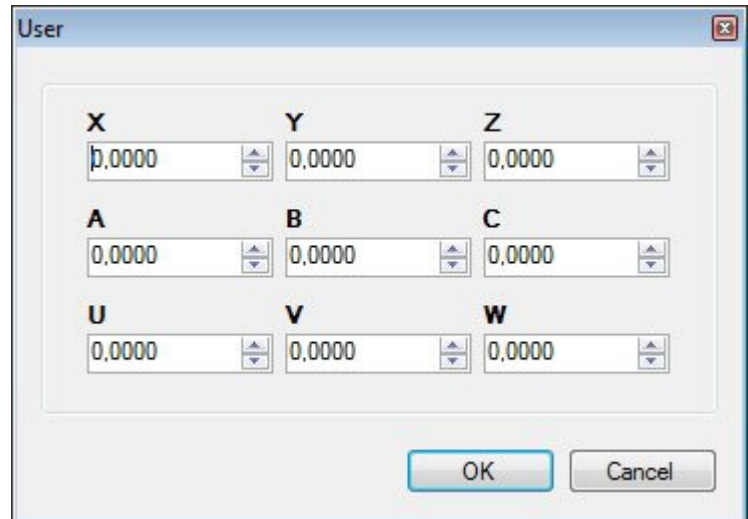
2.3.11.15.6 Measure Tool Offset Here

Measures Tool offset using a movable tool sensor at the current location. The movable tool sensor is placed by the operator.

NOTE: This is not the correct way of measuring tool offset.

2.3.11.15.7 Set Tool Offset

Opens the data entry dialog. Tool Offset can be defined for all 9 axes.



The image shows a software dialog box titled "User". It contains nine input fields for tool offsets, arranged in a 3x3 grid. The axes are labeled X, Y, Z in the first row; A, B, C in the second row; and U, V, W in the third row. Each input field contains the value "0.0000" and has a small up/down arrow button to its right. At the bottom right of the dialog are two buttons: "OK" and "Cancel".

2.3.11.15.8 Set Tool Offset from Active Tool

Tool offset is set from 'Tool Table' in Settings.

2.3.11.15.9 Measure Tool Length

Measure tool length and set absolute position using a fixed tool sensor. The machine moves to tool sensor position and measures tool length.

NOTE: This does not measure offset, and it is not the correct way to measure tool length.

2.3.11.15.10 Measure Tool Length Here

Measure tool length and set absolute position at current location using a movable tool sensor. The movable tool sensor is positioned by the operator. The machine performs the measurement.

NOTE: This does not measure offset, and it is not the correct way to measure tool length.

2.3.11.15.11 Sensor Setup

Size:

If we use sensor or touching probe to measure X and Y coordinates then this value represent radius of stylus ball at the point of touching probe. This is the same as Height parameter for measuring Z coordinates.

Return Distance:

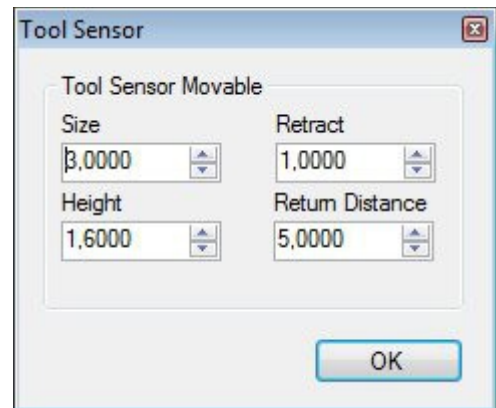
Value represents distance for which tool returns once it has touched the surface of sensor.

Height:

Thickness of material that is used for sensor.

Retract:

Value represents distance for which machine ascends (from height to which previously returned to).



Tool Sensor Movable	
Size	Retract
3,0000	1,0000
Height	Return Distance
1,6000	5,0000

OK

2.3.11.16 Home

Initiates automatic homing procedure, configured in *Settings/Axes/Homing*.

Homing procedure gives machine absolute coordinates. Only now machine becomes “self aware” and therefore is capable of knowing where its position is at any given moment. (See example in 'Settings' chapter of this user manual and at our homepage under FAQ)

2.3.11.17 Camera

2.3.11.17.1 Show Camera

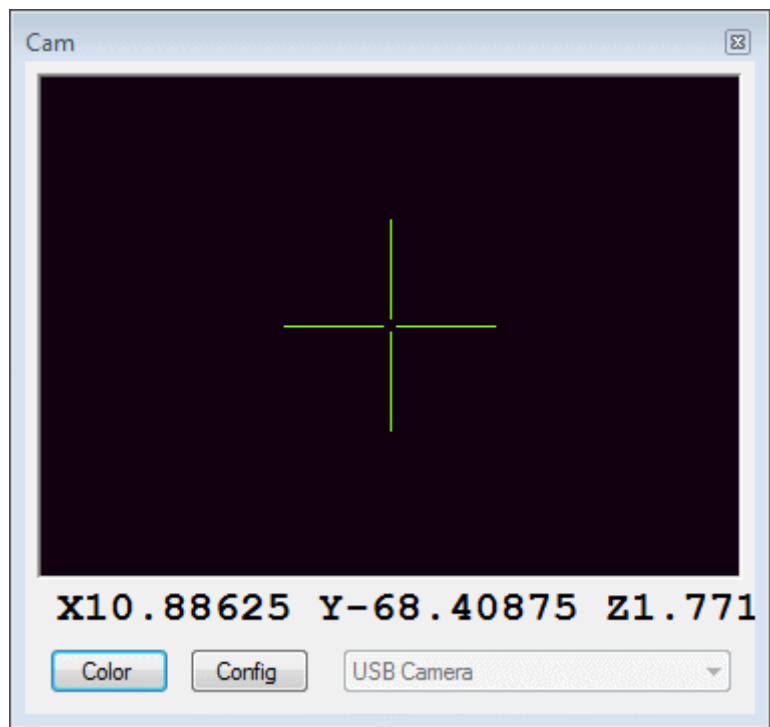
Camera view displays the view visible to a USB web-cam, mounted alongside the spindle, looking down on the workpiece.

The large cross-hair marker in the center of the view is used for precise ‘targeting’ of points on the stock, for measurement or capture purposes.

Available USB web-cams are listed in a drop-down in the center of the panel. Depending on attached hardware, options to use camera filters may also be shown. CNC USB software provides adjustment, so the basic web-cam option is selected. Basic web-cam hardware is suitable. It is much less costly to replace in event of failure or damage.

Mark	/4801
Read 1	/4802
Read 2	/4803
Set Offset ...	/4804

Camera hardware mounted close to spindle experiences much vibration. Robust construction and the ability to mount hardware securely are the most important considerations when choosing web-cam tool sensors. Camera and camera cable might also be source of electrical interference.



2.3.11.17.2 Set Offset

Camera Offset is set here. To obtain correct camera offset, we must first measure reference point at two different positions of Z axis.

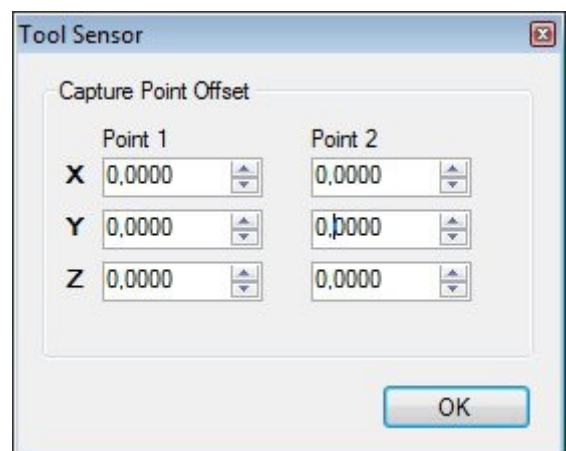
Capture Point Offset:

Point 1:

Point 1 represents XY coordinates of reference point when Z axis is at minimal height

Point2:

Point 2 represents XY coordinates of reference point when Z axis is at maximal height



How to set Camera two point offset:

1. Open camera window with "Machine/Camera/Show Camera ..."
2. Mark position on machine table. You can use marker, small drill or tool, ... Whatever you find useful.
3. Position tool exactly over this marker and do "Machine/Camera/Set Offset/Mark"
4. Move machine and locate marker with camera. Use lowest Z level possible.
5. When your mark is in camera center do "Machine/Camera/Set Offset/Read 1"
6. Now move Z to highest possible level where marker is still visible on camera and center it.
7. When your mark is in camera center do "Machine/Camera/Set Offset/Read 2"

2.3.11.18 Transformation

When our workpiece coordinate system is not aligned with coordinate system of our machine, we can help ourselves with transformations.

Transformation Matrix ...	/481
Calculate Transformation Matrix ...	/482
Calculate Transformation Matrix From Points...	/483

It is advised to clear all points and offsets that are currently set, before beginning with transformation. Red color indicates that certain parameters couldn't be calculated.

2.3.11.18.1 Transformation Matrix

Applies matrix based transformation to current G-Code program.

Enabled:

Enables/disables transformation.

A0, B0, C0 and **A1, B1, C1** are transformation matrix parameters. You can enter and change these parameters also manually.

Enabled ☐					
A0	0.9097	B0	0.4377	C0	-24.6579
A1	-0.4152	B1	0.8991	C1	93.0967

2.3.11.18.2 Calculate Transformation Matrix

Calculates transformation matrix from measured points. Measured points should define your workpiece position and optionally size.

Points X-:

Points X+:

X+ and X- coordinate point count obtained by 'Measure' or 'Capture' feature is shown here.

Points Y-:

Points Y+:

Y+ and Y- coordinate point count obtained by 'Measure' or 'Capture' feature is shown here.

Transformation - Points: 0

Points X-: 0 Points Y-: 0
Points X+: 0 Points Y+: 0

From

Position
X 0,0000 Y 0,0000 Z 0,0000

Size
X 0,0000 Y 0,0000 Z 0,0000

Calculate

To

Angle
α 0,0000

Position
X 0,0000 Y 0,0000 Z 0,0000

Size
X 0,0000 Y 0,0000 Z 0,0000

Transformation

A0 0,0000 B0 0,0000 C0 0,0000
A1 0,0000 B1 0,0000 C1 0,0000

OK Cancel

From:

Position:

Original position from where we want to make transformation.

Size:

Original size from where we want to make transformation. If we measure opposite edge of our workpiece, and therefore obtain correct dimensions of workpiece, transformation matrix applies scaling to the transformed toolpath, if sizes are not identical. You set 'Size' to zero if you don't want scaling.

To:

Based on Measured/Captured points, transformation matrix calculates the **Angle**, **Position** and **Size** of our workpiece. After transformation is applied, machines coordinate system is aligned with coordinate system of workpiece.

2.3.11.18.3 Calculate Transformation Matrix From Points

“Calculate Transformation Matrix From Points” calculates transformation from first group of points to second group of points. Names 'From' and “To” are used because we calculate transformation matrix “from” first group of points “to” another group of points. 'From' group of points uses X and Y point coordinates that are usually taken from G-code, and 'To' group of points uses X and Y point coordinates that are usually machine position coordinates. Each row in the table represents one 'From' point and one 'To' point.

Transformation - Points: 3

From		To	
X	Y	X	Y

Points

From

X: 0,0000 Y: 0,0000

Buttons: Open... Update Add Remove Capture Line Save...

To

X: 0,0000 Y: 0,0000

Buttons: Open... Update Add Remove Capture Line Save...

Calculate

Transformation

A0: 0,0000 B0: 0,0000 C0: 0,0000

A1: 0,0000 B1: 0,0000 C1: 0,0000

OK Cancel

Points:

From:

'From' group of points uses X and Y coordinates from G-code line or some other known coordinate. You can enter these coordinate values by hand, using the numerical box, or you can use buttons below the numerical box.

Button description:

Update:

Existing values of 'From' points can be updated to new values. Whenever you wish to update the X or Y values of certain 'From' point, you must click on the row to select it first.

Add:

With 'Add' button, we add new row, in which now we can insert the X and Y coordinate values for new 'From' and 'To' points. For each new 'From' and 'To' point, 'Add' button must be pressed.

Remove:

Removes selected point row.

Capture:

Captures the X and Y coordinate values of current machine position and copies them in 'From' numerical box. These are absolute position coordinate values. You can insert these values to new point row with 'Add' button or with 'Update' button to overwrite existing 'From' X and Y values in selected point row.

Line:

Copies X and Y coordinate values from selected line of G-code and copies them in 'From' numerical box. You can insert these values to new point row with 'Add' button or with 'Update' button to overwrite existing 'From' X and Y values in selected point row.

Inserting points in to the table can be time-consuming task and since you should never exclude the possibility of something going wrong, you can save your group of points to a file with **Save** button and also restore them with **Open** button.

To:

'To' group of points uses X and Y coordinates of current machine position. You can enter these coordinate values by hand, using the numerical box, or you can use buttons below the numerical box.

Button description:**Update:**

Existing values of 'To' points can be updated to new values. Whenever you wish to update the X or Y values of certain 'To' point, you must click on the row to select it first.

Add:

With 'Add' button, we add new row, in which now we can insert the X and Y coordinate values for new 'From' and 'To' points. For each new 'From' and 'To' point, 'Add' button must be pressed.

Remove:

Removes selected point row.

Capture:

Captures the X and Y coordinate values of current machine position and copies them in 'To' numerical box. These are absolute position coordinate values. You can insert these values to new point row with 'Add' button or with 'Update' button to overwrite existing 'To' X and Y values in selected point row.

Line:

Copies X and Y coordinate values from selected line of G-code and copies them in 'To' numerical box. You can insert these values to new point row with 'Add' button or with 'Update' button to overwrite existing 'To' X and Y values in selected point row.

Inserting points in to the table can be time-consuming task and since you should never exclude the possibility of something going wrong, you can save your group of points to a file with **Save** button and also restore them with **Open** button.

You can use previously obtained points with the **'Reload'** button.

Points obtained with 'Store' feature will populate the group of 'From' points.

Points obtained with 'Capture' feature will populate the group of 'To' points.

All points can be cleared from the list with the **'Clear'** button.

With button **'Calculate'** we calculate the transformation matrix for current groups of 'From' and 'To' points.

2.3.11.19 Capture & Measure Points

Measured points are obtained with the help of semi-automated process. Measured points are usually used for Warp and Digitize 3D features and calculation of Transformation matrix.

Measure procedure requires configuration of parameters such as tool radius, touching probe end ball radius, copper sensor thickness, direction of edge measurement, grid point density etc..

Software needs these parameters in order to compensate them.

Captured points are machine position coordinates that are obtained without semi-automated process. Capture points group include plain machine position coordinates without offsets, points with direction, points captured with camera and Stored points.

Measure	▶
Capture	▶
Clear	▶
Load Points ...	/4841
Save Points ...	/4842
Save Points To STL ...	/4843
Save Points To G-Code ...	/4844
Save Points To DXF ...	/4845

2.3.11.19.1 Measure

Initiates measuring procedure using 'movable' tool sensor/touching probe.

Measure Edge X-	(0) /48001
Measure Edge X+	(0) /48002
Measure Edge Y-	(0) /48003
Measure Edge Y+	(0) /48004
Set Edge ...	/48005
Measure Point Z	(0) /48006
Measure Grid Z	(0) /48007
Measure Grid Z Offset	/48008
Set Grid ...	/48009
Digitize 3D	(0) /48010
Set Digitize 3D ...	/48011

Measure Edge group of features, are usually used for calculating transformation matrix.

Measure Edge X-:

Moves the X axis in positive direction, stops on contact with tool sensor/ touching probe and captures point. Then it moves for return distance, retracts and moves in Y positive direction for distance defined with Length value. This sequence of moves repeats for value of 'Count' times.

Measure Edge X+:

Moves the X axis in the negative direction, stops on contact with tool sensor/ touching probe and captures point. Then it moves for return distance, retracts and moves in Y negative direction for distance defined with Length value. This sequence of moves repeats for value of 'Count' times.

Measure Edge Y-:

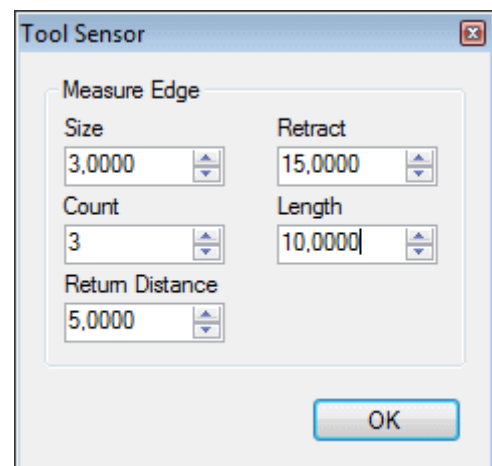
Moves the Y axis in the positive direction, stops on contact with tool sensor/ touching probe and captures point. Then it moves for return distance, retracts and moves in X positive direction for distance defined with Length value. This sequence of moves repeats for value of 'Count' times.

Measure Edge Y+:

Moves the Y axis in the negative direction, stops on contact with tool sensor/ touching probe and captures point. Then it moves for return distance, retracts and moves in X negative direction for distance defined with Length value. This sequence of moves repeats for value of 'Count' times.

Set Edge:

Here we set measure 'Edge' procedure parameters.



Measure Edge	
Size	Retract
3.0000	15.0000
Count	Length
3	10.0000
Return Distance	
5.0000	

OK

Size:

Size is distance from the Z axis center to point of contact. If tool and movable tool sensor are used, then this distance is Tool radius + sensor thickness. If touching probe is used, this distance is radius of touching probes end ball.

Return Distance:

Value represents distance for which tool returns when sensor/touching probe is triggered.

Count:

Number of measuring points.

Length:

Distance between two neighboring measuring points.

Retract:

Value represents distance for which machine retracts (from position to which previously returned to).

Measure Grid Z group of features are usually used for 'Warp'.

Measure Point Z:

If you need to measure single point on surface of material, you can use 'Measure Point Z'.

Measure Grid Z:

If you need to measure grid of points on surface of your material, you can use 'Measure Grid Z' feature. Usually this feature is used in production of PCB's, where we measure unevenness of the surface, but it can be used for other applications also.

The whole Measure Grid Z procedure goes like this:

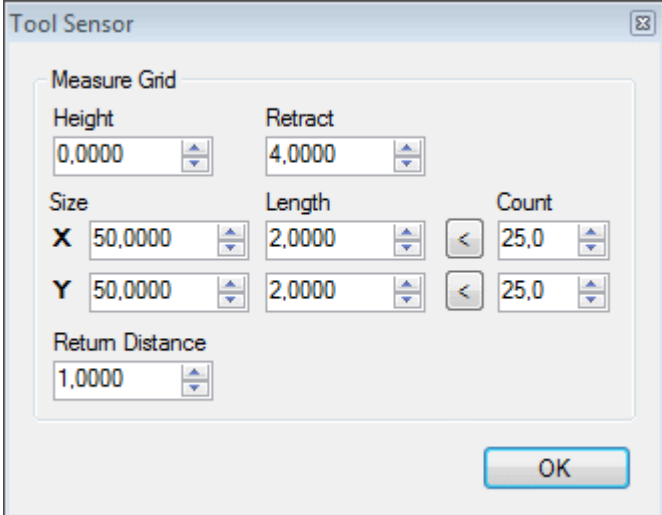
1. Put PCB on machine table
2. Connect PCB to sensor
3. Move machine to start position, usually corner of PCB
4. "Machine/Offset/Current XY" to set X0 Y0
5. Move to approximate center of PCB
6. "Capture & Measure Points/Measure/Measure Grid Z Offset" to set Z0
7. Move back to X0 Y0
8. "Capture & Measure Points/Measure/Set Grid ..." to set grid size and point density
9. "Capture & Measure Points/Measure/Measure Grid Z" to start measuring

Measure grid Z Offset:

Before you start measuring your grid, you need to measure Grid Z Offset. This is the surface height of your workpiece material.

Set Grid:

Here we set the grid of measuring points over copper board.



The screenshot shows a 'Tool Sensor' dialog box with a 'Measure Grid' tab. The dialog contains the following fields and controls:

- Height:** 0.0000
- Retract:** 4.0000
- Size X:** 50.0000
- Size Y:** 50.0000
- Length:** 2.0000
- Count:** 25.0 (two instances)
- Return Distance:** 1.0000
- OK** button

Height:

Height is distance between surface and contact point . If we want to measure grid Z over copper board that will be used for PCB, then value of 'Height' = 0.

Retract:

Value represents distance for which machine retracts (from position to which previously returned to).

Size:

XY Dimensions of copper board

Length:

Calculated distance between two adjacent measuring points.

Count:

Number of desired measuring points in X or Y axis of copper board.

Example:

First we set dimensions of the board (XY=50x50) and how many points we want to measure in X and Y coordinates (25). With 'Count' button length between two adjacent points is calculated(result is 2mm).

Return Distance:

Value represents distance for which tool returns once it has touched and made contact.

Digitize 3D:

Initiates measuring procedure using a touching probe.

Set Digitize 3D:

Here we set parameters of Digitize 3D feature.

Raster:

With Raster we define the distance between two neighboring scan lines.

Angle:

Surface is scanned in lines at defined Angle.

X,Y:

With X and Y values we set the dimensions of Digitize 3D feature.

Inside these dimensions constraints scanning will be applied.

Depth:

With Depth we set the value of lowest height at which scanning will still be possible.

