User manual

2017/11/22

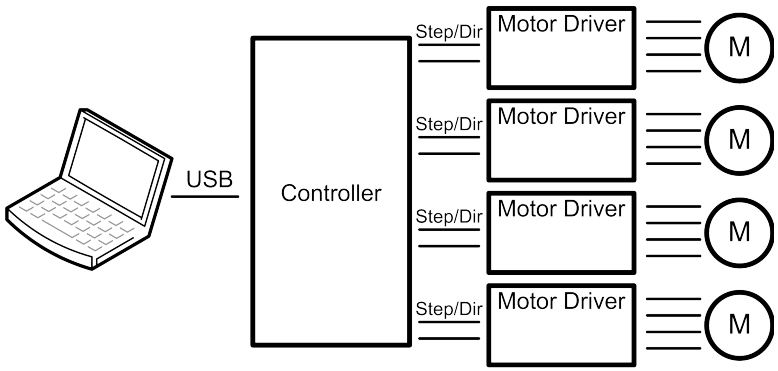
Disclaimer

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 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 or Ethernet 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 different models available. PlanetCNC controllers provide a complete, fully integrated software/hardware solution. The PlanetCNC TNG software is a dedicated application, designed to fully exploit the features of the purpose-built CNC hardware. It has many advanced features to assist day-to-day CNC machine operation.

1.2 Features and specifications:

- PC/Laptop running Windows XP, Vista, Windows 7, 8, 8.1 or Windows 10 (32-bit or 64-bit)
- PC/Laptop running Linux 64-bit OS
- PC/Laptop with USB (V2.x) or Ethernet port (Mk3 controller only)
- advanced motion interpolation and kinematic algorithms
- start, stop, pause and resume execution of program on your machine
- standard RS274/NGC G-code with extensions to achieve full LinuxCNC G-code compatibility
- support for user defined M-codes
- customizable M-codes (e.g.: custom M6, M3,... behavior)
- tested with DeskProto, SheetCAM, SolidCAM, MasterCAM, ArtCAM, Vectric, CamBam, MeshCAM ... generated G-code
- foam cutting 4-axes G-code supported
- lathe G-codes supported
- plasma with optional THC G-code supported
- rotational axes G-code supported
- 5-axes G-code supported
- measuring and probing supported
- spindle synchronization supported
- canned cycles supported
- transformations, different coordinate systems and offsets supported
- PWM, I2C, SPI, USART communication with external devices
- 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
- export toolpath to different formats
- simulation
- automatic and fully configurable homing procedure
- fully configurable toolchange procedure
- automatic tool length measuring

1.3 System Requirements

PlanetCNC TNG is a high performance CNC system. It is designed with flexibility in mind and it can be used for mills, routers, lathes, plasma or laser machines as well as any other machine or system where coordinated movement of servo or stepper motors is needed.

PlanetCNC TNG software works with Mk3 series of motion controllers and PC running windows 7, 8, 8.1, 10 or Linux.

For best performance of PlanetCNC TNG software, PC with 4 virtual processors(cores) CPU is recommended. However, PC's with 2 virtual processors will do just fine. Various services running in the background, antivirus software and program updates can interfere with PlanetCNC TNG performance and that is why dedicated computer is recommended.
If using USB, controller should be connected directly to computers root USB port. We recommend that you connect controller to computer root USB port via USB HUB device. Note that controller should be the only USB device connected on this HUB.
You see, all devices connected to the same HUB device share available bandwidth. Because data traffic is prioritized by the OS, it would not be uncommon if another device connected to same HUB would interfere with controller and therefore compromise the communication between controller and PC.

2 Software

2.1 Overview

PlanetCNC TNG 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 ease.

Configuration options allow for maximum flexibility, integration and customization.

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.

2.2 Installation

PlanetCNC TNG software is compatible with Linux (tested with Ubuntu MATE distribution), Windows XP, Vista, Windows 7, Windows 8, 8.1 and Windows 10(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.

Requirements:

Microsoft .NET 3.5 SP1 Framework

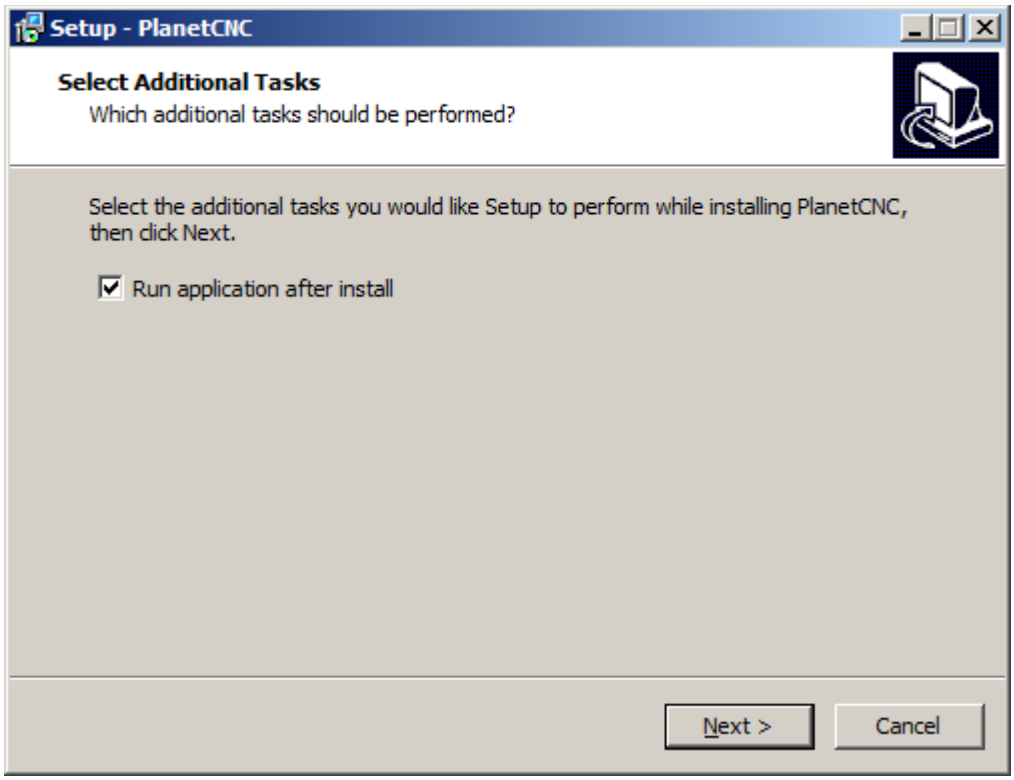
2.2.1 Software installation on Windows

Download installation files from PlanetCNC (www.planet-cnc.com) homepage:

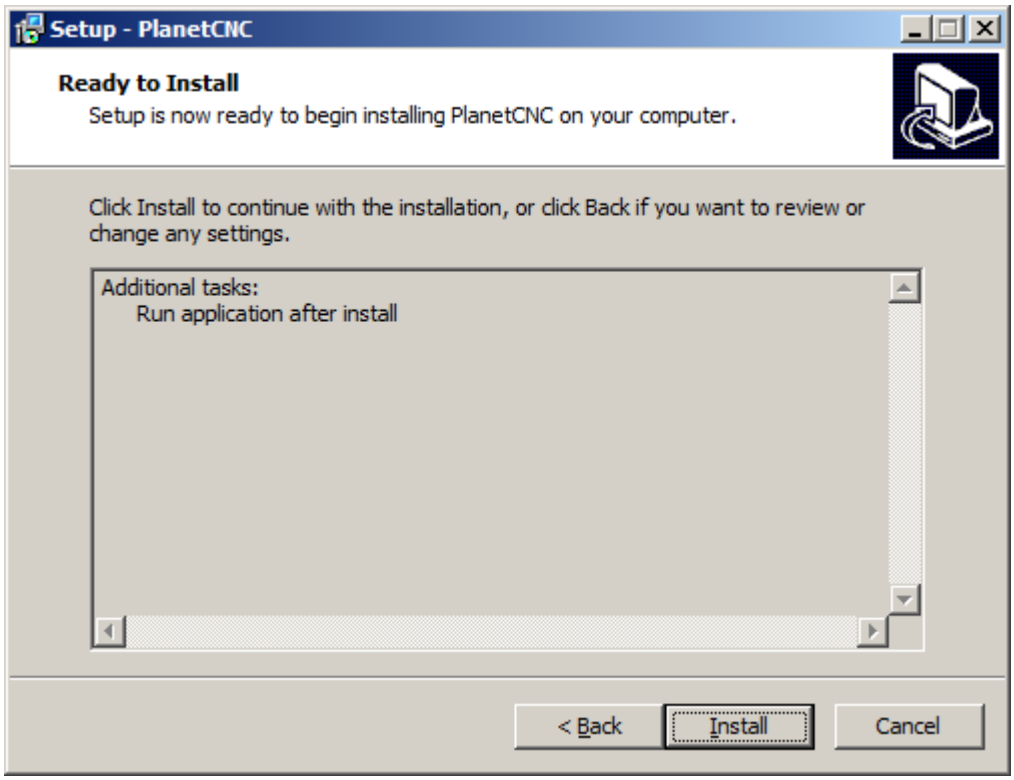
https://planet-cnc.com/software/

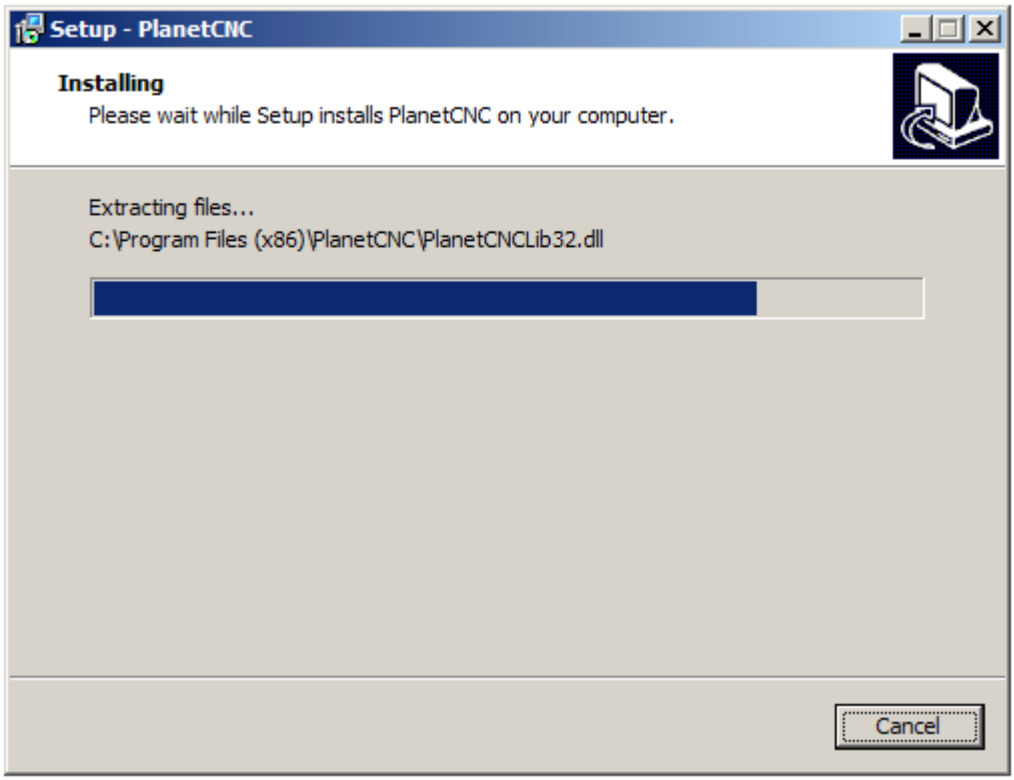
From *Choose your download* drop-down menu select *PlanetCNC TNG 2017 – Windows* and click *Download.* Double click on downloaded *PlanetCNC_Install.exe* file to begin with installation.

Setup-PlanetCNC dialogue will be displayed:

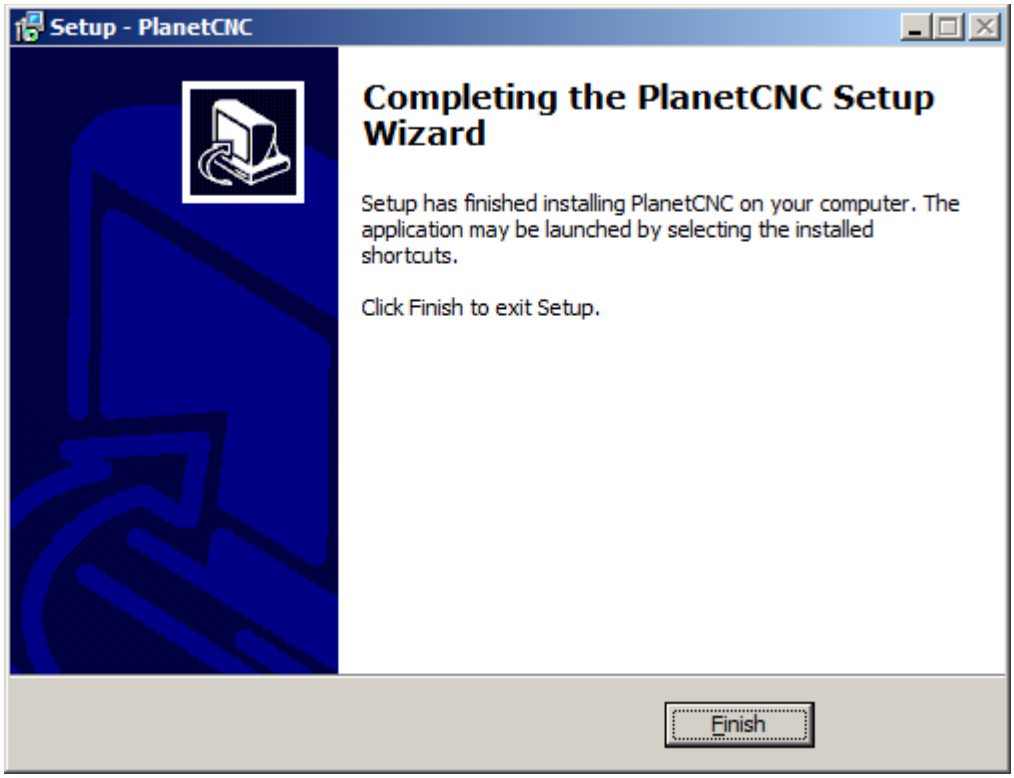


Setup-PlanetCNC dialogue will ask if you are ready to install, click *Install*:





Complete PlanetCNC Setup wizard by clicking *Finish*:



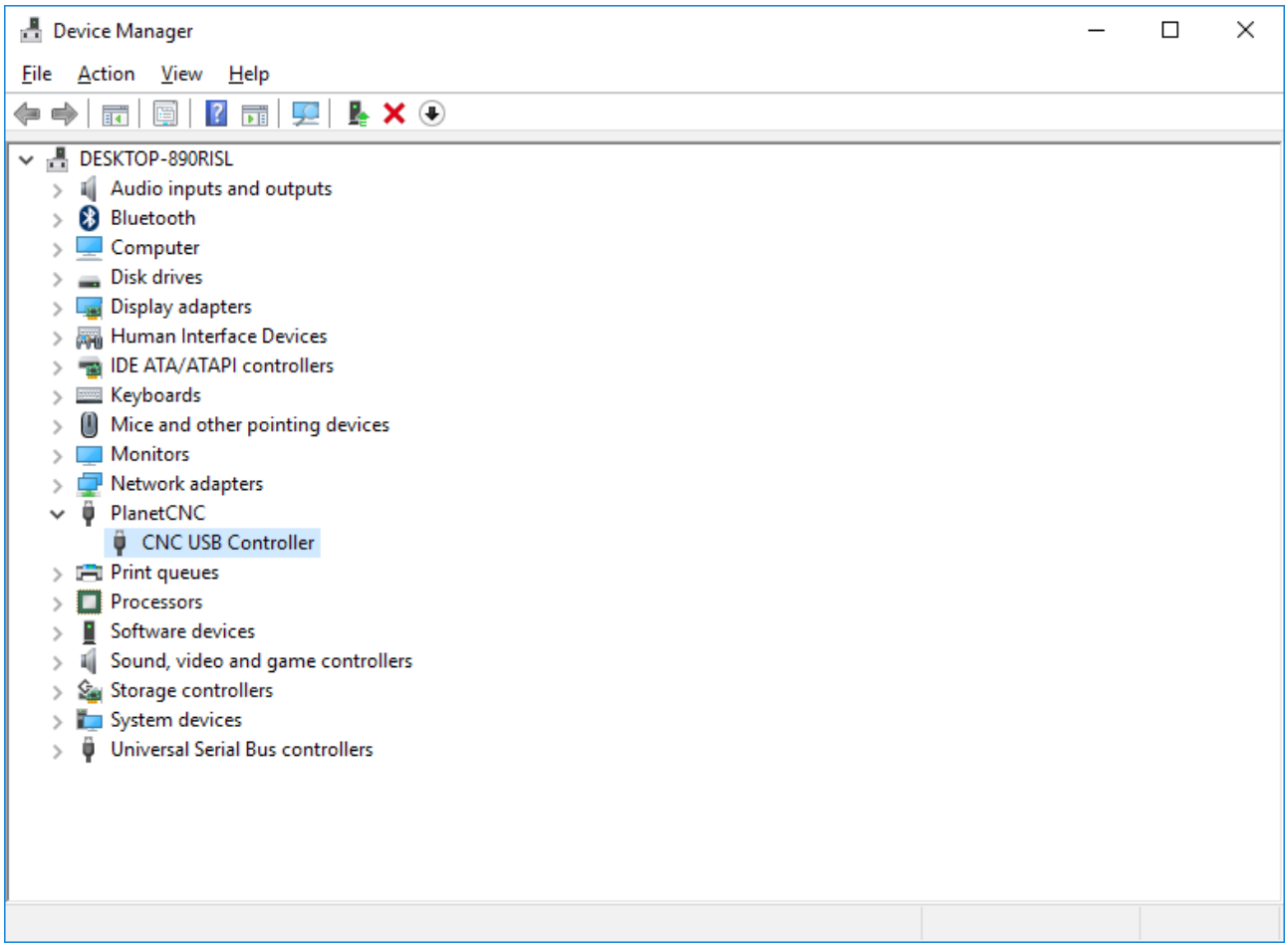
2.2.2 PlanetCNC USB driver installation

PlanetCNC TNG software uses new and optimized USB driver.

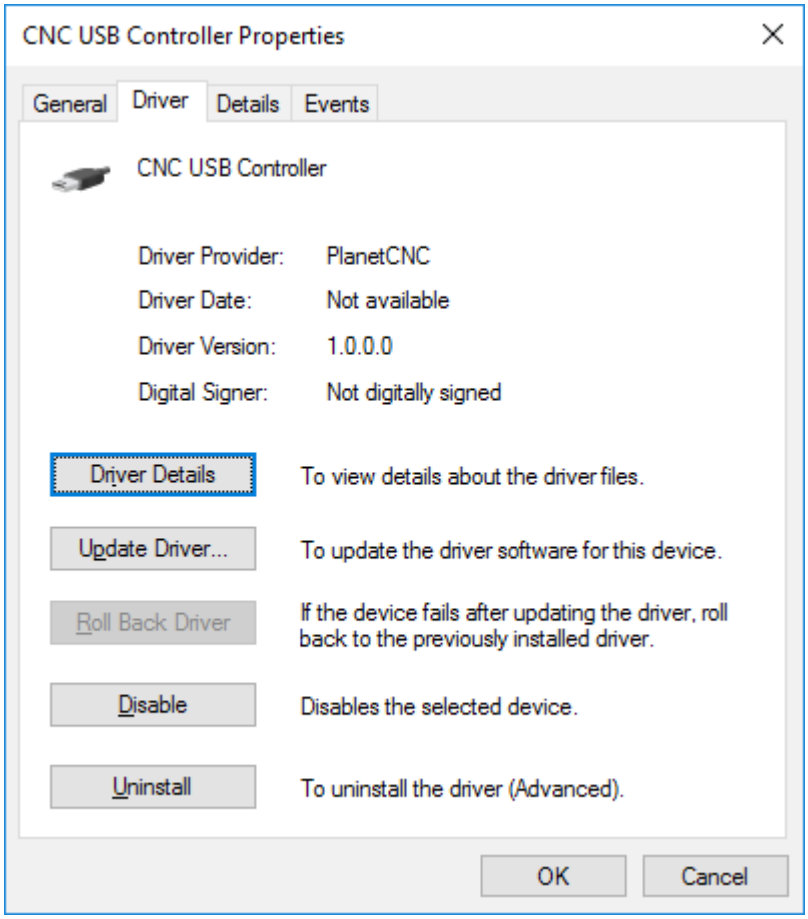
In order to update and use your PlanetCNC Mk3 series controller with PlanetCNC TNG software, you would need to use latest PlanetCNC USB driver.

To check your USB driver version click: Win Key/Control Panel/Device Manager

Under connected devices you will notice CNC USB controller:



Right click on it and choose Properties, and under Driver tab you will see USB driver version:



You will notice that USB driver version is 1.0.0.0. and that it is not digitally signed.

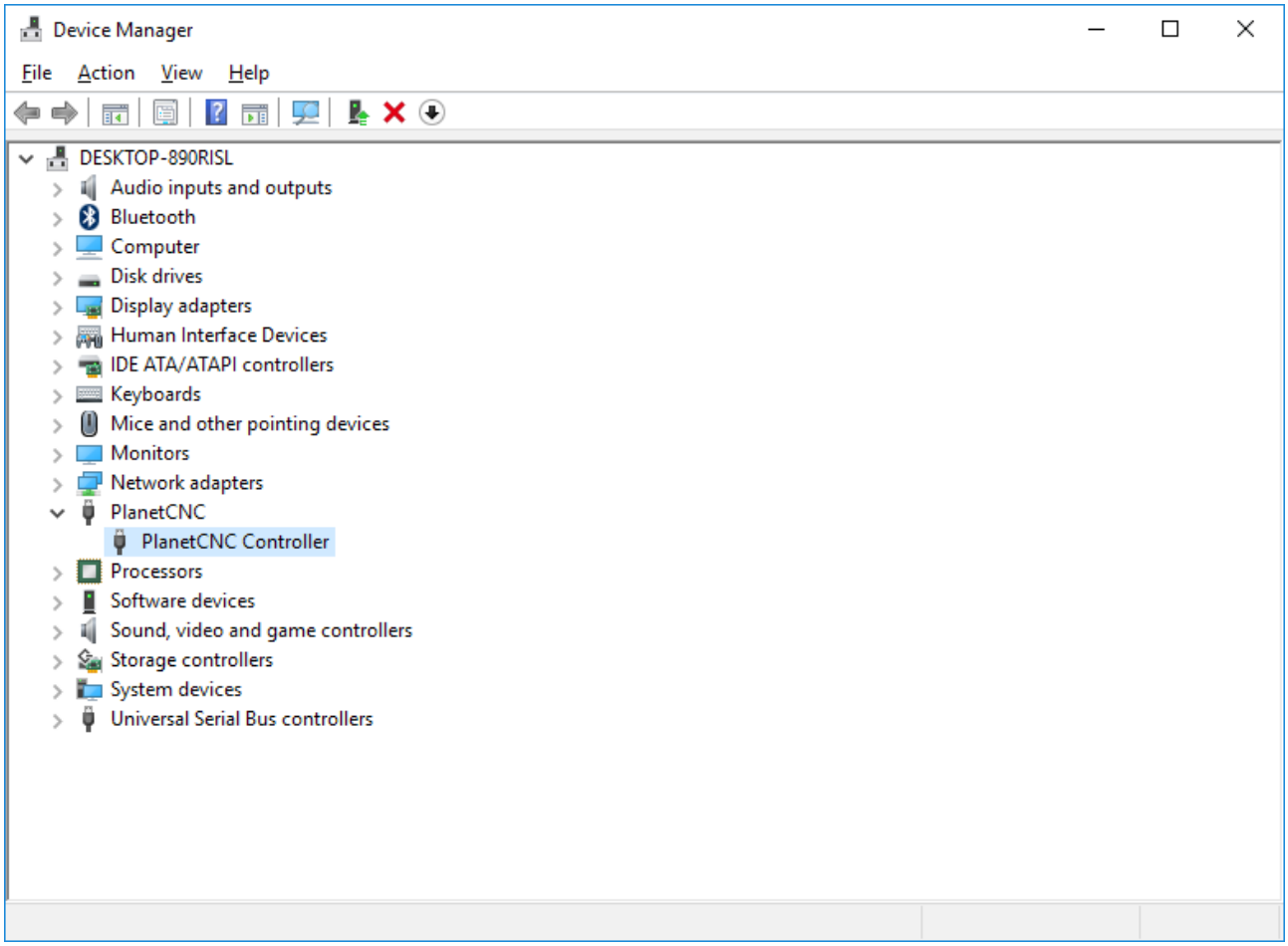
You can get latest PlanetCNC USB driver here: https://planet-cnc.com/software/

After you download file double click on the .exe file and follow installation wizard.

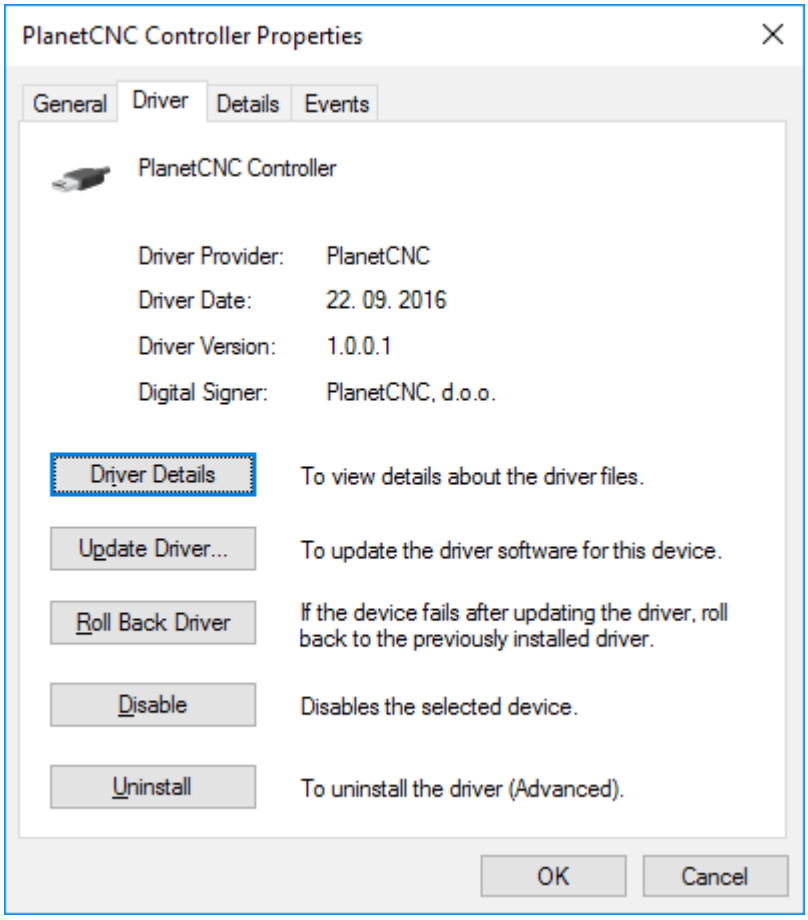
After installation is complete, it would be best to restart your computer.

Now check if your USB driver has been updated to latest version, click:
Win Key/Control Panel/Device Manager

You will notice that PlanetCNC device has a new name: *PlanetCNC controller*



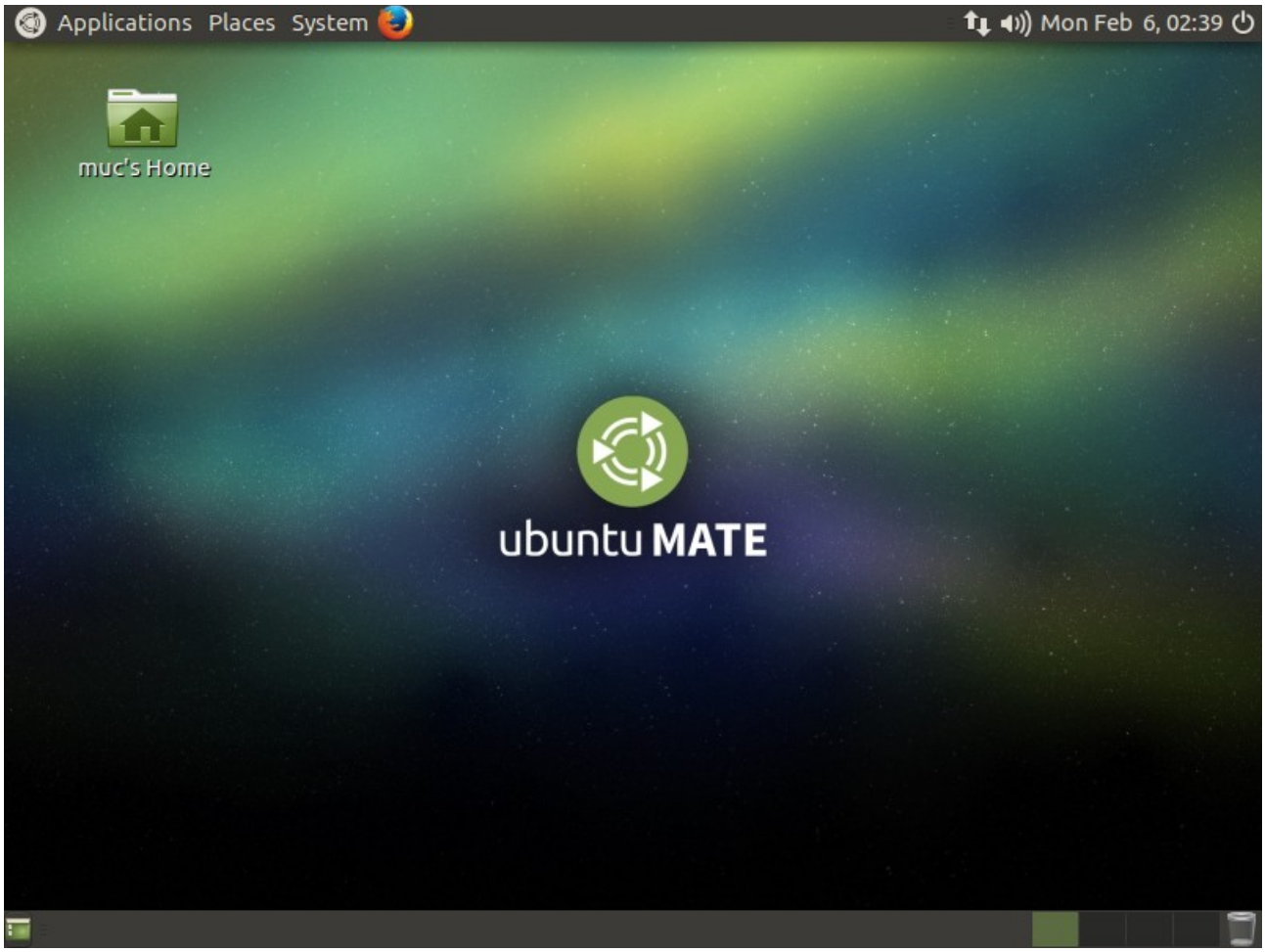
If you check properties of this device you will see that under Driver tab driver version is now 1.0.0.1. and is digitally signed by PlanetCNC d.o.o.:



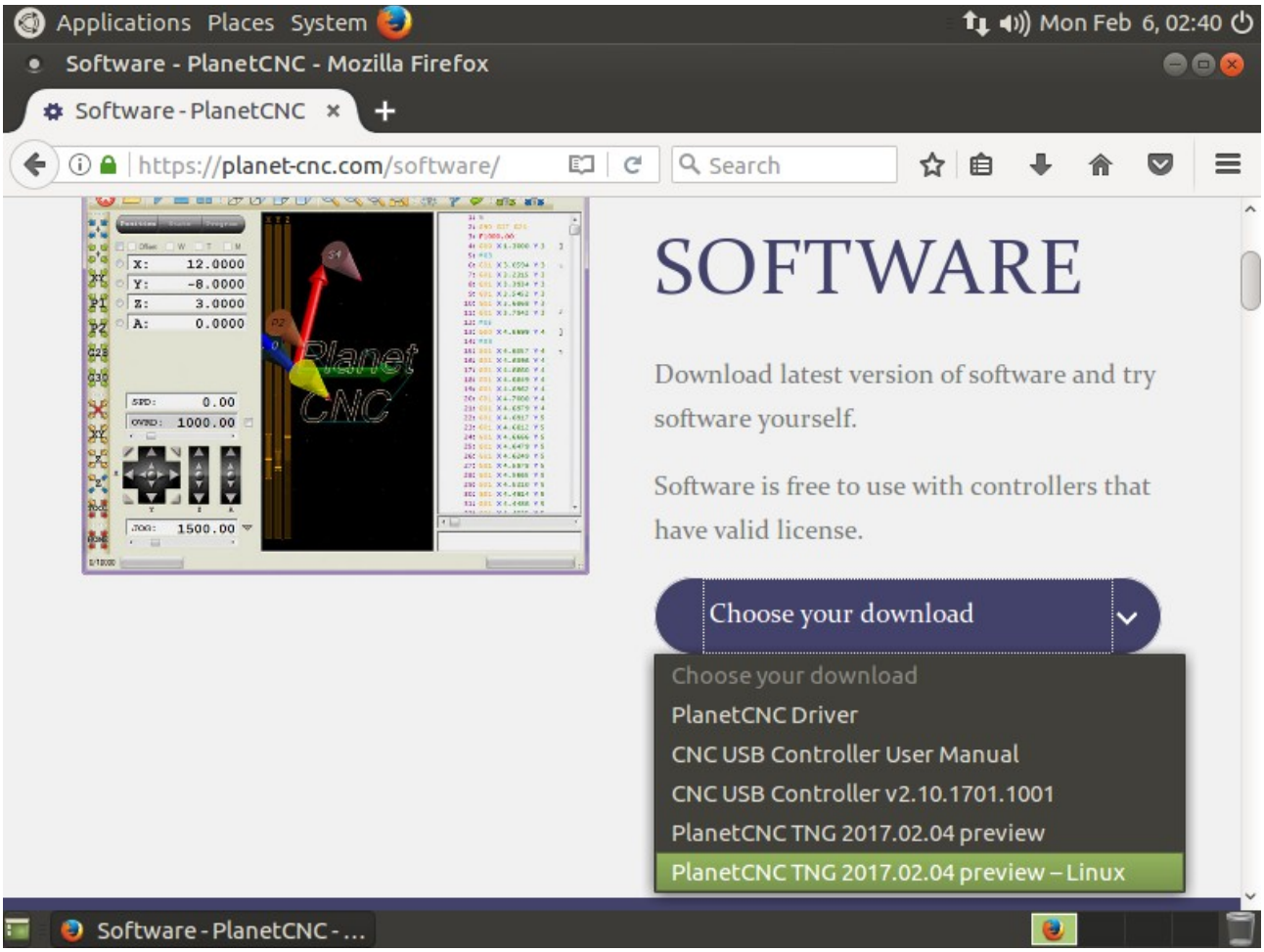
2.2.3 Software installation on Linux

We used freshly installed Linux – Ubuntu MATE distribution for this guide. Please note that distributions differ one from another so these steps may not be suitable for all distributions and installation methods may vary.

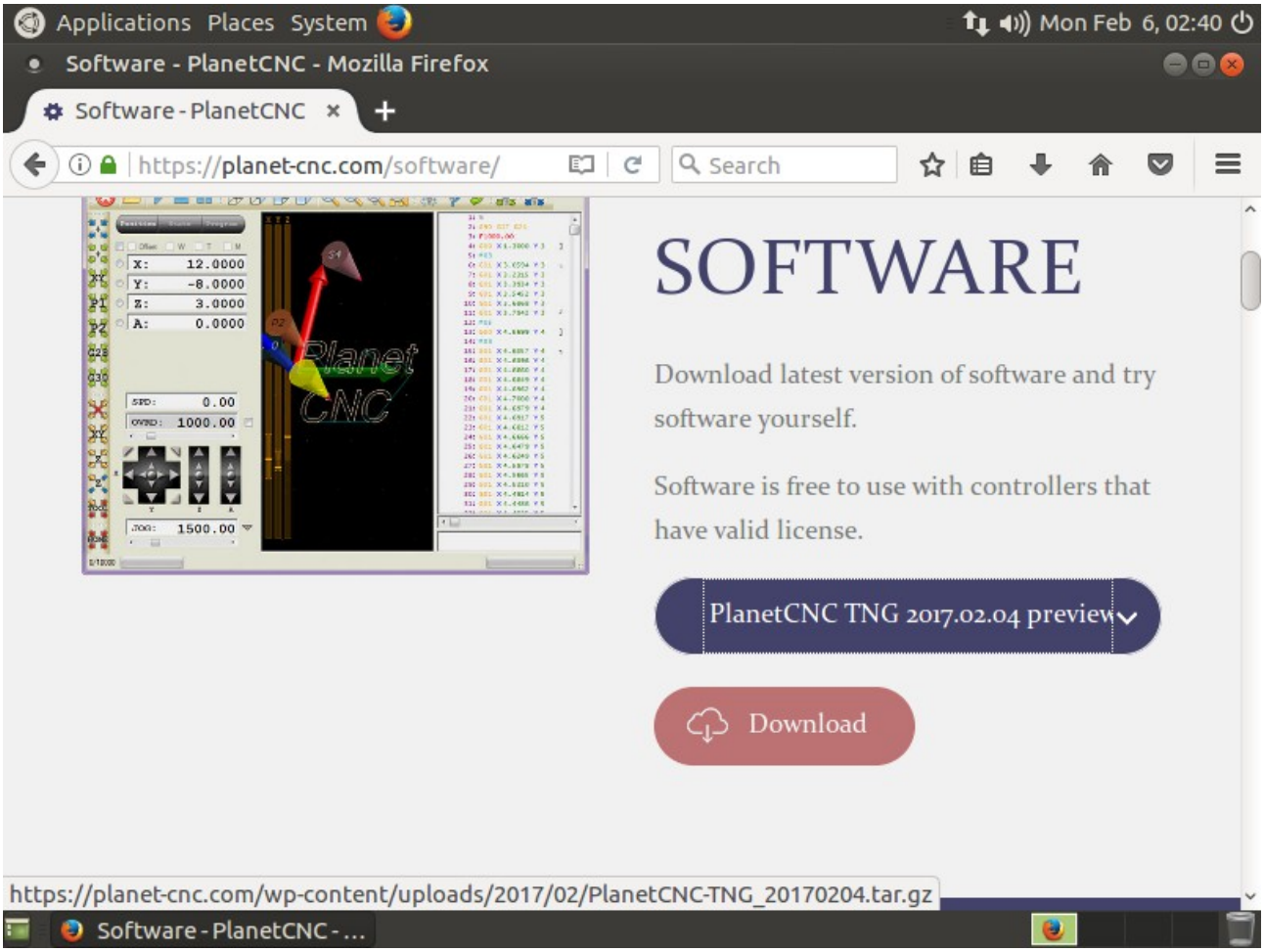
Start your Ubuntu MATE system:



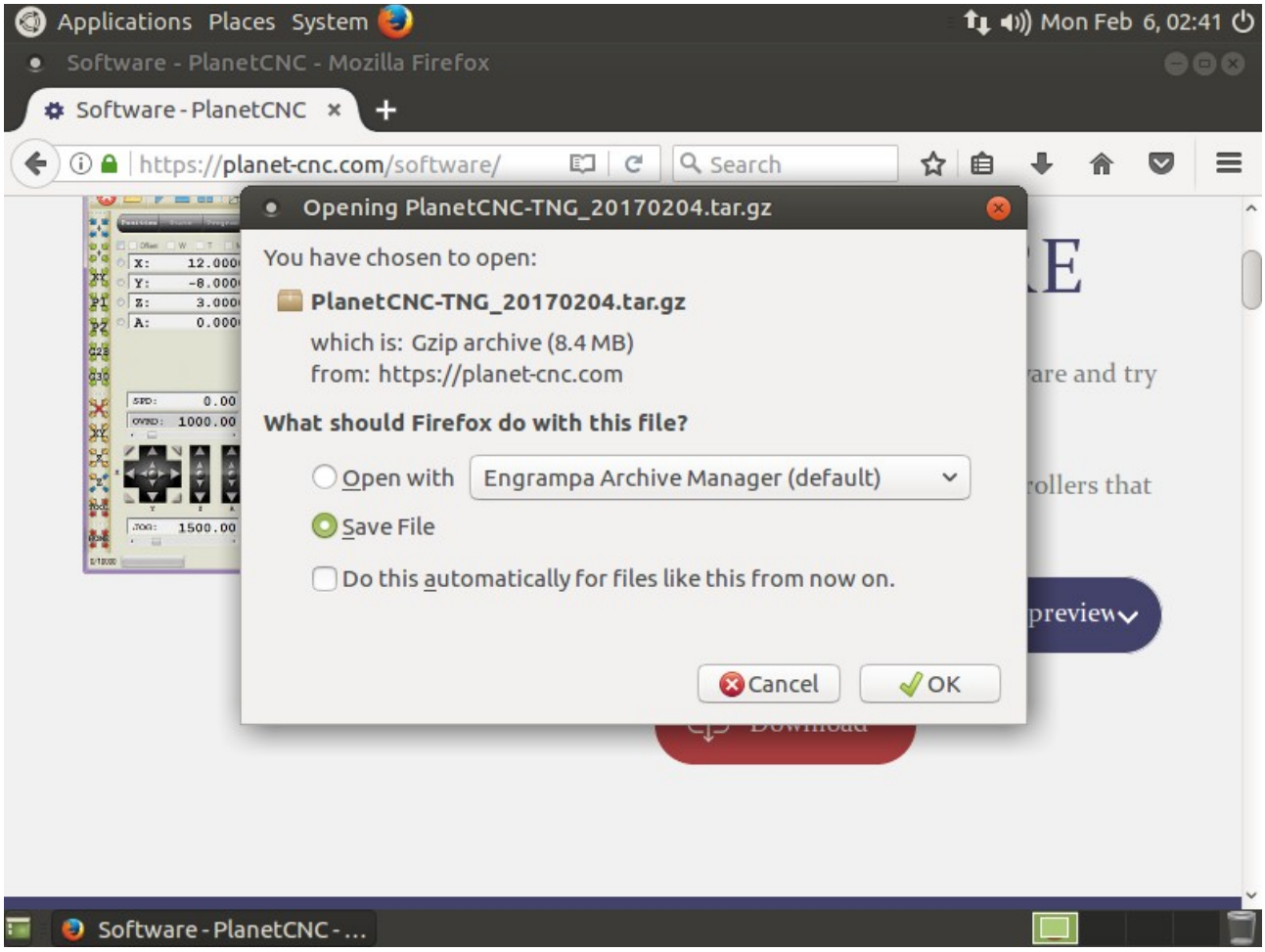
Using your web browser, download PlanetCNC TNG version from PlanetCNC download page: *https://planet-cnc.com/software/*



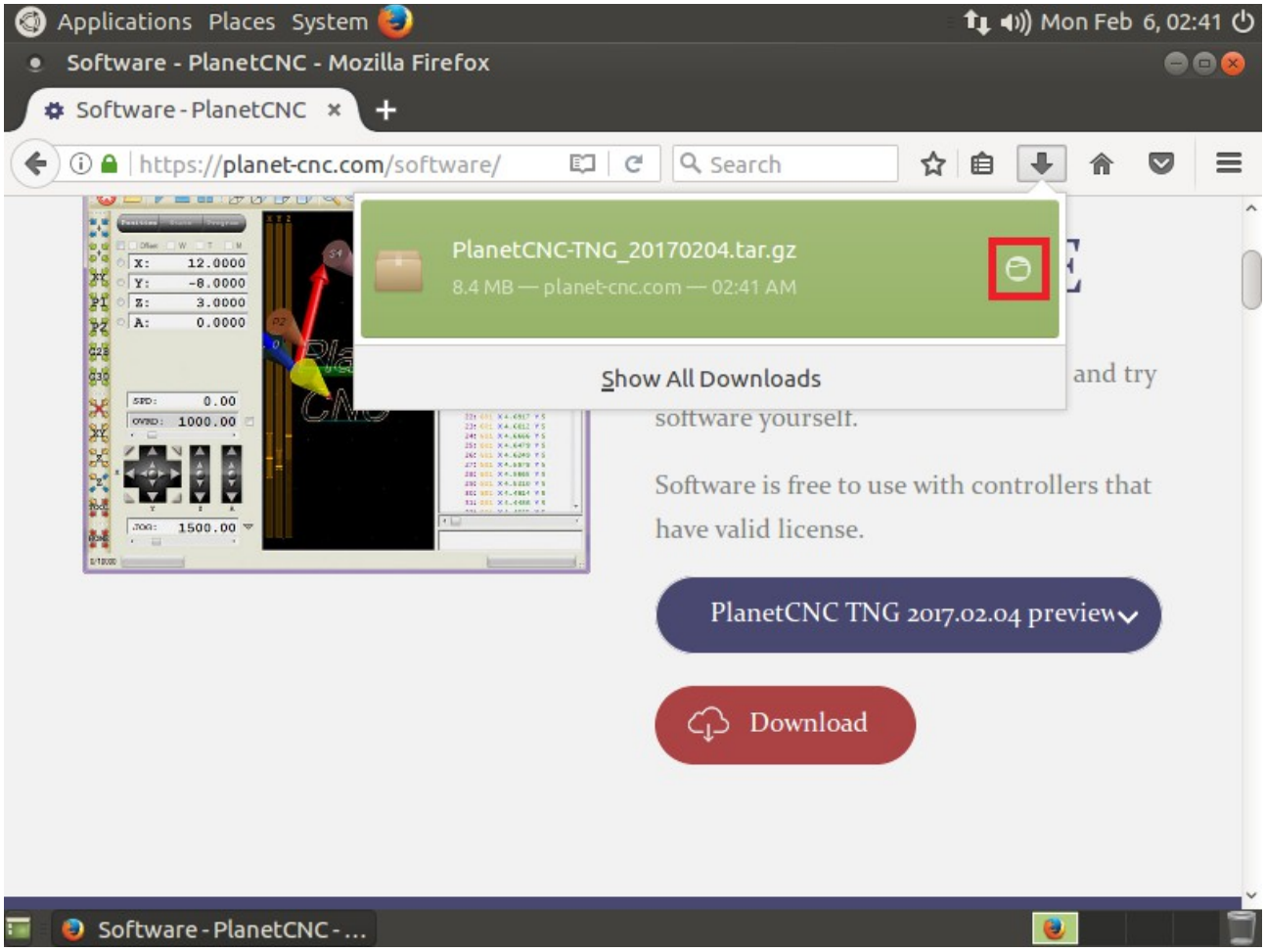
Under download options choose *PlanetCNC TNG preview-Linux* and click *Download* button:



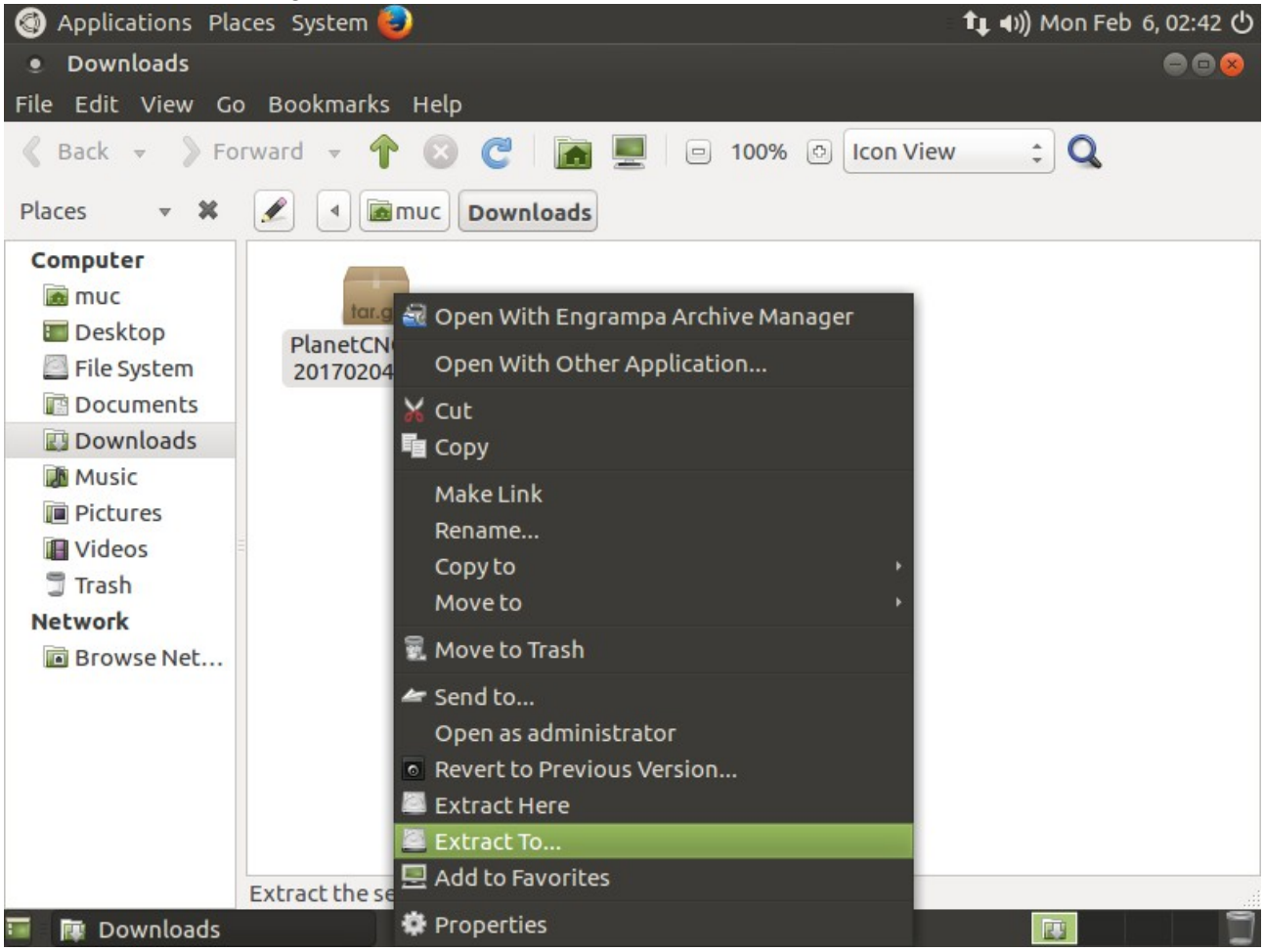
When download dialogue appears, select *Save File* and hit *OK* button:



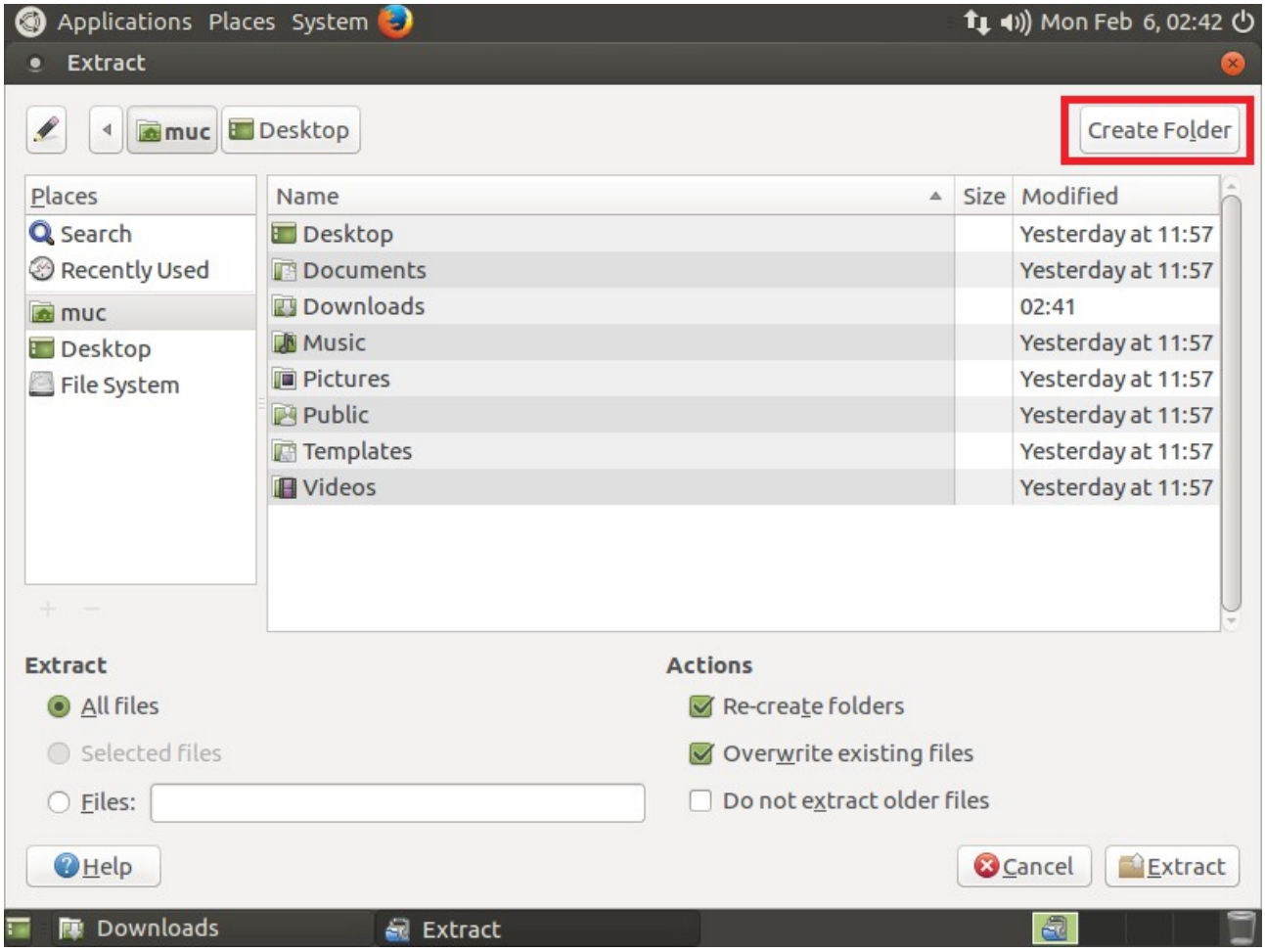
When download is complete, click *Open folder* button:



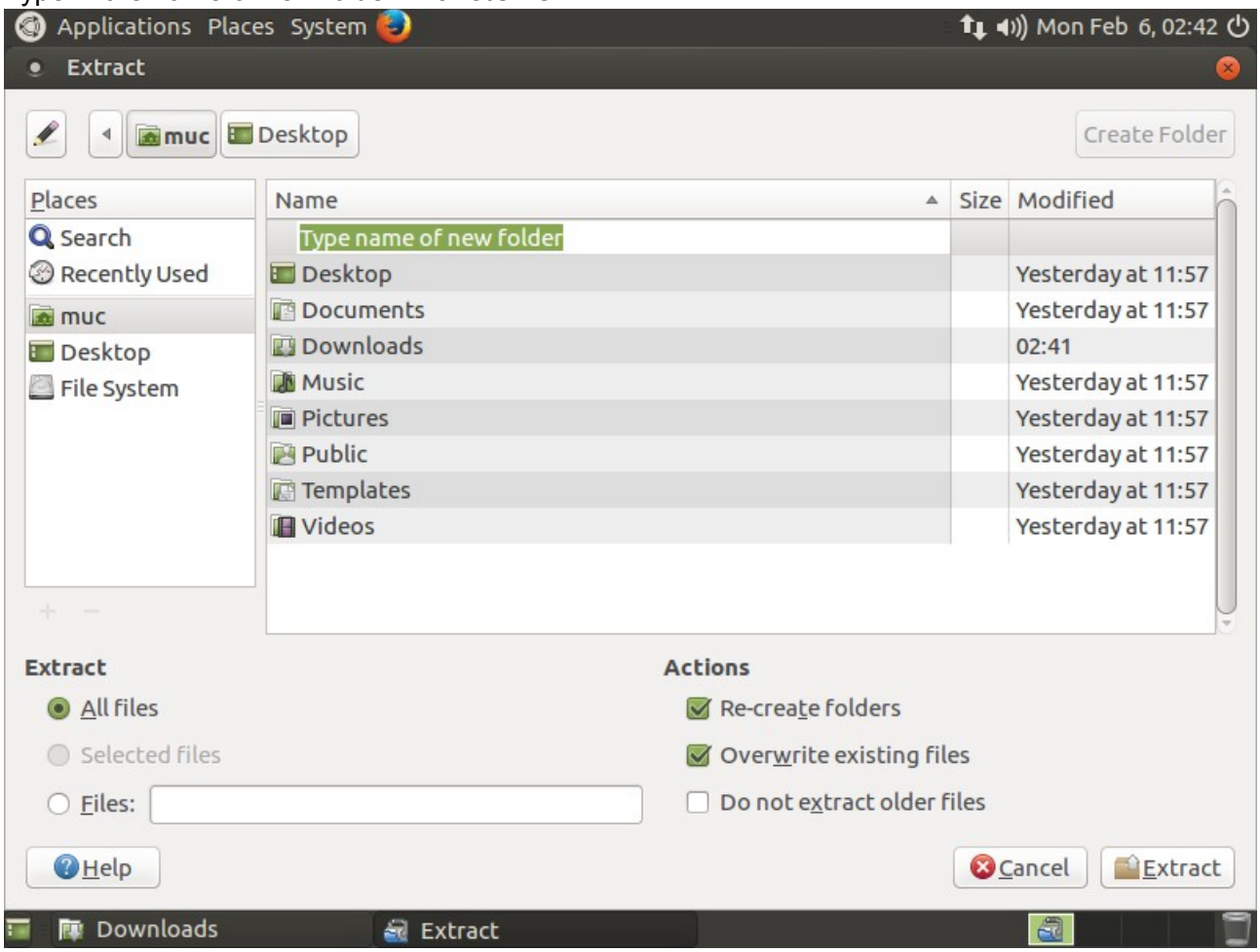
In *Downloads* folder, right click on downloaded file and click: *Extract To…*:

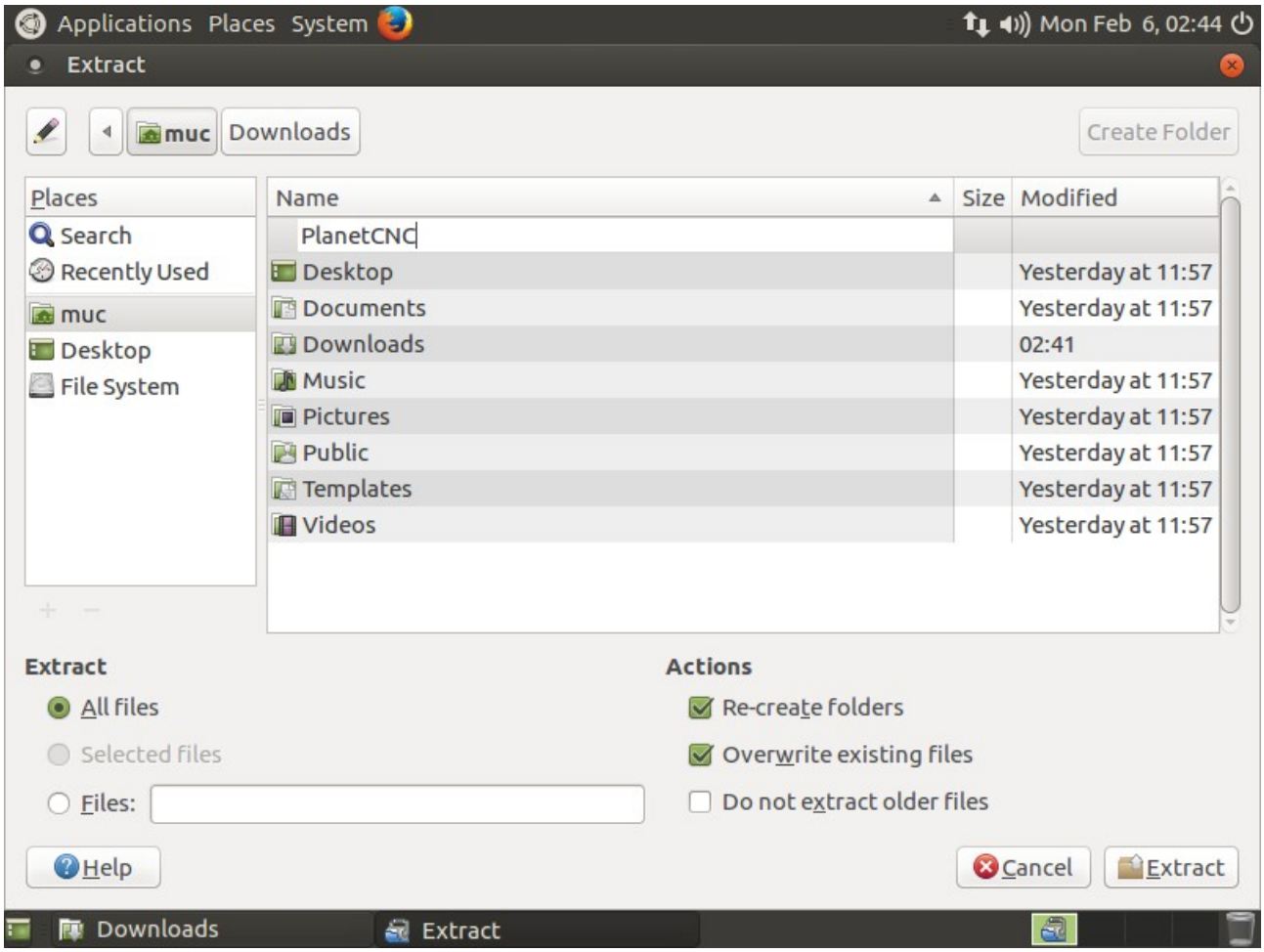


Extract dialogue will appear, click: *Create Folder* button:

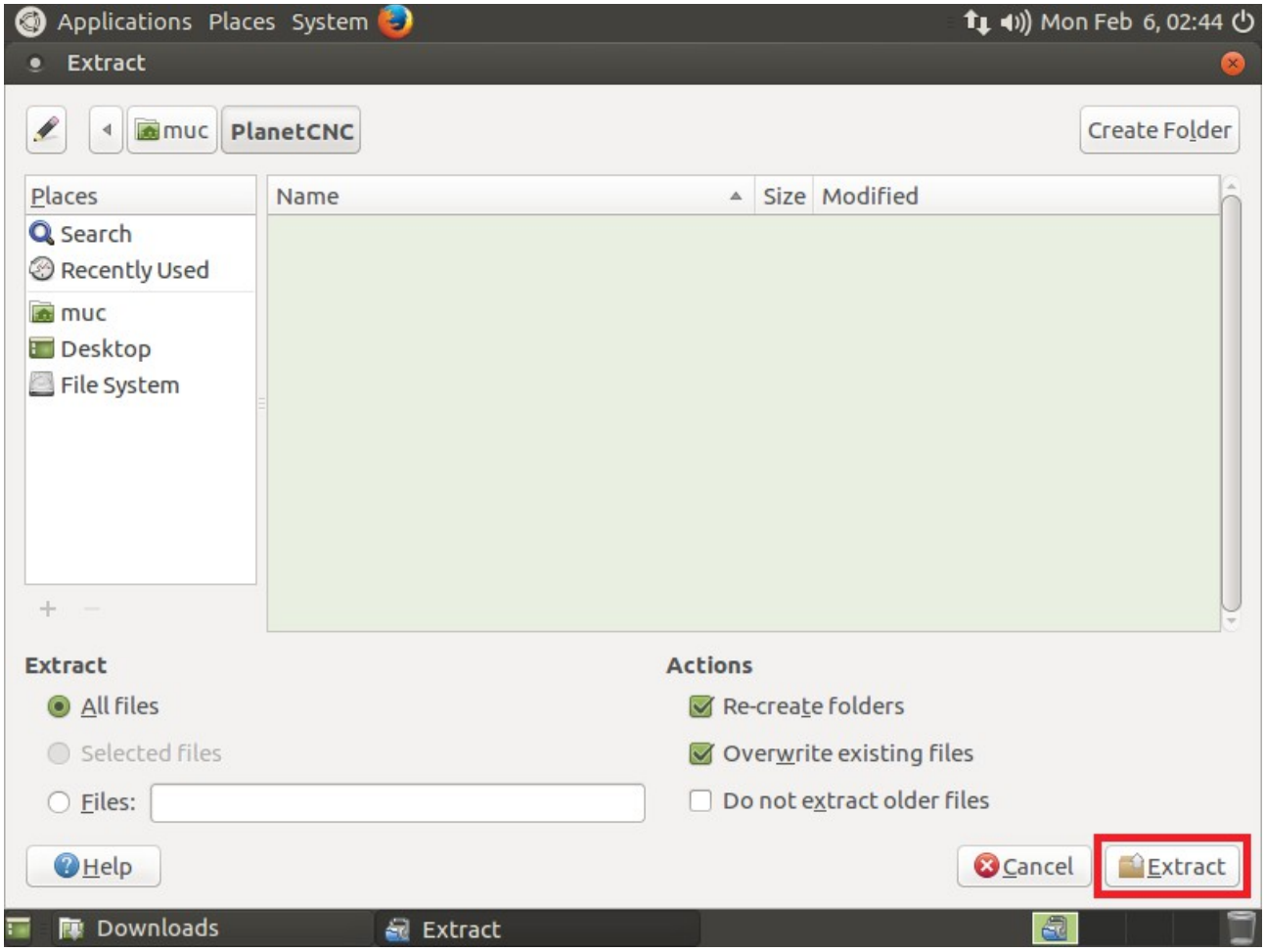


Type in the name of new folder: PlanetCNC

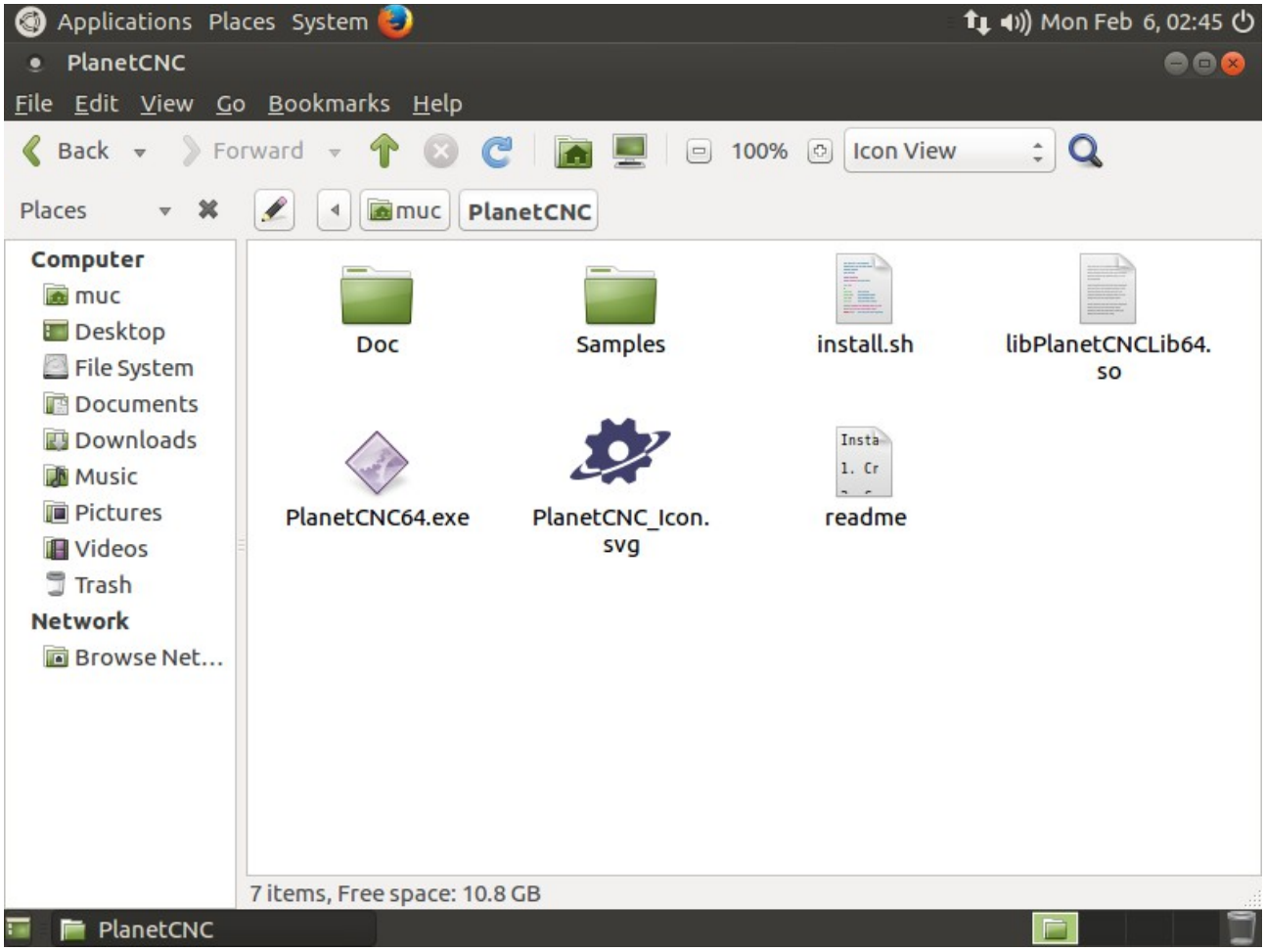



Applications Places System
Mon Feb 6, 02:44
Extract
muc
Downloads
Create Folder
Places
Search
Recently Used
muc
Desktop
File System
Name
Size
Modified
PlanetCNC
Desktop
Yesterday at 11:57
Documents
Yesterday at 11:57
Downloads
02:41
Music
Yesterday at 11:57
Pictures
Yesterday at 11:57
Public
Yesterday at 11:57
Templates
Yesterday at 11:57
Videos
Yesterday at 11:57
Extract
All files
Selected files
Files:
Actions
Re-create folders
Overwrite existing files
Do not extract older files
Help
Cancel
Extract
Downloads
Extract

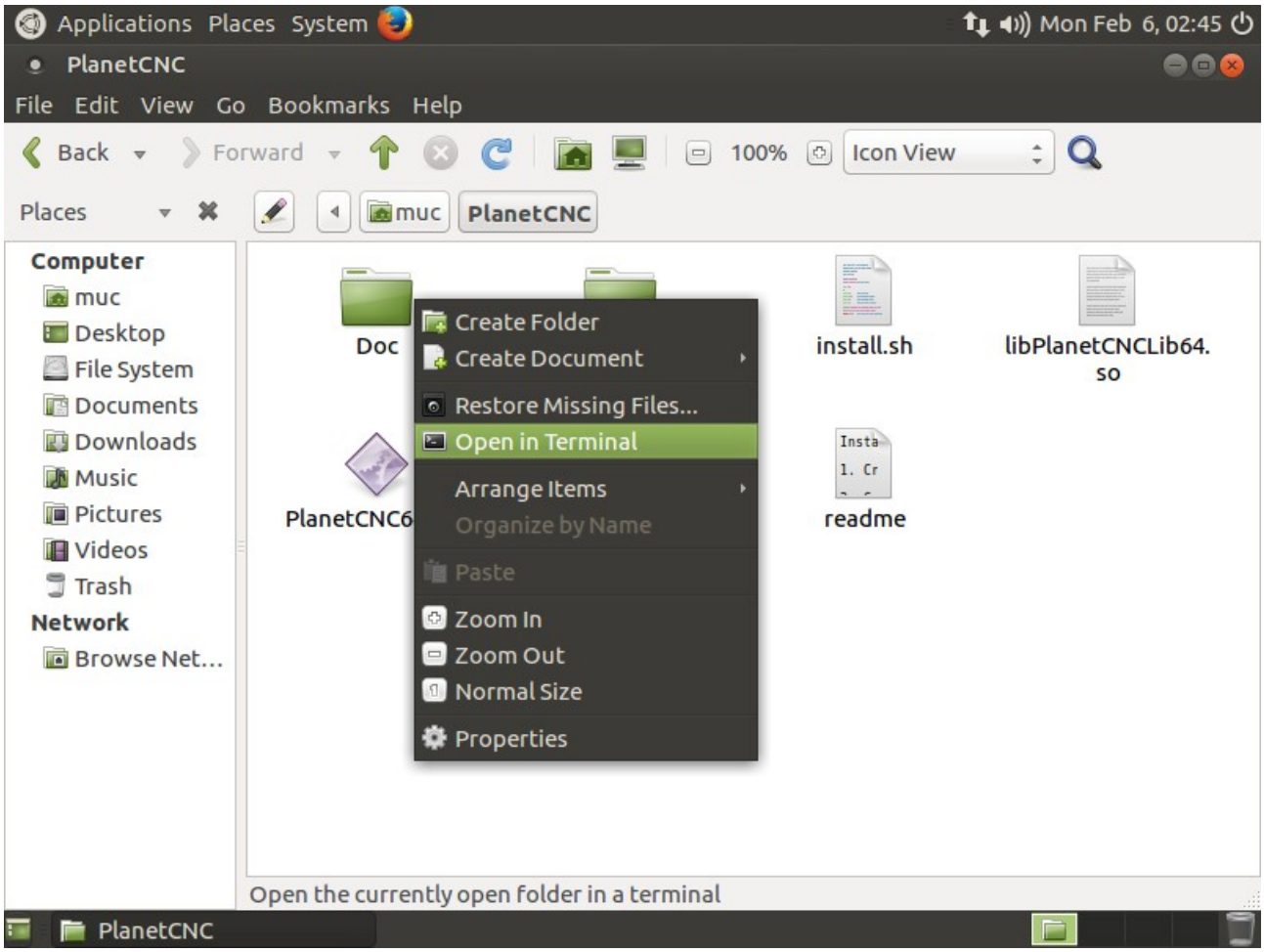
Open PlanetCNC folder and click *Extract* button:



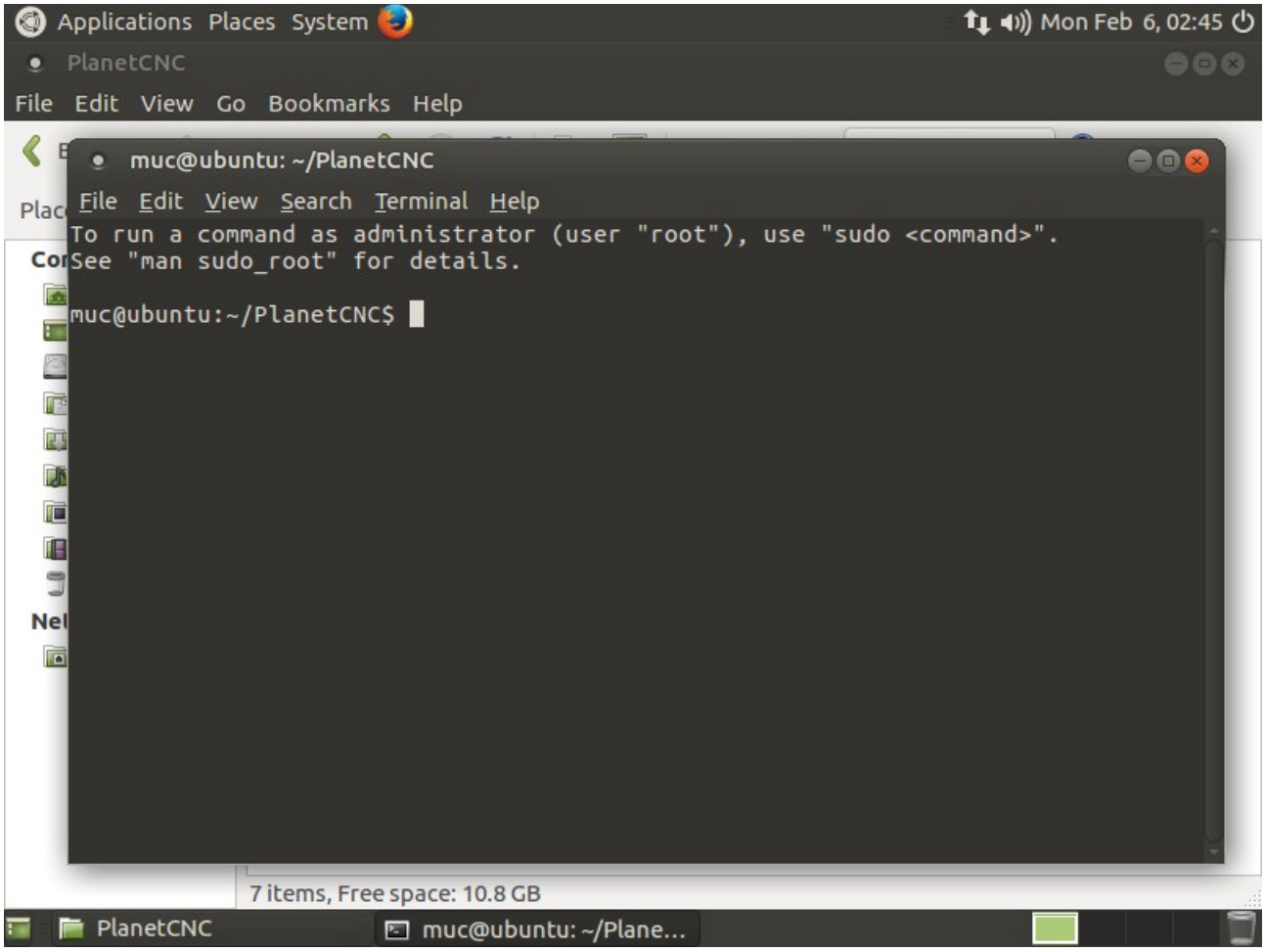
Extracted files will now populate PlanetCNC folder:



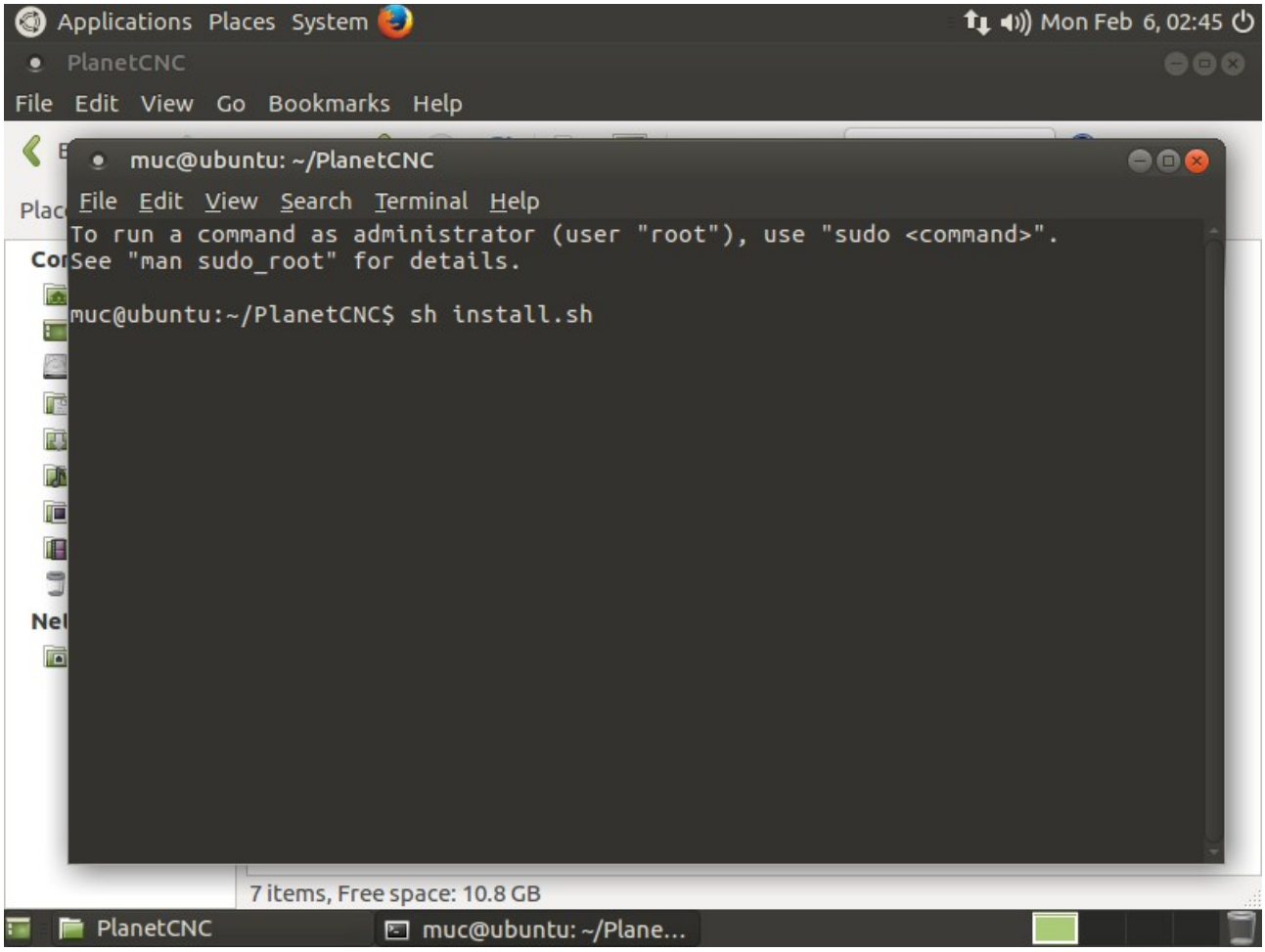
Right mouse click on blank space and click: *Open in Terminal*



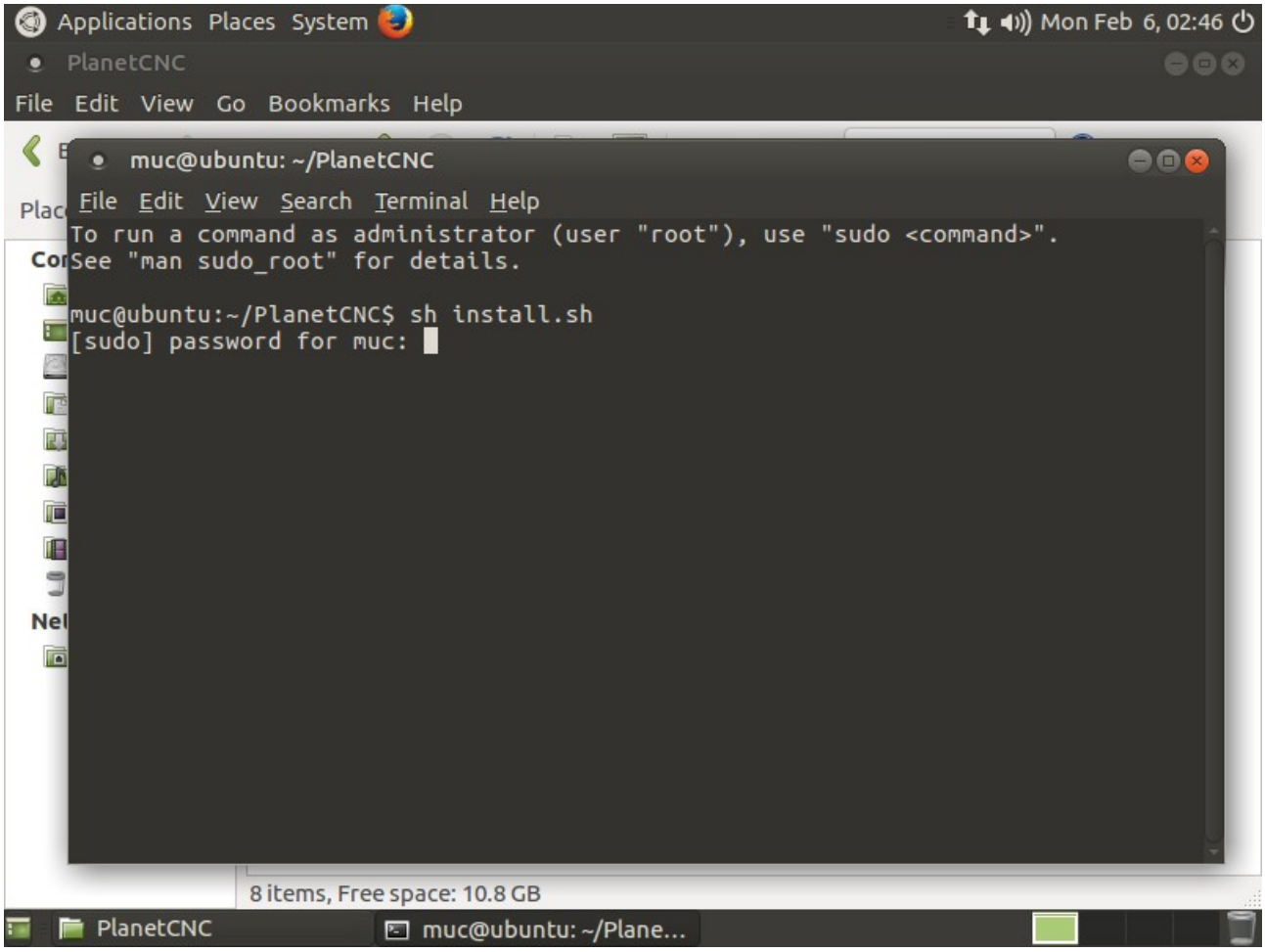
Terminal window will appear:



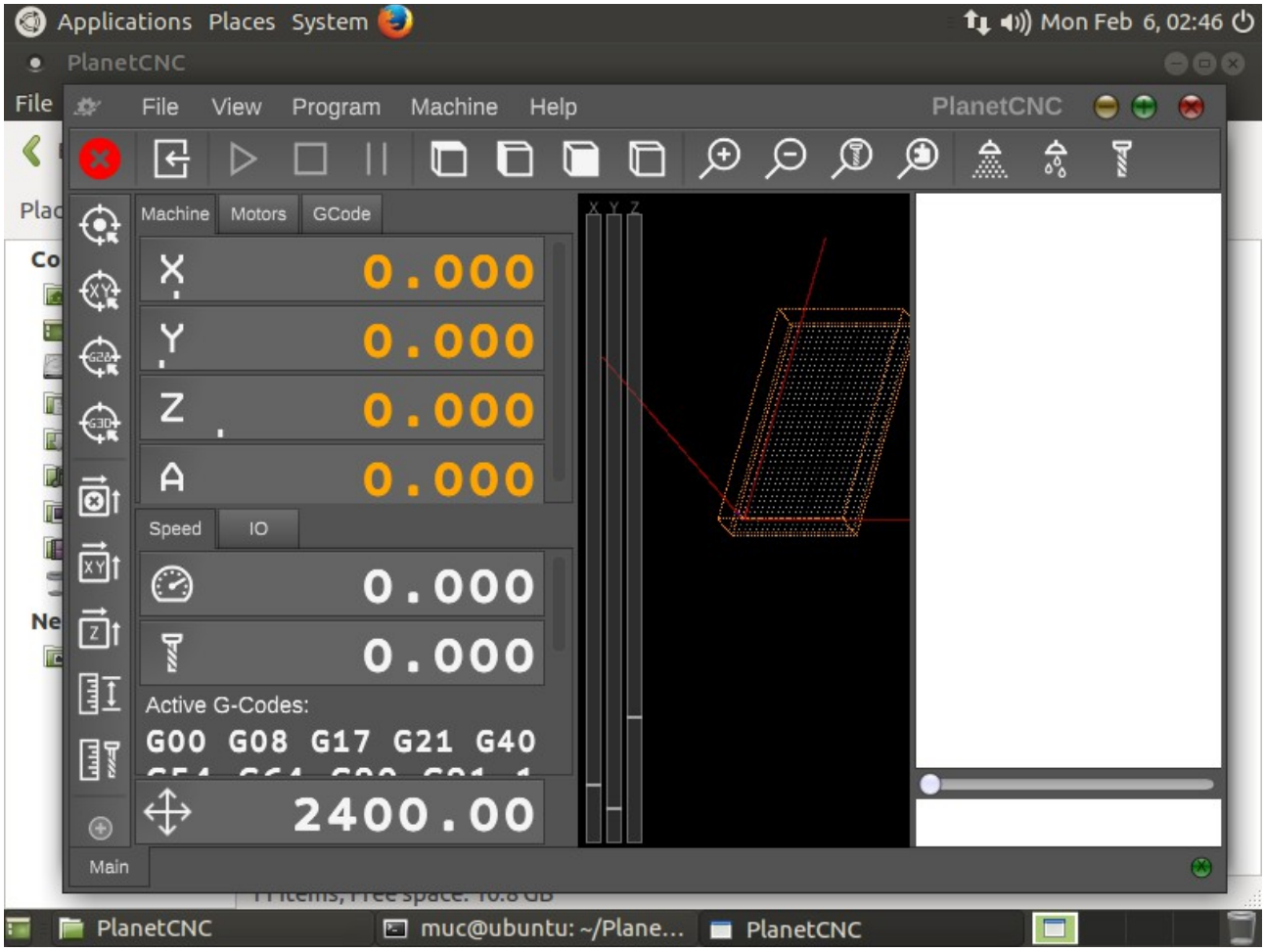
Write: *sh install.sh*



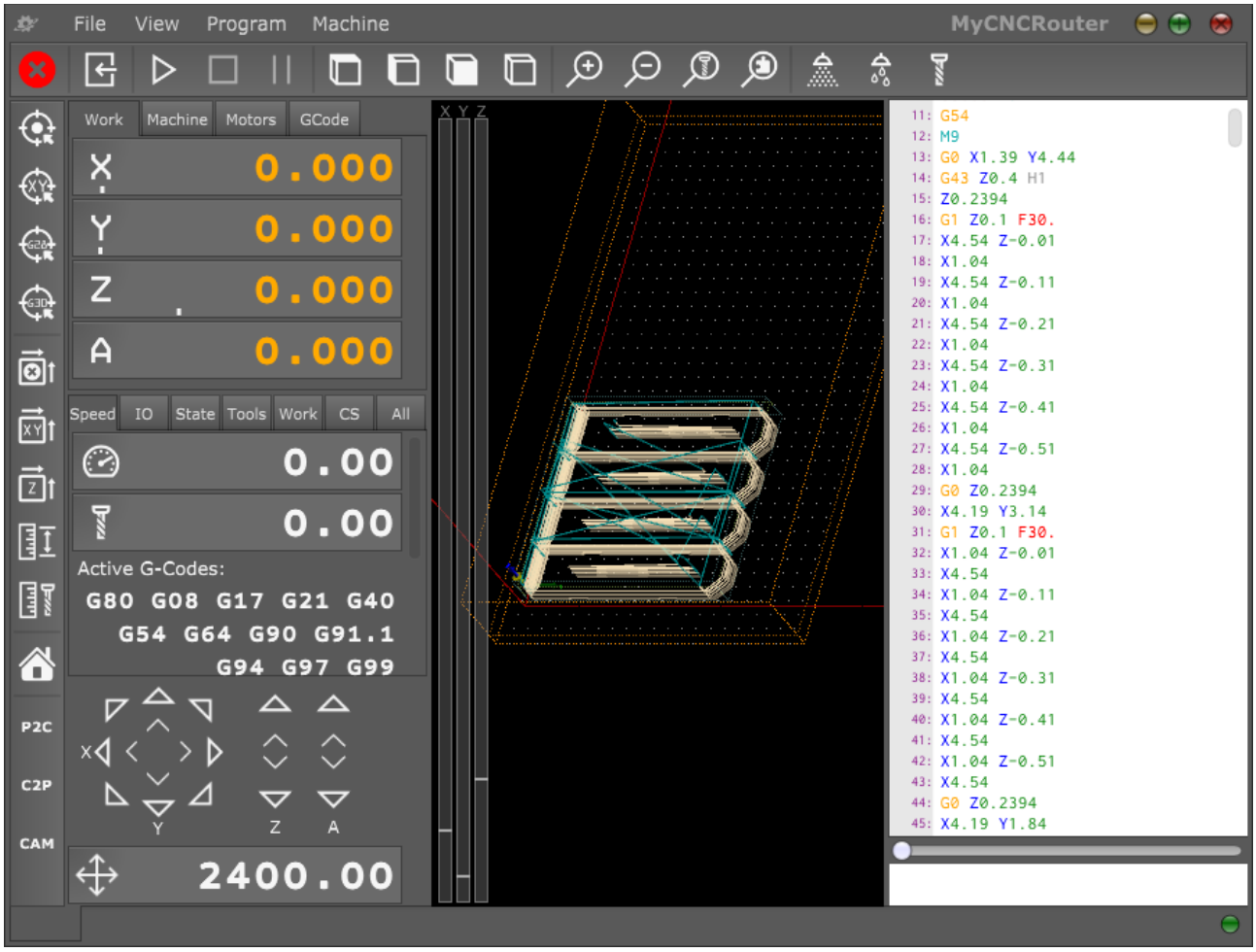
Type in your root password and hit enter.