Distance:

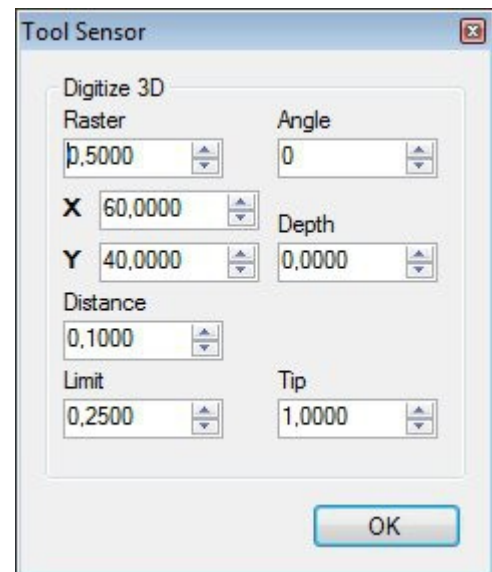
Distance between two scanning points.

Limit:

Density of scanning points is limited with 'Limit' value.

Tip:

Radius of the end ball of touching probe.



2.3.11.19.2 Capture

Captures machine position coordinates.

Capture Point	(0) /48101
Capture Camera Point	(0) /48102
Capture Point X-	(0) /48111
Capture Point X+	(0) /48112
Capture Point Y-	(0) /48113
Capture Point Y+	(0) /48114
Capture Point Z	(0) /48115
Store Position Point	(1) /48121
Store Camera Point	(1) /48122
Store Selected Line Start Point	(1) /48123
Store Selected Line End Point	(1) /48124

Capture Point:

Stores the current machine position as an entry in the 'points' list.

Capture Camera Point:

Stores the current camera view (current machine position with camera offset) as an entry for XY position points in the 'points' list.

Capture Point X-:

Captures the current machine position as an entry for "Edge X-" in the 'points' list.

Capture Point X+:

Captures the current machine position as an entry for "Edge X+" in the 'points' list.

Capture Point Y-:

Captures the current machine position as an entry for "Edge Y-" in the 'points' list.

Capture Point Y+:

Captures the current machine position as an entry for "Edge Y+" in the 'points' list.

Capture Point Z:

Captures the current machine position as an entry for "Grid Z" in the 'points' list.

Store Position Point:

Stores current machine position as an entry for "Stored points" in the points list.

Store Camera Point:

Stores current Camera position as an entry for "Stored points" in the points list.

Store Selected Line Start Point:

Stores currently selected G-code lines starting point as an entry for "Stored points" in the points list.

Store Selected Line End Point:

Stores currently selected G-code lines ending point as an entry for "Stored points" in the points list.

2.3.11.19.3 Clear

Clear Points:

Clears not edge related points from the 'points' list.

Clear Points X-:

Clears "Edge X-" points from the 'points' list.

Clear Points X+:

Clears "Edge X+" points from the 'points' list.

Clear Points Y-:

Clears "Edge Y-" points from the 'points' list.

Clear Points Y+:

Clears "Edge Y+" points from the 'points' list.

Clear Points Z:

Clears "Edge Z" points from the 'points' list.

Clear Points	(0) /48201
Clear Points X-	(0) /48202
Clear Points X+	(0) /48203
Clear Points Y-	(0) /48204
Clear Points Y+	(0) /48205
Clear Points Z	(0) /48206
Clear Points 3D	(0) /48207
Clear Stored Points	(0) /48208
Clear All Points	(0) /48209

Clear points 3D:

Clears 3D Digitize related points from the 'points' list.

Clear Stored Points:

Clears all 'Store' related points from the 'points' list.

Clear All Points:

Clears all points from the 'points' list.

2.3.11.19.4 Load Points

Loads points to file.

2.3.11.19.5 Save Points

Saves points to file.

2.3.11.19.6 Save Points To STL

Generates STL file format from points.

2.3.11.19.7 Save Points To G-Code

Generates G-code file format from points.

2.3.11.19.8 Save Points To DXF

Generates DXF file format from points.

2.3.11.20 Soft Limits

Soft limits are used as safety precaution so that machine doesn't make hard stop or crash into the limit switches with full speed. When machine activates 'Soft Limits', it decelerates and stops without any "stress". Soft limits can be enabled or disabled with this feature. 'Soft Limits' parameters can be defined in *File/Settings/Axes/Soft Limits*.

2.3.11.21 Firmware

Firmware management functions for update and verification.

Update:

Updates controller firmware with version embedded in software. It is not possible to destroy or damage controller with firmware update. If firmware update fails it is always possible to update it again.

Verify:

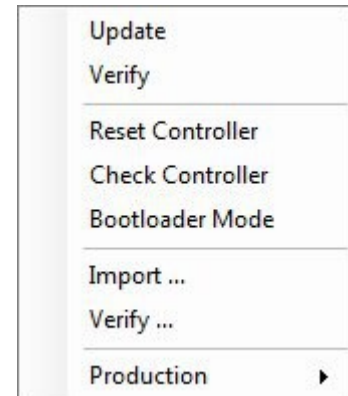
Verifies if firmware is correct for current software release.

Reset Controller:

Performs controller reset. Controller will be rebooted. This will not affect settings and license.

Check Controller:

This will verify controller's internal memory. If problem is found, then LED's on controller will start blinking in specific pattern.



2.3.12 Help menu

The help menu provides access to offline and internet assistance for CNC USB software. Reference content related to supported G-Code is also linked.

Software license activation and management are also catered for along with information regarding start-up, update status and software version.

2.3.12.1 Help

Launch PDF reader with this document loaded.

PDF reader software must be installed on the system or the document cannot be viewed.



2.3.12.2 Planet CNC Forum

PlanetCNC forum offers help for all hardware, software related problems and provides answers for all PlanetCNC related questions.

2.3.12.3 G-Code wiki

Link to G-code Wikipedia page.

2.3.12.4 G-Code Reference

Link to G-code reference page.

2.3.12.5 Activate License

Displays the license activation panel, for entry of the software 'registration key'. When software is purchased, a 'registration key', matching controller hardware / software, is sent via email. The bottom line of 'registration key' is copied from email and pasted into the large panel area.

Be attentive about the shape of your license key, because some e-mail clients parse the code in more lines and add additional characters which, in result, make registration faulty.

2.3.12.6 Import License

Imports license file, allowing licensed software to transfer to alternative computer hardware or to restore from backup.

2.3.12.7 Export License

Exports license file, allowing software license to transfer to alternative computer hardware or to make backup.

2.3.12.8 Log

Software keeps a log of various system files and settings related to software compatibility and performance. If problems are encountered, this information can assist trouble-shooting and problem solving.

2.3.12.9 Check For Updates

Checks for software updates. An active internet connection is required. CNC USB Software is regularly updated as features are advanced and refined. It's of much benefit to check for updates on a regular basis.

2.3.12.10 About

Displays the software logo screen along with software and hardware version information. As with log data, this can be helpful if trouble-shooting or problem solving.

2.4 Settings

The panel allows setup of all software options and features. Settings allow automation and integration of many features to provide advanced functions and simplify or speed-up many common tasks. Required options are dependent on user machinery, ancillary hardware and application.

It's MOST IMPORTANT that software is appropriately configured before any attempt is made to control CNC machinery. Failure to do so may result in serious injury or damage to machinery. With unknown or untested machinery, it's safer to keep initial motion rates below anticipated maximums.

Emergency stop and limit switch hardware should be configured and tested before higher traverse or 'rapid' move rates are performed.

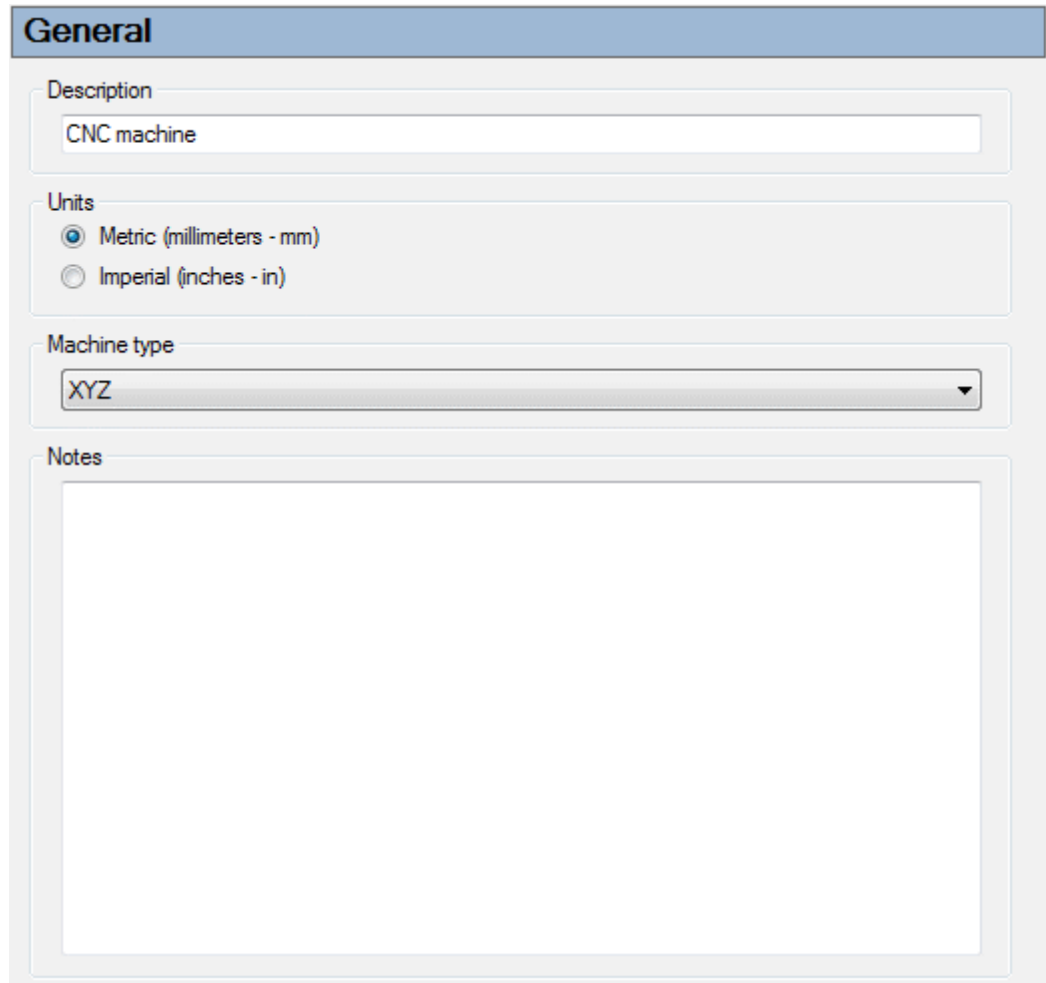
Settings shown in this section are examples. It's the users responsibility to ensure that safe and appropriate settings are applied to suit the controlled machine. Mk3 controller settings are used in examples. Most settings are applicable to other controllers. If specific settings for certain model of controller are required, this is noted in the text.

Some settings relate to hardware. These might be items like tool sensors, jog keyboards, MPG Pendants, tool-changers and so forth. If hardware is not available when software is configured, it can be installed and configured separately. It may be helpful to have hardware in place and operational before configuring related options. This allows testing or adjustment to determine safe and suitable parameters. Again, for reasons of safety, this is particularly the case with emergency stop and limit switch hardware.

General
Connection
User Interface
Axes
Setup
Options
Backlash
Limits
Soft Limits
Homing
Speed
Lookahead
Options
Output
Pins
Jog
Limit
Input
Input Ext
Ctrl
MPG Pendant
Spindle Synch
Tool table
Tool Change
Tool Change ATC
Tool Sensor
Materials
Parameters
Accessories

2.4.1 General

In 'General' tab we set measurement units, description of machine or settings used, and notes that could notify or aware the user.



The 'General' settings window is shown with a blue header. It contains four sections: 'Description' with a text input field containing 'CNC machine'; 'Units' with two radio buttons, 'Metric (millimeters - mm)' which is selected, and 'Imperial (inches - in)'; 'Machine type' with a dropdown menu showing 'XYZ'; and 'Notes' with a large empty text area.

2.4.1.1 Description

Anything written in description bar, is displayed in top left corner of CNC USB software window. This is helpful when indicating which setting file is used.



2.4.1.2 Units

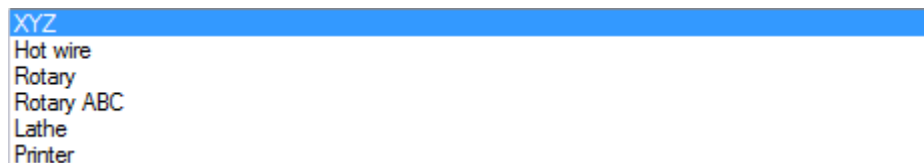
Sets the measurement units used in software.

Metric (millimeters-mm) or **Imperial** (inches-in) units can be used.

If you change this settings, all values in settings will be recalculated to different units.

2.4.1.3 Machine Type

You can choose between different machine types. Each machine type has specific layout that in some cases of chosen type of machine can be seen in 3D display.



XYZ:

Typical three axis machine.

Hot wire:

Foam-cutter which uses X and Y axes on one tower and U and V on second tower.

Rotary:

Machine with rotary 4th axis set as axis A.

Rotary ABC:

Machine with rotational axes A, B, C.

Lathe:

Lathe with axes Z and X.

Printer:

3D printer with E axis as extruder. E axis is assigned to axes A, B or C with tool number.

Tool 0 : A axis, Tool 1: B axis, Tool 2: C axis

2.4.1.4 Notes

Here you can write anything you like.

2.4.2 Connection

Controller with Ethernet connector can communicate with CNC USB software via USB of Network connection.

The screenshot shows a 'Connection' dialog box with a blue header. It contains two radio buttons: 'USB' (selected) and 'Network'. To the right of the 'Network' option is an 'Automatic Reconnect' checkbox. Below the radio buttons are four input fields for an IP address, showing '192', '168', '1', and '200'. Underneath is a section titled 'Available Network Controllers' with a 'Scan' checkbox and a large empty rectangular box. To the right of this box are input fields for 'IP', 'Mask', 'MAC', and a 'DHCP' checkbox. A 'Change' button is located at the bottom right of the 'Available Network Controllers' section.

USB:

When enabled, USB connection is used.

Network:

When enabled, Network connection is used.

IP:

Default IP of controller

Automatic reconnect:

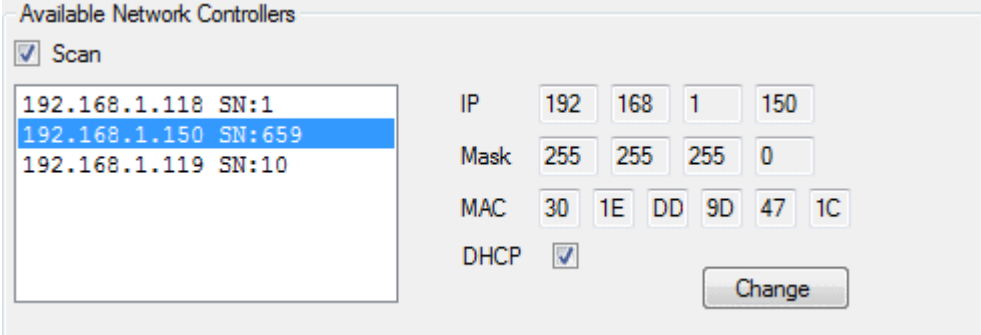
When connection fails, controller automatically reconnects with computer.

(Recommended: disabled)

Available Network Controllers:

Scan:

When enabled, you will be able to see all controllers that are connected to your network. IP and serial number are displayed for each controller visible in scan view list.



The 'Available Network Controllers' dialog box features a 'Scan' checkbox which is checked. Below it is a list box containing three entries: '192.168.1.118 SN:1', '192.168.1.150 SN:659' (which is highlighted in blue), and '192.168.1.119 SN:10'. To the right of the list box are input fields for network parameters: IP (192, 168, 1, 150), Mask (255, 255, 255, 0), MAC (30, 1E, DD, 9D, 47, 1C), and a DHCP checkbox which is also checked. A 'Change' button is located at the bottom right of the dialog.

IP:

IP of selected controller from scan view list.

Mask:

Mask of selected controller from scan view list.

MAC:

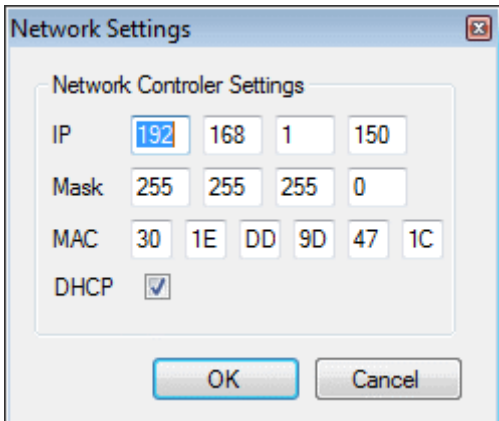
MAC address of selected controller from scan view list.

DHCP:

DHCP status of selected controller from scan view list.

Change:

You can change network parameters of selected controller from scan view list.



The 'Network Settings' dialog box contains a section titled 'Network Controller Settings'. This section includes input fields for IP (192, 168, 1, 150), Mask (255, 255, 255, 0), MAC (30, 1E, DD, 9D, 47, 1C), and a DHCP checkbox which is checked. The '192' field in the IP address is highlighted with a blue border. At the bottom of the dialog are 'OK' and 'Cancel' buttons.

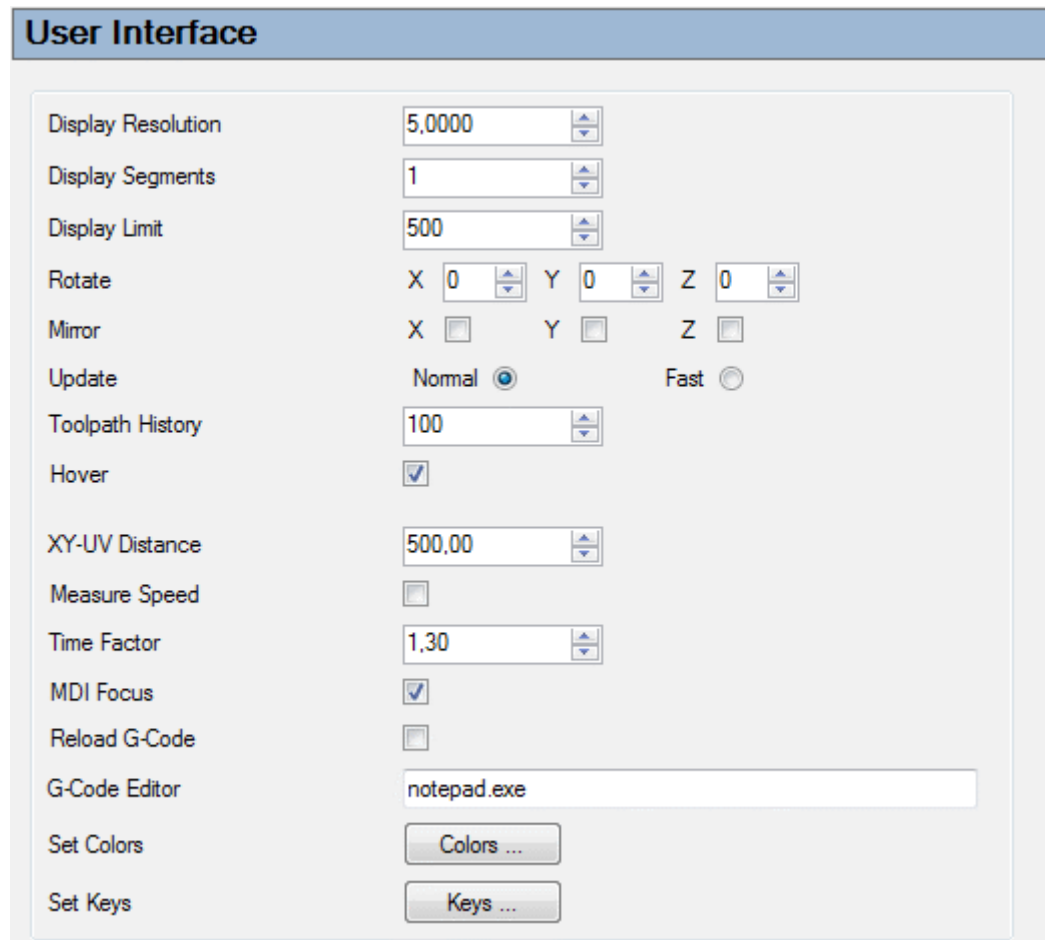
If your network device does not use DHCP, then controller will not be provided with the IP from DHCP. Meaning, controller will not be visible in scan list. That is why controller has inner built timer, which times out after certain amount of time and assign's controller its IP. Controller is then visible in scan view, where we can disable DHCP option so we can avoid this in the future.

2.4.3 User Interface

2.4.3.1 Display Resolution:

The distance between two points in display arc interpolation.

(Recommended: 1mm-5mm)



The screenshot shows a 'User Interface' settings window with a light blue header. The settings are organized into a list on the left and their corresponding controls on the right. The controls include spinners for numerical values, checkboxes for boolean options, and radio buttons for selection. At the bottom, there are two buttons: 'Colors ...' and 'Keys ...'.

Setting	Value/Option
Display Resolution	5,0000
Display Segments	1
Display Limit	500
Rotate	X: 0, Y: 0, Z: 0
Mirror	X: ☐ , Y: ☐ , Z: ☐
Update	Normal (selected), Fast
Toolpath History	100
Hover	☒
XY-UV Distance	500,00
Measure Speed	☐
Time Factor	1,30
MDI Focus	☒
Reload G-Code	☐
G-Code Editor	notepad.exe
Set Colors	Colors ...
Set Keys	Keys ...

2.4.3.2 Display Segments:

Minimal number of points in display arc interpolation.

(Recommended: 6-12)

2.4.3.3 Display Limit:

Limit number of points on display. Huge G-code files can generate more points than your computer can handle.

(Recommended: 50-500)

2.4.3.4 Rotate:

Rotates 3D display.

(Recommended for typical CNC : 0°, 0°, 0° ; Recommended for lathe: 0°, 90°, 0°)

2.4.3.5 Mirror:

Mirrors axes on 3D display.

2.4.3.6 Update:

Screen update rate.

(Recommended for Mk3: Fast for other controllers is Normal)

2.4.3.7 Tool-path History:

Length of tool footprint. Tool footprint is gray line that shows where tool has moved. This line is inaccurate and only shows approximate position.

(Recommended: 0-10000)

2.4.3.8 Hover:

This enables toolpath highlighting when mouse hovers over. Right mouse double click on highlighted line selects corresponding g-code line.

2.4.3.9 XY-UV Distance:

Distance between towers on Hot-wire machine types.

2.4.3.10 Measure Speed:

If enabled displayed screen is measured from movement, if disabled, display speed is calculated from step frequency.

(Recommended: Enabled)

2.4.3.11 Time Factor:

Value is used when calculating cutting time estimate.

(Recommended: 1.3)

2.4.3.12 MDI Focus:

If enabled, MDI box stays focused after MDI command.

2.4.3.13 Reload G-Code:

If enabled, reloads G-code if g-code file was modified with external editor.

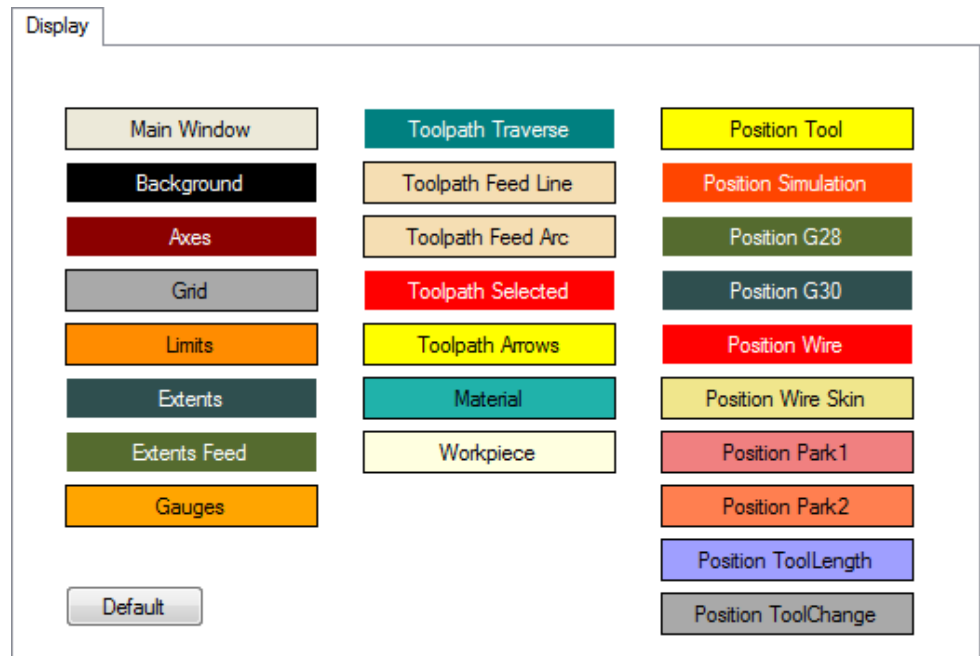
2.4.3.14 G-Code Editor:

External editor for 'Edit G-code' feature.

2.4.3.15 Set Colors:

Display:

Named swatches identify visualization elements and applied colors. Color is adjusted by selection of an element. A windows color selection panel will display. Adjustment or selection of an alternate color is made and the software applies this to the element. Visualization colors can be reset to initial values using the 'Default' button.



G-Code:

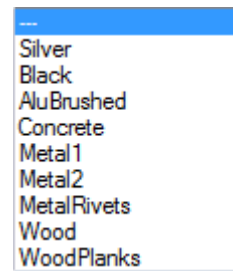
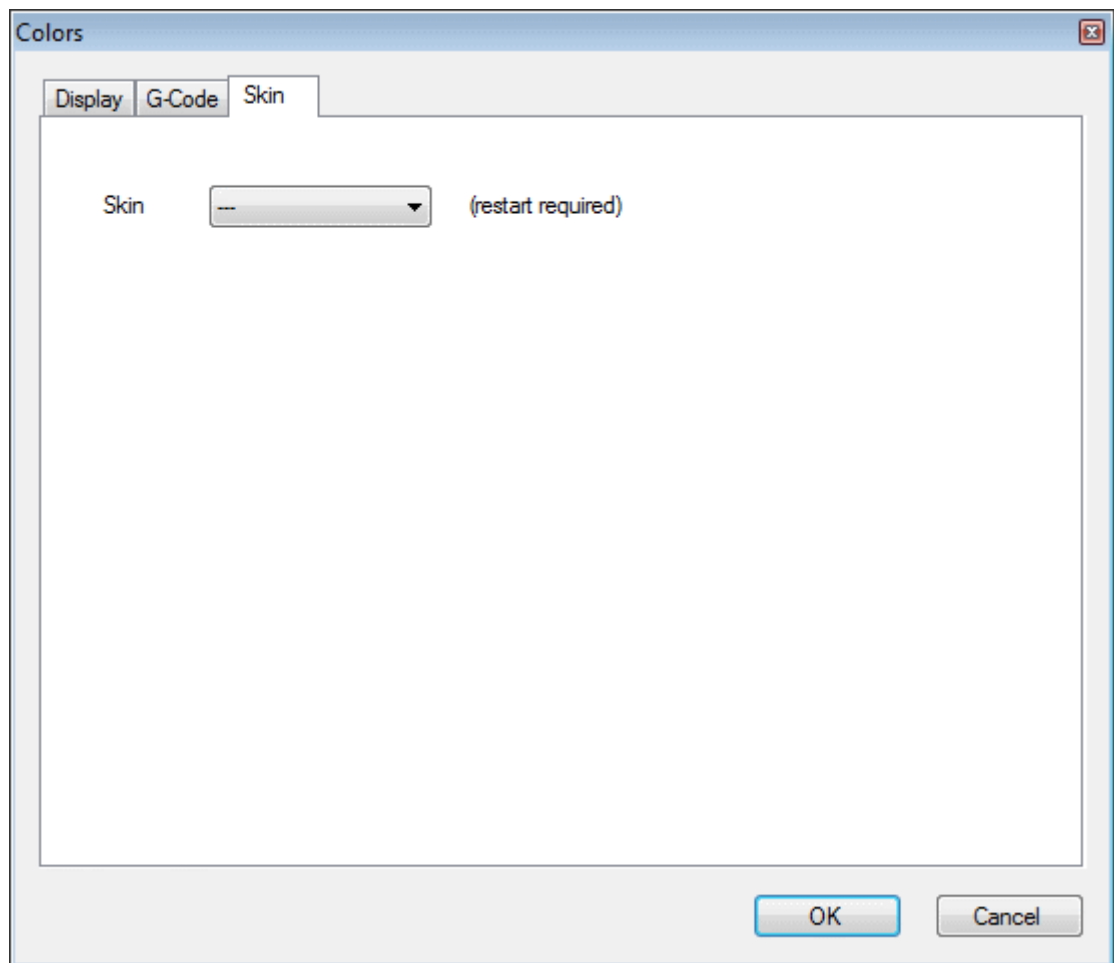
Named swatches identify G-Code words and other listing elements, displaying the applied color.

Color assignment is sadjusted as previously described. The 'G-Code' tab has an additional 'Enable' option for use of colors in the G-Code panel. 'Default 1' and 'Default 2' buttons provide two preconfigured color options.



Skin:

You can set different look of main window.

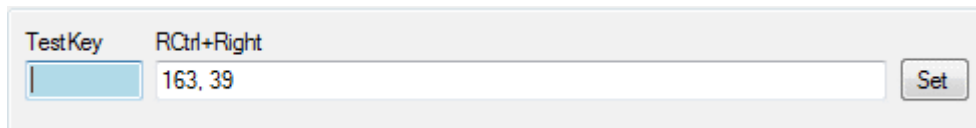


2.4.3.16 Set Keys:

You can set keyboard key combinations for jogging sets and all other features of CNC USB software. This is faster and easier way of accessing desired features and their parameters then through menus.

Shortkey or key combination can be assign to certain feature by clicking the 'TestKey' box and pressing desired key or combination of keys. In the box to the right you can see how for each key, specific key code corresponds.

With 'Set' button we assign key combination to selected software feature selected from the list below.



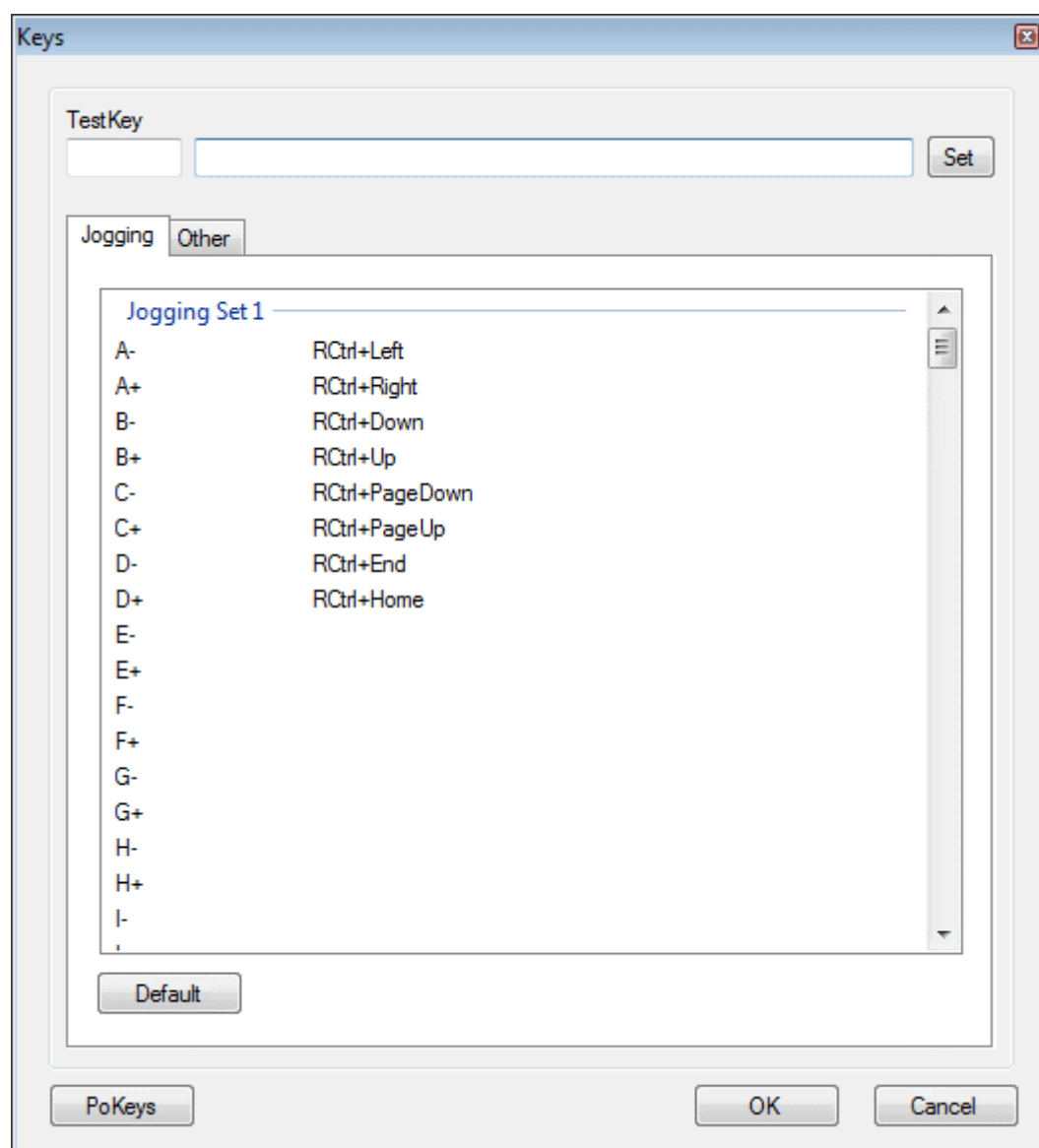
Jogging tab:

Each jogging direction can have three different modes: Normal, Step and Sync.

Each jogging direction/mode can have three different sets of key combinations assigned. For jogging modes 'Jogging', 'Jogging STEP' and 'Jogging SYNC' we can configure key combinations in three different shortcut sets: **Set 1**, **Set 2** and **Set3**.

Other tab:

Features are sectioned by menus to which they are affiliated to. For each feature you can set key combination.



2.4.4 Axes

This tab is used to specify the number of axes used on the controlled machine as well as axis names and other options.

Axes

Number of Axes3R

Dir/Step Delay30µs

Axis SPU CalibrationCalibration ...

Motor Enable

Use Motor Enable☒

Invert☐

On Exit☐

Dynamic☐

	Name	Function	Reverse	Invert Pulse
Axis 1	X	----	☐	☐
Axis 2	Y	----	☐	☐
Axis 3	Z	----	☐	☐
Axis 4	----	----	☐	☐
Axis 5	----	----	☐	☐
Axis 6	----	----	☐	☐
Axis 7	----	----	☐	☐
Axis 8	----	----	☐	☐
Axis 9	----	----	☐	☐

Number of Axes:

The number of axes on controlled CNC machine.

R:

Resets all axis names to defaults.

Dir/Step Delay:

The value of delay after direction change. This is intended for older motor drivers. Available for Mk2 controller only. Mk3 controller uses maximum possible value by default.

Axes SPU Calibration:

Motor Enable:

Some motor drivers use Enable signal input. PlanetCNC controllers have Enable pin on their axis connectors.

Use Motor Enable:

'Motor Enable' signal can be enabled or disabled.

Invert:

'Motor Enable' signal can be inverted.

On Exit:

Motors are disabled on exit of CNC USB software.

Dynamic:

Motors are enabled only when they are moving.

Available for Mk3 controller only.

Name:

Specifies the G-Code name used for each axis. Standard names (X, Y, Z,...) should be used. If not, some software features will not be available.

Function:

Assigns 'Slave' function to selected axis. For example: Classic machine with X,Y and Z axes and two motors for X axis. 'Number of Axes' is set to 3: Axis 1 is X, Axis 2 is Y, Axis 3 is Z. Axis 4 is second motor for X axis and is set as 'Slave 1'.

Reverse:

Reverses motor direction.

Invert Pulse:

Inverts step signal (pulse) from 0V to 5V or from 5V to 0V.

2.4.4.1 Setup

Setup				
	Steps/Unit	Acceleration Ramp		
		Init speed	Maximum speed	Acceleration
X	800,0000	0,00	2000,00	100,000
Y	800,0000	0,00	2000,00	100,000
Z	800,0000	0,00	2000,00	100,000

Steps/Unit:

The number of steps the controller must generate to move the machine axis for distance of one unit (mm or inch).

Example:

A stepper motor using 200 steps per revolution, motor driver setting 1/16 micro-stepping and 4mm per revolution lead screw, requires a Steps per Unit value of: $200 * 16 / 4 = 800$.

Acceleration Ramp:

Init speed:

Motor starting speed on the acceleration curve.

(Recommended: 0 mm/min)

Maximum speed:

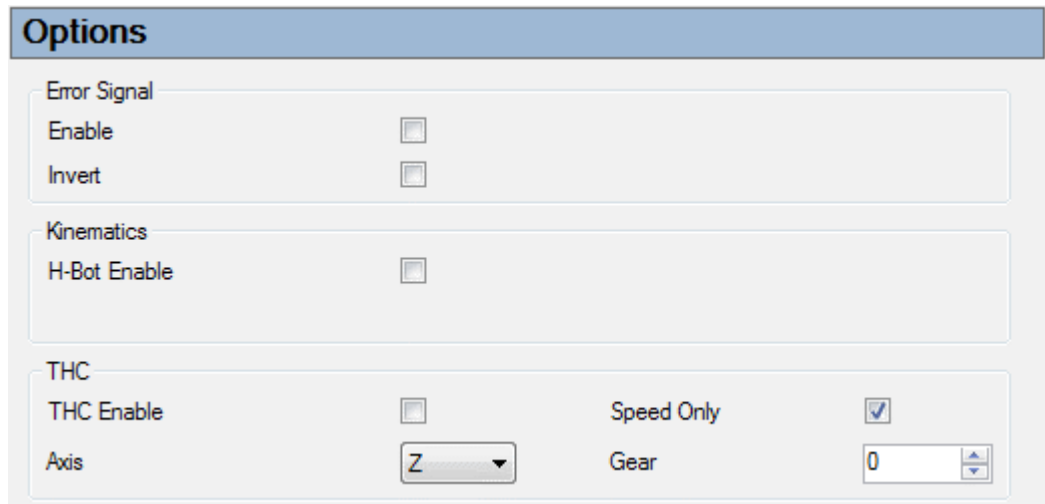
Maximum axis speed on acceleration curve. Value 0 disables maximum speed limitations.

Acceleration:

Rate of acceleration.

(Recommended: 10-500 mm/sec²)

2.4.4.2 Options



Options			
Error Signal			
Enable	☐		
Invert	☐		
Kinematics			
H-Bot Enable	☐		
THC			
THC Enable	☐	Speed Only	☒
Axis	Z ▼	Gear	0

Error Signal:

Some motor drivers support error signal. Error signal “notifies” controller of faulty machine movement and stops the machine. Available on Mk3 controller only.

Enable:

Enables/disables Error signal.

Invert:

Inverts Error signal.

Kinematics:

H-Bot Enable:

This enables H-bot kinematics.

THC:

THC stands for 'Torch Height Control'. It is used for plasma type machines where distance between the workpiece surface and the torch must be constant at all times of cutting for best results.

THC unit measures the distance between the torch and the material in any given moment of cutting. THC outputs are sending signals which controller interprets as control signals for Z axis.

Note: THC device control output pins are connected to Jogging keyboards A axis jog buttons.

THC Enable:

Enables/disables the use of THC device.

Speed Only:

Height is compensated only when machine is at full speed and not during accelerations and decelerations.

Axis:

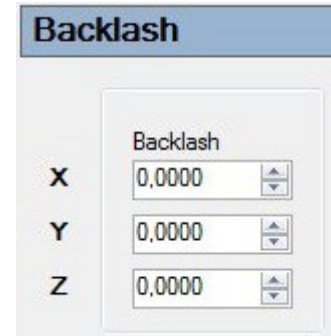
Choose between Z or W axis. Available on Mk3 controller only.

Gear:

Reduces speed of height compensation.

2.4.4.3 Backlash

Backlash compensation value. If value is not set to zero, it is recommended to enable 'Split arcs' and 'Direction Change' in *Setting/Lookahead*.



Backlash	
X	0,0000
Y	0,0000
Z	0,0000

2.4.4.4 Limits

Limits			
	Limit -	Limit +	
X	0,000	0,000	Limit Switches Stop ☐ - ☐ + ☐ - ☐ + ☐ - ☐ +
Y	0,000	0,000	
Z	0,000	0,000	

Limit Switches:**Stop-:**

If checked, trigger of a limit switch in negative direction of selected axis stops the machine.

Stop+:

If checked, trigger of a limit switch in positive direction of selected axis stops the machine

Limit-:

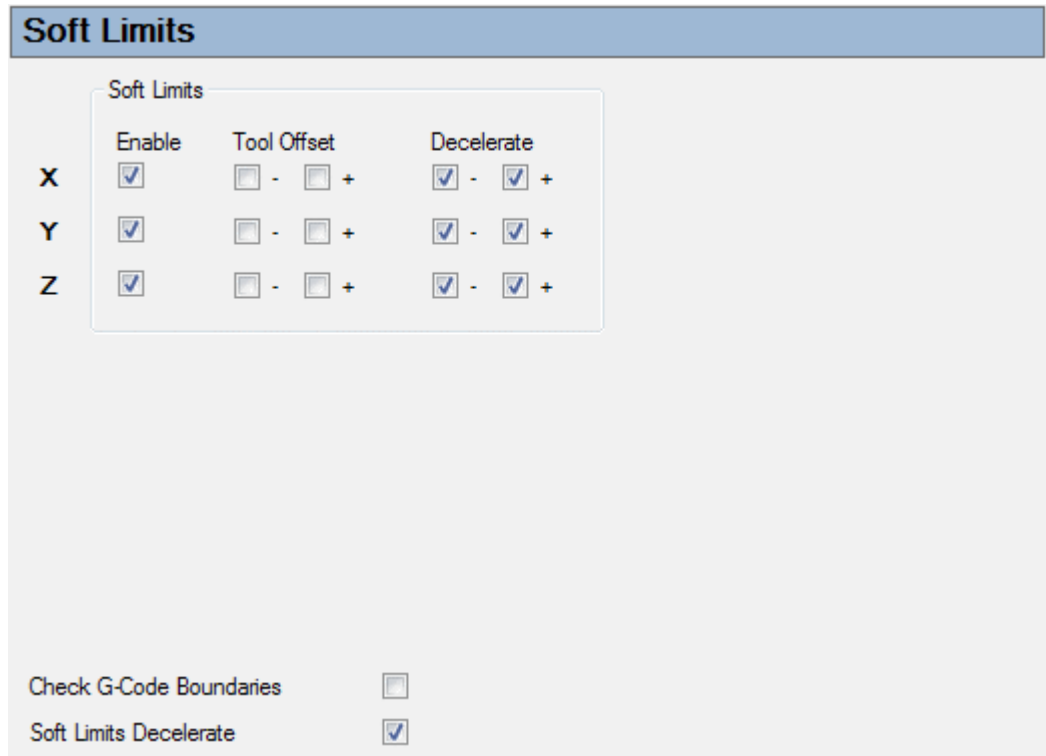
Negative limit value of selected axis of machine working area.