PlanetCNC TNG software will automatically launch



2.3 Main Window



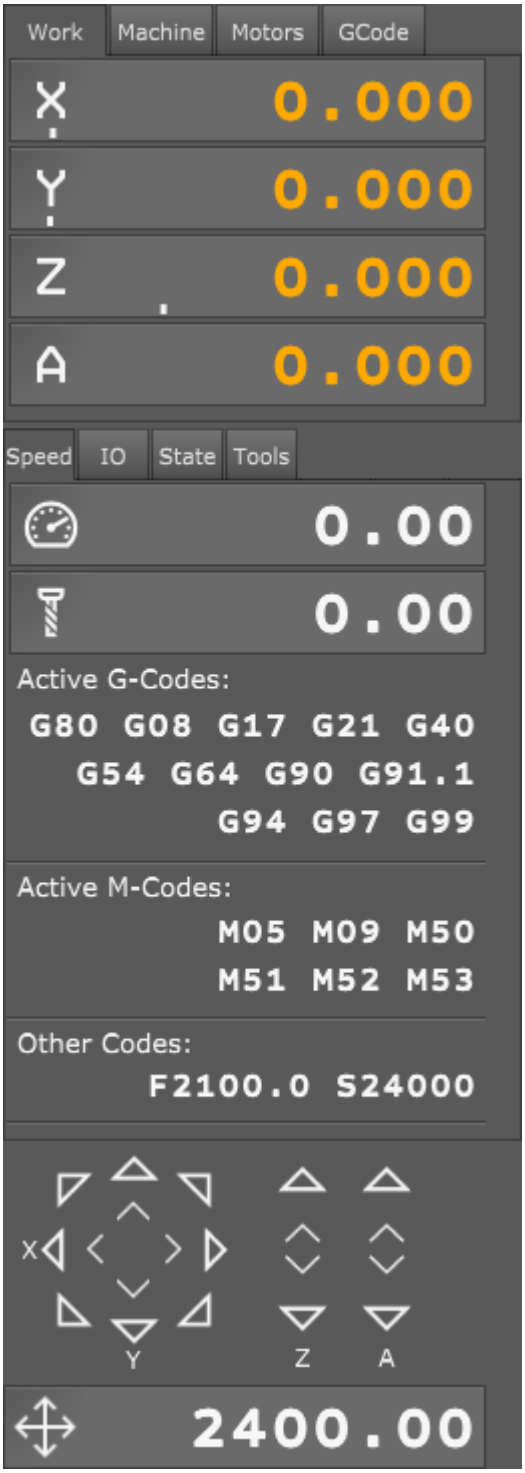
On launch of PlanetCNC TNG software *Main Window* will display.

There are seven areas within the window. Each provides information or function concerning specific aspects of G-Code program execution, purposes of illustration or displaying machines or controllers state.

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

2.3.1 Position/Status Panel

Position panel layout consists of position panel, status panel and jogging panel.



2.3.1.1 Position Panel

Position panel uses 4 tabs: *Work, Machine, Motors, Gcode.*

2.3.1.1.1 Work

This tab displays work or relative position coordinates of machine.

2.3.1.1.2 Machine

This tab displays machine or absolute position coordinates of machine.

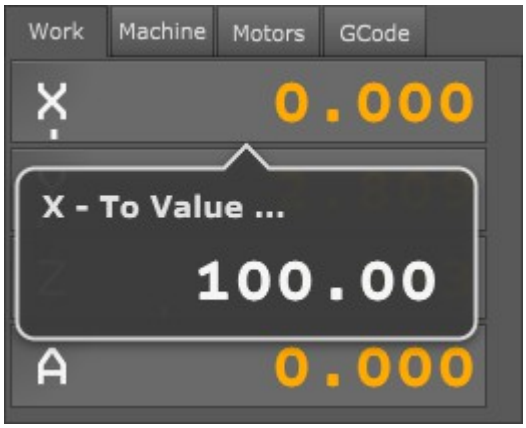
2.3.1.1.3 Motors

This tab displays motor position coordinates.

2.3.1.1.4 GCode

This tab displays positions coordinates of selected g-code line.

User can set new *Work* or *Machine* position value by double clicking on the axis value of Work or Machine tab. *To Value...* insert filed will appear:



2.3.1.2 Status Panel

Status panel uses two tabs by default, *Speed* and *IO*.

2.3.1.2.1 Speed Tab

Speed tab displays machine speed parameters and active G-codes, M-codes and Other codes.

Speed display:

By clicking speed dial button  additional feed speed options appear:

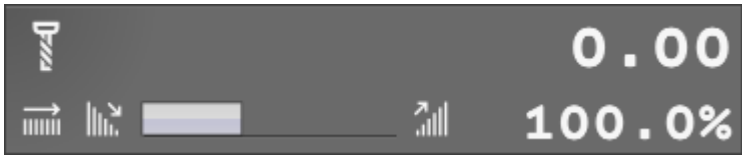
User can set new feed speed override value using buttons:

Decrease  **:** Decreases speed override value for 10%.

Increase  **:** Increases speed override value for 10%.

Reset  **:** Resets any speed override.

By clicking spindle button  , additional spindle speed options appear:



User can set new spindle speed override value using buttons:

Decrease  **:** Decreases speed override value for 10%.

Increase  **:** Increases speed override value for 10%.

Reset  **:** Resets any speed override.

Regarding override please also read chapter: *2.8.3.23.3.*

Active G-Codes:

Displays all currently active G-codes.

Active M-Codes:

Displays all currently active M-codes.

Other Codes:

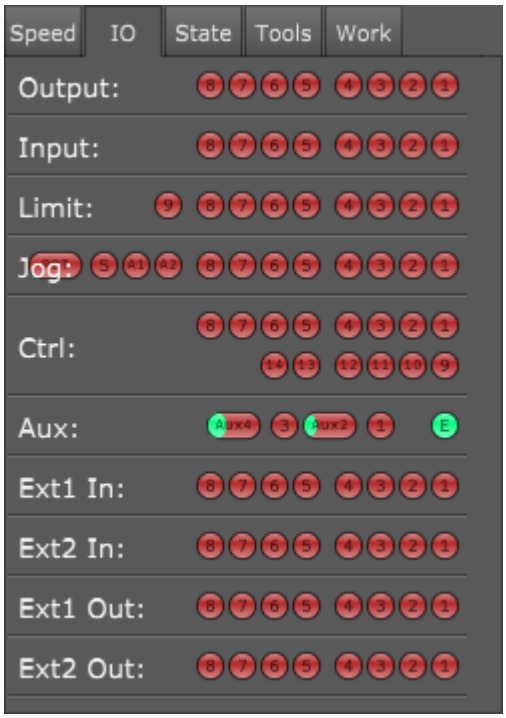
Displays all currently active G-codes such as F-word, S-word g-codes and current tool number.

2.3.1.2.2 IO Tab

IO tab displays status of controllers input and output pins.

Additional 5 tabs can be added(in picture below 3 additional tabs were added).

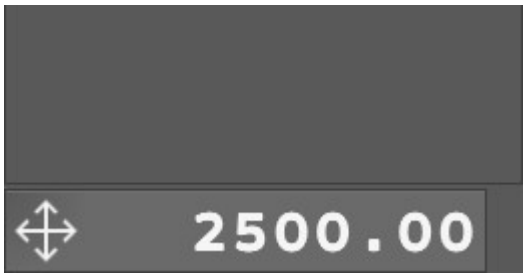
User can populate tabs with desired parameters.



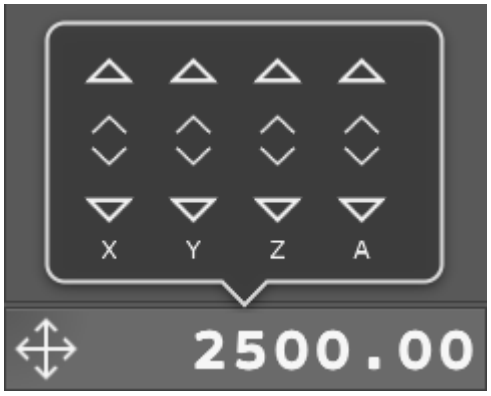
2.3.1.3 Jogging Panel

Jogging panel offers jog buttons to move machine to desired position.

Jogging layout is hidden by default:



by clicking cross-arrow button, jogging panel will appear:



Mouse click+hold on large arrow jog buttons to jog machine in desired direction. Machine will move 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.

New jogging speed can be set by double clicking on the jogging speed value, *Set Jog Speed* insert filed will appear:

In settings (chapter *2.8.3.11*), user can change jogging panel layout as also change hidden behavior to fixed(jogging panel is always displayed on main window):

2.3.2 Toolbars

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

Move/To Zero:

Moves the machine XY and Z axis to zero position.

Move/Axis To Zero/XY:

Moves the machine X and Y axis to zero position.

Move/To G28:

Moves machine to the absolute G28 position as set with *Machine/Absolute Position/As G30*.

Move/ To G30:

Moves machine to the absolute G30 position as set with *Machine/Absolute Position/As G30.*

Work Offset/To Zero:

Sets working offset to zero.

Work Position/Axis To Zero/XY:

Sets the current XY position of machine as zero XY work position.

Work Position/Axis To Zero/Z:

Sets current Z position of machine as zero Z work position.

Work Position/Measure:

Measures Z working position at current machine position, using movable tool sensor.

Tool Offset/Measure:

Measures tool length using a fixed tool sensor.

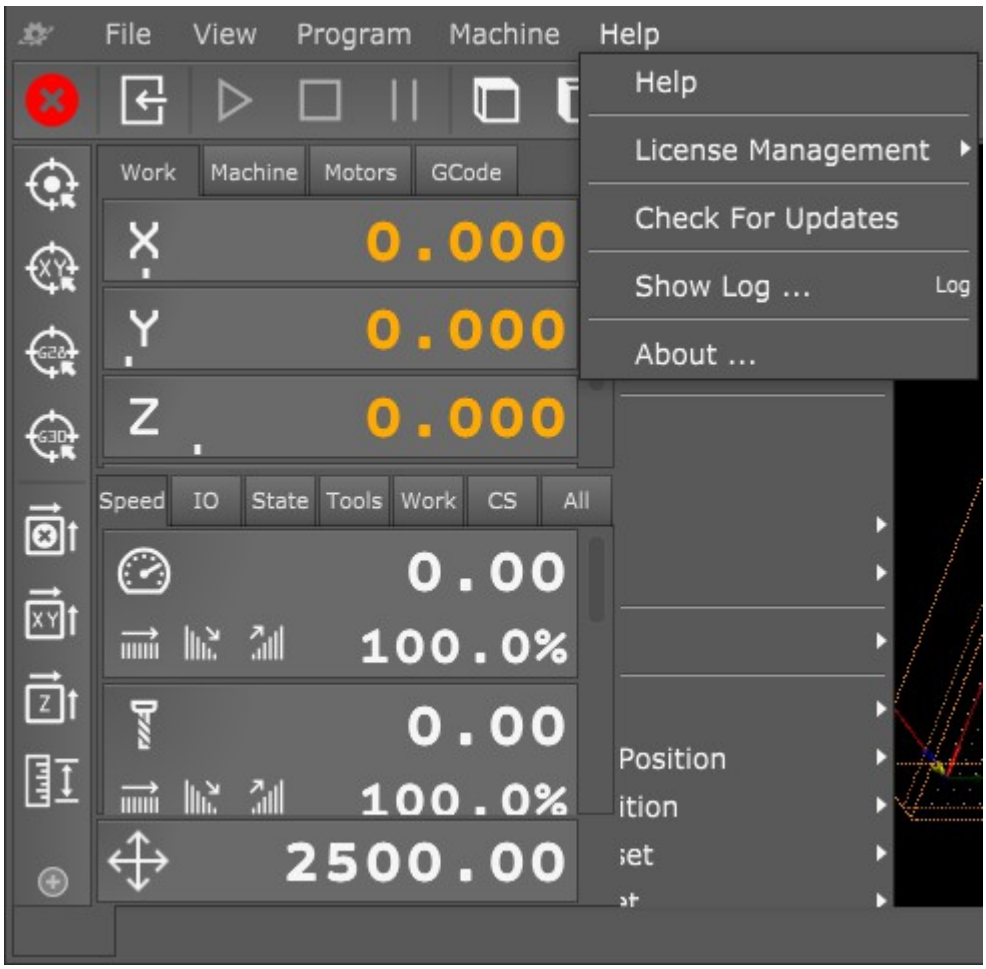
Home:

Initiate automatic homing procedure.

2.3.3 Menu bar

Menu bar populates *File*, *View*, *Program*, *Machine* and *Help* menus.

To activate specified menu, click on the menu bar item.



2.3.4 Manual data input (MDI) window

MDI window allows user of manual G-code input and execution of MDI shortcuts.

Example (manual G-code input):

```
G53 X0 Y0 Z10
```

Typing G53 X0 Y0 Z10 will move machine to its absolute position of X0 Y0 Z10.

The MDI window can also be used for execution of shortcuts using MDI codes(see chapter *2.8.3.23.3*).

Example (MDI code):

```
/Log
```

In settings, shortcut MDI code named *Log* was set for *Show Log* action.

Typing */Log* into MDI window will open *Show Log* dialog.

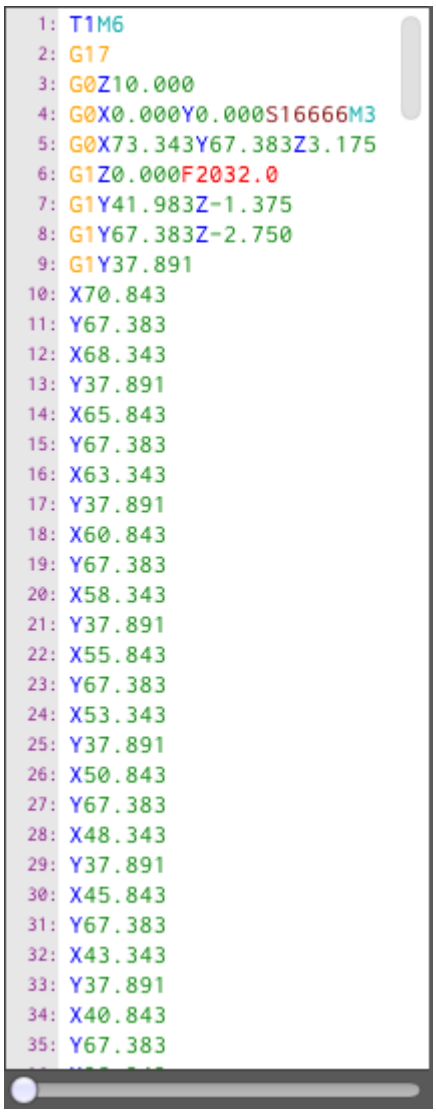
Please note: Symbol “/” before MDI code needs to be used.

Line break can be created with keyboard key combination*Shift+Enter.* This way user can input multi-line g-code command:

```
G53 X0 Y0 Z10
G00 X10 Y10
G01 Z-5
```

2.3.5 G-Code panel

G-code panel displays current program g-code lines.



2.3.5.1 Vertical Slider

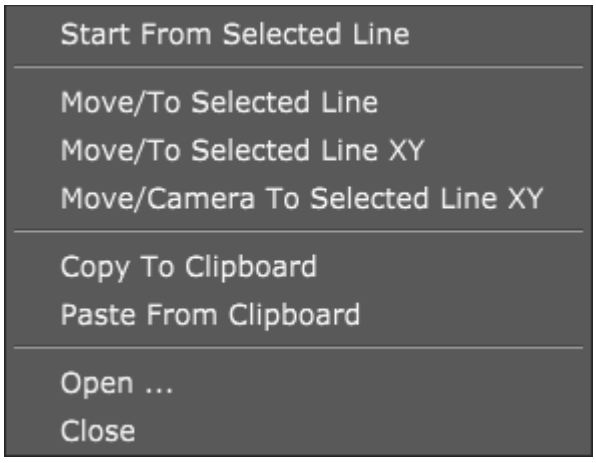
At the right side of G-code panel is vertical slider. Dragging slider up or down user can navigate trough pages of current g-code program.

2.3.5.2 Position Slider

At the bottom of G-Code panel is position slider. Dragging slider left or right, user can navigate trough program lines and observes progress of toolpath simulation in program visualization display.

2.3.5.3 Additional G-code panel options

Right mouse click on selected g-code line opens a dialog with additional options:



2.3.5.3.1 Start From Selected Line

Program will start at position of selected program g-code line.

2.3.5.3.2 Move/To Selected Line

See chapter *2.6.12.6*

2.3.5.3.3 Move To Selected Line XY

See chapter *2.6.12.7*

2.3.5.3.4 Move/Camera To Selected Line XY

See chapter *2.6.12.10*

2.3.5.3.5 Copy To Clipboard

See chapter *2.5.16*

2.3.5.3.6 Paste From Clipboard

See chapter *2.5.17*

2.3.5.3.7 Open...

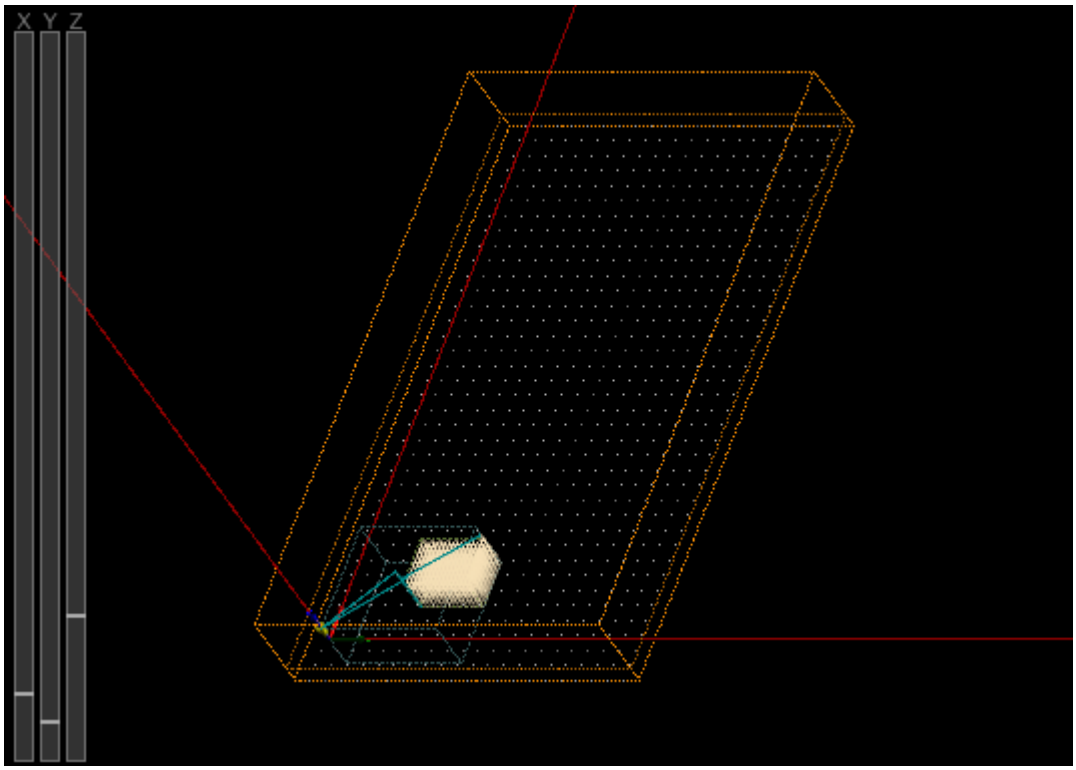
See chapter *2.3.2*

2.3.5.3.8 Close

See chapter *2.3.1*

2.3.6 Program visualization

Shows 3D display of g-code program.



2.3.7 Indication Light

In the bottom right corner of main window is connection light that indicates controllers status concerning license activation and connection with PlanetCNC TNG software.

2.3.7.1 Connection light color description:

Green light: 

Indicates that controller is updated to correct firmware version and controller is activated meaning license is found by software:

Green light with X: 

Indicates that software does not find proper license for connected controller. Make sure that your PlanetCNC TNG license is correctly imported.

Orange light with X: 

Indicates that controllers firmware version is not correct.

Gray light with X: 

Indicates no communication between controller and software. Click *Machine/Controller/Reconnect* and make sure that correct controller is set as *Primary controller* in settings.

Red light with status bar: 

Indicates that software is processing motion commands and is sending them to controller.

2.4 File menu

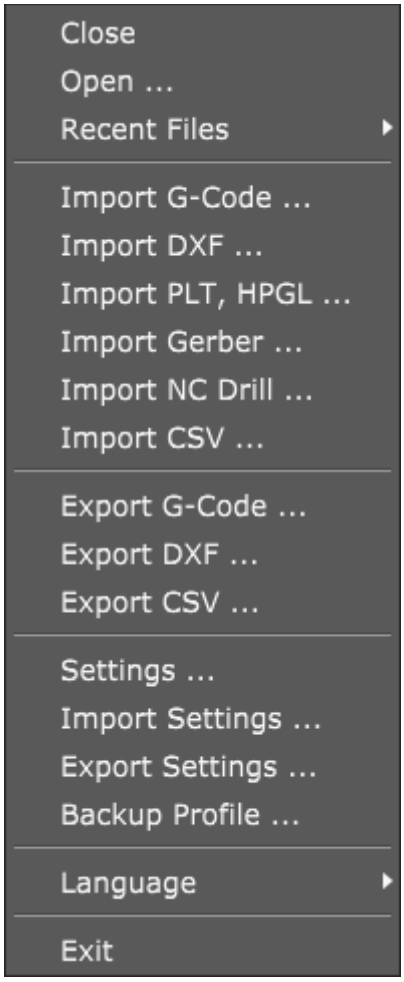
File menu offers a group of methods for opening, importing and exporting of machine programs.

Programs can be used for visualization, simulation, generating toolpath and can be in various formats.

Import features display user dialogues for entry of user parameters. Description of features is provided below.

G-Code can be exported using an option best suited to user requirement.

‘Settings’ is where we configure and set machines main parameters.



2.4.1 Close

Closes program that is currently opened.

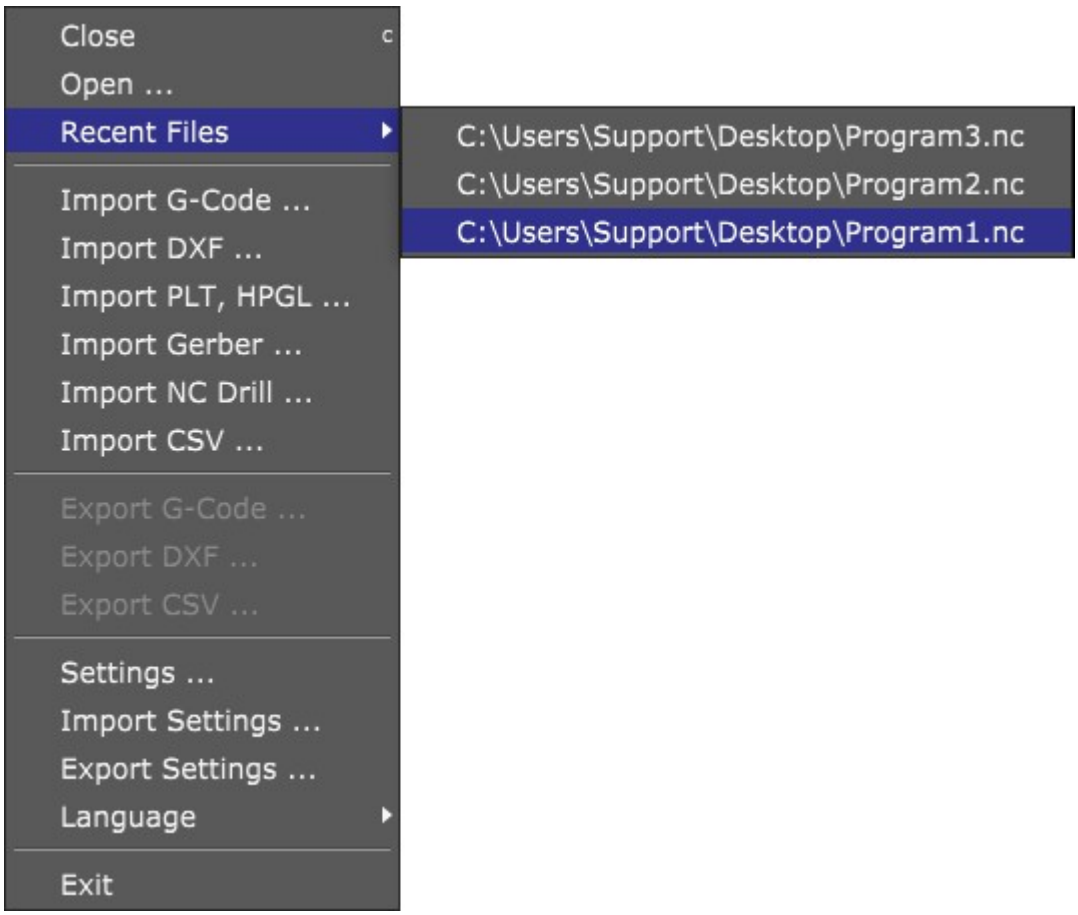
2.4.2 Open

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

Example: If you want to open .dxf file, TNG software will automatically recognize the DXF file format and launch DXF import dialog.