Limit+:

Positive limit value of selected axis of machine working area.

2.4.4.5 Soft Limits:

Soft limits are used as safety precaution so that machine doesn't make hard stop or crash into the limit switches with full speed. When machine activates 'Soft Limits', it decelerates and stops without any "stress". Soft limits use machine working area values.



	Enable	Tool Offset	Decelerate
X	☒	☐ - ☐ +	☒ - ☒ +
Y	☒	☐ - ☐ +	☒ - ☒ +
Z	☒	☐ - ☐ +	☒ - ☒ +

Check G-Code Boundaries ☐

Soft Limits Decelerate ☒

Soft limits are not available on Mk1 controller.

Enable:

Enables or disables soft limits.

Tool Offset:

If this option is enabled then working area is compensated with active tool offset. This is usually used in Z- direction to prevent the tool crashing to tool table. If tool is changed and its tool offset is measured, soft limits will consider offset of new tool in soft limit boundaries.

Available on Mk3 controller only.

Tool Offset-:

Soft Limits will consider tool offset when moving in negative direction of selected axis.

Tool Offset+:

Soft Limits will consider tool offset when moving in positive direction of selected axis.

Decelerate:

Machine decelerates from current speed and stops. Available on Mk3 controller only.

Decelerate-:

Deceleration is applied when soft limits are activated in negative direction of selected axis.

Decelerate+:

Deceleration is applied only when soft limits are activated in positive direction of selected axis.

Usually tool offset is not enabled and deceleration is enabled, however in Z- direction enabled tool offset and disabled deceleration might prevent tool from crashing to tool table. Mk2 controller has this feature enabled by default.

Check G-code Boundaries:

Software sends warning when imported toolpath boundaries exceed working area (soft limit boundaries).

Soft Limits Decelerate:

Enables deceleration of machine when soft limits are activated.

2.4.4.6 Homing

Homing

	Sequence	Speed	Direction	Set Position	Go To
X	3	0,00	☒ - ☐ +	0,00	0,00
Y	2	0,00	☐ - ☒ +	0,00	0,00
Z	1	0,00	☐ - ☒ +	0,00	0,00

Enable

☐

Return Distance

0,0000

Go To First

☐

Enable:

Enables use of the automatic homing procedure.

Return Distance:

When a limit switch is triggered, the machine will move back for 'Return Distance' value, and release the triggered switch.

Go To First:

When enabled, axes with defined 'Go To' value will finish its homing with 'Go To' move, before next axis homing sequence starts.

Sequence:

The order in which axes are homed. It's normal practice to first 'home' the Z axis, to ensure the tool is clear of work, prior to homing moves of other axes.

Speed:

Speed of the machine during homing. A low homing speed is advised. Machine axes will stop without deceleration when a limit switch is triggered.

Direction:

Defines the direction of motion applied in homing moves - negative or positive direction.

Set Position:

This value sets limit switch position when it is triggered. Together with 'Limits' settings this defines working area and absolute position.

Go To:

Moves axis to position as final step of its homing.

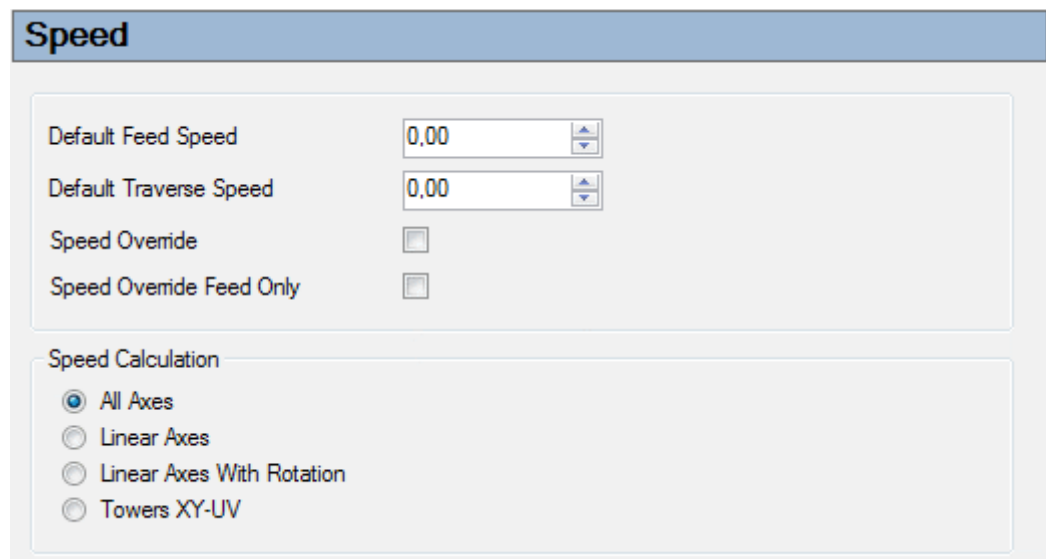
Example:

A machine has two X axis limit switches 500 mm apart. To give tolerance of 10mm on each side, the working area is set as 'Limit- = 0' and 'Limit+= 480'. Without this tolerance the X axis has absolute value range of -10 mm to +490 mm = 500 mm.

Homing in the negative X direction ('Direction = -'), the machine triggers a limit switch at location -10 mm. Because of 'Return Distance = 3', the X axis moves back 3 mm to location -7 mm, releasing the switch. This is limit switch position and 'Set position' value of = -7 mm is set.

Finally 'Go To = 0' initiates a traverse of 7 mm in X positive direction, stopping at 0 mm.

2.4.5 Speed



The screenshot shows a window titled "Speed" with a light blue header. Inside, there are four settings: "Default Feed Speed" and "Default Traverse Speed" both set to "0,00" with up/down arrows; "Speed Override" and "Speed Override Feed Only" both with unchecked checkboxes. Below these is a "Speed Calculation" section with four radio button options: "All Axes" (selected), "Linear Axes", "Linear Axes With Rotation", and "Towers XY-UV".

Default Feed Speed:

Default feed rate (G01,G02,G03). Used if 'F' word is unspecified in G-Code.

Default Traverse Speed:

Traverse rate (G00). Speed at which machine travels at traverse moves.

Speed Override:

Overrides G-Code 'F' word values. Also set or adjusted in the main GUI using a checkbox next to the speed slider.

Speed Override Feed Only:

If enabled, override affects feed rate only and leaves traverse speed at same rate.

Speed Calculation:

To calculate the speed, software needs to know the distance machine has traveled. Because some machines use rotational and tower axes, distance is calculated differently.

All Axes:

All axes are treated as linear axes in distance calculation.

Linear Axes:

Only linear axes X, Y, Z, U, V and W are used in distance calculation.

Linear Axes With Rotation:

Linear axes X, Y, Z, U, V and W are used in distance calculation. Rotational axes A, B and C are used to rotate X, Y and Z in distance calculation.

Towers XY-UV:

X, Y, Z distance and U, V, W distance is calculated separately. Larger value is used.

Towers with rotation:

X, Y, Z distance and U, V, W distance is calculated separately. Rotational axes A, B and C are used to rotate X, Y and Z in distance calculation. Larger value is used.

2.4.6 Lookahead

Lookahead			
Angle	170		
Radius	0,5000	Angle	20
Acceleration			
Radial Acceleration	0,0000		
Path Tolerance	0,2500		
Path Blend	☒	Speed Adjust	☒
Path Range	5,00		175,00
Other			
Split Arcs	☐		
Direction Change	☐	Force	☐
Dominant Axis Change	☐		

Angle:

The angle the CNC machine can travel without reducing speed.

Example: The value of 150 degrees means that a dodecagon (a twelve-sided polygon) having internal angles of 150 degrees, can be cut without reducing speed at corners.

(Recommended: 150-178 deg)

Radius:

Minimum arc radius that machine can travel without reducing speed.

Value zero disables this feature.

(Recommended: 0 units)

Angle:

If arc angle is less than this value then 'Radius' feature is disabled.

(Recommended: 20 deg)

2.4.6.1 Acceleration:

Radial Acceleration:

Maximum radial acceleration that your machine can handle. This parameter is used to calculate maximum speed that machine can travel in arcs and blended corners.

(Recommended: 0-200 mm/sec²)

Path Tolerance:

Maximum distance that blended toolpath can deviate from source toolpath.

Path Blend:

With Path Blend enabled *G64 Continuous Path Blend mode* is enabled by default. Otherwise *G61 Exact Path mode* is enabled. This can also be changed in G-code.

Speed Adjust:

Enables early deceleration if machine has to travel with lower speed.

(Recommended: Enabled)

Path Range:

Path Range is range of angle values that will be “blended” if 'Path Blend' is enabled.

E.g.: If Path range is defined from 15 deg to 175 deg then all angle values included in this angle range will be blended with tolerance defined with 'Path Tolerance'.

(Recommended Low: 5-45 deg ; High: 150-179.99 deg)

2.4.6.2 Other:

Split Arcs:

This feature splits arcs to smaller segments not larger than 45 deg. This is important if machine has backlash set.

(Recommended: Enabled)

Direction Change:

If backlash is set, speed is reduced if axis direction is changed.

(Recommended: Enabled)

-Force:

Speed is reduced if axis direction is changed. Backlash feature of direction change is ignored.

(Recommended: Disabled)

Dominant Axis Change:

Speed is reduced if dominant axis is changed. Dominant axis is axis that needs highest number of steps.

(Recommended: Disabled unless difference in SPU settings between axes is high.)

2.4.7 Options

Options

Interpolation	0.0010
Optimization	0.0250

Pause

Show Dialog	☐
Decelerate	☐

Stop

Decelerate	☐
Outputs Off	☐

Speed Tolerance

Speed Tolerance	0.00
Disable Helix Interpolation	☐
Disable List Move	☐

Interpolation:

Value used for interpolating arcs and other curves.

(Recommended: Mk3: 0.1-0.05mm ; Mk2: 0.5-0.1mm)

Optimization:

Moves with length shorter then this value are ignored.

(Recommended: 0)

2.4.7.1 Pause:

Show Dialog:

If enabled, when pause is activated dialog is displayed on screen.

Decelerate:

If enabled, when pause is activated machine decelerates and stops.

2.4.7.2 Stop:

Decelerate:

If enabled, when 'Stop' is activated machine decelerates and stops.

Outputs Off:

If enabled, when 'Stop' is activated all outputs are turned OFF.

Speed Tolerance:

This value sets how much can new speed value differentiate from previous one. This value is in percentage of speed value.

(Recommended: Mk3: 0)

Disable Helix Interpolation:

Disables helix interpolation.

(Recommended: Enabled)

Disable List Move:

Disables list move.

(Recommended: Disabled)

2.4.8 Output

Inverts signal on output pin.



The screenshot shows a software window titled "Output" with a light blue header. Below the header is a list of eight items, each labeled "Invert Output" followed by a number from 1 to 8. To the right of each label is a small, empty square checkbox. All checkboxes are currently unchecked.

Output	
Invert	
Invert Output 1	☐
Invert Output 2	☐
Invert Output 3	☐
Invert Output 4	☐
Invert Output 5	☐
Invert Output 6	☐
Invert Output 7	☐
Invert Output 8	☐

Invert output 1-8:**Mk3:**

Inverts output pins 1-8.

Mk2:

Inverts output pins 1-7.

Mk2/4:

Inverts output pins 1,2 and 7.

Mk1:

Inverts output pins 1-3.

2.4.8.1 Pins

Here we define output pins for 'Spindle' and 'Coolant' control. If you use Planet CNC Output board, you can follow this link to help you properly set output pins and connect output board to your controller.

http://www.planet-cnc.com/faq/output_board/

Pins

Spindle (M3, M4, M5)

Output pin - On/Off: [---]

Output pin - Direction: [---]

Output pin - Speed: [---] Min: 300 Max: 30000

Pause: ☐

Delay: On: 0.0 Off: 0.0

Use RC Controller: ☐ Lo: 400 Hi: 2500

Use PWM: ☐ Freq: 12000

Coolant (M7, M8, M9)

Output pin - Mist (M7): [---]

Output pin - Flood (M8): [---]

Pause

Output Pin: [---]

2.4.8.1.1 Spindle (M3, M4, M5)

Output pin On/Off:

Assigns output pin for spindle ON/OFF control (M3/M5).

Output pin Direction:

Assigns output pin for spindle direction control (M4).

Output pin Speed:

Assigns output pin for spindle speed control.

Min/Max:

Minimum and maximum speeds for spindle (RPM).

Pause:

If checked, spindle ON/OFF command will pause the machine, allowing manual control of spindle ON/OFF.

Delay On/Off:

Delay period allowing the spindle to spin-up or stop.

Individual time period settings for spin-up (ON) and stop (OFF) are assignable.

Use RC controller:

Generates RC servo or ESC compatible signal on the spindle speed pin.

Lo/Hi:

Minimum / Maximum values of RC servo speed range.

Use PWM:

Generates PWM signal.

Freq:

PWM base frequency.

Supported only for Mk3 controller.

2.4.8.1.2 Coolant (M7, M8, M9)**Output pin Flood:**

Assigns output pin for coolant 'Flood' ON/OFF control. (M8/M9).

Output pin Mist :

Assigns output pin for coolant 'Mist' ON/OFF control. (M7/M9).

2.4.8.2 Pause**Output pin:**

Assigns output pin that triggers when 'Pause' is generated.

2.4.9 Jog

'Jog' options configure use of software and hardware jogging keyboards.

The screenshot shows a software window titled "Jog" with a light blue header. The window contains two main sections. The top section has four rows of controls: "Reverse" with eight checkboxes, "Accelerate" with one checkbox, "Decelerate" with one checkbox, and "Step Distance" with a text box containing "0,1000" and a vertical spinner. The bottom section is titled "Jogging Keyboard" and contains six rows: "Enable" with a checked checkbox, "Invert" with one checkbox, "Swap" with four checkboxes, "Max Speed" with a text box containing "1800,00" and a vertical spinner, "Speed Adjust" with one checkbox, and "Shift Is Step" with one checkbox.

Reverse:

Reverses movement for selected axis. Axes order depends on your axes names and usually goes like this:

X, Y, Z, A, B, C, U, V, W

Accelerate:

If enabled, axis at each move accelerates.

Decelerate:

If enabled, axis at each move decelerates.

Step Distance:

The distance machine will travel on a single press of a jog button, in jog 'STEP' mode.

2.4.9.1 Jogging Keyboard

Enable:

Enables / disables use of hardware jogging keyboard.

Invert:

Allows use of Normally Open (NO) or Normally Closed (NC) switch hardware on jogging keyboards.

Swap:

Swaps the axis keys

(Assuming axes order is X, Y, Z, A, B, C, U, V, W)

First box: X axis and Z axis keys are swapped.
B axis and U axis keys are swapped (shift key active).

Second box: Y axis and A axis keys are swapped.
C axis and V axis keys are swapped (shift key active).

Third box: X axis and Y axis keys are swapped.
B axis and C axis keys are swapped (shift key active).

Fourth box: Z axis and A axis keys are swapped.
U axis and V axis keys are swapped (shift key active).

Fifth box: Y axis and Z axis keys are swapped.
C axis and U axis keys are swapped (shift key active).

Max Speed:

Maximum traverse rate in jog moves when using hardware jogging keyboards.

Software keyboard jog speeds are NOT affected by this option.

Speed Adjust:

When enabled, you can adjust speed in-between jogging moves with potentiometer on keyboard.

When not enabled, speed can be adjusted only between jog moves.

Shift is Step:

When checked, the 'SHIFT' key toggles 'STEP' jog mode. Unchecked, 'SHIFT' key toggles axis 5-8 jogging.

2.4.10 Limit

Limit input pins can be inverted and 'Normal' or 'Single Input' mode can be selected.

Limit	
Invert	
Invert Limit 1	☐
Invert Limit 2	☐
Invert Limit 3	☐
Invert Limit 4	☐
Invert Limit 5	☐
Invert Limit 6	☐
Invert Limit 7	☐
Invert Limit 8	☐
Invert Limit Shift	☐
Single Input ☐	
Lock Other Axes	☐

Invert Limit 1-9:

Invert options are for use of Normally Open (NO) or Normally Closed (NC) switch hardware.

Controllers can use 'Normal' or 'Single Input' mode:

Normal:

Each switch (for positive and negative axis direction) is connected to its own pin.

Single Input:

Both axis limit switches are connected to one pin. Direction of travel determines if positive or negative switch is triggered.

Lock Other Axes:

When axis limits switch is activated, movement of all other axes is locked.

2.4.11 Input

Mk2/4, Mk2 controllers:

NOTE: Mk2/4 has only Input 5 and Input 1 pin.

Input	
Use Pause	☐
Use Start	☐
Use Stop	☐
Use E-Stop	☐
Use Sensor	☐
Invert	
Invert Input 1	☐
Invert Input 2	☐
Invert Input 3	☐
Invert Input 4	☐
Invert Input 5	☐

Use Pause:

When enabled, switching of input toggles Pause/Resume of NC program. (INPUT 1)

Use Start:

When enabled, switching input triggers G-Code execution start. (INPUT 2)

Use Stop:

When enabled, switching input triggers NC program stop execution using a 'Normal' stop. (INPUT 3)

Use E-Stop:

When enabled, switching input triggers machine E-Stop. (INPUT 4)
(Essential safety feature!)

Use Sensor:

Input is used for 'Tool Sensor' connection. (INPUT 5)

Invert Input 1-5:

Invert options are for use of Normally Open (NO) or Normally Closed (NC) switch hardware.

Mk3 controller:

Mk3 has eight assignable input pins.

Input	
Input 1	---
Input 2	---
Input 3	---
Input 4	---
Input 5	---
Input 6	---
Input 7	---
Input 8	---

Invert	
Invert Input 1	☐
Invert Input 2	☐
Invert Input 3	☐
Invert Input 4	☐
Invert Input 5	☐
Invert Input 6	☐
Invert Input 7	☐
Invert Input 8	☐

Input 1-8:

Each input pin can be assigned for specific task:

Start
Stop
Pause Toggle
Pause Push
Pause On
Pause Off
E-Stop Toggle
E-Stop Push
E-Stop On
E-Stop Off
Spindle Toggle
Spindle Push
Spindle On
Spindle Off
Spindle Dir Toggle
Spindle Dir CW
Spindle Dir CCW
Spindle Speed 0%
Spindle Speed 25%
Spindle Speed 50%
Spindle Speed 75%
Spindle Speed 100%
Coolant M7 Toggle
Coolant M7 Push
Coolant M7 On
Coolant M7 Off
Coolant M8 Toggle
Coolant M8 Push
Coolant M8 On
Coolant M8 Off
Tool Sensor
Sensor
Probe

Invert Input 1-8:

Invert options are for use of Normally Open (NO) or Normally Closed (NC) switch hardware.

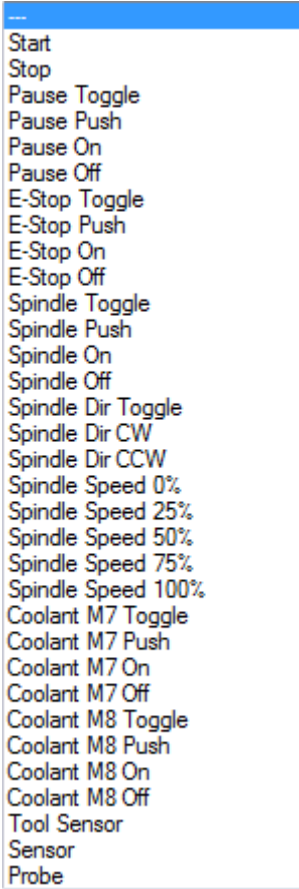
2.4.11.1 Input Ext

With external input board you can greatly increase the number of input pins.

Input Ext	
Ext 1	
Ext 2	
Ext 3	
Ext 4	
Ext 5	
Ext 6	
Ext 7	
Ext 8	
Ext 9	
Ext 10	
Ext 11	
Ext 12	
Ext 13	
Ext 14	
Ext 15	
Ext 16	
Ext 17	
Ext 18	
Ext 19	
Ext 20	
Ext 21	
Ext 22	
Ext 23	
Ext 24	
Ext 25	
Ext 26	
Ext 27	
Ext 28	
Ext 29	
Ext 30	
Ext 31	
Ext 32	

Ext 1-32:

Each external input pin can be assigned for specific task:



- Start
- Stop
- Pause Toggle
- Pause Push
- Pause On
- Pause Off
- E-Stop Toggle
- E-Stop Push
- E-Stop On
- E-Stop Off
- Spindle Toggle
- Spindle Push
- Spindle On
- Spindle Off
- Spindle Dir Toggle
- Spindle Dir CW
- Spindle Dir CCW
- Spindle Speed 0%
- Spindle Speed 25%
- Spindle Speed 50%
- Spindle Speed 75%
- Spindle Speed 100%
- Coolant M7 Toggle
- Coolant M7 Push
- Coolant M7 On
- Coolant M7 Off
- Coolant M8 Toggle
- Coolant M8 Push
- Coolant M8 On
- Coolant M8 Off
- Tool Sensor
- Sensor
- Probe

2.4.12 Ctrl

The screenshot shows a configuration window titled 'Ctrl'. It contains several settings for a control system:

Option	Checkbox	Label	Value	Option	Checkbox	Label	Value	Option	Checkbox
E-Stop	☐	Invert	☐						
Use MPG	☐	PPR	100	Reverse	☐				
Use Rev. Index	☐	PPR	1	Invert	☐	Fast	☐		
Use Rev. Encoder	☐	PPR	100	Reverse	☐				

E-Stop:

Enable 'E-Stop' to use E-Stop switch connected to CTRL1 pin.