2.4.3 Recent files

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



2.4.4 Import G-code

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

2.4.5 Import DXF...

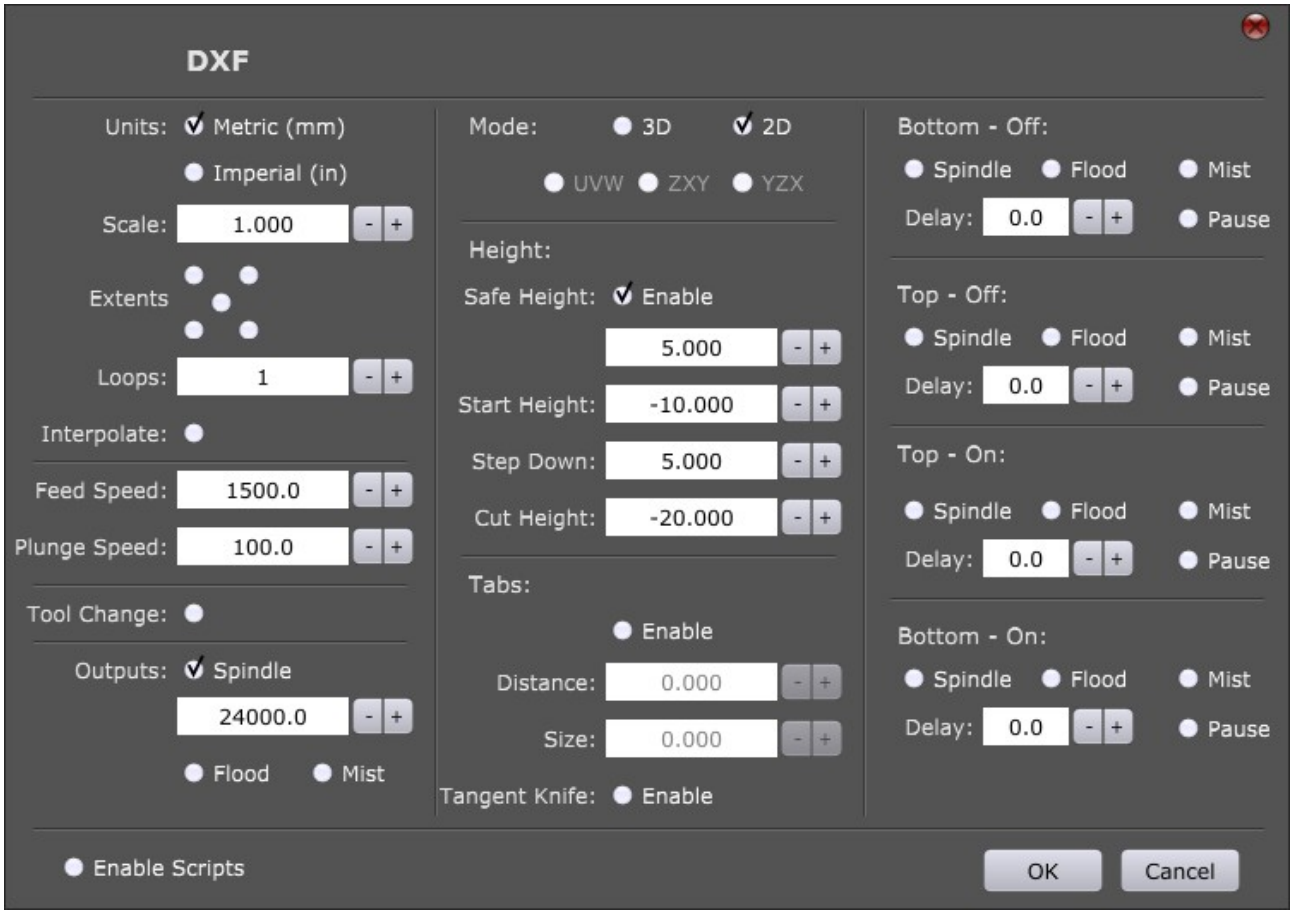
and

2.4.6 Import PLT, HPGL...

Imports program in DXF and PLT/HPGL format. Most software's for CAD drawing or vector images have option to save design in DXF format.

These types of format usually contain vectors, that can be converted to toolpath. PlanetCNC TNG software will automatically generate g-code program based on your imported DXF file.

When using **Import DXF** feature, user dialog will be displayed. User has option to configure program parameters to suit his machining needs.



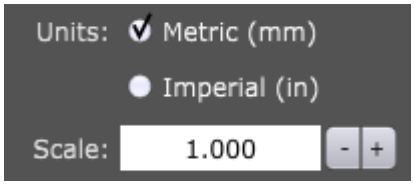
2.4.6.1 Units

You can set units as **Metric(mm)** or **Imperial (in)** for your DXF design. You can fine tune your units using *Scale* option.

2.4.6.2 Scale

Sets scale of your imported DXF design. This comes handy when you need your toolpath to be re-sized or some other units are used.

E.g.: If you DXF design is drawn in centimeters (cm) then select *Units* as *Metric* and set S*cale* to 10.

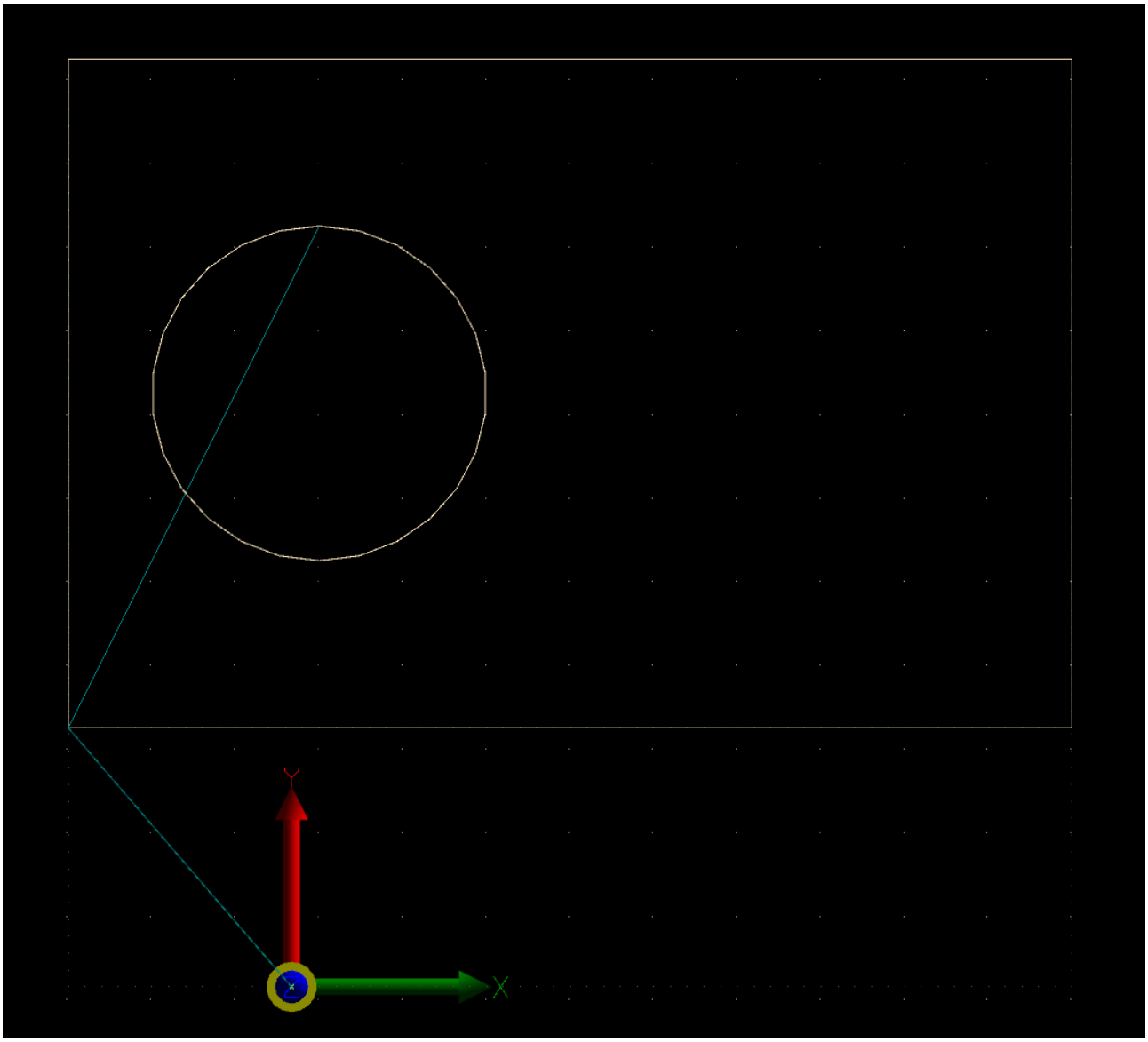


2.4.6.3 Extents

Example:

Let say we import DXF file of which we do not know its absolute XY 0,0 coordinates.

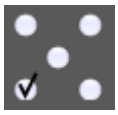
Generated toolpath will be positioned accordingly, but not necessarily in a way that would suit us :



Generated toolpath is positioned in relation to its absolute XY 0,0 coordinates (the way it was drawn). But what if we want that toolpath extents are aligned with working position XY0,0?

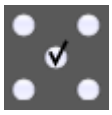
For this purpose you can use one of five options under *Extents:*

-Bottom left :



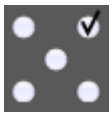
-Top Left:

-Center:



-Bottom right:

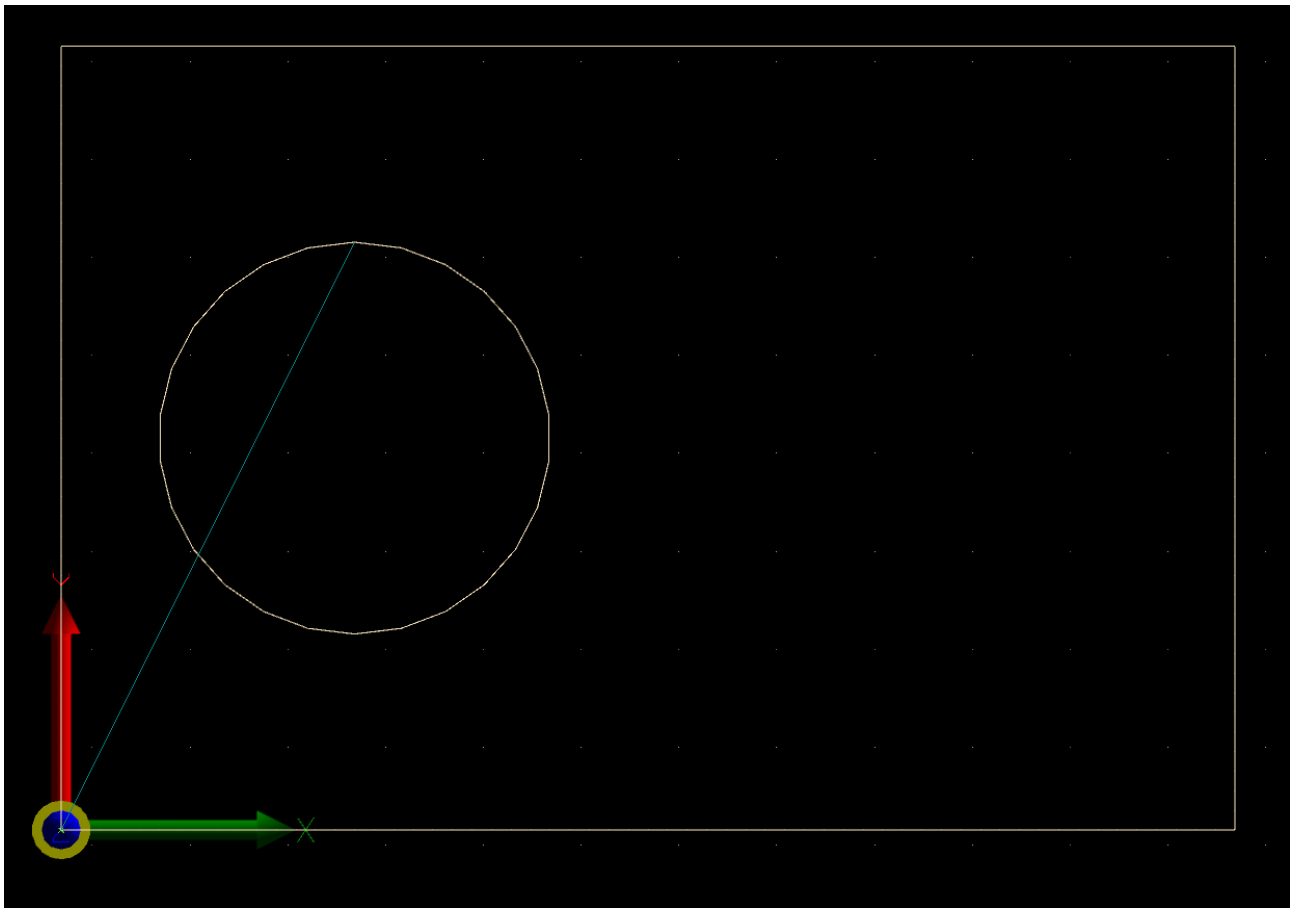
-Top right:



Example:

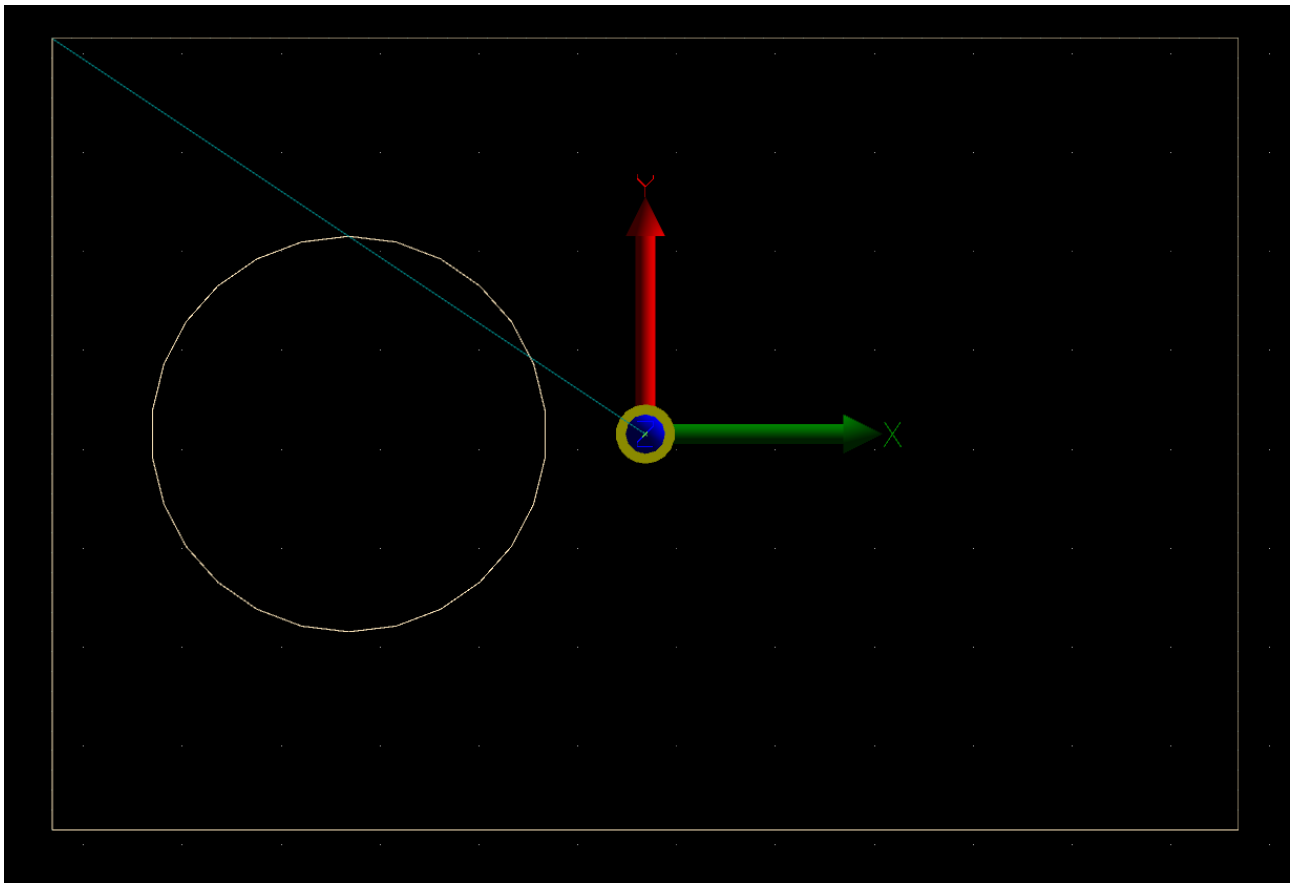
Bottom Left: 

With this option enabled, toolpath XY0,0 point will be aligned with our working offset XY0,0:



Center: 

With this option enabled, toolpath center point will be aligned with our working offset XY0,0:



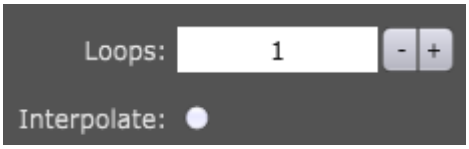
2.4.6.4 Loops

You can set number of loops of your program. Each generated toolpath pass will be repeated for inserted value of loops.

Example: If you insert *Loop:3,* pass will be repeated for 3 iterations.

2.4.6.5 Interpolate

If your DXF file contains elements such as circles, arcs etc.. you can interpolate these elements into short lines by enabling *Interpolate* option.

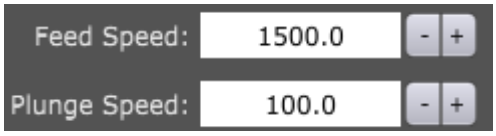


2.4.6.6 Feed Speed

Sets feed speed for generated toolpath. F-word g-code will be generated. Each feed move will be performed at this speed.

2.4.6.7 Plunge Speed

Set feed speed of plunge moves for generated toolpath. Each feed move in Z- direction (plunge) will be performed at this speed.



2.4.6.8 Tool change

Generated program will include tool change commands.

Please note: If DXF file uses layers, each layer will represent different tool number in generated program. Tn M6 and G43 Hn g-codes will be generated for specified tool.

2.4.6.9 Outputs

Generated program will include *M3/M5;M7,M8/M9* spindle and coolant g-code commands, depending on options selected.

Please note: If DXF file uses layers then outputs will be turned ON at beginning and turned OFF at end of layers toolpath.

Spindle:

With this option enabled, generated program will include M3/M5 g-codes.

If layers are used in DXF file, layers toolpath will include M3 g-code at start and M5 g-code at the end.

If no layers are used in DXF design, spindle g-codes will be generated only at the beginning of program and at the end.

Flood:

With this option enabled, generated program will include M7/M9 g-codes.

If layers are used, layers toolpath will include M7 g-code at the start and M9 g-code at the end.

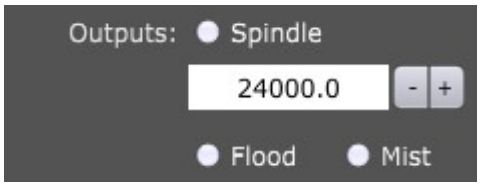
If no layers are used, Flood g-codes will be generated only at the beginning of program and at the end.

Mist:

With this option enabled, generated program will include M8/M9 g-codes.

If layers are used, layers toolpath will include M8 g-code at the start and M9 g-code at the end.

If no layers are used, Mist g-codes will be generated only at the beginning of program and at the end.



2.4.6.10 Enable scripts

Enable Scripts provides means to personalize and achieve advanced g-code generation when using Import DXF feature.

Automatically generated g-code of *DXF Import* feature is not necessarily within users requirements. User can tweak the DXF Import importation script to achieve desired results.

2.4.6.11 Mode

Your DXF design can be in 2D or 3D. If **2D** mode is selected, *Height* options will be enabled and you will be able to configure height cutting parameters of generated toolpath.

If **3D** mode is enabled, Height options will be disabled since it is assumed that are already defined in original DXF (however, Safe Height option is available).

3D mode also enables user to translate g-code from conventional XYZ plane to UVW, ZXY or YZK plane:

XYZ-> UVW

With this option selected, generated g-code program will translate XY coordinates to UV coordinates. This feature is useful for foam cutters, where second tower uses UV coordinates for its motion.

```
G01 X28.573216 Y59.949042 U28.573216 V59.949042 F1500.0
G01 X27.153703 Y59.796429 U27.153703 V59.796429
G01 X25.748694 Y59.542937 U25.748694 V59.542937
G01 X24.365349 Y59.189859 U24.365349 V59.189859
G01 X23.010716 Y58.738994 U23.010716 V58.738994
G01 X21.6917 Y58.19264 U21.6917 V58.19264
G01 X20.41502 Y57.55358 U20.41502 V57.55358
G01 X19.187184 Y56.825071 U19.187184 V56.825071
G01 X18.014447 Y56.010825 U18.014447 V56.010825
```

XYZ->ZXY

With this option selected, generated g-code program will be in ZX plane. XYZ coordinates from DXF will be translated to ZXY.

```
G01 X59.949042 Z28.573216 F1500.0
G01 X59.796429 Z27.153703
G01 X59.542937 Z25.748694
G01 X59.189859 Z24.365349
G01 X58.738994 Z23.010716
G01 X58.19264 Z21.6917
G01 X57.55358 Z20.41502
G01 X56.825071 Z19.187184
G01 X56.010825 Z18.014447
```

XYZ->YZX

With this option selected, generated g-code program will be in YZ plane. XYZ coordinates from DXF will be translated to YZX.

```
G01 Y28.573216 Z59.949042 F1500.0
G01 Y27.153703 Z59.796429
G01 Y25.748694 Z59.542937
G01 Y24.365349 Z59.189859
G01 Y23.010716 Z58.738994
G01 Y21.6917 Z58.19264
G01 Y20.41502 Z57.55358
G01 Y19.187184 Z56.825071
G01 Y18.014447 Z56.010825
```

2.4.6.12 Height

2.4.6.12.1 Safe Height

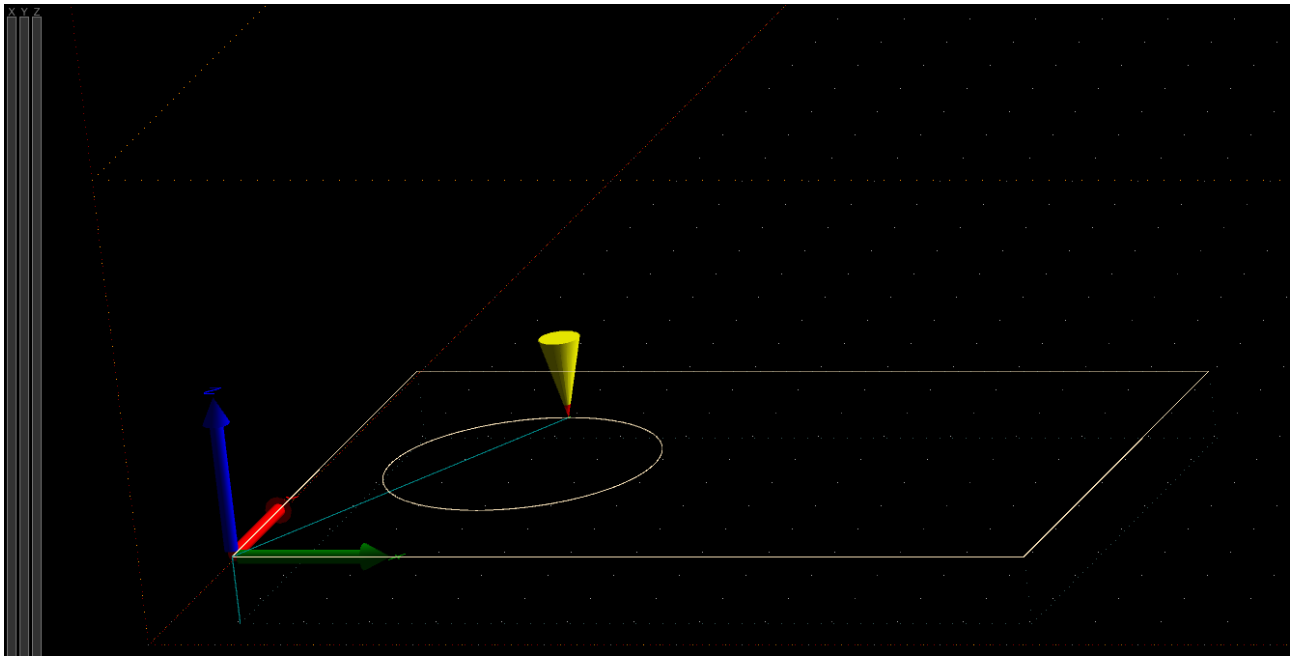
When *2D* mode is used then *Height* parameters can be set.

Safe height is a safety feature which helps with prevention of machine crashing into obstacles that may interfere with machines toolpath. Obstacles could be screws, fixtures, vises etc..

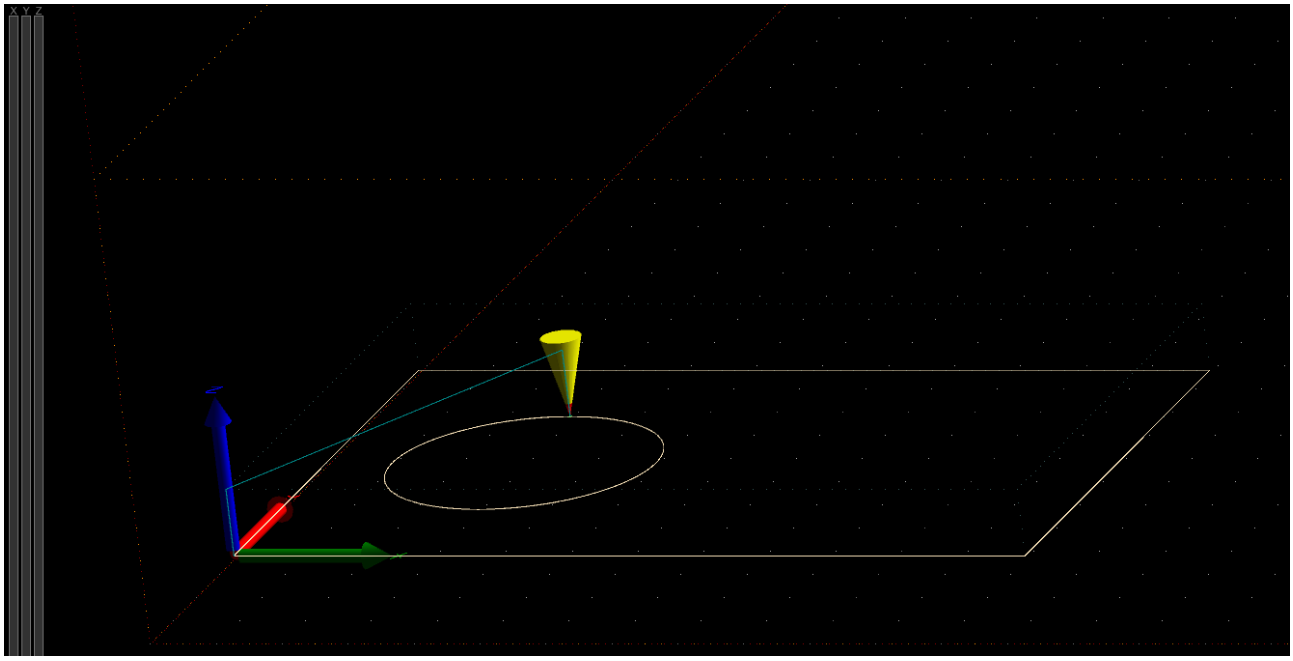
When machine is finished with cutting toolpath A, it will ascend to safe height and move to next cutting location of toolpath B.

With this option enabled, generated toolpath will include traverse moves performed at safe height.

Toolpath without safe height:



Toolpath with safe height:



2.4.6.12.2 Start Height

Start height is usually surface of workpiece material. To this height, machine will descend at traverse rate.

2.4.6.12.3 Step Down

Depth of first cutting pass. Each new cutting pass will be deeper for this value. To this height, machine will descend at plunge rate.

2.4.6.12.4 Cut Height

Deepest cutting depth that machine will cut at.

NOTE: Software will automatically calculate number of passes to achieve “Cut Height” depth at “Step Down” value per pass.

Example:

Let's say we want to cutout a square out of a solid wooden block of thickness 30mm.

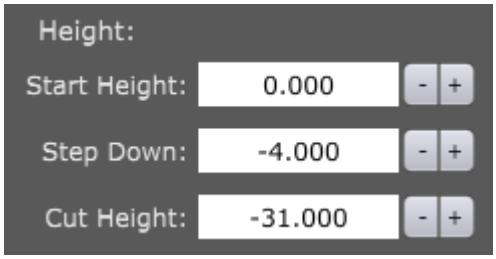
We would like to cut in passes, with each pass being 4mm deeper than the previous one. To successfully cutout our square, deepest cut should be performed at 31mm.

Set work position Z = 0 is at surface of wooden block.

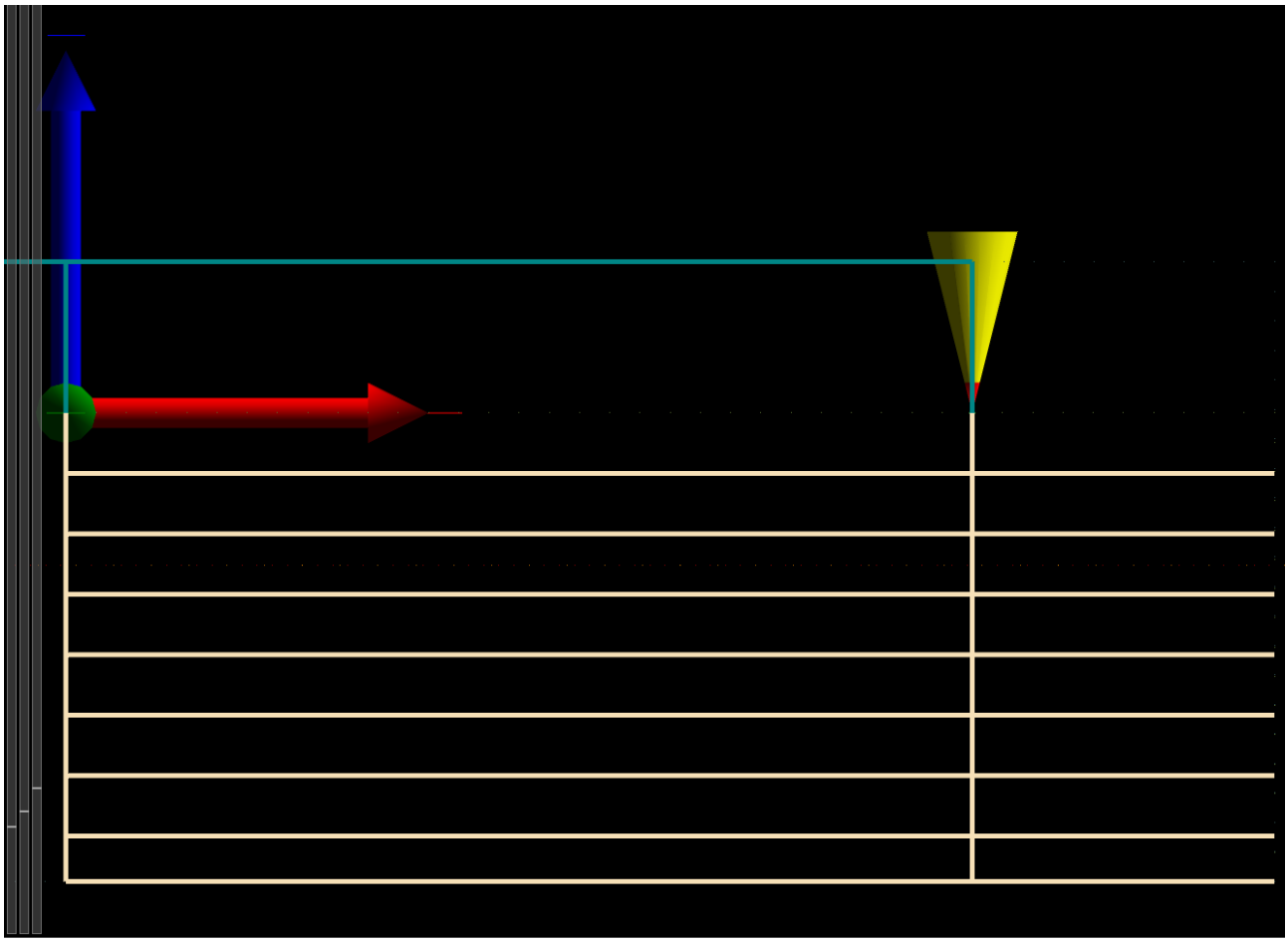
Start Height = 0

Step Down = -4

Cut Height= 31

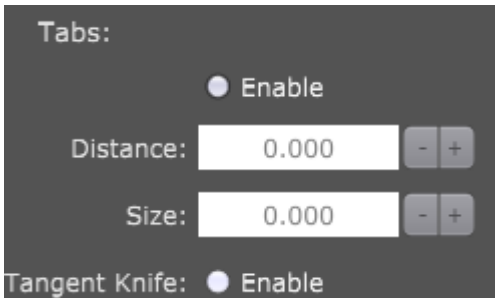


Bottom picture displays side view of generated toolpath based on parameter configuration described above:



2.4.6.13 Tabs

Tabs are used for holding element in place during cut.