Invert:

Invert option is for use of Normally Open (NO) or Normally Closed (NC) switch hardware.

Use MPG:

Enable "Use MPG" to use MPG pendant.

PPR:

Pulses per revolution.

Reverse:

Enable "Reverse" to set reverse MPG direction.

Use Rev. Index:

Enable "Use Rev. Index" to use index signal for measuring spindle RPM and spindle position.

PPR:

Pulses per revolution.

Invert:

Enable "Invert" option if your signal is active low type.

Fast:

With higher values you can enable 'Fast' options to increase response time

Use Rev. Encoder:

Enable "Use Rev. Encoder" to use encoder for measuring spindle RPM.

PPR:

Set "PPR" to your encoder pulses per revolution value.

Reverse:

Enable "Reverse" to set reverse direction.

2.4.12.1 MPG Pendant

Mode	Factor	Step	Options		
Mode 0	1	☒	☐	☐	☐
Mode 1	1	☐	☐	☐	☐
Mode 2	1	☐	☐	☐	☐
Mode 3	1	☐	☐	☐	☐
Mode 4	1	☒	☐	☐	☐
Mode 5	1	☒	☐	☐	☐
Mode 6	1	☒	☐	☐	☐

Max Speed:

Maximum speed that machine can travel when operated with MPG.

Distance:

Length of one step.

Mode 0-6:

Seven modes are possible for Mk3 controllers and four modes are possible for Mk2 controller. Standard MPG pendant has switch for Mode 1, 2 and 3. X1 on MPG selects "Mode1", X10 "Mode2" and X100 "Mode3". For each mode you can set 'Factor' value and if "Step" mode is enabled.

It is best to experiment to get best mode settings for your machine.

Step:

Step mode or speed mode behavior can be selected. Because 'Step' distance depends on the 'Distance' and 'Factor' value and also on the version of controller, 'Step' will be described for Mk3 and Mk2 separately.

Mk2:**Step mode enabled:**

Factor value is used to calculate distance multiplier. Step distance = "Distance" * (1000 / "Factor")

For example:

If "Distance" is set to 0.01mm and factor is 1000, then step is 0.01mm long.

If "Distance" is set to 0.01mm and factor is 100, then step is 0.1mm long.

If "Distance" is set to 0.01mm and factor is 10, then step is 1mm long.

Step mode not enabled:

Factor value is set to calculate speed multiplier. If factor value is higher, then you need to turn MPG encoder faster to achieve same speed.

Mk3:

Step mode enabled: Factor value is distance multiplier.

For example:

If "Distance" is set to 0.01mm and factor is 1000, then step is 10mm long.

If "Distance" is set to 0.01mm and factor is 100, then step is 1mm long.

If "Distance" is set to 0.01mm and factor is 10, then step is 0.1mm long.

Step mode not enabled: Factor value is speed multiplier in percent

For example:

If "Max Speed" is set to 2000mm/min and factor is 100, then "Max Speed" used is 2000mm/min.

If "Max Speed" is set to 2000mm/min and factor is 200, then "Max Speed" used is 4000mm/min.

If "Max Speed" is set to 2000mm/min and factor is 10, then "Max Speed" used is 200mm/min.

Options:

This feature is supported only by Mk3 controller:

Step mode enabled:

If first "Option" is enabled, then steps are buffered.

Step mode not enabled:

No "Option" enabled: Normal behavior.

Only first "Option" enabled: Potentiometer from Jogging keyboard controls the speed.

Only second "Option" enabled: Machine speed depends on speed of encoder rotation.

Both "Options" enabled: Machine speed is synchronized with encoder rotation.

2.4.13 Spindle Synch

Motion is synchronized with spindle rotation.

A screenshot of a control panel titled "Spindle Synch". The panel has a blue header bar with the title. Below the header, there is a light gray rectangular area containing two labels and two checkboxes. The first label is "Wait For Index" and the second is "Change Direction". Both labels are aligned to the left, and their corresponding checkboxes are to their right. Both checkboxes are currently unchecked.

Spindle Synch	
Wait For Index	☐
Change Direction	☐

Wait For Index:

Motion always starts at same spindle position.

Change Direction:

Motion changes direction if motion direction of spindle is changed. (Mk3 controller only)

2.4.14 Tool Table

In 'Tool Table' you can create tool list for needs of Automatic Tool Change (ATC), and set position coordinates of populated tools in tool table.

Tool table

General

Tool Change

Number

0

Name

Shape

Description

Diameter

0,0000

Flutes

0

Offset

X

0,0000

Y

0,0000

Z

0,0000

A

0,0000

B

0,0000

C

0,0000

U

0,0000

V

0,0000

W

0,0000

Tool Sensor Offset

X

0,0000

Y

0,0000

Update

Add

Remove

General Tab:

In this tab you can create tools that will be included in tool list. Each created tool gets assigned its parameters. When tool is created you add it to tool list with 'Add' button.

Number:

Assigns number to tool.

Name:

Assigns name to tool.

Shape:

You can choose between eight shapes:

- Unknown
- End Mill
- Ball Nose
- Vbit
- Conical
- Ogee
- Drill
- Other

The screenshot shows the 'General' tab of a tool management interface. It includes the following fields and controls:

- Number:** A numeric input field with the value '0'.
- Name:** A text input field.
- Shape:** A dropdown menu.
- Description:** A text input field.
- Diameter:** A numeric input field with the value '0.0000'.
- Flutes:** A numeric input field with the value '0'.
- Offset:** A group of nine numeric input fields for X, Y, Z, A, B, C, U, V, and W, all with the value '0.0000'.
- Tool Sensor Offset:** A group of two numeric input fields for X and Y, both with the value '0.0000'.
- Buttons:** 'Update', 'Add', and 'Remove' buttons at the bottom.

Description:

You can add a description of tool.

Diameter: Diameter of tool

Flutes: Number of flutes.

Offset:

Tool offset.

Tool Sensor Offset:

Offset added to fixed tool sensor position when this tool is measured.

Tool Change tab:

Tools created in 'General Tab' get their position coordinates assigned in this tab. When tool position coordinates are entered, you assign them to selected tool with 'Update' button.

Tool Change

Tool Change

X	0,0000	Y	0,0000	Z	0,0000
A	0,0000	B	0,0000	C	0,0000
U	0,0000	V	0,0000	W	0,0000

Update

2.4.14.1 Tool Change

Tool Change settings are divided into three setting groups: 'Tool Change', 'Tool Offset' and 'Position'.

Tool Change

Tool Change

Enable

☐

Z Axis First

☐

Stop

☐

Pause

☐

Pause For Spindle

☐

☐

Leave Spindle On

☐

Skip Already Active Tool

☐

Reset Active Tool

☐

Use Default Tool

☒

Auto Return

☒

Auto Compensate

☒

Tool Offset

Not Used

☒

Measure Tool Length

☐

Tool Offset From Tool Table

☐

Position

☒ Not Set

☐ From Tool Table

☐ At Park 1

☐ At Park 2

☐ At G28

☐ At G30

☐ User Defined

Abs ☐

X

Y

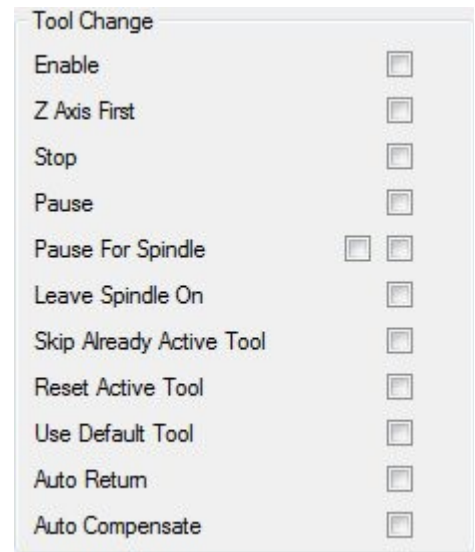
Z

Z Axis Only

☐

Tool Change settings group:

Here we set how we want our machine to “behave” during Tool change.



Tool Change	
Enable	☐
Z Axis First	☐
Stop	☐
Pause	☐
Pause For Spindle	☒ ☐
Leave Spindle On	☐
Skip Already Active Tool	☐
Reset Active Tool	☐
Use Default Tool	☐
Auto Return	☐
Auto Compensate	☐

Enable:

Enables tool change procedure.

Z Axis First:

Usually you want to move Z axis first, before X axis and Y axis.

Stop:

Machine will stop at tool change position and will not resume.

Pause:

Machine will pause at tool change position. During this pause you manually change tool.

Pause For Spindle:

Machine will pause after cutting (before tool change). During this pause you can turn OFF spindle manually. Second pause is before cutting (after tool change). During this pause You can turn spindle ON manually. If spindle is controlled through controller then this pause is not needed.

Skip Already Active Tool:

If tool with number N is already mounted then TN M6 g-code will be ignored and tool change will not occur.

Reset Active Tool:

Resets currently active tool at beginning of program. This will set active tool number to 0.

Use Default Tool:

If G-Code tool numbers are not in the tool table an error is reported. This option avoids this error. Default values from tool with number 0 are used instead. When tool 0 is not available, all 0 parameters are used. This option is useful, to load G-Code from another machine without error.

Auto Return:

This depends on your g-code. By g-code standard g-code return moves should be in g-code program but are often not.

Here is example:

This is correct g-code:

```
G01 X0 Y0(start position)
G01 X10 Y0(cut to X10 Y0with first tool)
T2 M6 (change tool tool with auto return off)
G01 X10 Y0(return to last position)
G01 X20 Y0
G01 X30 Y0
```

This is wrong but often used:

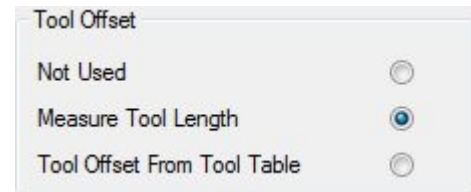
```
G01 X0 Y0(start position)
G01 X10 Y0 (cut to X10 Y0 with first tool)
T2 M6 (change tool with with auto return on)
(no need for G01 X10 Y0 line because of auto return)
G01 X20 Y0
G01 X30 Y0
```

Auto Compensate:

By g-code standard G43 offset will not change if tool is changed. If this option is enabled and G43 is active then it will automatically adjust to changed tool.

Tool Offset settings group:

We define if Tool Offset will be set or measured for each newly changed tool.



Not Used:

Tool offset will not be measured.

Measure Tool Length:

After tool change, tool length offset is measured using fixed tool length sensor.

Tool Offset From Tool Table:

After tool change, tool length offset value is “taken” from tool table. This is often used with ATC where tool length offset is known in advance.

Position settings group:

We define position coordinates of tool change.

Not set:

Position of tool change will not be set, machine will pause when tool change occurs. Machine waits at position where current tool-path ends.

From Tool Table:

Position defined in Tool Table under Tool Change. Each tool in tool table has its own position. This is often used with ATC where each tool has its own position in tool magazine.

At Park 1:

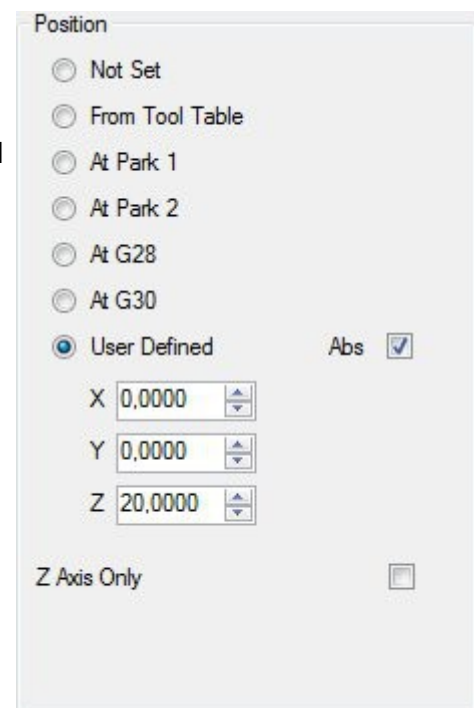
Position defined at 'Park 1' coordinates.

At Park 2:

Position defined at 'Park 2' coordinates.

At G28:

Position defined at 'G28' coordinates.



At G30:

Position defined at 'G30' coordinates.

User defined:

User defines position coordinates. With 'Abs' option we select if this coordinates are absolute or relative.

Z Axis Only:

If any of previous tool change positions is set and 'Z Axis Only' is enabled, machine stops at position where certain tools tool-path ends and ascends to Z coordinate value of tool change position previously selected. (Z coordinate value of Park1, Park 2, G28....)

2.4.14.2 Tool Change ATC

The screenshot displays the 'Tool Change ATC' control panel. It is divided into two main sections: 'Lock' and 'Move'.

Lock Section:

- Safe Height:** A numeric input field set to 0,0000.
- Speed:** A numeric input field set to 0,00.
- Output pin:** A dropdown menu.
- Delay:** A numeric input field set to 1,0.
- Hold:** A checkbox.
- Blow:** A dropdown menu.
- Output pin:** A dropdown menu.
- Delay:** A numeric input field set to 1,0.
- Hold:** A checkbox.

Move Section:

- Move Unloaded:** A sub-section with three rows (X, Y, Z) and two columns of numeric input fields, all set to 0,0000.
- Move Loaded:** A sub-section with three rows (X, Y, Z) and two columns of numeric input fields, all set to 0,0000.

Safe Height:

Value of height to/from which machine ascends/descends when it is in tool change procedure.

Speed:

Speed at which machine descends/ascends to lock or unlock tool.

The screenshot shows a configuration window with the following settings:

Parameter	Value
Safe Height	43,0000
Speed	200,00
Lock Output pin	2
Lock Delay	2,0
Lock Hold	☐
Blow Output pin	3
Blow Delay	0,0
Blow Hold	☐

Lock:

Lock output pin will activate/deactivate pneumatic tool-holder.

Output pin:

Define Output pin number for Lock.

Delay:

Delay value is amount of seconds that machine waits after Lock output is activated.

Hold:

Leaves Lock output activated when machine changes between two tools.

Blow:

Blow output pin will activate/deactivate blowing of air.

Output pin:

Define Output pin number for Blow.

Delay:

Delay value is amount of seconds that machine waits after Blow output is activated.

Hold:

Leaves Blow output activated when machine changes between two tools.

Move Unloaded:

We can set sequence of moves when machine does not have any tool loaded in the tool-holder. First coordinate column represents first move and second column represents second move. These are offset coordinates.

Move Unloaded		
X	0,0000	30,0000
Y	0,0000	0,0000
Z	20,0000	0,0000

This coordinate setup will do the following:

When machine unlocks tool, it ascends for 20mm in Z+ and then moves 30mm in X+ direction. Then continues at default traverse rate.

Move Loaded:

We can set sequence of moves when machine does have tool loaded in the tool-holder. First coordinate column represents first move and second column second move. This are offset coordinates.

Move Loaded		
X	0,0000	0,0000
Y	0,0000	30,0000
Z	10,0000	0,0000

This coordinate setup will do the following:

After machine locks tool, it ascends for 10mm in Z+ and then moves for 30mm in Y+ direction at Speed value. Then continues at default traverse rate.

2.4.15 Tool Sensor

Tool Sensor

Enable☐

Speed

Tool Sensor Fixed

Location

Move

X

X

Y

Y

Safe Height

Direction☐ - ☐ +

Set Position Z

Return Distance

Return☐

Tool Sensor Movable

Size

Retract

Height

Return Distance

Mk1 controller users connect the tool sensor to LIMIT pin 5 (Z-).

Mk2 and Mk2/4 controller users can use a dedicated sensor connection on INPUT pin 5.

Mk3 controller has assignable input pins.

Enable:

Enables tool sensor.

Speed:

This is the speed value at which machine descends/ascends when it is in tool offset measurement procedure and is directly above the tool sensor.

Enable☐

Speed

2.4.15.1 Tool Sensor Fixed

With fixed tool sensor we measure tool's length at fixed position on the tool table. When tool activates tool sensor tool offset is calculated and set.

The screenshot shows a software interface titled "Tool Sensor Fixed". It contains several input fields and controls:

- Location**: X (0,0000) and Y (0,0000) with up/down arrows.
- Move**: X (0,0000) and Y (0,0000) with up/down arrows.
- Safe Height**: (0,0000) with up/down arrows.
- Direction**: Two radio buttons, one selected (labeled '-') and one labeled '+', with a '-' sign between them.
- Set Position Z**: (0,0000) with up/down arrows.
- Return Distance**: (0,0000) with up/down arrows.
- Return**: A checkbox that is currently unchecked.

Location X,Y:

Absolute location coordinates of the tool sensor. Automatic 'Measure Tool Offset' procedure moves the machine to this position.

Move X,Y:

A lead in/out offset move. 'Move' has two values. XY values are offset coordinates that "tell" the machine from which initial point machine moves further to Fixed Tool Sensor location. This is usually used if you want to move to sensor location from desired direction to avoid any obstacles.

Set Position Z:

Actual Z position of the tool sensor.

Direction:

The direction (up / down) the Z axis will be moved when the machine reaches 'Location XY'.

Safe Height:

Safe Z value used for traverse machine move to 'Location XY'.

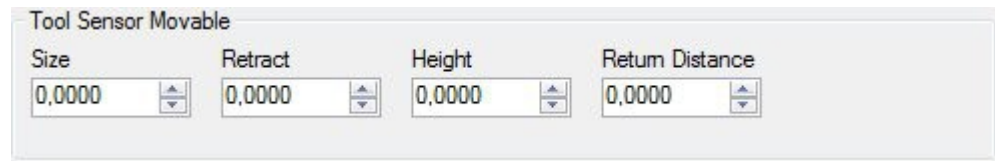
Return Distance:

Distance for which machine ascends (returns) when tool sensor is activated (pressed). Machine further ascends to Safe Height value.

Return:

If enabled, the machine will auto-return to the initial location.

2.4.15.2 Tool Sensor Movable



The screenshot shows a dialog box titled "Tool Sensor Movable". It contains four input fields, each with a numeric value of 0.0000 and up/down arrow controls:

Size	Retract	Height	Return Distance
0.0000	0.0000	0.0000	0.0000

Size:

Total distance from tool center to workpiece when sensor is triggered. (When measuring 'XY Edges'.)

Example:

If the tool sensor is a simple piece of PCB of thickness 1.6mm and tool radius is 4mm (8mm diameter), 'Size' value = $1.6 + 4 = 5.6\text{mm}$

NOTE: When measuring 'Current working Z offset', this parameter does not affect the measurement.

Retract:

Value represents distance for which machine retracts (from position to which previously returned to).

Height:

If the tool sensor is a simple piece of PCB of thickness 1.6mm, Height = 1.6mm

Return Distance:

Value represents distance for which tool returns when sensor/touching probe is triggered.

2.4.16 Materials

To visualize your raw work piece in CNC USB software 3D display, you can create the model of your work piece in Materials.

Materials

Materials

Number

0

Name

Kind

Description

X Size

0,0000

Y Size

0,0000

Z Size

0,0000

X Position

0,0000

Y Position

0,0000

Z Position

0,0000

Update

Add

Remove

Number:

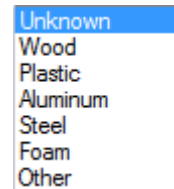
Assign number of material.

Name:

Assign name of material.

Kind:

You can choose type of material from drop-down menu.

**Description:**

You can add description of your material.

X,Y,Z Size:

Insert dimensions of your workpiece.

X,Y,Z Position:

Insert position coordinates of your work piece. These are absolute coordinates.

Buttons:**Update:**

Updates any changes made to selected material from list.

Add:

Adds new material to list.

Remove:

Removes selected material from list.

2.4.17 Parameters

You can assign position coordinates to selected parameter from parameters list.

Parameters

Offset
Park 1
Park 2
Stored Position 1
Stored Position 2
Stored Position 3
Stored Position 4
Stored Position 5
Stored Position 6
Stored Position 7
Stored Position 8
Stored Position 9
Coordinate System P1
Coordinate System P2
Coordinate System P3
Coordinate System P4
Coordinate System P5
Coordinate System P6
Coordinate System P7
Coordinate System P8
Coordinate System P9
Position G28
Position G30
Offset G92

Offset

X

0,0000

Y

0,0000

Z

0,0000

A

0,0000

B

0,0000

C

0,0000

U

0,0000

V

0,0000

W

0,0000

Update

Reset

2.4.18 Accessories

Mk2 and Mk3 support I2C port.

Current Position:

With **I2C** or **UART** enabled you can receive data of current position.



The screenshot shows a software interface titled "Accessories". It contains two main sections:

- I2C Test:** This section includes a numeric input field set to "60", a "Data" label followed by a text field containing the hexadecimal string "50;18;0;0;9;80;108;97;110;101;116;73;50;67;0;0;0;0", and a "Send" button.
- Current Position:** This section features several controls:
 - A checkbox labeled "I2C" which is currently checked.
 - A numeric input field set to "54".
 - A checkbox labeled "UART" which is currently unchecked.
 - A numeric input field set to "460800".
 - A label "Axes" followed by a numeric input field set to "4".
 - A checkbox labeled "RPM" which is currently unchecked.

3 G-Code

G-code interpreter used by CNC USB controller is implemented according to National Institute of Standards and Technology RS274NGC Interpreter - Version 3 specification.

Full documentation of RS274NGC is available at web page:

http://www.isd.mel.nist.gov/personnel/kramer/pubs/RS274NGC_3.web/RS274NGC_3TOC.html

3.1 Overview

The RS274/NGC language is based on lines of code. Each line (also called a "block") may include commands to a machining center to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more "words." A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, "G1 X3" is a valid line of code with two words. "G1" is a command meaning "move in a straight line at the programmed feed rate," and "X3" provides an argument value (the value of X should be 3 at the end of the move). Most RS274/NGC commands start with either G or M (for miscellaneous). The words for these commands are called "G codes" and "M codes."

The RS274/NGC language has no indicator for the start of a program. The Interpreter, however, deals with files. A single program may be in a single file, or a program may be spread across several files. A file may demarcated with percents in the following way. The first non-blank line of a file may contain nothing but a percent sign, "%", possibly surrounded by white space, and later in the file (normally at the end of the file) there may be a similar line. Demarcating a file with percents is optional if the file has an M2 or M30 in it, but is required if not. An error will be signaled if a file has a percent line at the beginning but not at the end. The useful contents of a file demarcated by percents stop after the second percent line. Anything after that is ignored.

The RS274/NGC language has two commands (M2 or M30), either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed.

3.2 Format of a Line

A permissible line of input RS274/NGC code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

1. an optional block delete character, which is a slash "/" .
2. an optional line number.
3. any number of words, parameter settings, and comments.
4. an end of line marker (carriage return or line feed or both).

Any input not explicitly allowed is illegal and will cause the Interpreter to signal an error.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. The line "g0x +0. 12 34y 7" is equivalent to "g0 x+0.1234 y7", for example.

Blank lines are allowed in the input. They are to be ignored.

Input is case insensitive, except in comments. Any letter outside a comment may be in upper or lower case without changing the meaning of a line.

3.3 Line Number

A line number is the letter N followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not OK, for example). Line numbers may be repeated or used out of order, although normal practice is to avoid such usage. Line numbers may also be skipped, and that is normal practice. A line number is not required to be used, but must be in the proper place if used.

3.4 Word

A word is a letter other than N followed by a real value.

Words may begin with any of the letters shown below. Several letters (I, J, K, L, P, R) may have different meanings in different contexts. Permitted letters are:

A, B, C, D, F, G, H, I, J, K, L, M, N, P, Q, R, S, T, X, Y, Z

3.5 Number

A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression, or a unary operation value. Definitions of these follow immediately. Processing characters to come up with a number is called "evaluating". An explicit number evaluates to itself.

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

A number consists of an optional plus or minus sign, followed by zero to many digits, followed, possibly, by one decimal point, followed by zero to many digits - provided that there is at least one digit somewhere in the number.

There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it, a decimal does.

Numbers may have any number of digits, subject to the limitation on line length. Only about seventeen significant figures will be retained, however (enough for all known applications).

A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros will have the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes in RS274/NGC are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M codes, and G codes multiplied by ten. A decimal number which is supposed to be close to an integer is considered close enough if it is within 0.0001 of an integer.

3.6 Parameters

A parameter value is the pound character # followed by a real value. The real value must evaluate to an integer between 1 and 5399. The integer is a parameter number, and the value of the parameter value is whatever number is stored in the numbered parameter. Parameters can be used as variables.

The # character takes precedence over other operations, so that, for example, "#1+2" means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, #[1+2] does mean the value found in parameter 3. The # character may be repeated; for example ##2 means the value of the parameter whose index is the (integer) value of parameter 2.

A parameter setting is the following four items one after the other: (1) a pound character #, (2) a real value which evaluates to an integer between 1 and 5399, (3) an equal sign =, and (4) a real value. For example "#3 = 15" is a parameter setting meaning "set parameter 3 to 15."

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line "#3=6 G1 x#3" is interpreted, a straight move to a point where x equals 15 will occur and the value of parameter 3 will be 6.

3.7 Expressions and Binary Operations

An expression is a set of characters starting with a left bracket [and ending with a balancing right bracket]. In between the brackets are numbers, parameter values, mathematical operations, and other expressions. An expression may be evaluated to produce a number. The expressions on a line are evaluated when the line is read, before anything on the line is executed. An example of an expression is [1 + cos[0] - [#3 ** [4.0/2]]].

Binary operations appear only inside expressions. Nine binary operations are defined. There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*), and division (/). There are three logical operations: non-exclusive or (OR), exclusive or (XOR), and logical and (AND). The eighth

operation is the modulus operation (MOD). The ninth operation is the "power" operation (**) of raising the number on the left of the operation to the power on the right.

The binary operations are divided into three groups. The first group is: power. The second group is: multiplication, division, and modulus. The third group is: addition, subtraction, logical non-exclusive or, logical exclusive or, and logical and. If operations are strung together (for example in the expression $[2.0 / 3 * 1.5 - 5.5 / 11.0]$), operations in the first group are to be performed before operations in the second group and operations in the second group before operations in the third group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first. Thus, the example is equivalent to: $[(2.0 / 3) * 1.5] - (5.5 / 11.0)$, which simplifies to $[1.0 - 0.5]$, which is 0.5.

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

3.8 Unary Operation Value

A unary operation value is either "ATAN" followed by one expression divided by another expression (for example "ATAN[2]/[1+3]") or any other unary operation name followed by an expression (for example "SIN[90]"). The unary operations are: ABS (absolute value), ACOS (arc cosine), ASIN (arc sine), ATAN (arc tangent), COS (cosine), EXP (e raised to the given power), FIX (round down), FUP (round up), LN (natural logarithm), ROUND (round to the nearest whole number), SIN (sine), SQRT (square root), and TAN (tangent). Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in degrees. Values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that $\text{FIX}[2.8] = 2$ and $\text{FIX}[-2.8] = -3$, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; $\text{FUP}[2.8] = 3$ and $\text{FUP}[-2.8] = -2$, for example.

3.9 Comments and Messages

Printable characters and white space inside parentheses is a comment. A left parenthesis always starts a comment. The comment ends at the first right parenthesis found thereafter. Once a left parenthesis is placed on a line, a matching right parenthesis must appear before the end of the line. Comments may not be nested; it is an error if a left parenthesis is found after the start of a comment and before the end of the comment. Here is an example of a line containing a comment: "G80 M5 (stop motion)". Comments do not cause a machining center to do anything.

A comment contains a message if "MSG," appears after the left parenthesis and before any other printing characters. Variants of "MSG," which include white space and lower case characters are allowed. The rest of the characters before the right parenthesis are considered to be a message. Messages should be displayed on the message display device. Comments not containing messages need not be displayed there.

3.10 Item Repeats

A line may have any number of G words, but two G words from the same modal group may not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, "#3=15 #3=6", for example, only the last setting will take effect. It is silly, but not illegal, to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one will be used; each of the other comments will be read and its format will be checked, but it will be ignored thereafter. It is expected that putting more than one comment on a line will be very rare.

3.11 Item Order

The three types of item whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of item are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there will be no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter will take effect. For example, after the line "#3=15 #3=6" has been interpreted, the value of parameter 3 will be 6. If the order is reversed to "#3=6 #3=15" and the line is interpreted, the value of parameter 3 will be 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment will be used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line. For example, the line "g40 g1 #3=15 (foo) #4=-7.0" has five items and means exactly the same thing in any of the 120 possible orders (such as "#4=-7.0 g1 #3=15 g40 (foo)") for the five items.

3.12 Modal and Non-modal modes

In RS274/NGC, many commands cause a machining center to change from one mode to another, and the mode stays active until some other command changes it implicitly or explicitly. Such commands are called "modal". For example, if coolant is turned on, it stays on until it is explicitly turned off. The G codes for motion are also modal. If a G1 (straight move) command is given on one line, for example, it will be executed again on the next line if one or more axis words is available on the line, unless an explicit command is given on that next line using the axis words or canceling motion.

"Non-modal" codes have effect only on the lines on which they occur. For example, G4 (dwell) is non-modal.

3.13 Modal Groups

Modal commands are arranged in sets called "modal groups", and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time - like measure in inches vs. measure in millimeters. A machining center may be in many modes at the same time, with one mode from each modal group being in effect.

The modal groups for G codes are:

group 1 = {G0, G1, G2, G3, G38.2, G80, G81, G82, G83, G84, G85, G86, G87, G88, G89} motion

group 2 = {G17, G18, G19} plane selection

group 3 = {G90, G91} distance mode

group 5 = {G93, G94} feed rate mode

group 6 = {G20, G21} units

group 7 = {G40, G41, G42} cutter radius compensation

group 8 = {G43, G49} tool length offset

group 10 = {G98, G99} return mode in canned cycles

group 12 = {G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3} coordinate system selection

group 13 = {G61, G61.1, G64} path control mode

The modal groups for M codes are:

group 4 = {M0, M1, M2, M30, M60} stopping

group 6 = {M6} tool change

group 7 = {M3, M4, M5} spindle turning

group 8 = {M7, M8, M9} coolant (special case: M7 and M8 may be active at the same time)

group 9 = {M48, M49} enable/disable feed and speed override switches

In addition to the above modal groups, there is a group for non-modal G codes:

group 0 = {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

For several modal groups, when a machining center is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining center is turned on or otherwise re-initialized, the default values are automatically in effect.

Group 1, the first group on the table, is a group of G codes for motion. One of these is always in effect. That one is called the current motion mode.

It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line), and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30, and G92.

3.14 G - Codes

G-codes of the RS274/NGC language are shown and described below.

3.14.1 G00 - Rapid Move

For rapid linear motion, program G0 X- Y- Z- A- B- C-, where all the axis words are optional, except that at least one must be used. The G0 is optional if the current motion mode is G0. This will produce coordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G0 command is executing.

It is an error if:

- all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above.

3.14.2 G01 - Linear Move

For linear motion at feed rate (for cutting or not), program G1 X- Y- Z- A- B- C-, where all the axis words are optional, except that at least one must be used. The G1 is optional if the current motion mode is G1. This will produce coordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).

It is an error if:

- all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above.

3.14.3 G02 - Arc/Helix Move (CW)

3.14.4 G03 - Arc/Helix Move (CCW)

A circular or helical arc is specified using either G2 (clockwise arc) or G3 (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane), or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G2 or G3 is optional if it is the current motion mode.

- **Radius Format Arc**

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program G2 X- Y- Z- A- B- C- R- (or use G3 instead of G2). R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees. If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified.

It is an error if:

- both of the axis words for the axes of the selected plane are omitted,
- the end point of the arc is the same as the current point.

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point will produce a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

Here is an example of a radius format command to mill an arc: G17 G2 x 10 y 15 r 20 z 5.

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=15, and Z=5, with a radius of 20. If the starting value of Z is 5, this is an arc of a circle parallel to the XY-plane; otherwise it is a helical arc.

- **Center Format Arc**

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

- when the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimeter (if millimeters are being used).

When the XY-plane is selected, program G2 X- Y- Z- A- B- C- I- J- (or use G3 instead of G2). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location (in the X and Y directions, respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

- X and Y are both omitted,
- I and J are both omitted.

When the XZ-plane is selected, program G2 X- Y- Z- A- B- C- I- K- (or use G3 instead of G2). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location (in the X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used. It is an error if:

- X and Z are both omitted,
- I and K are both omitted.

When the YZ-plane is selected, program G2 X- Y- Z- A- B- C- J- K- (or use G3 instead of G2). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location (in the Y and Z directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used.

It is an error if:

- Y and Z are both omitted,
- J and K are both omitted.

Here is an example of a center format command to mill an arc: G17 G2 X10 Y16 I3 J4 Z9.

That means to make a clockwise (as viewed from the positive z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=16, and Z=9, with its center offset in the X direction by 3 units from the current X location and offset in the Y direction by 4 units from the current Y location. If the current location has X=7, Y=7 at the outset, the center will be at X=10, Y=11. If the starting value of Z is 9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 5.

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

3.14.5 G04 - Dwell

For a dwell, program G4 P- . This will keep the axes unmoving for the period of time in seconds specified by the P number. It is an error if:

- the P number is negative.

3.14.6 G7 - Lathe Diameter Mode

Program G7 to enter the diameter mode for axis X on a lathe. When in the diameter mode the X axis moves on a lathe will be 1/2 the distance to the center of the lathe. For example X1 would move the cutter to 0.500" from the center of the lathe thus giving a 1" diameter part.

3.14.7 G8 - Lathe Radius Mode

Program G8 to enter the radius mode for axis X on a lathe. When in Radius mode the X axis moves on a lathe will be the distance from the center. Thus a cut at X1 would result in a part that is 2" in diameter. G8 is default at power up.

3.14.8 G10 L1 - Set Tool Table

G10 L1 P[tool number] R[radius] X[offset] Z[offset] Q[orientation]

Program a G10 L1 to set a tool table entry from a program. G10 L1 reloads the tool table.

It is an error if:

- cutter compensation is on

3.14.9 G10 L2 - Set Coordinate System

You can set the offsets of the nine program coordinate systems using G10 L2 Pn (n is the number of the coordinate system) with values for the axes in terms of the absolute coordinate system.

To set the coordinate values for the origin of a coordinate system, program

G10 L2 P - X- Y- Z- A- B- C-, where the P number must evaluate to an integer in the range 1 to 9 (corresponding to G54 to G59.3) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

It is an error if:

- the P number does not evaluate to an integer in the range 1 to 9.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

Example: G10 L2 P1 x 3.5 y 17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

3.14.10 G10 L6 - Set Offset

Machine will set offset.

Examples:

G10 L6 X- Y- Z- A- B- C-

3.14.11 G10 L7 - Measure and Store Offset

Machine will measure and store offset.

Use R to specify height of sensor.

Examples:

G10 L7 R-

3.14.12 G10 L8 - Home Machine Axes

Machine will home axes.

Use P1 is for X-, P2 for X+, P4 for Y-, P8 for Y+, P16 for Z-, P32 for Z+
P64 for A-, P128 for A+, P256 for B-, P512 for B+, P1024 for C-, P2048 for C+
P4096 for U-, P8192 for U+, P16384 for V-, P32768 for V+, P65536 for W-, P131072 for W+

Examples:

G10 L8 Pn X- Y- Z- A- B- C-

3.14.13 G10 L9 - Set Machine Position Without Move

Machine will set position to coordinates defined with axis words

Examples:

G10 L9 X- Y- Z- A- B- C-

3.14.14 G17 - XY-Plane Selection

3.14.15 G18 - XZ-Plane Selection

3.14.16 G19 - YZ-Plane Selection

Program G17 to select the XY-plane.

Program G18 to select the XZ-plane.

Program G19 to select the YZ-plane.

3.14.17 G20 - Inch Units Selection

3.14.18 G21 - Millimeter Units Selection

Program G20 to use inches for length units.

Program G21 to use millimeters for length units.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units.

3.14.19 G28 – Go To Predefined Position

3.14.20 G30 – Go To Predefined Position

Two predefined positions are defined (by parameters 5161-5166 for G28 and parameters 5181-5186 for G30). The parameter values are in terms of the absolute coordinate system, but are in unspecified length units.

To return to predefined position by way of the programmed position, program G28 X- Y- Z- A- B- C- (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

3.14.21 G33 - Spindle Synchronized Motion

G33 X- Y- Z- K-

K - distance per revolution

For spindle-synchronized motion in one direction, code G33 X- Y- Z- K- where K gives the distance moved in XYZ for each revolution of the spindle. For instance, if starting at Z=0, G33 Z-1 K.0625 produces a 1 inch motion in Z over 16 revolutions of the spindle. This command might be part of a program to produce a 16TPI thread. Another example in metric, G33 Z-15 K1.5 produces a movement of 15mm while the spindle rotates 10 times for a thread of 1.5mm.

Spindle-synchronized motion waits for the spindle index and spindle at speed pins, so multiple passes line up. G33 moves end at the programmed endpoint. G33 could be used to cut tapered threads or a fusee.

All the axis words are optional, except that at least one must be used.

Note: K follows the drive line described by X- Y- Z-. K is not parallel to the Z axis if X or Y endpoints are used for example when cutting tapered threads.

3.14.22 G40 - Compensation Off

To turn cutter radius compensation off, program G40. It is OK to turn compensation off when it is already off.

3.14.23 G41 - Cutter Compensation Left

3.14.24 G42 - Cutter Compensation Right

Cutter radius compensation may be performed only if the XY-plane is active.

To turn cutter radius compensation on left (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program G41 D- . To turn cutter radius compensation on right (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program G42 D- . The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.

It is an error if:

- the D number is not an integer, is negative or is larger than the number of carousel slots,
- the XY-plane is not active,
- cutter radius compensation is commanded to turn on when it is already on.

3.14.25 G41.1 - Dynamic Cutter Compensation Left

3.14.26 G42.1 - Dynamic Cutter Compensation Right

G41.1 & G42.1 function the same as G41 & G42 with the added scope of being able to program the tool diameter. D word is tool diameter.

3.14.27 G43 - Tool Length Offset

To use a tool length offset, program G43 H-, where the H number is the desired index in the tool table. It is expected that all entries in this table will be positive. The H number should be, but does not have to be, the same as the slot number of the tool currently in the spindle. It is OK for the H number to be zero; an offset value of zero will be used.

It is an error if:

- the H number is not an integer, is negative, or is larger than the number of carousel slots.

It is OK to program using the same offset already in use. It is also OK to program using no tool length offset if none is currently being used.

3.14.28 G43.1 - Dynamic Tool Length Offset

3.14.29 G49 - Cancel Tool Length Offset

To use no tool length offset, program G49.

3.14.30 G52 - Coordinate System Offset

To offset the current point by a given positive or negative distance (without motion), program G52 X- Y- Z- A- B- C- U- V- W- , where the axis words contain the offsets you want to provide.

All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

It is an error if:

- all axis words are omitted.

When G52 is executed, the origin of the currently active coordinate system moves by the values given.

The effect of G52 is canceled by programming G52 X0 Y0 etc.

Example:

Suppose the current point is at X=4 in the currently specified coordinate system, then G52 X7 sets the X-axis offset to 7, and so causes the X-coordinate of the current point to be -3.

3.14.31 G53 - Move in Machine Coordinates

You can make straight moves in the absolute machine coordinate system by using G53 with either G0 or G1.

For linear motion to a point expressed in absolute coordinates, program G1 G53 X- Y- Z- A- B- C- (or use G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point. If G1 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- G53 is used without G0 or G1 being active,
- G53 is used while cutter radius compensation is on.

- 3.14.32 G54 - Use Preset Work Coordinate System 1**
- 3.14.33 G55 - Use Preset Work Coordinate System 2**
- 3.14.34 G56 - Use Preset Work Coordinate System 3**
- 3.14.35 G57 - Use Preset Work Coordinate System 4**
- 3.14.36 G58 - Use Preset Work Coordinate System 5**
- 3.14.37 G59 - Use Preset Work Coordinate System 6**
- 3.14.38 G59.1 - Use Preset Work Coordinate System 7**
- 3.14.39 G59.2 - Use Preset Work Coordinate System 8**
- 3.14.40 G59.3 - Use Preset Work Coordinate System 9**

You can select one of the nine systems by using G54, G55, G56, G57, G58, G59, G59.1, G59.2, or G59.3. It is not possible to select the absolute coordinate system directly.

To select coordinate system 1, program G54, and similarly for other coordinate systems.

The system-number-G-code pairs are:

(1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59), (7-G59.1), (8-G59.2), and (9-G59.3).

It is an error if:

- one of these G-codes is used while cutter radius compensation is on.

3.14.41 G61 - Set Path Control Mode: Exact Path

3.14.42 G61.1 - Set Path Control Mode: Exact Stop

3.14.43 G64 - Set Path Control Mode: Path Blending

Program G61 to put the machining center into exact path mode, G61.1 for exact stop mode, or G64 for continuous blend mode. It is OK to program for the mode that is already active.

3.14.44 G73 - Drilling Cycle with Chip Breaking

The G73 cycle is intended for deep drilling or milling with chip breaking. The retracts in this cycle cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z axis.

Program G73 X- Y- Z- A- B- C- R- L- Q-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
3. Rapid up a bit.
4. Repeat steps 2 and 3 until the Z position is reached at step 2.
5. Retract the Z-axis at traverse rate to clear Z.

It is an error if:

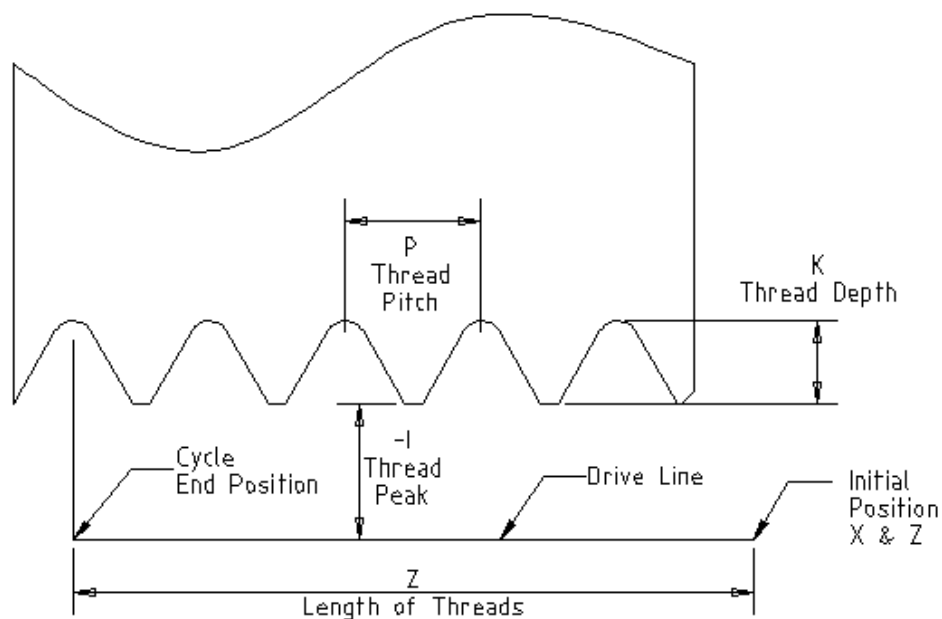
- the Q number is negative or zero.