Enable:

Enables tabs.

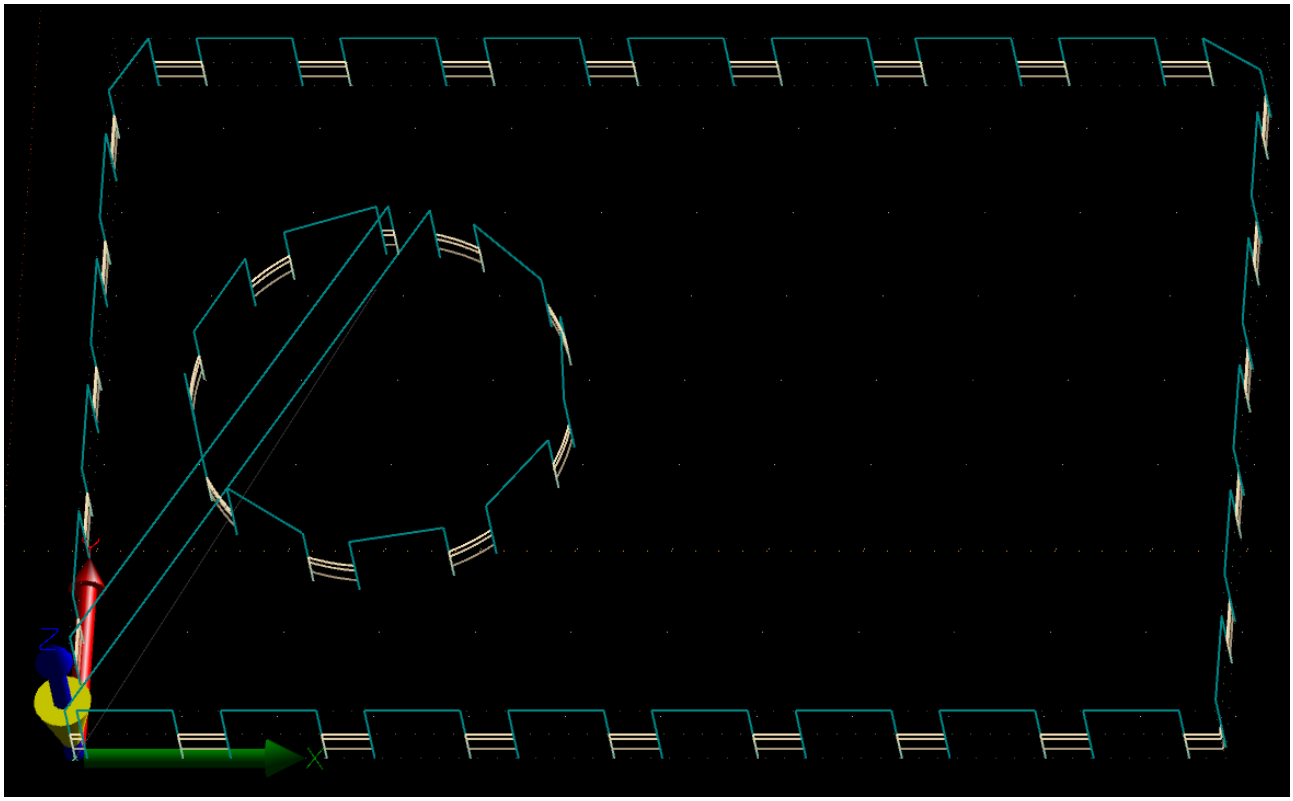
Distance:

Distance between two tabs.

Size:

Size of tabs.

Example of toolpath with tabs enabled:



2.4.6.14 Tangent Knife

Enable:

Enables C axis movement in direction of toolpath for use with tangential knives. Safe Height moves are generated if required.

2.4.6.15 Bottom - Off

Inserts OFF g-codes for Spindle, Flood, Mist (M5,M9), Delay and Pause at the end of cut before moving up to Safe Height

Spindle:

Inserts OFF g-code for Spindle M5.

Flood:

Inserts OFF g-code for Flood M9.

Mist:

Inserts OFF g-code for Mist M9.

Delay:

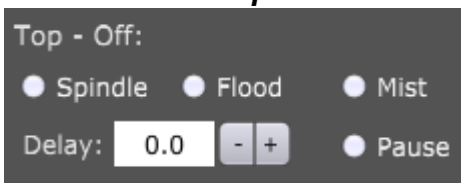
Inserts Delay G04 P g-code.

Pause:

Inserts Pause M00 g-code.

```
17: G01 Z-5.0 F100.0
18: G01 X120.0 F1500.0
19: G01 Y80.0
20: G01 X0.0
21: G01 Y0.0
22: M05
23: M09
24: G01 Z0.0
25: G00 Z10.0
```

2.4.6.16 Top Off



Inserts OFF g-codes for Spindle, Flood, Mist (M5,M9), Delay and Pause at the end of cut after moving up to Safe Height.

Spindle:

Inserts OFF g-code for Spindle M5.

Flood:

Inserts OFF g-code for Flood M9.

Mist:

Inserts OFF g-code for Mist M9.

Delay:

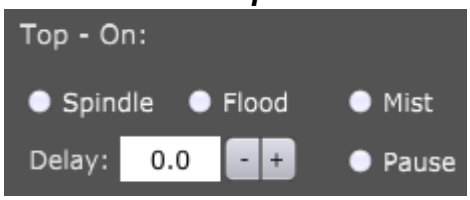
Inserts Delay G04 P g-code

Pause:

Inserts Pause M00 g-code

```
17: G01 Z-5.0 F100.0
18: G01 X120.0 F1500.0
19: G01 Y80.0
20: G01 X0.0
21: G01 Y0.0
22: G01 Z0.0
23: G00 Z10.0
24: M05
25: M09
```

2.4.6.17 Top On



Inserts ON g-codes for Spindle, Flood, Mist (M3,M7,M8), Delay and Pause before cut, before moving down from Safe Height to cut (or pass) height.

Spindle:

Inserts ON g-code for Spindle M3.

Flood:

Inserts ON g-code for Flood M7.

Mist:

Inserts ON g-code for Mist M8.

Delay:

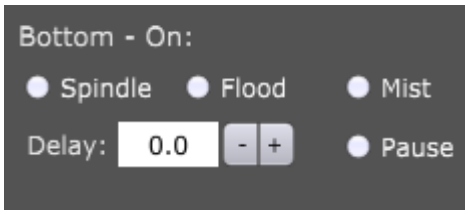
Inserts Delay G04 P g-code

Pause:

Inserts Pause M00 g-code

```
5: G00 Z10.0
6: ;  Layer: 0
7: ;  Feed Speed: 1500.0000
8: ;  Plunge Speed: 100.0000
9: ;  Safe Height: 10.0000
10: ;  Start Height: 0.0000
11: ;  Cut Height: -5.0000
12: G00 X0.0 Y0.0
13: M03 S24000.0
14: M07
15: M08
16: G00 Z0.0
17: G01 Z-5.0 F100.0
```

2.4.6.18 Bottom On



Inserts ON g-codes for Spindle, Flood, Mist (M3,M7,M8), Delay and Pause before cut, after moving down from Safe Height to cut (or pass) height.

Spindle:

Inserts ON g-code for Spindle M3.

Flood:

Inserts ON g-code for Flood M7.

Mist:

Inserts ON g-code for Mist M8.

Delay:

Inserts Delay G04 P g-code

Pause:

Inserts Pause M00 g-code

```
5: G00 Z10.0
6: ;  Layer: 0
7: ;  Feed Speed: 1500.0000
8: ;  Plunge Speed: 100.0000
9: ;  Safe Height: 10.0000
10: ;  Start Height: 0.0000
11: ;  Cut Height: -5.0000
12: G00 X0.0 Y0.0
13: G00 Z0.0
14: G01 Z-5.0 F100.0
15: M03 S24000.0
16: M07
17: M08
```

2.4.7 Import Gerber

Imports Gerber file. Gerber files are generated with software for design of printed circuit boards(PCB's). With gerber files you can also mill printed circuit boards with your CNC machine.

2.4.7.1 Feed Speed

Sets feed speed for generated toolpath. F-word g-code will be generated. Each G01 move will be performed at this speed.

2.4.7.2 Plunge Speed

Sets feed speed of plunge moves for generated toolpath. Each G01 move in Z- direction will be performed at this speed.

2.4.7.3 Safe Height

Safe height is a safety feature which helps with prevention of machine crashing into obstacles that may interfere with machines toolpath. Obstacles could be screws, fixtures, vises etc..

When machine is finished with cutting toolpath A, it will ascend to safe height and move to next cutting location of toolpath B.

With this option enabled, generated toolpath will include traverse moves performed at safe height.

2.4.7.4 Start Height

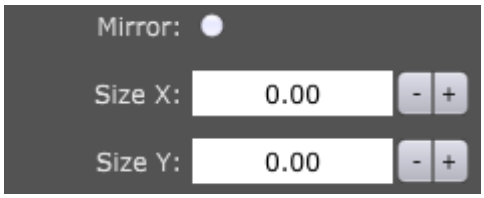
Start height is usually surface of workpiece material. To this height, machine will descend at traverse rate.

2.4.7.5 Tool change

Enables tool change for: Mark Pads, Cutting Path, Drill Pads or Clear Copper options.

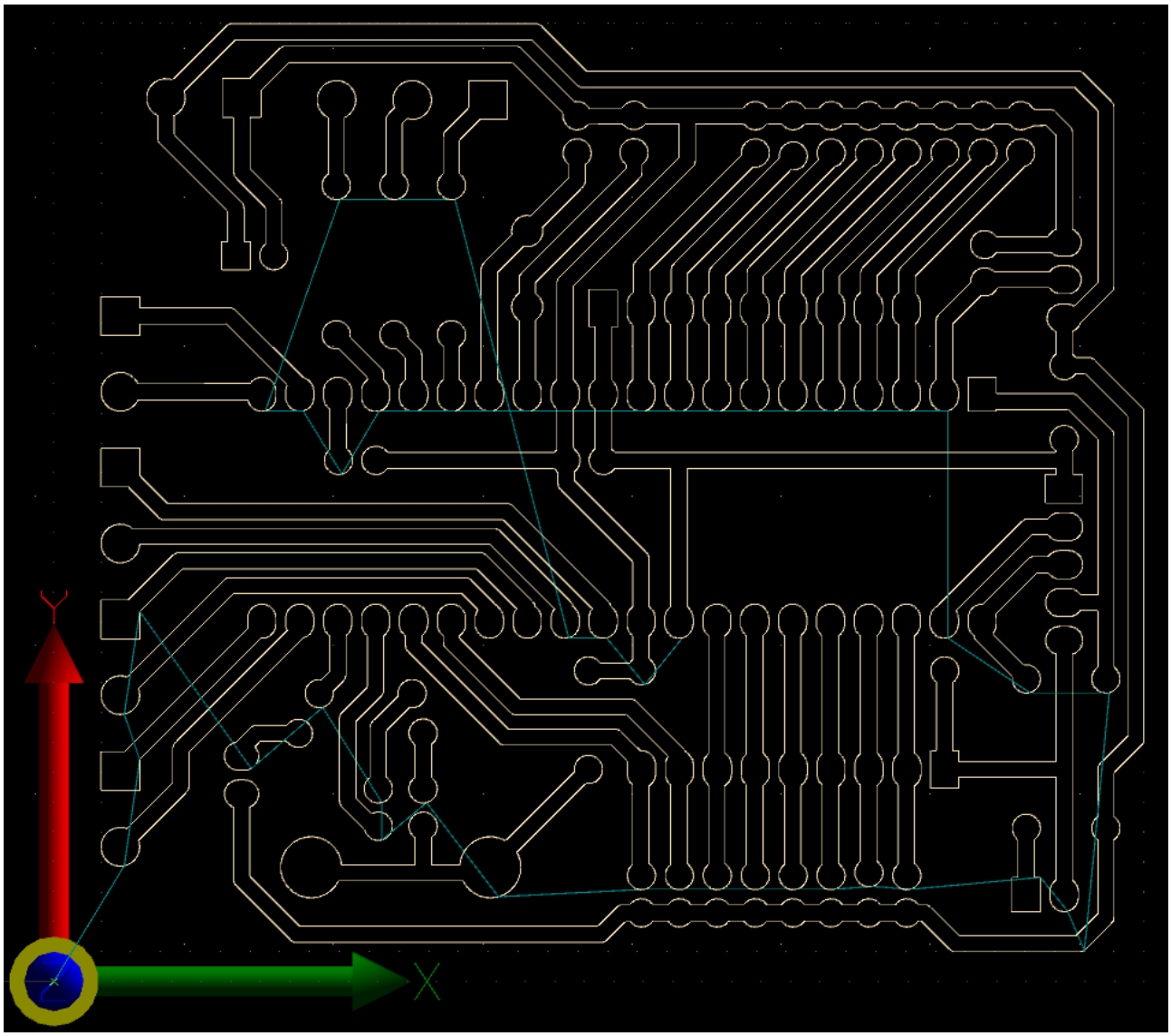
Please note: When configuring options Mark Pads, Cutting Path, Drill Pads or Clear Copper you need to enable them and define tool number.

2.4.7.6 Mirror

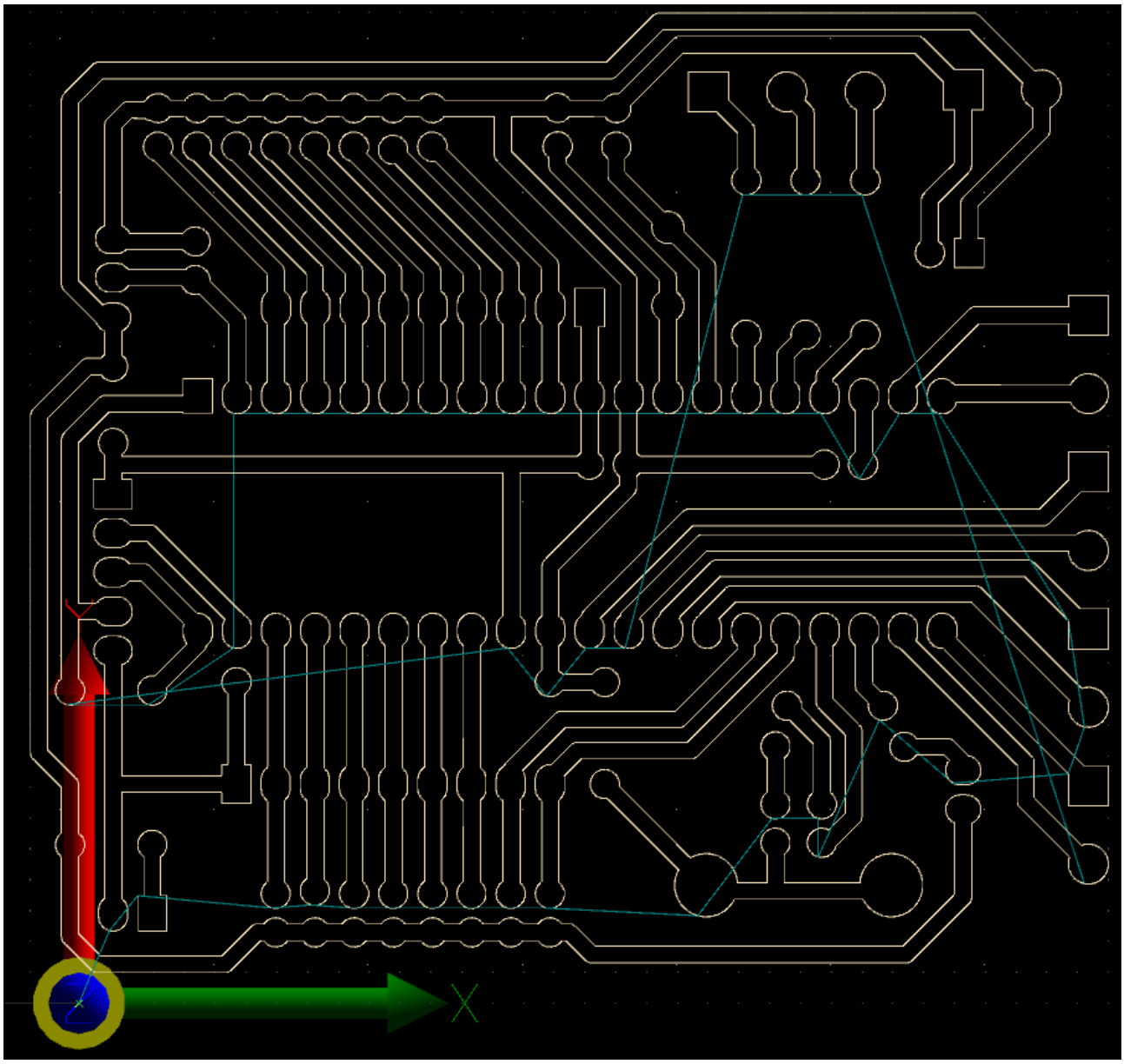


Mirrors program in XY.

Normal toolpath:



Mirrored toolpath:



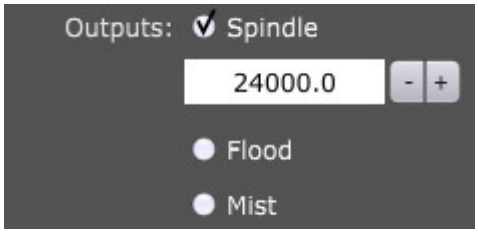
2.4.7.7 Size X

X coordinate approximate dimension of PCB board. This comes very useful when drilling holes of PCB using NC drill file. Make sure that same value is used with NC drill import.

2.4.7.8 Size Y

Y coordinate approximate dimension of PCB board.

2.4.7.9 Outputs



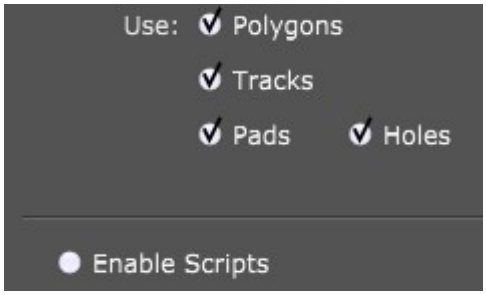
Generated program will include *M3/M5;M7,M8/M9* g-code commands, depending on options selected.

Spindle:
With this option enabled, generated program will include M3/M5 g-codes. If tool change is enabled M3/M5 g-codes will be generated for each toolchange.

Flood:
With this option enabled, generated program will include M7/M9 g-codes. If tool change is enabled M7/M9 g-codes will be generated for each toolchange.

Mist:
With this option enabled, generated program will include M8/M9 g-codes. If tool change is enabled M8/M9 g-codes will be generated for each toolchange.

2.4.7.10 Use



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

You can select which element(s) will be converted to toolpath.

Polygons:
Use 'Polygons' in toolpath calculation. Only polygons will be visible.

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

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

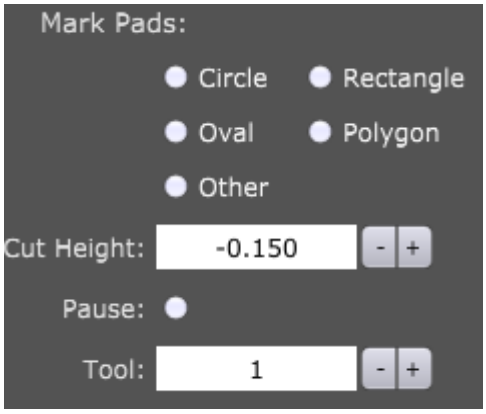
Holes:
Use 'holes' in toolpath calculation. Only holes will be visible.

2.4.7.11 Enable Scripts

Enable Scripts provides means to personalize and achieve advanced g-code generation when using *Import Gerber* feature.

Automatically generated g-code of *Import Gerber* feature is not necessarily within users requirements. User can tweak the *Import Gerber* importation script to achieve desired results.

2.4.7.12 Mark Pads



You can mark selected pad shapes to ease manual drilling later. Software will recognize shapes: Circle, Oval, Rectangle, Polygon or Other(custom).

Circle:
Use circle pads for marking. Only circled pads will be marked.

Rectangle:
Use circle pads for marking. Only circled pads will be marked.

Oval:
Use oval pads for marking. Only oval pads will be marked.

Polygon:
Use polygon pads for marking. Only polygon pads will be marked.

Other:
Use other(custom) pads for marking. Only other(custom) pads will be marked.

2.4.7.13 Cut Height

Depth of marking point.

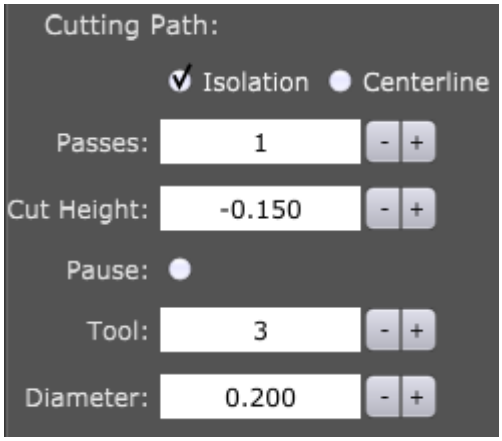
2.4.7.14 Pause

Insert 'Pause' (M00) G-Code before marking pads.

2.4.7.15 Tool

Number of tool used for marking pads. If tool change is enabled, tool with this number will be used.

2.4.7.16 Cutting Path



Isolation:
Enable to mill electrical isolation toolpath.

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

2.4.7.17 Passes

Number of milling passes of electrical isolation toolpath. Each milling pass is distanced from previous one for value of tool radius (tool used for Cutting Path).

2.4.7.18 Cut Height

Depth of milling.

2.4.7.19 Pause

Insert 'Pause' (M00) G-Code before milling electrical isolation toolpath.

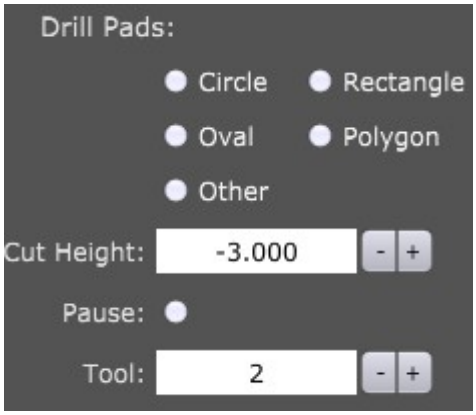
2.4.7.20 Tool

Number of tool used for milling electrical isolation toolpath. If tool change is enabled, tool with this number will be used.

2.4.7.21 Diameter

Diameter of tool used for milling electrical isolation toolpath. Radius of this value is used for *Passes* toolpath.

2.4.7.22 Drill Pads



You can drill selected pad shapes. Software will recognize shapes: Circle, Oval, Rectangle, Polygon or Other(custom).

You might want to drill hole on circled pad for trough hole component but not for rectangle pad for surface mount component.

Circle:
Use circle pads for drilling. Only circled pads will be drilled.

Rectangle:
Use circle pads for drilling. Only circled pads will be drilled.

Oval:
Use oval pads for drilling. Only oval pads will be drilled.

Polygon:
Use polygon pads for drilling. Only polygon pads will be drilled.

Other:
Use other(custom) pads for drilling. Only other(custom) pads will be drilled.

2.4.7.23 Cut Height

Depth of drilling.

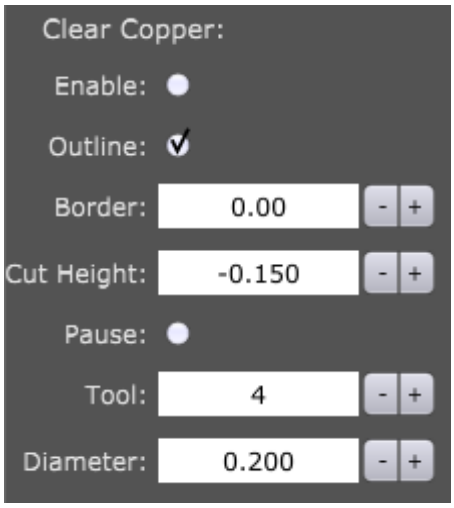
2.4.7.24 Pause

Insert 'Pause' (M00) G-Code before drilling.

2.4.7.25 Tool

Number of tool used for drilling. If tool change is enabled, tool with this number will be used.

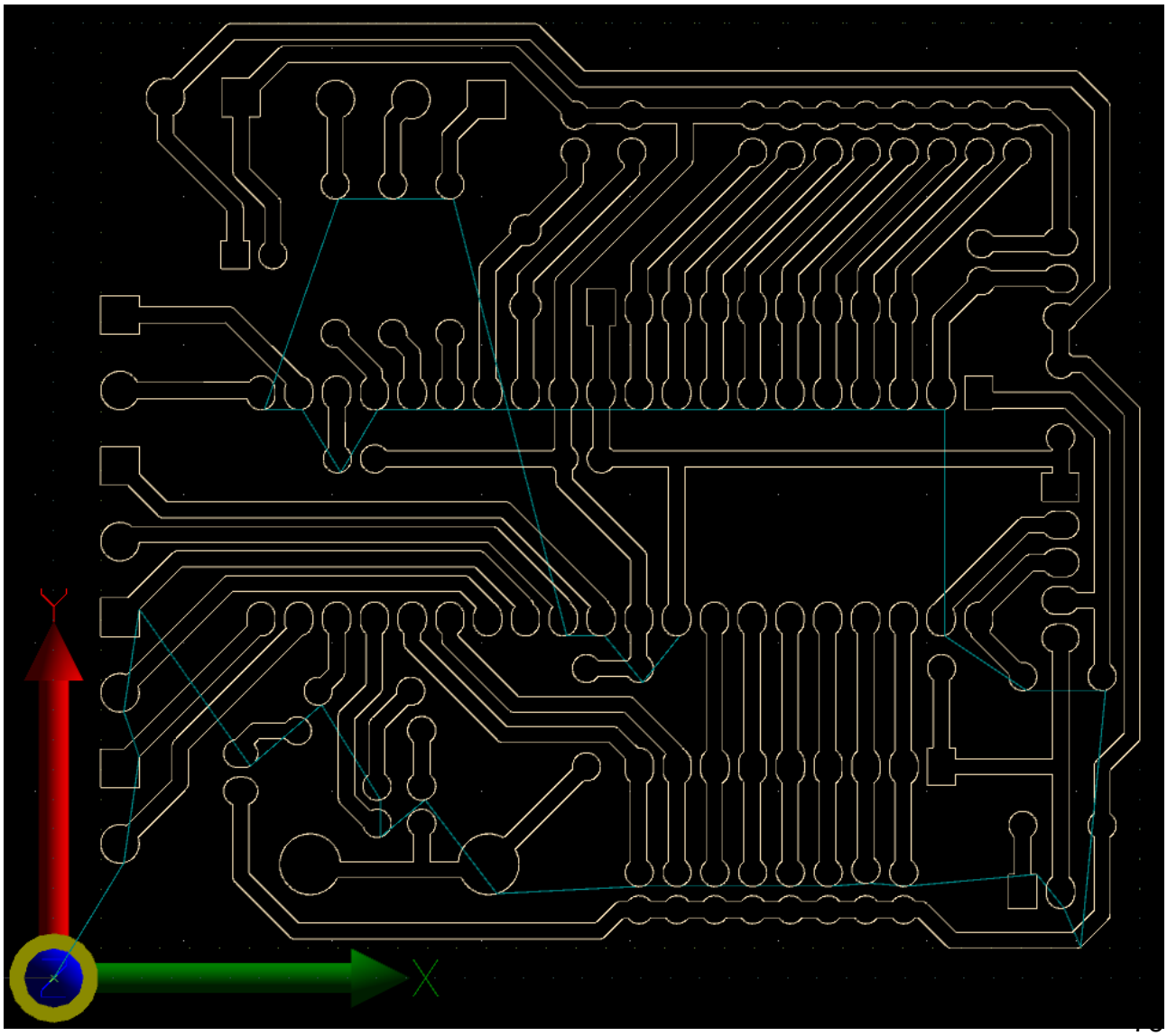
2.4.7.26 Clear Copper



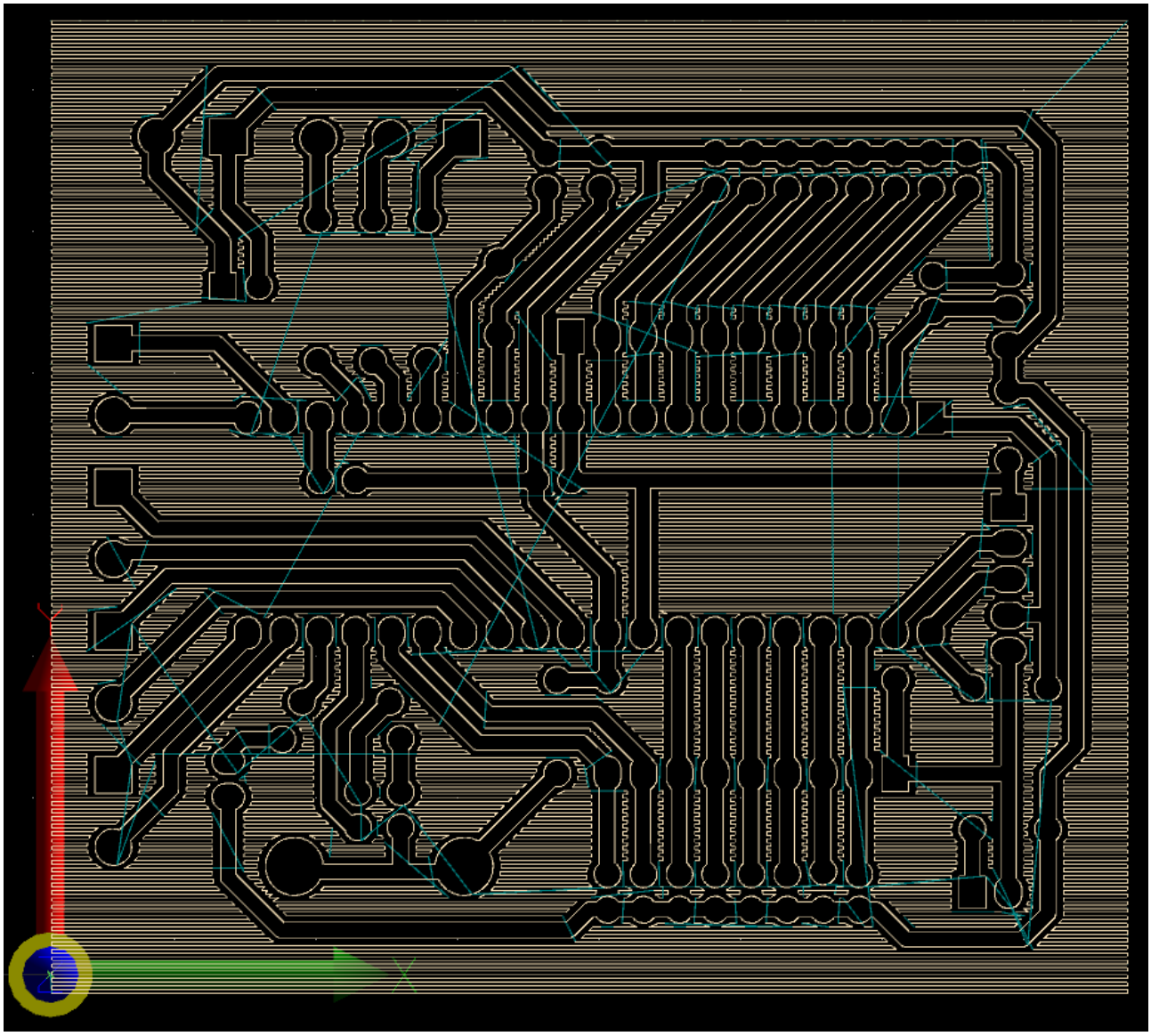
2.4.7.27 Enable

Generates toolpath that mills(“clears”) unused copper.

Toolpath without clear copper:



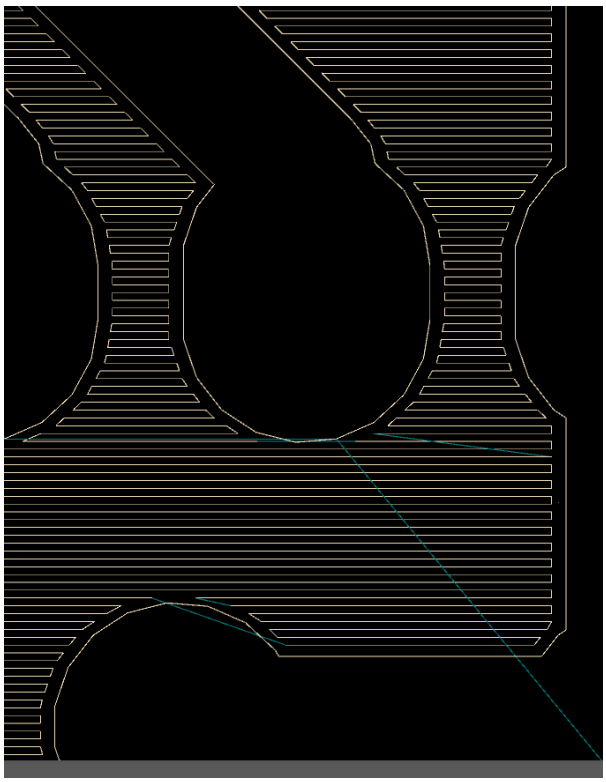
Toolpath with clear copper:



2.4.7.28 Outline

Finishes clear copper toolpath with additional outline to prevent edges.

Without outline:

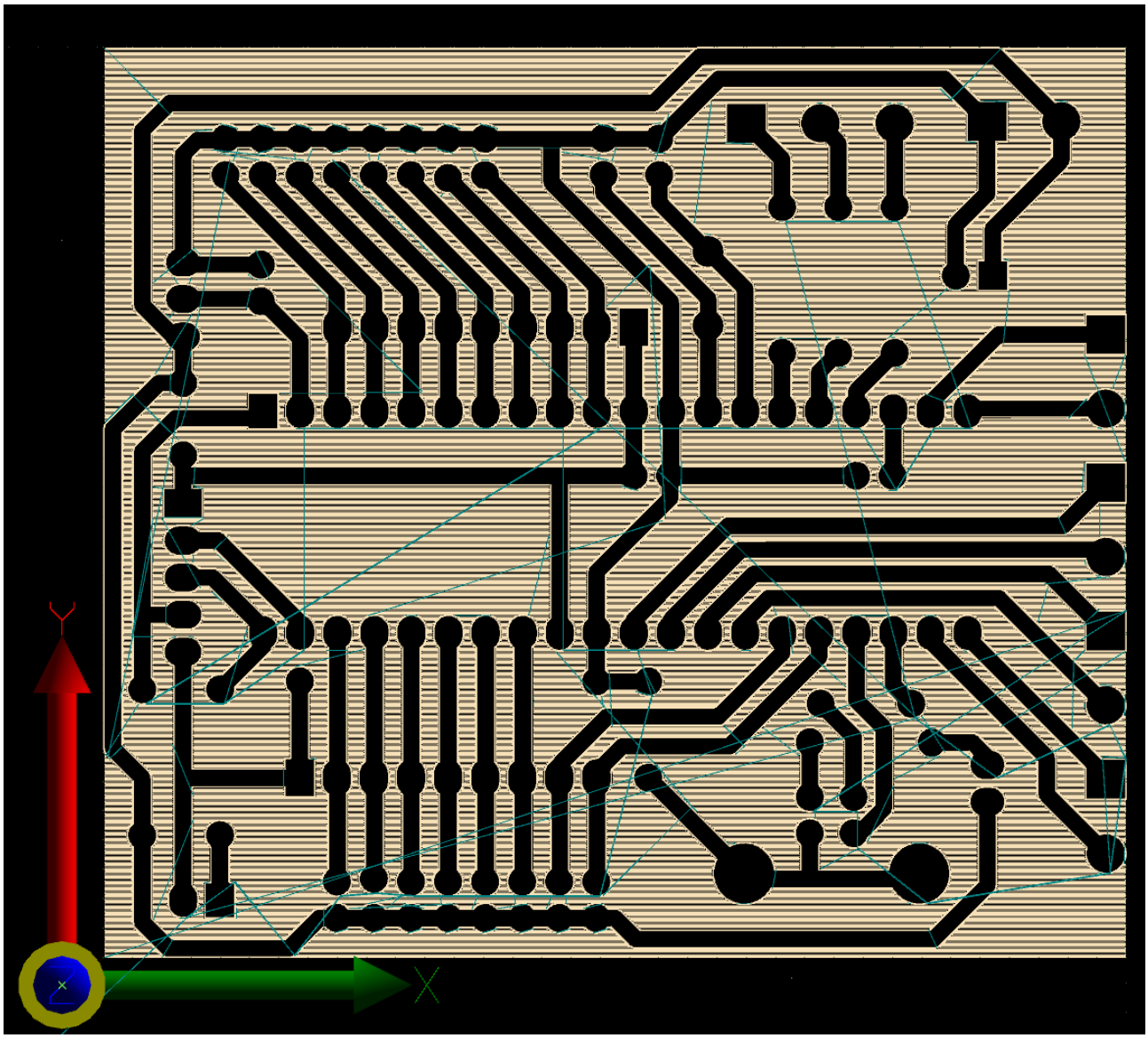


With outline:

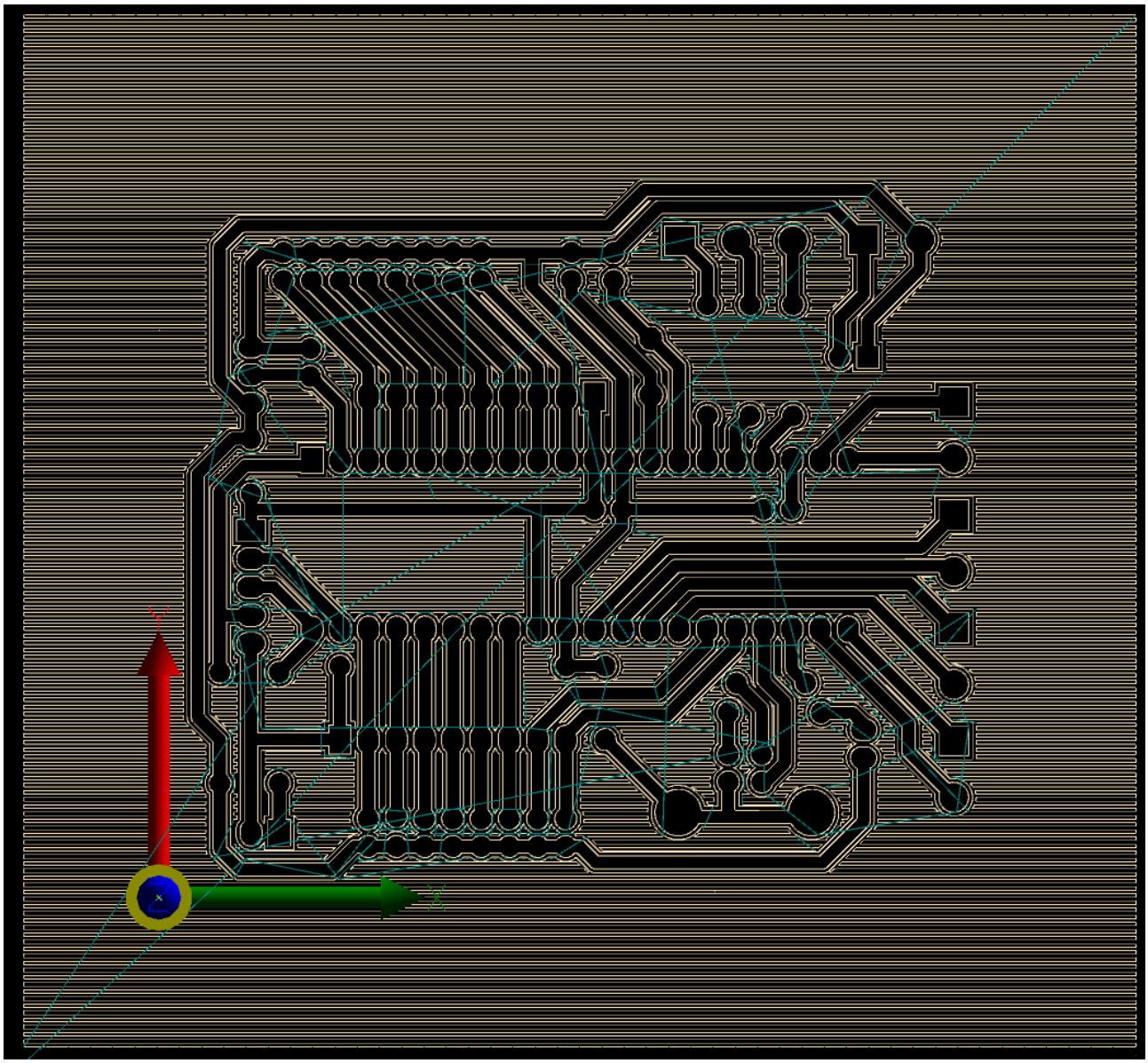
2.4.7.29 Border

Distance value for area beyond PCB max extents where copper will also be milled.

Border zero:



Border value set to 15mm:



2.4.7.30 Cute Height

Depth of clear copper milling.

2.4.7.31 Pause

Insert 'Pause' (M00) G-Code before clearing copper.

2.4.7.32 Tool

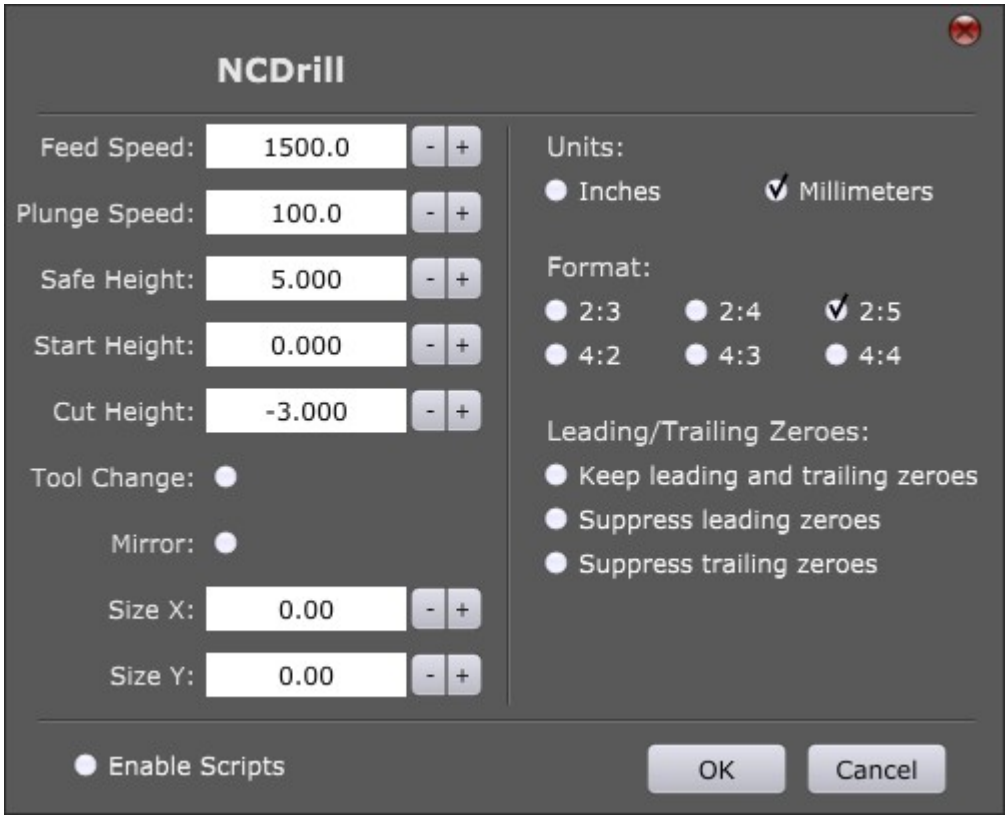
Number of tool used for clearing copper. If tool change is enabled, tool with this number will be used.

2.4.7.33 Diameter

Diameter of tool used for clearing copper.

2.4.8 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.



2.4.8.1 Feed Speed

Sets feed speed for generated toolpath. F-word g-code will be generated. Each G01 move will be performed at this speed.

'Feed Speed' is usually the speed that is used for cutting or milling, and since there will be no cutting involved in the drilling procedure, you can set this value the same as your 'Traverse Speed' in settings.

'Feed speed' is the speed at which machine will descend from 'Safe height' to 'Start height'.

2.4.8.2 Plunge Speed

Sets feed speed of plunge moves for generated toolpath. Each G01 move in Z- direction will be performed at this speed.

2.4.8.3 Safe Height

With this option enabled, generated toolpath will include traverse moves performed at safe height.

Safe height is a safety feature which helps with prevention of machine crashing into obstacles that may interfere with machines toolpath. Obstacles could be screws, fixtures, vises etc..

When machine is finished with cutting toolpath A, it will ascend to safe height and move to next cutting location of toolpath B.

2.4.8.4 Start Height

Start height is usually surface of workpiece material. Machine will descend from 'Safe Height' to 'Start Height' at feed rate.

2.4.8.5 Cut Height

Cut height is depth of drilling.

Holes should be drilled in its entirety, meaning, hole should not be drilled half way. Cut Height value is basically the thickness of your PCB with some added safe distance just to be sure that the holes will be drilled “clean”.

2.4.8.6 Tool change

Enables tool change for drilling. Tool number used for drilling should be set in NC drill file.

2.4.8.7 Mirror

Mirrors program in XY.

2.4.8.8 Size X

X coordinate approximate dimension of PCB board. Make sure that same value is used with Gerber import.

2.4.8.9 Size Y

Y coordinate approximate dimension of PCB board. Make sure that same value is used with Gerber import.

2.4.8.10 Enable Scripts

Enable Scripts provides means to personalize and achieve advanced g-code generation when using *Import NC Drill* feature.

Automatically generated g-code of *Import NC Drill* feature is not necessarily within users requirements. User can tweak the *Import NC Drill* importation script to achieve desired results.

2.4.8.11 Units

Specify millimeter or inch units.

2.4.8.12 Format

Specify number decimal digit format.

2.4.8.13 Leading/Trailing Zeroes

Specify leading and trailing zeroes.

2.4.9 Import CSV

Imports .CSV file.

2.4.10 Export G-code

Export tool-path to G-code file.

2.4.11 Export DXF

Export tool-path to DXF file.

2.4.12 Export CSV

Export tool-path to CSV file.

2.4.13 Settings

In 'Settings' are all main configurations of parameters that are crucial for proper functionality and correct behavior of machine and attached hardware. See chapter: ***2.7 Settings***.

2.4.14 Import Settings

Imports ‘Settings’ file to quickly restore a known working configuration or to re-configure PlanetCNC TNG software to suit different machines types or applications.

2.4.15 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.

2.4.16 Backup Profile

Some users can face an obstacle during their time learning and using PlanetCNC TNG software.

While PlanetCNC always tries to give fast and effective support, sometimes more info is needed about the problem user is facing in order to come up with solution as fast as possible.

Backup Profile creates archive file that includes necessary files for PlanetCNC support to start investigating an issue quicker and more effectively.

Backup profile includes:

-Data files:

gcode.txt: G-code program file that is loaded in TNG when you export backup profile

Log files: log_computer, log_device, log_usb

params.txt: file with all parameter values

paramsn.txt:

trans.txt:

warp.txt:

-Script files

-Source: source g-code file loaded in PlanetCNC TNG when you export backup profile

-PlanetCNC TNG settings file

-keys file

-Lic. File

-state files

-Tool table file

-.var file

-toolbar button files

2.4.17 Language

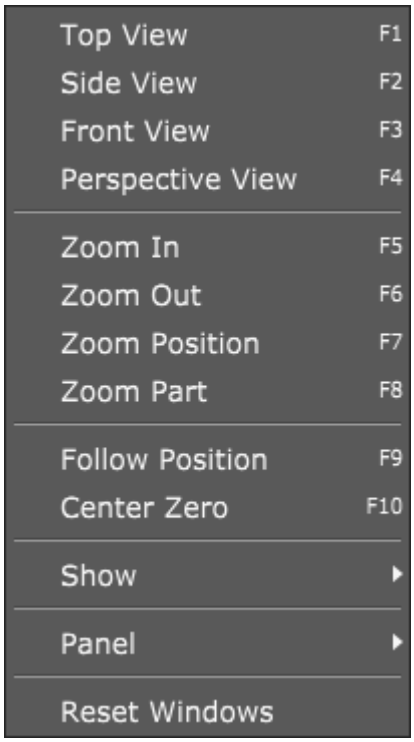
Select language of application from list of supported languages.

2.4.18 Exit

Closes PlanetCNC TNG software. If the ‘E-Stop’ on exit feature is active, 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.

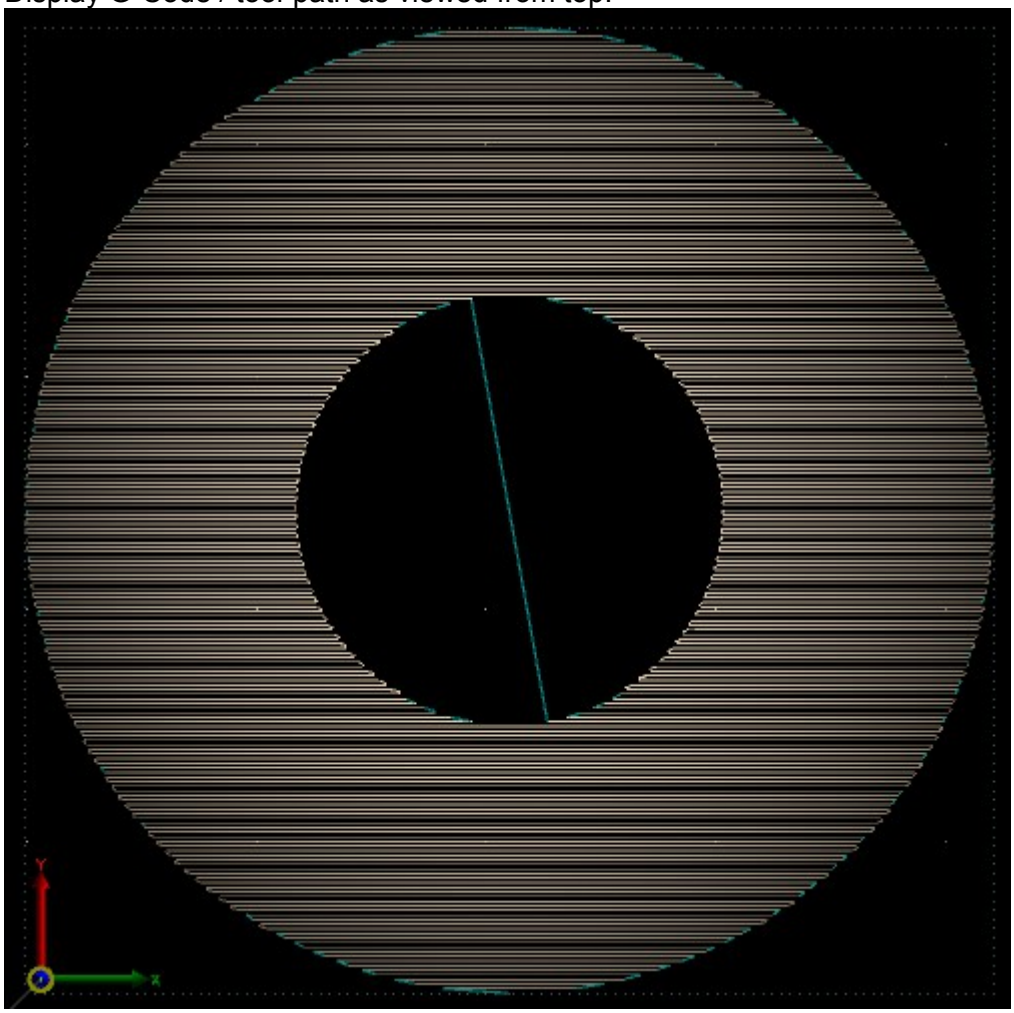
2.5 View menu

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.



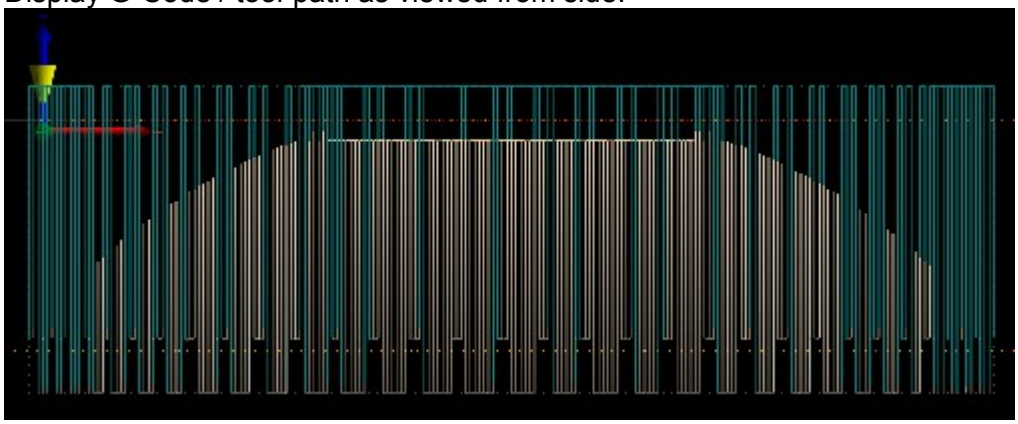
2.5.1 Top view

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



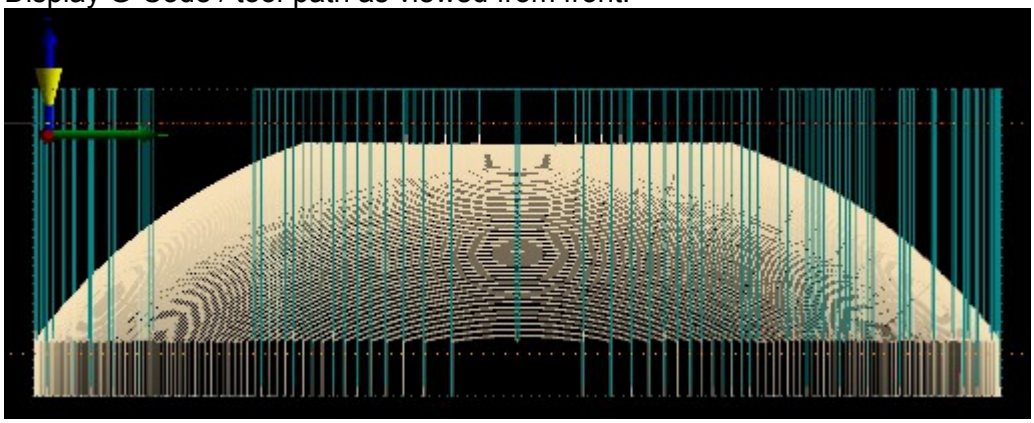
2.5.2 Side View

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



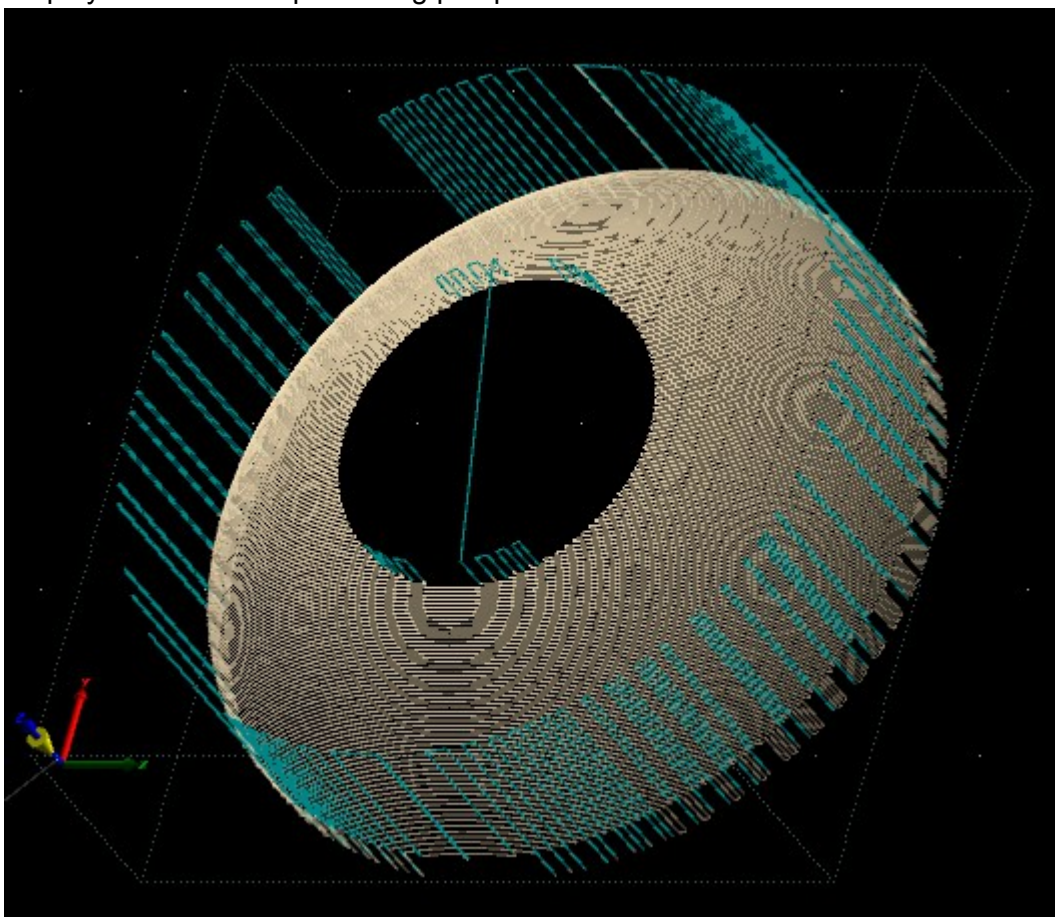
2.5.3 Front View

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



2.5.4 Perspective view

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



2.5.5 Zoom In

Zooms-in the display for close view of smaller details.

2.5.6 Zoom Out

Zooms-out the display for overview of a larger area.

2.5.7 Zoom Position

Zooms-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.

2.5.8 Zoom Part

Zooms-in for a closer view of displayed toolpath.