3.14.45 G76 - Threading Cycle

Program G76 P- Z- I- J- R- K- Q- H- E- L-

It is an error if:

- The active plane is not the ZX plane
- Other axis words, such as X- or Y-, are specified
- The R- depression value is less than 1.0.
- All the required words are not specified
- P-, J-, K- or H- is negative
- E- is greater than half the drive line length



Drive Line - A line through the initial X position parallel to the Z.

P- The "thread pitch" in distance per revolution.

Z- The final position of threads. At the end of the cycle the tool will be at this Z position.

I- The "thread peak" offset from the "drive line". Negative I values are external threads, and positive I values are internal threads. Generally the material has been turned to this size before the G76 cycle.

J- A positive value specifying the "initial cut depth". The first threading cut will be J beyond the "thread peak" position.

K- A positive value specifying the "full thread depth". The final threading cut will be K beyond the "thread peak" position.

Optional settings

R- The "depth degression". R1.0 selects constant depth on successive threading passes. R2.0 selects constant area. Values between 1.0 and 2.0 select decreasing depth but increasing area. Values above 2.0 select decreasing area. Beware that unnecessarily high degression values will cause a large number of passes to be used. (degression = a descent by stages or steps.)

Q- The "compound slide angle" is the angle (in degrees) describing to what extent successive passes should be offset along the drive line. This is used to cause one side of the tool to remove more material

than the other. A positive Q value causes the leading edge of the tool to cut more heavily. Typical values are 29, 29.5 or 30.

H- The number of "spring passes". Spring passes are additional passes at full thread depth. If no additional passes are desired, program H0.

Tapered entry and exit moves can be programmed using E- and L-.

E-

Specifies the distance along the drive line used for the taper. The angle of the taper will be so the last pass tapers to the thread crest over the distance specified with E. E0.2 will give a taper for the first/last 0.2 length units along the thread. For a 45 degree taper program E the same as K.

L- Specifies which ends of the thread get the taper. Program L0 for no taper (the default), L1 for entry taper, L2 for exit taper, or L3 for both entry and exit tapers. Entry tapers will pause at the drive line to synchronize with the index pulse then feed in to the beginning of the taper. No entry taper and the tool will rapid to the cut depth then synchronize and begin the cut.

The tool is moved to the initial X and Z positions prior to issuing the G76. The X position is the "drive line" and the Z position is the start of the threads.

The tool will pause briefly for synchronization before each threading pass, so a relief groove will be required at the entry unless the beginning of the thread is past the end of the material or an entry taper is used.

Unless using an exit taper, the exit move (traverse to original X) is not synchronized to the spindle speed. With a slow spindle, the exit move might take only a small fraction of a revolution. If the spindle speed is increased after several passes are complete, subsequent exit moves will require a larger portion of a revolution, resulting in a very heavy cut during the exit move. This can be avoided by providing a relief groove at the exit, or by not changing the spindle speed while threading.

The final position of the tool will be at the end of the "drive line". A safe Z move will be needed with an internal thread to remove the tool from the hole.

Example:

G0 Z-.5 X .2

G76 P0.05 Z-1 I-.075 J0.008 K0.045 Q29.5 L2 E0.045

3.14.46 G80 - Cancel Canned Cycle

Program G80 to ensure no axis motion will occur. It is an error if:

- axis words are programmed when G80 is active, unless a modal group 0 G code is programmed which uses axis words.

3.14.47 Canned Cycles (G81 - G89)

The canned cycles G81 through G89 have been implemented as described in this section. Two examples are given with the description of G81 below.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R, and Z numbers in the NC code. These numbers are used to determine X, Y, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times," Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98, OLD_Z), or otherwise to the R position.

It is an error if:

- X, Y, and Z words are all missing during a canned cycle,
- a P number is required and a negative P number is used,
- an L number is used that does not evaluate to a positive integer,
- rotational axis motion is used during a canned cycle,

- inverse time feed rate is active during a canned cycle,
- cutter radius compensation is active during a canned cycle.

When the XY plane is active, the Z number is sticky, and it is an error if:

- the Z number is missing and the same canned cycle was not already active,
- the R number is less than the Z number.

When the XZ plane is active, the Y number is sticky, and it is an error if:

- the Y number is missing and the same canned cycle was not already active,
- the R number is less than the Y number.

When the YZ plane is active, the X number is sticky, and it is an error if:

- the X number is missing and the same canned cycle was not already active,
- the R number is less than the X number.

Preliminary and In-Between Motion

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

1. a straight traverse parallel to the XY-plane to the given XY position,
2. a straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

3.14.48 G81 - Canned Cycle: Drilling

The G81 cycle is intended for drilling. Program G81 X- Y- Z- A- B- C- R- L-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Retract the Z-axis at traverse rate to clear Z.

Example 1. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G90 G81 G98 X4 Y5 Z1.5 R2.8

This calls for absolute distance mode (G90) and OLD_Z retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. Old Z is 3. The following moves take place.

1. a traverse parallel to the XY-plane to (4,5,3)
2. a traverse parallel to the Z-axis to (4,5,2.8)
3. a feed parallel to the Z-axis to (4,5,1.5)

4. a traverse parallel to the Z-axis to (4,5,3)

Example 2. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3

This calls for incremental distance mode (G91) and OLD_Z retract mode (G98) and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 ($=1+4$), the initial Y position is 7 ($=2+5$), the clear Z position is 4.8 ($=1.8+3$), and the Z position is 4.2 ($=4.8-0.6$). Old Z is 3.

The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.

The first repeat consists of 3 moves.

1. a traverse parallel to the XY-plane to (5,7,4.8)
2. a feed parallel to the Z-axis to (5,7, 4.2)
3. a traverse parallel to the Z-axis to (5,7,4.8)

The second repeat consists of 3 moves. The X position is reset to 9 ($=5+4$) and the Y position to 12 ($=7+5$).

1. a traverse parallel to the XY-plane to (9,12,4.8)
2. a feed parallel to the Z-axis to (9,12, 4.2)
3. a traverse parallel to the Z-axis to (9,12,4.8)

The third repeat consists of 3 moves. The X position is reset to 13 ($=9+4$) and the Y position to 17 ($=12+5$).

1. a traverse parallel to the XY-plane to (13,17,4.8)
2. a feed parallel to the Z-axis to (13,17, 4.2)
3. a traverse parallel to the Z-axis to (13,17,4.8)

3.14.49 G82 - Canned Cycle: Drilling, Dwell

The G82 cycle is intended for drilling. Program G82 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Retract the Z-axis at traverse rate to clear Z.

3.14.50 G83 - Canned Cycle: Peck Drilling

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program G83 X- Y- Z- A- B- C- R- L- Q-

1. Preliminary motion, as described above.

2. Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
3. Rapid back out to the clear_z.
4. Rapid back down to the current hole bottom, backed off a bit.
5. Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
6. Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

3.14.51 G84 - Canned Cycle: Right Hand Tapping

The G84 cycle is intended for right-hand tapping with a tap tool.

Program G84 X- Y- Z- A- B- C- R- L-

1. Preliminary motion, as described above.
2. Start speed-feed synchronization.
3. Move the Z-axis only at the current feed rate to the Z position.
4. Stop the spindle.
5. Start the spindle counterclockwise.
6. Retract the Z-axis at the current feed rate to clear Z.
7. If speed-feed synch was not on before the cycle started, stop it.
8. Stop the spindle.
9. Start the spindle clockwise.

The spindle must be turning clockwise before this cycle is used.

It is an error if:

- the spindle is not turning clockwise before this cycle is executed.

With this cycle, the programmer must be sure to program the speed and feed in the correct proportion to match the pitch of threads being made. The relationship is that the spindle speed equals the feed rate times the pitch (in threads per length unit). For example, if the pitch is 2 threads per millimeter, the active length units are millimeters, and the feed rate has been set with the command F150, then the speed should be set with the command S300, since $150 \times 2 = 300$.

If the feed and speed override switches are enabled and not set at 100%, the one set at the lower setting will take effect. The speed and feed rates will still be synchronized.

3.14.52 G85 - Canned Cycle: Boring, Feed Out

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling. Program G85 X- Y- Z- A- B- C- R- L-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Retract the Z-axis at the current feed rate to clear Z.

3.14.53 G86 - Canned Cycle: Boring, Spindle Stop, Rapid Move Out

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell.

Program G86 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Stop the spindle turning.
5. Retract the Z-axis at traverse rate to clear Z.
6. Restart the spindle in the direction it was going.

The spindle must be turning before this cycle is used.

It is an error if:

- the spindle is not turning before this cycle is executed.

3.14.54 G87 - Canned Cycle: Back Boring

The G87 cycle is intended for back boring.

Program G87 X- Y- Z- A- B- C- R- L- I- J- K-

The situation is that you have a through hole and you want to counter bore the bottom of hole. To do this you put an L-shaped tool in the spindle with a cutting surface on the UPPER side of its base. You stick it carefully through the hole when it is not spinning and is oriented so it fits through the hole, then you move it so the stem of the L is on the axis of the hole, start the spindle, and feed the tool upward to make the counter bore. Then you stop the tool, get it out of the hole, and restart it.

This cycle uses I and J numbers to indicate the position for inserting and removing the tool. I and J will always be increments from the X position and the Y position, regardless of the distance mode setting. This cycle also uses a K number to specify the position along the Z-axis of the controlled point top of the counter bore. The K number is a Z-value in the current coordinate system in absolute distance mode, and an increment (from the Z position) in incremental distance mode.

1. Preliminary motion, as described above.
2. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
3. Stop the spindle in a specific orientation.
4. Move the Z-axis only at traverse rate downward to the Z position.
5. Move at traverse rate parallel to the XY-plane to the X,Y location.
6. Start the spindle in the direction it was going before.
7. Move the Z-axis only at the given feed rate upward to the position indicated by K.
8. Move the Z-axis only at the given feed rate back down to the Z position.
9. Stop the spindle in the same orientation as before.
10. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
11. Move the Z-axis only at traverse rate to the clear Z.
12. Move at traverse rate parallel to the XY-plane to the specified X,Y location.
13. Restart the spindle in the direction it was going before.

When programming this cycle, the I and J numbers must be chosen so that when the tool is stopped in an oriented position, it will fit through the hole. Because different cutters are made differently, it may take some analysis and/or experimentation to determine appropriate values for I and J.

3.14.55 G88 - Canned Cycle: Boring, Spindle Stop, Manual Out

The G88 cycle is intended for boring. This cycle uses a P word, where P specifies the number of seconds to dwell. Program G88 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Stop the spindle turning.
5. Stop the program so the operator can retract the spindle manually.
6. Restart the spindle in the direction it was going.

3.14.56 G89 - Canned Cycle: Boring, Dwell, Feed Out

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. program G89 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Retract the Z-axis at the current feed rate to clear Z.

3.14.57 G90 - Absolute Distance Mode

Interpretation of RS274/NGC code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program G90. In absolute distance mode, axis numbers (X, Y, Z, A, B, C) usually represent positions in terms of the currently active coordinate system.

3.14.58 G91 - Incremental Distance Mode

Interpretation of RS274/NGC code can be in one of two distance modes: absolute or incremental.

To go into incremental distance mode, program G91. In incremental distance mode, axis numbers (X, Y, Z, A, B, C) usually represent increments from the current values of the numbers.

I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments in all but one usage (see G87), where the meaning changes with distance mode.

3.14.59 G92 - Coordinate System Offset

You can offset the current coordinate system using G92 or G92.3. This offset will then apply to all nine program coordinate systems. This offset may be canceled with G92.1 or G92.2.

To make the current point have the coordinates you want (without motion), program G92 X- Y- Z- A- B- C- , where the axis words contain the axis numbers you want. All axis words are optional, except that at

least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- all axis words are omitted.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 x7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the nine coordinate systems (those designated by G54 - G59.3). Thus all nine coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may be already be in effect when the G92 is called. If this is the case, the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 x9 is then programmed, the new X-axis offset is -5, which is calculated by $[(7-9) + -3]$.

3.14.60 G92.1 - Reset Coordinate System Offsets

To reset axis offsets to zero, program G92.1 or G92.2. G92.1 sets parameters 5211 to 5216 to zero.

3.14.61 G92.2 - Reset Coordinate System Offsets Parameters

To reset axis offsets to zero, program G92.1 or G92.2. G92.2 leaves parameters values in memory.

3.14.62 G92.3 - Restore Axis Offsets

To set the axis offset to the values saved in memory, program G92.3.

3.14.63 G93 - Inverse Time Feed Rate Mode

Three feed rate modes are recognized: units per minute, inverse time and units per revolution. Program G93 to start the inverse time mode.

In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

When the inverse time feed rate mode is active, an F word must appear on every line which has a G1, G2, or G3 motion, and an F word on a line that does not have G1, G2, or G3 is ignored. Being in inverse time feed rate mode does not affect G0 (rapid traverse) motions.

It is an error if:

- inverse time feed rate mode is active and a line with G1, G2, or G3 (explicitly or implicitly) does not have an F word.

3.14.64 G94 - Units Per Minute Feed Rate Mode

Three feed rate modes are recognized: units per minute, inverse time and units per revolution. Program G94 to start the units per minute mode.

In units per minute feed rate mode, an F word is interpreted to mean the controlled point should move at a certain number of inches per minute, millimeters per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

It is an error if:

- a new feed rate is not specified after switching to G94 or G95

3.14.65 G95 - Units Per Revolution Feed Rate Mode

Three feed rate modes are recognized: units per minute, inverse time and units per revolution. Program G95 to start the units per revolution mode.

In units per revolution mode, an F word is interpreted to mean the controlled point should move a certain number of inches per revolution or millimeters per revolution of the spindle, depending on what length units are being used and which axis or axes are moving.

It is an error if:

- a new feed rate is not specified after switching to G94 or G95

3.14.66 G98 - Canned Cycle Return Level: Initial level

When the spindle retracts during canned cycles, there is a choice of how far it retracts. Program G98 to retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).

3.14.67 G99 - Canned Cycle Return Level:

When the spindle retracts during canned cycles, there is a choice of how far it retracts. Program G99 to retract perpendicular to the selected plane to the position indicated by the R word.

3.15M - Codes

3.15.1 M0 - Program Stop

To stop a running program temporarily (regardless of the setting of the optional stop switch), program M0.

3.15.2 M1 - Optional Program Stop

To stop a running program temporarily (but only if the optional stop switch is on), program M1.

3.15.3 M2 - Program End

To end a program, program M2.

3.15.4 M30 - Program End, Pallet Shuttle, and Reset

To exchange pallet shuttles and then end a program, program M30.

3.15.5 M60 - Pallet Shuttle and Program Stop

To exchange pallet shuttles and then stop a running program temporarily (regardless of the setting of the optional stop switch), program M60.

3.15.6 M3 - Turn Spindle Clockwise

To start the spindle turning clockwise at the currently programmed speed, program M3.

3.15.7 M4 - Turn Spindle Counterclockwise

To start the spindle turning counterclockwise at the currently programmed speed, program M4.

3.15.8 M5 - Stop Spindle Turning

To stop the spindle from turning, program M5.

3.15.9 M7 - Mist Coolant On

To turn mist coolant on, program M7.

3.15.10 M8 - Flood Coolant On

To turn flood coolant on, program M8.

3.15.11 M9 - Mist and Flood Coolant Off

To turn all coolant off, program M9.

It is always OK to use any of these commands, regardless of what coolant is on or off.

3.15.12 M87 – Turn Spindle Clockwise and Mist Coolant On (M3M7)

To turn spindle clockwise and mist coolant on program M87.

3.15.13 M88 – Turn Spindle Clockwise and Flood Coolant On (M3M8)

To turn spindle clockwise and flood coolant on program M88.

3.15.14 M89 – Stop Spindle Turning and Coolant Off (M5M9)

To stop spindle and set coolant off program M89.

3.15.15 M6 - Tool Change

To change a tool in the spindle from the tool currently in the spindle to the tool most recently selected (using a T word), program M6.

3.15.16 M61 – Select Tool

To select a tool in the spindle without toolchange program M61. Active tool is set with Q word.

Example:

M61 Q3 – this will set value active tool number to 3

3.15.17 M48 - Enable Speed and Feed Overrides

3.15.18 M49 - Disable Speed and Feed Overrides

3.15.19 M50 - Disable Speed and Feed Overrides

To enable the speed and feed override switches, program M48. To disable both switches, program M49. It is OK to enable or disable the switches when they are already enabled or disabled.

Example:

M48 – this will set feed override enabled

M49 – this will set feed override disabled

M50 P1 – this will set feed override enabled

M50 P0 – this will set feed override disabled

3.15.20 M62 – Digital Output

M62 Qx - Set digital outputs to value 'x'

M62 Pn Qx - Set digital output 'n' to value 'x'

M62 H1 Pn Qx - Set digital output 'n' to frequency 'f'

M62 H2 Pn Qx - Set digital output 'n' to rc servo position 'x'°

M62 H3 Pn Qx - Set digital output 'n' to pwm with duty cycle 'x'°

M62 H3 Pn Qx,f - Set digital output 'n' to pwm with duty cycle 'x'° at frequency 'f'

3.15.21 M63 – Ext Output

M63 Pn Qx - Set ext output 'n' to value 'x'

M63 Hn Qx - Set outputs on ext, port 'n' to value 'x'

M63 Hn Qx;x;x;x... - Send values 'x' to ext, port 'n'

3.15.22 M64 – TX Output

M64 Hn Qx;x;x;x... - Send values 'x' to TX on port 'n'

3.15.23 M65 – I2C Output

M65 Ha Qx;x;x;x... - Send values 'x' to I2C address 'a'

Example:

M65 H60 Q50;18;0;0;9;80;108;97;110;101;116;73;50;67;0;0;0;0

3.16 Other Codes

3.16.1 F - Set Feed Rate

To set the feed rate, program F. The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the user. For motion involving one or more of the X, Y, and Z axes (with or without simultaneous rotational axis motion), the feed rate means length units per minute along the programmed XYZ path, as if the rotational axes were not moving.

3.16.2 S - Set Spindle Speed

To set the speed in revolutions per minute (rpm) of the spindle, program S-. The spindle will turn at that speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program S0; the spindle will not turn if that is done.

It is an error if:

- the S number is negative

3.16.3 T - Select Tool

To select a tool, program T-, where the T number is the carousel slot for the tool. The tool is not changed until an M6 is programmed. The T word may appear on the same line as the M6 or on a previous line. It is OK, but not normally useful, if T words appear on two or more lines with no tool change. The carousel may move a lot, but only the most recent T word will take effect at the next tool change. It is OK to program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change.

It is an error if:

- a negative T number is used,
- a T number larger than the number of slots in the carousel is used.

3.17 Sample G-code programs

3.17.1 Square

```
%  
M3  
G01 X0 Y0  
G01 Z-3  
G01 Z3  
G01 X-150 Y-150  
G01 Z-3  
G01 X150 Y-150  
G01 X150 Y150  
G01 X-150 Y150  
G01 X-150 Y-150  
G01 Z3  
G01 X0 Y0  
M5  
%
```

3.17.2 Circle

```
%  
M3  
G01 X0 Y0  
G01 Z-3  
G01 Z3  
G01 X0 Y-150  
G01 Z-3  
G02 X0 Y-150 I0 J150  
G01 Z3  
G01 X0 Y0  
G01 Z-3  
G01 Z3  
M5  
%
```

3.17.3 One side of a ball in cage

```
%  
G90  
G21  
#1=50 (width of box)  
#2=2 (tool radius)  
#3=4 (height of box)  
#4=5 (resolution in degrees)  
#5=#4 (counter)  
#6=[[#1/2]+#2] (actual radius of circle - cutter radius + radius)  
G01 X0 Y0  
o140 do  
  #8 = [[SIN[#5]*#6]*SIN[45]] ( X and Y position)  
  #9 = [0-[[1-COS[#5]*#6]] (Z position)  
  G1 X[#8] Y[#8]  
  Z[#9]  
  G18 G02 X[0-#8] Z[#9] I[0-#8] K[0-[#9+#6]]  
  G19 G03 Y[0-#8] Z[#9] J[0-#8] K[0-[#9+#6]]  
  G18 G03 X[#8] Z[#9] I[#8] K[0-[#9+#6]]  
  G19 G02 Y[#8] Z[#9] J[#8] K[0-[#9+#6]]  
  #5 = [#5+#4]  
o140 while [#8 LT [#1/2-#2-#3]]  
G1 Z1  
X0 Y0  
%
```

Table of Contents

1	Introduction.....	3
1.1	Overview.....	3
1.2	Features and specifications:.....	4
1.3	System Requirements.....	5
2	Software.....	6
2.1	Overview.....	6
2.2	Installation.....	7
2.2.1	Software installation.....	7
2.2.2	Driver installation.....	10
2.2.3	Completing driver installation (Windows XP).....	12
2.2.4	Completing driver installation (Windows 7).....	14
2.2.5	Completing driver installation (Windows 8 and Windows 8.1).....	15
2.3	Main window.....	18
2.3.1	Position panel.....	19
2.3.2	State panel.....	21
2.3.2.1	Current:.....	21
2.3.2.2	Axes:	21
2.3.2.3	Origin:.....	21
2.3.2.4	Tool:.....	21
2.3.3	Program panel.....	23
2.3.4	Graphical program visualization display.....	24
2.3.5	G-code panel.....	24
2.3.6	Manual data input (MDI) panel.....	24
2.3.7	Toolbar.....	25
2.3.8	File menu.....	27
2.3.8.1	Open.....	28
2.3.8.2	Close.....	28
2.3.8.3	Recent Programs.....	28
2.3.8.4	Import programs.....	28
2.3.8.4.1	Import G-code.....	28
2.3.8.4.2	Import DXF	28
2.3.8.4.3	Import PLT/HPGL.....	28
2.3.8.4.3.1	Advanced / General.....	29
2.3.8.4.3.1.1	General.....	30
2.3.8.4.3.2	Advanced / Safe Height.....	32
2.3.8.4.3.2.1	Safe Height.....	32
2.3.8.4.3.2.2	Base Height.....	34

2.3.8.4.3.2.3 Traverse.....	34
2.3.8.4.3.3 Advanced / Stages.....	36
2.3.8.4.3.3.1 Stage 1	36
2.3.8.4.3.3.2 Stage 2.....	36
2.3.8.4.3.3.3 End.....	37
2.3.8.4.4 Import NC Drill.....	38
2.3.8.4.4.1 Units:.....	39
2.3.8.4.4.2 Format:.....	39
2.3.8.4.4.3 Leading / Trailing Zeroes:.....	39
2.3.8.4.5 Import Gerber.....	40
2.3.8.4.5.1 Use:.....	41
2.3.8.4.5.2 Mark Pads:.....	41
2.3.8.4.5.3 Drill Pads:.....	42
2.3.8.4.5.4 Mode:.....	42
2.3.8.4.5.5 Clear copper.....	43
2.3.8.4.6 Import Image 2D.....	44
2.3.8.4.7 Import Image 3D.....	46
2.3.8.4.8 Import Text.....	47
2.3.8.5 G-Code Wizard.....	48
2.3.8.6 Export program.....	48
2.3.8.6.1 Export Tool-path To G-code.....	48
2.3.8.6.2 Export Tool-path to DXF.....	48
2.3.8.6.3 Export Tool-path to CSV.....	48
2.3.8.6.4 Export Tool-path to RAW.....	48
2.3.8.7 Settings.....	48
2.3.8.7.1 Import Settings.....	48
2.3.8.7.2 Export Settings.....	48
2.3.8.8 Language.....	48
2.3.8.9 Exit.....	48
2.3.9 View menu.....	49
2.3.9.1 Origin.....	50
2.3.9.2 Top View.....	50
2.3.9.3 Side View.....	50
2.3.9.4 Front View.....	50
2.3.9.5 Perspective View.....	50
2.3.9.6 Zoom In.....	50
2.3.9.7 Zoom Out.....	50
2.3.9.8 Zoom Tool.....	50
2.3.9.9 Zoom Extents.....	50
2.3.9.10 Center Tool.....	50
2.3.9.11 Center View.....	50
2.3.9.12 Change View.....	50
2.3.9.13 Simulate.....	51
2.3.9.14 Window	51

2.3.9.14.1	Jog.....	51
2.3.9.14.2	Reset.....	51
2.3.9.14.3	Large.....	51
2.3.9.15	Show.....	52
2.3.9.15.1	Show Grid.....	53
2.3.9.15.2	Show Axes.....	53
2.3.9.15.3	Show Working Area.....	53
2.3.9.15.4	Show Tool.....	53
2.3.9.15.5	Show Tool Handle.....	53
2.3.9.15.6	Show G28, G30.....	53
2.3.9.15.7	Show Park.....	53
2.3.9.15.8	Show Tool Length.....	53
2.3.9.15.9	Show Tool Change.....	53
2.3.9.15.10	Show Stored Positions.....	53
2.3.9.15.11	Show Labels.....	53
2.3.9.15.12	Show Extents.....	53
2.3.9.15.13	Show Direction.....	53
2.3.9.15.14	Show Completed.....	53
2.3.9.15.15	Show Path History.....	54
2.3.9.15.16	Show Gauge.....	54
2.3.9.15.17	Large Tool Handle.....	54
2.3.9.15.18	Tool Radius.....	54
2.3.9.15.19	Follow Tool.....	54
2.3.9.15.20	Coordinates.....	54
2.3.9.15.21	3D.....	54
2.3.9.15.22	G-Code.....	54
2.3.9.15.23	Perspective.....	54
2.3.9.16	Material.....	55
2.3.9.17	Show IO.....	55
2.3.9.17.1	Input.....	56
2.3.9.17.2	Aux.....	56
2.3.9.17.3	Limit.....	56
2.3.9.17.4	Active Limit Switch.....	56
2.3.9.17.5	Active soft Limit Switch.....	56
2.3.9.17.6	Jog.....	56
2.3.9.17.7	Ctrl.....	56
2.3.9.17.8	ExtIn.....	56
2.3.9.17.9	Output.....	56
2.3.9.17.10	ExtOut.....	56
2.3.10	Program menu.....	57
2.3.10.1	Lines.....	58
2.3.10.1.1	Select Next:.....	58
2.3.10.1.2	Select Next Page:.....	58
2.3.10.1.3	Select Previous:.....	58

2.3.10.1.4	Select Previous Page:	58
2.3.10.1.5	Select Next Pause:	58
2.3.10.1.6	Select Previous Pause:	58
2.3.10.1.7	Select Next Tool change:	58
2.3.10.1.8	Select Previous Tool Change:	58
2.3.10.1.9	Select Next Comment:	58
2.3.10.1.10	Select Previous Comment:	58
2.3.10.2	Bookmarks:	59
2.3.10.2.1	Select Next:	59
2.3.10.2.2	Select Previous:	59
2.3.10.2.3	Select All:	59
2.3.10.2.4	Toggle:	59
2.3.10.2.5	Clear All:	59
2.3.10.3	Shift:	60
2.3.10.3.1	User:	60
2.3.10.3.2	Extents to Zero:	60
2.3.10.3.3	Extents to Position:	60
2.3.10.3.4	Center to Zero:	60
2.3.10.3.5	Center to Position:	60
2.3.10.3.6	Extents to Zero XY:	61
2.3.10.3.7	Extents to Position XY:	61
2.3.10.3.8	Center to Zero XY:	61
2.3.10.3.9	Center to Position XY:	61
2.3.10.4	Scale:	62
2.3.10.5	Mirror:	63
2.3.10.6	Rotate XY:	63
2.3.10.7	Advanced:	64
2.3.10.7.1	Clone:	64
2.3.10.7.2	Wrap:	65
2.3.10.7.3	Transform:	65
2.3.10.7.4	Transform from points:	67
2.3.10.7.5	Warp:	68
2.3.10.8	Set Speed:	69
2.3.10.8.1	Feed	69
2.3.10.8.2	Traverse:	69
2.3.10.9	Calculate Speed:	70
2.3.10.9.1	Material	70
2.3.10.9.2	Tool:	70
2.3.10.9.3	Spindle Speed	70
2.3.10.9.4	Chip Load:	70
2.3.10.9.5	Feed Rate:	70
2.3.10.10	Copy XYZ → UVW:	71
2.3.10.11	Swap XYZ ↔ UVW:	71
2.3.10.12	Convert Arcs To Lines:	71

2.3.10.13	Convert All To Lines.....	71
2.3.10.14	Edit G-Code.....	71
2.3.10.15	Edit DXF.....	71
2.3.10.16	Copy To Clipboard.....	71
2.3.10.17	Paste from Clipboard.....	71
2.3.11	Machine menu.....	72
2.3.11.1	Emergency Stop.....	72
2.3.11.2	Start.....	73
2.3.11.3	Start Loop.....	73
2.3.11.4	Start Raw File.....	73
2.3.11.5	Stop.....	73
2.3.11.6	Pause.....	73
2.3.11.7	Bookmarks.....	73
2.3.11.7.1	Start – From Bookmark to Bookmark :	73
2.3.11.7.2	Start - From Beginning To Bookmark:.....	73
2.3.11.7.3	Start - From Bookmark To End:.....	74
2.3.11.8	Mist.....	74
2.3.11.9	Flood.....	74
2.3.11.10	Spindle.....	74
2.3.11.11	Outputs.....	74
2.3.11.11.1	Output Ext:.....	74
2.3.11.12	Go To.....	75
2.3.11.12.1	Use Homing Sequence:	75
2.3.11.12.2	Zero:	75
2.3.11.12.3	Zero XY:	75
2.3.11.12.4	Park1:	75
2.3.11.12.5	Park2:	75
2.3.11.12.6	G28:	76
2.3.11.12.7	G30:	76
2.3.11.12.8	User:	76
2.3.11.12.9	Stored Position:.....	76
2.3.11.12.10	Selected Line:.....	76
2.3.11.12.11	Selected Line XY:.....	76
2.3.11.12.12	Tool To Camera XY:.....	76
2.3.11.12.13	Camera To Tool XY:.....	76
2.3.11.13	Set Position.....	77
2.3.11.13.1	Enable:.....	77
2.3.11.13.2	Zero:	77
2.3.11.13.3	Zero Axis:.....	77
2.3.11.13.4	Park1:.....	78
2.3.11.13.5	Park2:	78
2.3.11.13.6	G28:	78
2.3.11.13.7	G30:	78
2.3.11.13.8	User:	78

2.3.11.13.9	Stored Position:.....	78
2.3.11.13.10	Selected Line:.....	78
2.3.11.13.11	Selected Line XY:	78
2.3.11.13.12	Camera XY:.....	78
2.3.11.13.13	Store Current Position To:.....	79
2.3.11.13.14	Store Camera Position To:.....	79
2.3.11.14	Offset.....	80
2.3.11.14.1	Zero:	80
2.3.11.14.2	Current:	80
2.3.11.14.3	Current XY:	80
2.3.11.14.4	Current Z:	80
2.3.11.14.5	User:	81
2.3.11.14.6	Selected Line:.....	81
2.3.11.14.7	Selected Line XY:.....	81
2.3.11.14.8	Camera XY:.....	81
2.3.11.14.9	From Tool Sensor.....	82
2.3.11.14.10	From Program.....	82
2.3.11.14.11	From Material.....	83
2.3.11.14.12	Surface Distance.....	84
2.3.11.14.13	Measure Offset XY:.....	84
2.3.11.14.14	Measure Offset Z.....	89
2.3.11.14.15	Sensor Setup.....	89
2.3.11.15	Tools.....	90
2.3.11.15.1	Active.....	90
2.3.11.15.2	Change.....	90
2.3.11.15.3	Select.....	90
2.3.11.15.4	Zero Tool Offset.....	90
2.3.11.15.5	Measure Tool Offset.....	90
2.3.11.15.6	Measure Tool Offset Here.....	90
2.3.11.15.7	Set Tool Offset.....	91
2.3.11.15.8	Set Tool Offset from Active Tool	91
2.3.11.15.9	Measure Tool Length.....	91
2.3.11.15.10	Measure Tool Length Here.....	91
2.3.11.15.11	Sensor Setup.....	92
2.3.11.16	Home.....	93
2.3.11.17	Camera.....	93
2.3.11.17.1	Show Camera.....	93
2.3.11.17.2	Set Offset.....	94
2.3.11.18	Transformation.....	96
2.3.11.18.1	Transformation Matrix.....	96
2.3.11.18.2	Calculate Transformation Matrix.....	97
2.3.11.18.3	Calculate Transformation Matrix From Points.....	98
2.3.11.19	Capture & Measure Points.....	101
2.3.11.19.1	Measure.....	102

2.3.11.19.2	Capture.....	107
2.3.11.19.3	Clear.....	108
2.3.11.19.4	Load Points.....	108
2.3.11.19.5	Save Points.....	108
2.3.11.19.6	Save Points To STL.....	108
2.3.11.19.7	Save Points To G-Code.....	108
2.3.11.19.8	Save Points To DXF.....	108
2.3.11.20	Soft Limits.....	109
2.3.11.21	Firmware.....	109
2.3.12	Help menu.....	110
2.3.12.1	Help.....	110
2.3.12.2	Planet CNC Forum.....	110
2.3.12.3	G-Code wiki.....	110
2.3.12.4	G-Code Reference.....	110
2.3.12.5	Activate License.....	110
2.3.12.6	Import License.....	110
2.3.12.7	Export License.....	110
2.3.12.8	Log.....	111
2.3.12.9	Check For Updates.....	111
2.3.12.10	About.....	111
2.4	Settings.....	112
2.4.1	General.....	113
2.4.1.1	Description.....	113
2.4.1.2	Units.....	114
2.4.1.3	Machine Type.....	114
2.4.1.4	Notes.....	114
2.4.2	Connection.....	115
2.4.3	User Interface.....	117
2.4.3.1	Display Resolution:.....	117
2.4.3.2	Display Segments:.....	117
2.4.3.3	Display Limit:.....	117
2.4.3.4	Rotate:.....	118
2.4.3.5	Mirror:.....	118
2.4.3.6	Update:.....	118
2.4.3.7	Tool-path History:.....	118
2.4.3.8	Hover:.....	118
2.4.3.9	XY-UV Distance:.....	118
2.4.3.10	Measure Speed:.....	118
2.4.3.11	Time Factor:.....	118
2.4.3.12	MDI Focus:.....	118
2.4.3.13	Reload G-Code:.....	118
2.4.3.14	G-Code Editor:.....	118
2.4.3.15	Set Colors:.....	119
2.4.3.16	Set Keys:.....	121

2.4.4	Axes.....	123
2.4.4.1	Setup.....	125
2.4.4.2	Options.....	126
2.4.4.3	Backlash.....	127
2.4.4.4	Limits.....	127
2.4.4.5	Soft Limits:	128
2.4.4.6	Homing.....	130
2.4.5	Speed.....	131
2.4.6	Lookahead.....	133
2.4.6.1	Acceleration:.....	134
2.4.6.2	Other:.....	134
2.4.7	Options.....	135
2.4.7.1	Pause:.....	135
2.4.7.2	Stop:.....	135
2.4.8	Output.....	136
2.4.8.1	Pins.....	137
2.4.8.1.1	Spindle (M3, M4, M5).....	137
2.4.8.1.2	Coolant (M7, M8, M9).....	138
2.4.8.2	Pause.....	138
2.4.9	Jog.....	139
2.4.9.1	Jogging Keyboard.....	140
2.4.10	Limit.....	141
2.4.11	Input.....	142
2.4.11.1	Input Ext	144
2.4.12	Ctrl.....	146
2.4.12.1	MPG Pendant.....	147
2.4.13	Spindle Synch.....	149
2.4.14	Tool Table	150
2.4.14.1	Tool Change.....	153
2.4.14.2	Tool Change ATC.....	157
2.4.15	Tool Sensor.....	160
2.4.15.1	Tool Sensor Fixed.....	161
2.4.15.2	Tool Sensor Movable.....	162
2.4.16	Materials.....	163
2.4.17	Parameters.....	165
2.4.18	Accessories.....	166
3	G-Code.....	167
3.1	Overview.....	167
3.2	Format of a Line.....	167
3.3	Line Number	168
3.4	Word	168
3.5	Number.....	168
3.6	Parameters	169

3.7	Expressions and Binary Operations.....	169
3.8	Unary Operation Value.....	170
3.9	Comments and Messages.....	170
3.10	Item Repeats.....	171
3.11	Item Order.....	171
3.12	Modal and Non-modal modes.....	172
3.13	Modal Groups.....	172
3.14	G - Codes.....	173
3.14.1	G00 - Rapid Move.....	173
3.14.2	G01 - Linear Move.....	173
3.14.3	G02 - Arc/Helix Move (CW).....	174
3.14.4	G03 - Arc/Helix Move (CCW).....	174
3.14.5	G04 - Dwell.....	176
3.14.6	G7 - Lathe Diameter Mode.....	176
3.14.7	G8 - Lathe Radius Mode.....	176
3.14.8	G10 L1 - Set Tool Table.....	176
3.14.9	G10 L2 - Set Coordinate System.....	176
3.14.10	G10 L6 - Set Offset.....	177
3.14.11	G10 L7 - Measure and Store Offset.....	177
3.14.12	G10 L8 - Home Machine Axes.....	177
3.14.13	G10 L9 - Set Machine Position Without Move.....	177
3.14.14	G17 - XY-Plane Selection.....	178
3.14.15	G18 - XZ-Plane Selection.....	178
3.14.16	G19 - YZ-Plane Selection.....	178
3.14.17	G20 - Inch Units Selection.....	178
3.14.18	G21 - Millimeter Units Selection.....	178
3.14.19	G28 – Go To Predefined Position.....	178
3.14.20	G30 – Go To Predefined Position.....	178
3.14.21	G33 - Spindle Synchronized Motion.....	179
3.14.22	G40 - Compensation Off.....	180
3.14.23	G41 - Cutter Compensation Left.....	180
3.14.24	G42 - Cutter Compensation Right.....	180
3.14.25	G41.1 - Dynamic Cutter Compensation Left.....	180
3.14.26	G42.1 - Dynamic Cutter Compensation Right.....	180
3.14.27	G43 - Tool Length Offset.....	181
3.14.28	G43.1 - Dynamic Tool Length Offset.....	181
3.14.29	G49 - Cancel Tool Length Offset.....	181
3.14.30	G52 - Coordinate System Offset.....	182
3.14.31	G53 - Move in Machine Coordinates.....	182
3.14.32	G54 - Use Preset Work Coordinate System 1.....	183
3.14.33	G55 - Use Preset Work Coordinate System 2.....	183
3.14.34	G56 - Use Preset Work Coordinate System 3.....	183
3.14.35	G57 - Use Preset Work Coordinate System 4.....	183
3.14.36	G58 - Use Preset Work Coordinate System 5.....	183
3.14.37	G59 - Use Preset Work Coordinate System 6.....	183
3.14.38	G59.1 - Use Preset Work Coordinate System 7.....	183

3.14.39	G59.2 - Use Preset Work Coordinate System 8.....	183
3.14.40	G59.3 - Use Preset Work Coordinate System 9.....	183
3.14.41	G61 - Set Path Control Mode: Exact Path.....	183
3.14.42	G61.1 - Set Path Control Mode: Exact Stop.....	183
3.14.43	G64 - Set Path Control Mode: Path Blending.....	183
3.14.44	G73 - Drilling Cycle with Chip Breaking.....	183
3.14.45	G76 - Threading Cycle.....	184
3.14.46	G80 - Cancel Canned Cycle.....	185
3.14.47	Canned Cycles (G81 - G89).....	185
3.14.48	G81 - Canned Cycle: Drilling.....	187
3.14.49	G82 - Canned Cycle: Drilling, Dwell.....	188
3.14.50	G83 - Canned Cycle: Peck Drilling.....	188
3.14.51	G84 - Canned Cycle: Right Hand Tapping.....	189
3.14.52	G85 - Canned Cycle: Boring, Feed Out.....	189
3.14.53	G86 - Canned Cycle: Boring, Spindle Stop, Rapid Move Out.....	190
3.14.54	G87 - Canned Cycle: Back Boring.....	190
3.14.55	G88 - Canned Cycle: Boring, Spindle Stop, Manual Out.....	191
3.14.56	G89 - Canned Cycle: Boring, Dwell, Feed Out.....	191
3.14.57	G90 - Absolute Distance Mode.....	191
3.14.58	G91 - Incremental Distance Mode.....	191
3.14.59	G92 - Coordinate System Offset.....	191
3.14.60	G92.1 - Reset Coordinate System Offsets	192
3.14.61	G92.2 - Reset Coordinate System Offsets Parameters.....	192
3.14.62	G92.3 - Restore Axis Offsets.....	192
3.14.63	G93 - Inverse Time Feed Rate Mode.....	192
3.14.64	G94 - Units Per Minute Feed Rate Mode.....	193
3.14.65	G95 - Units Per Revolution Feed Rate Mode.....	193
3.14.66	G98 - Canned Cycle Return Level: Initial level.....	193
3.14.67	G99 - Canned Cycle Return Level:	193
3.15	M - Codes.....	194
3.15.1	M0 - Program Stop.....	194
3.15.2	M1 - Optional Program Stop.....	194
3.15.3	M2 - Program End.....	194
3.15.4	M30 - Program End, Pallet Shuttle, and Reset.....	194
3.15.5	M60 - Pallet Shuttle and Program Stop.....	194
3.15.6	M3 - Turn Spindle Clockwise.....	194
3.15.7	M4 - Turn Spindle Counterclockwise.....	194
3.15.8	M5 - Stop Spindle Turning.....	194
3.15.9	M7 - Mist Coolant On.....	194
3.15.10	M8 - Flood Coolant On.....	194
3.15.11	M9 - Mist and Flood Coolant Off.....	194
3.15.12	M87 – Turn Spindle Clockwise and Mist Coolant On (M3M7).....	195
3.15.13	M88 – Turn Spindle Clockwise and Flood Coolant On (M3M8).....	195
3.15.14	M89 – Stop Spindle Turning and Coolant Off (M5M9).....	195
3.15.15	M6 - Tool Change.....	195
3.15.16	M61 – Select Tool.....	195
3.15.17	M48 - Enable Speed and Feed Overrides.....	195

3.15.18	M49 - Disable Speed and Feed Overrides.....	195
3.15.19	M50 - Disable Speed and Feed Overrides.....	195
3.15.20	M62 – Digital Output.....	195
3.15.21	M63 – Ext Output.....	196
3.15.22	M64 – TX Output.....	196
3.15.23	M65 – I2C Output.....	196
3.16	Other Codes.....	197
3.16.1	F - Set Feed Rate.....	197
3.16.2	S - Set Spindle Speed.....	197
3.16.3	T - Select Tool.....	197
3.17	Sample G-code programs.....	198
3.17.1	Square.....	198
3.17.2	Circle.....	198
3.17.3	One side of a ball in cage.....	199