2.5.9 Follow Position

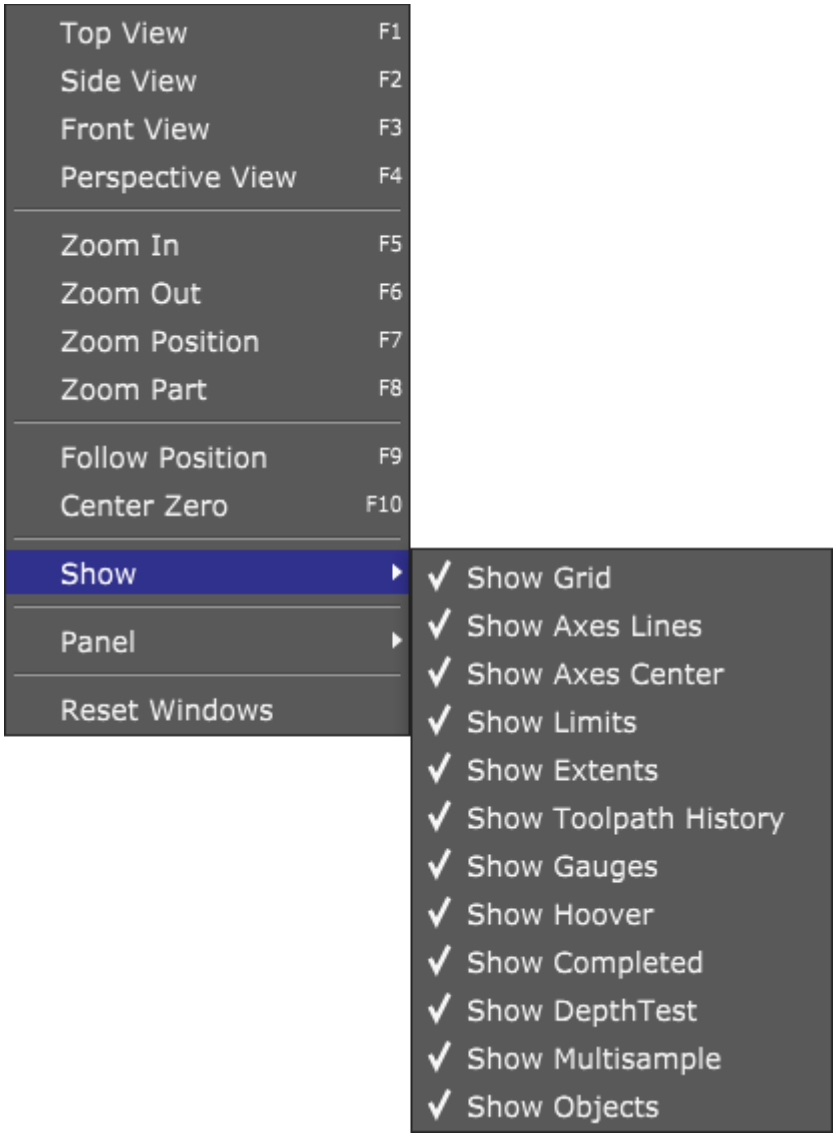
3D display follows tool over toolpath.

2.5.10 Center Zero

Sets absolute zero position of machine to center of 3D display.

2.5.11 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.



2.5.11.1 Show grid

Displays grid on screen.

2.5.11.2 Show Axes Lines

Displays coordinate system axes.

2.5.11.3 Show Axes Center

Displays center of axes.

2.5.11.4 Show Limits

Displays motion range limits of machine.

2.5.11.5 Show Extents

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

2.5.11.6 Show Toolpath History

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

2.5.11.7 Show Gauges

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.

2.5.11.8 Show Hover

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

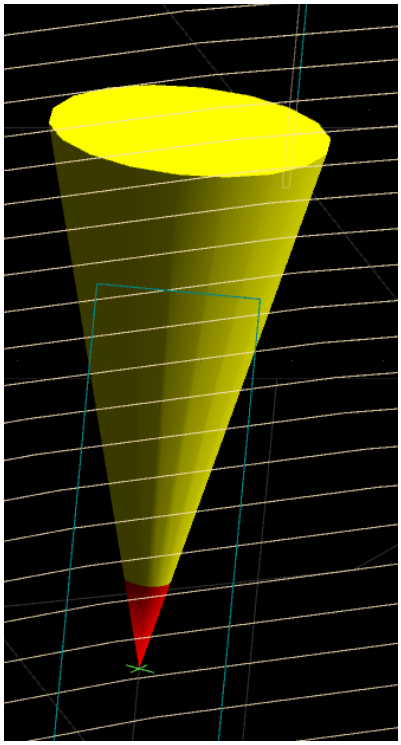
2.5.11.9 Show Completed

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

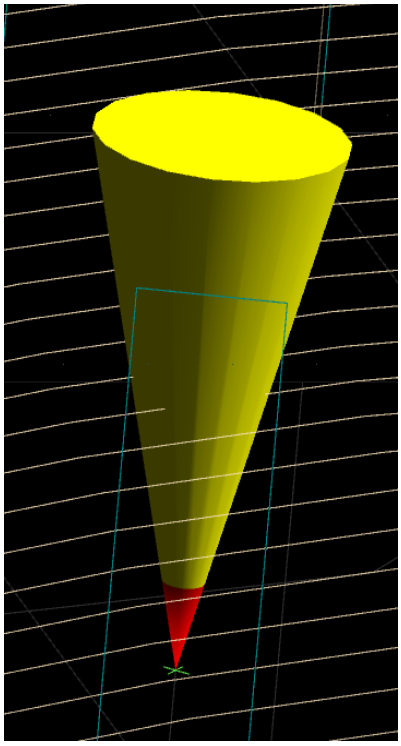
2.5.11.10 Show Depthtest

When enabled, toolpath follows depth rules in 3D visualization. When disabled toolpath is drawn over all other objects.

Disabled:



Enabled:



2.5.11.11 Show Multisample

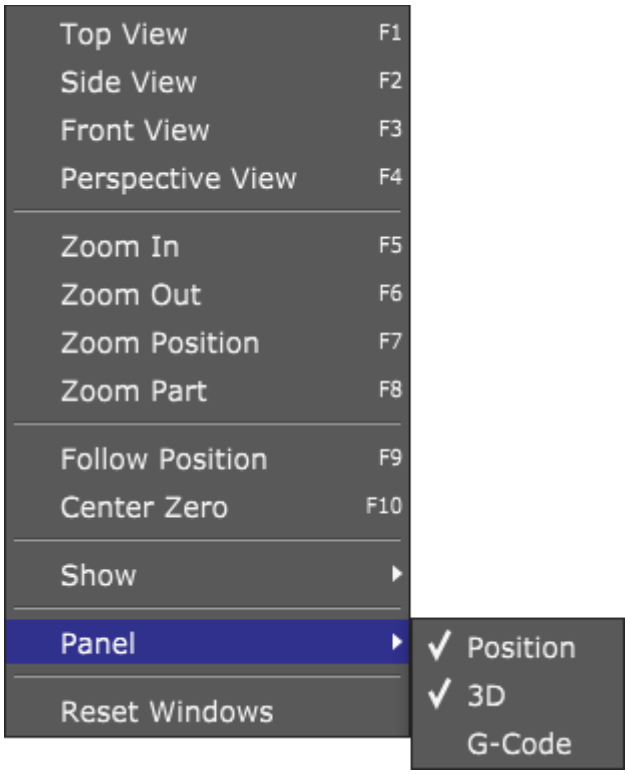
Enables high quality 3D display.

2.5.11.12 Show Objects

Enables display of 3D objects.

2.5.12 Panel

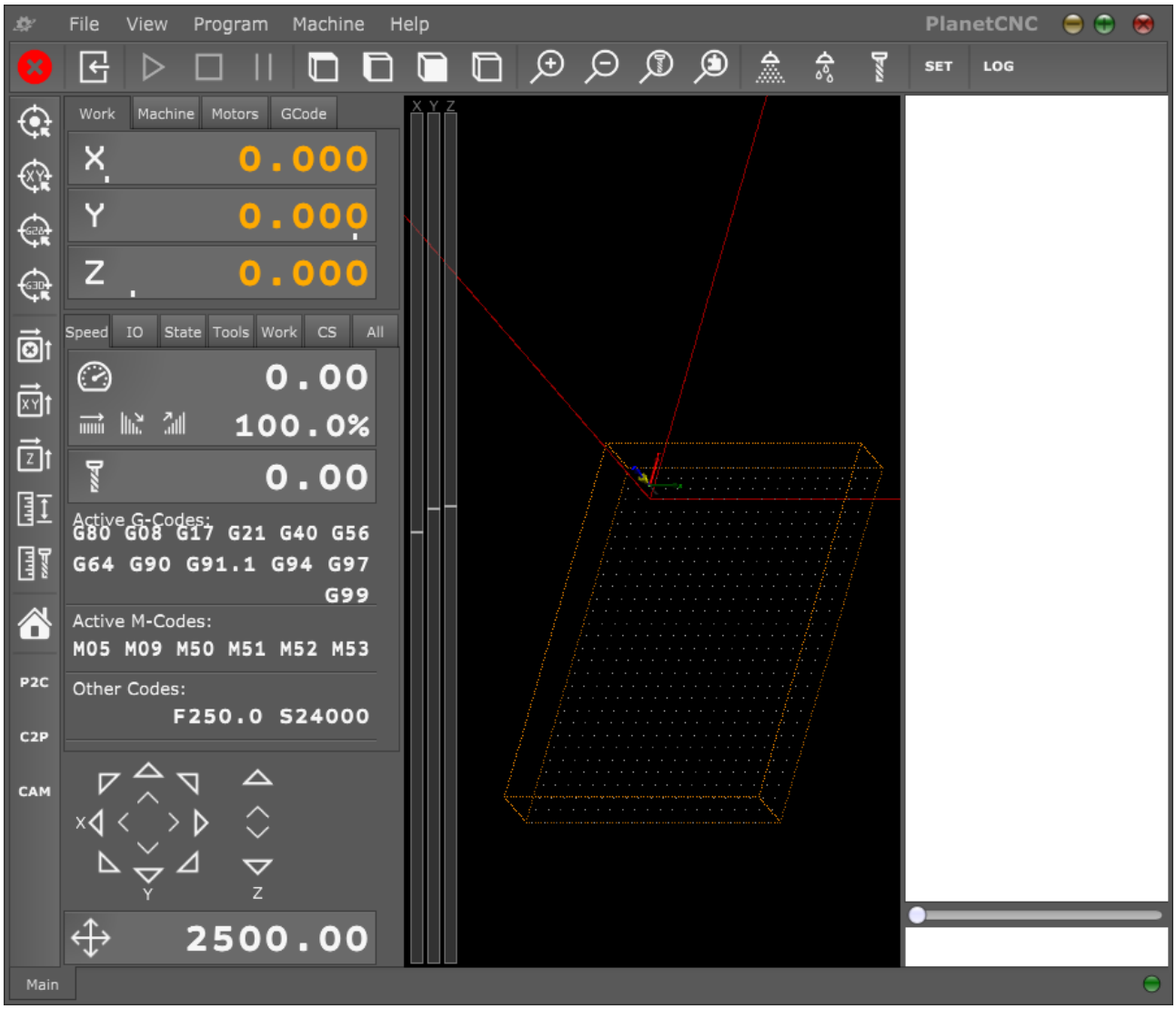
Enables/Disables display of Position panel, 3D toolpath display and G-code window.



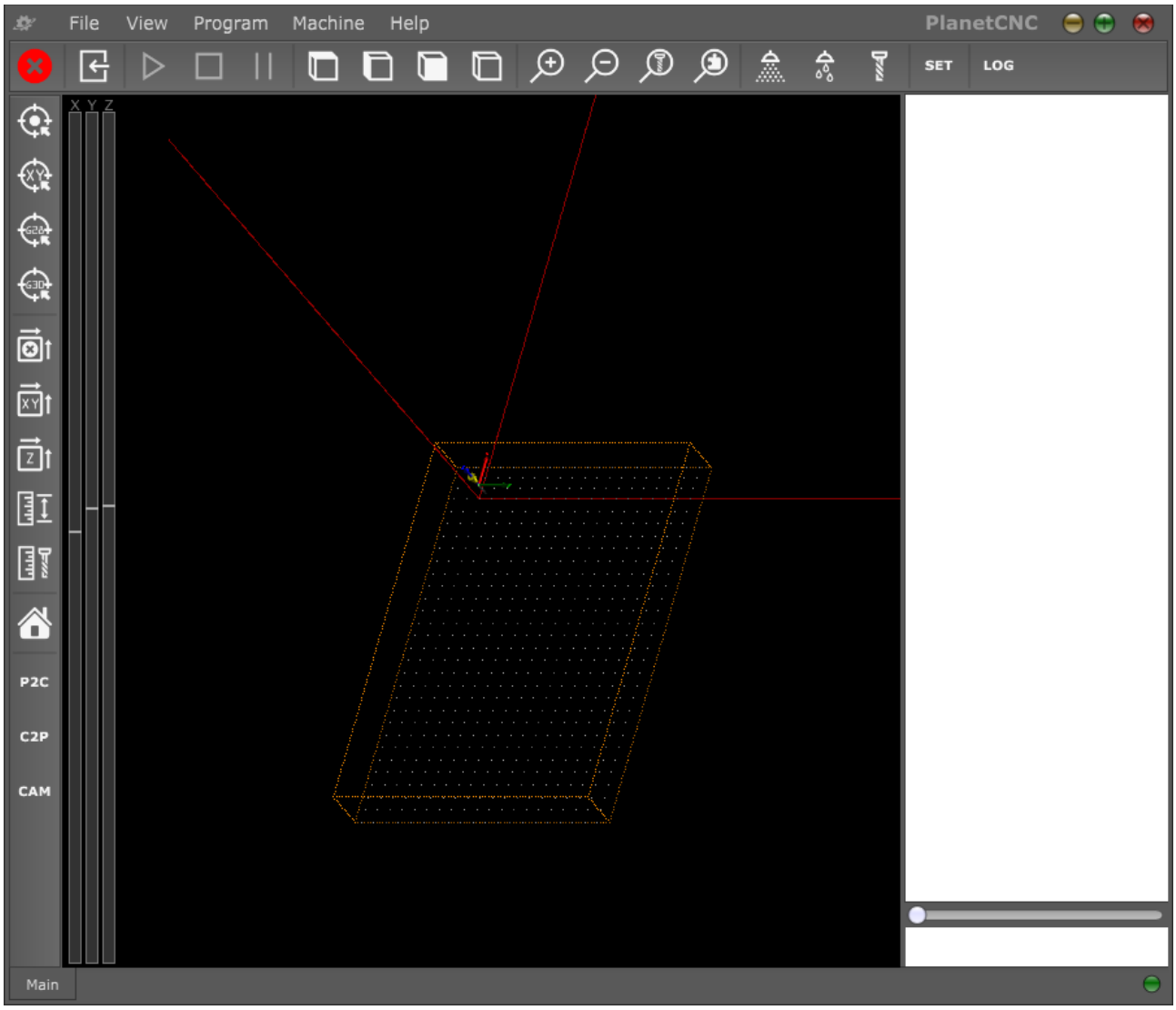
2.5.12.1 Position

Enables/Disables display of Position panel.

Position panel, 3D display panel and g-code panel enabled:



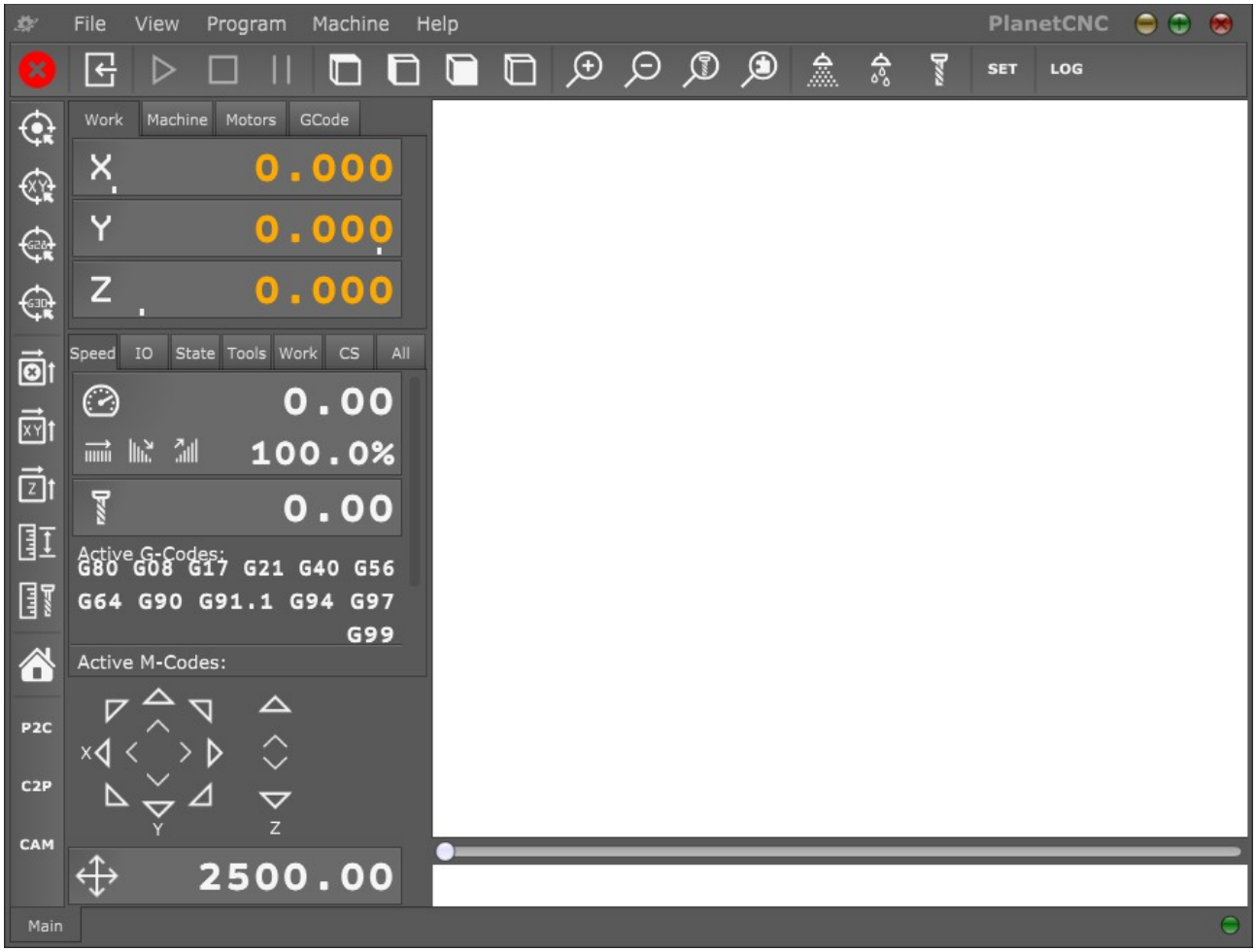
With position panel disabled:



2.5.12.2 3D

Enables/Disables 3D display panel.

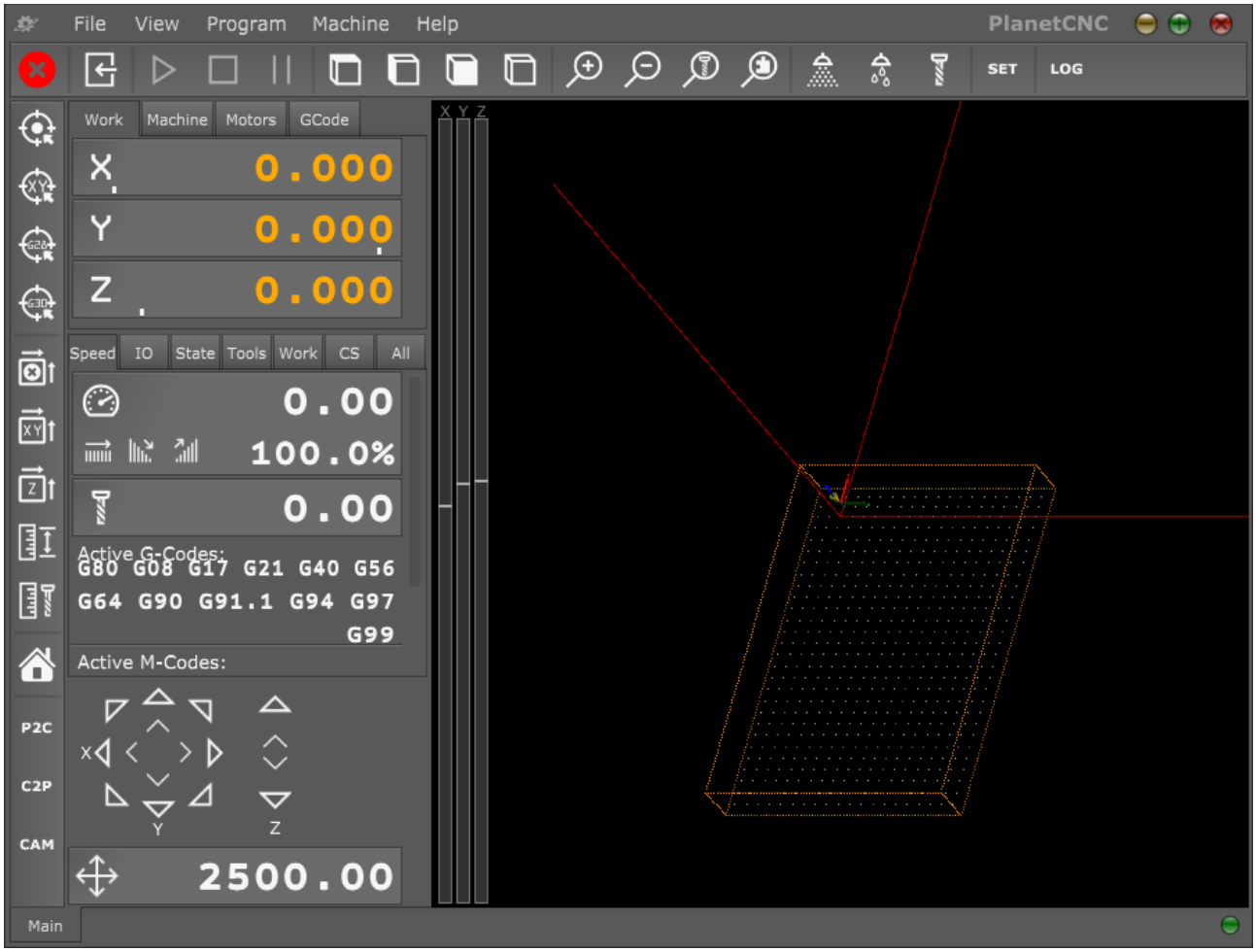
With 3D display panel disabled:



2.5.12.3 G-code

Enables/Disables G-code panel.

With g-code panel disabled:

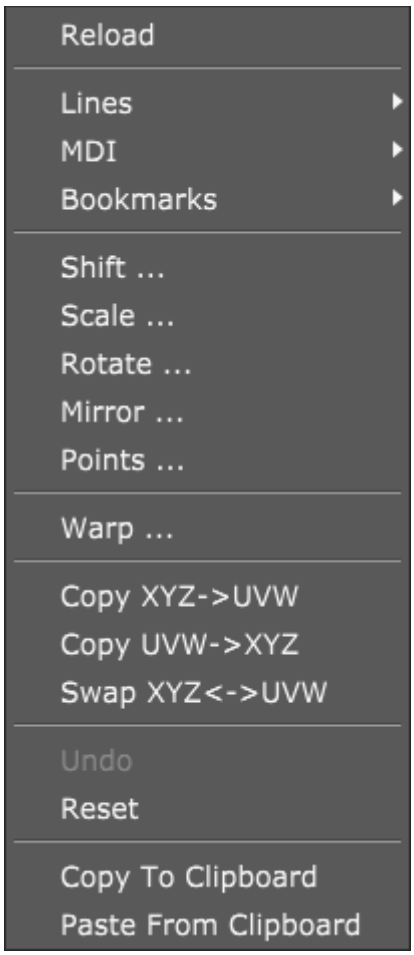


2.6 Program menu

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.

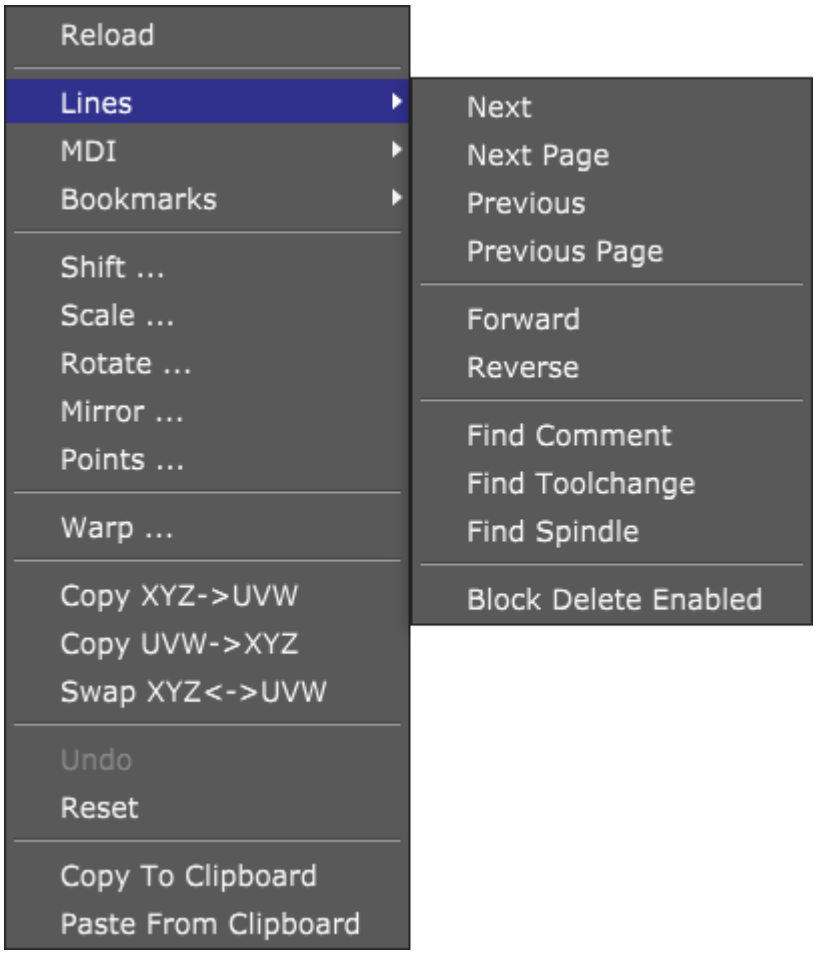


2.6.1 Reload

Reloads currently imported g-code program.

2.6.2 Lines

‘Lines’ functions allow easy navigation trough G-Code.



2.6.2.1 Next

Selects next G-code line in G-code window.

2.6.2.2 Next Page

Next page in G-code window is displayed.

2.6.2.3 Previous

Previous G-code line in G-code window.

2.6.2.4 Previous Page

Previous page in G-code window is displayed.

2.6.2.5 Forward

Forwards toolpath position slider(see chapter *2.2.5.2*).

Recommended use with keyboard shortcut (see chapter *2.7.3.23.2 Code*) for easier navigation.

2.6.2.6 Reverse

Reverses toolpath position slider(see chapter *2.2.5.2*).

Recommended use with keyboard shortcut (see chapter *2.7.3.23.2 Code*) for easier navigation.

2.6.2.7 Find Comment

Selects next line containing comment.

2.6.2.8 Find Tool-change

Selects G-code line containing 'M6' G-code.

2.6.2.9 Find Spindle

Selects G-code line containing 'M3' G-code.

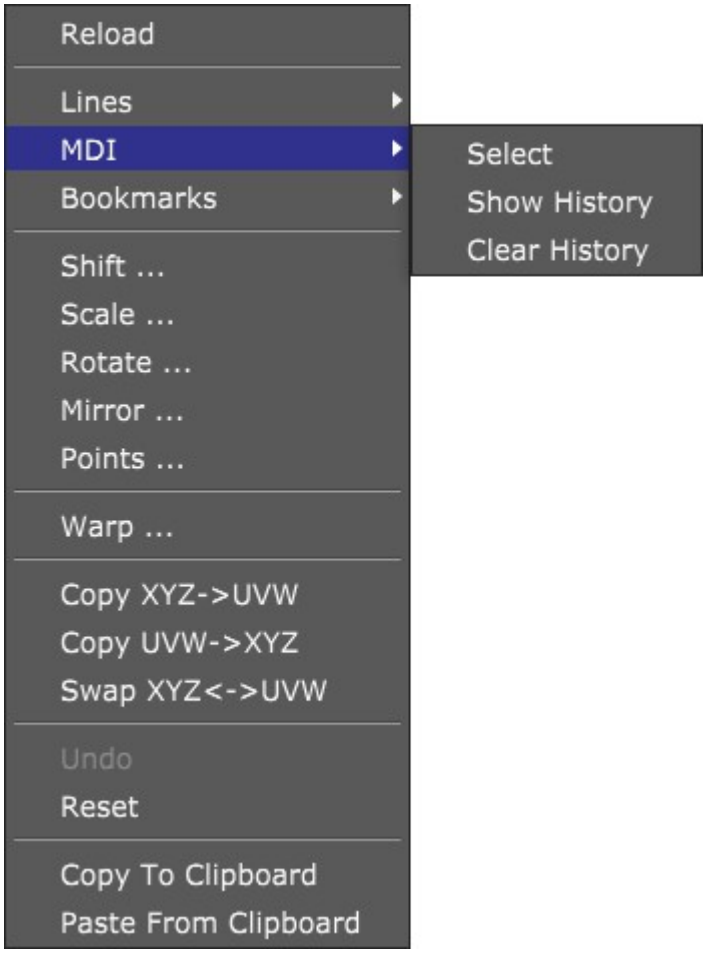
2.6.2.10 Block Delete Enabled

Block delete feature executes or skips lines of program that have character “/” at the beginning of the program line.

This feature comes useful when you need to test proof your program or if you have two versions of the same workpiece with minor changes etc…

In such case you can use “Block delete” feature which allows you to skip or execute marked lines of g-code of your program.

2.6.3 MDI



2.6.3.1 Select

Focuses MDI command window.

2.6.3.2 Show history

Displays list of MDI commands previously used.

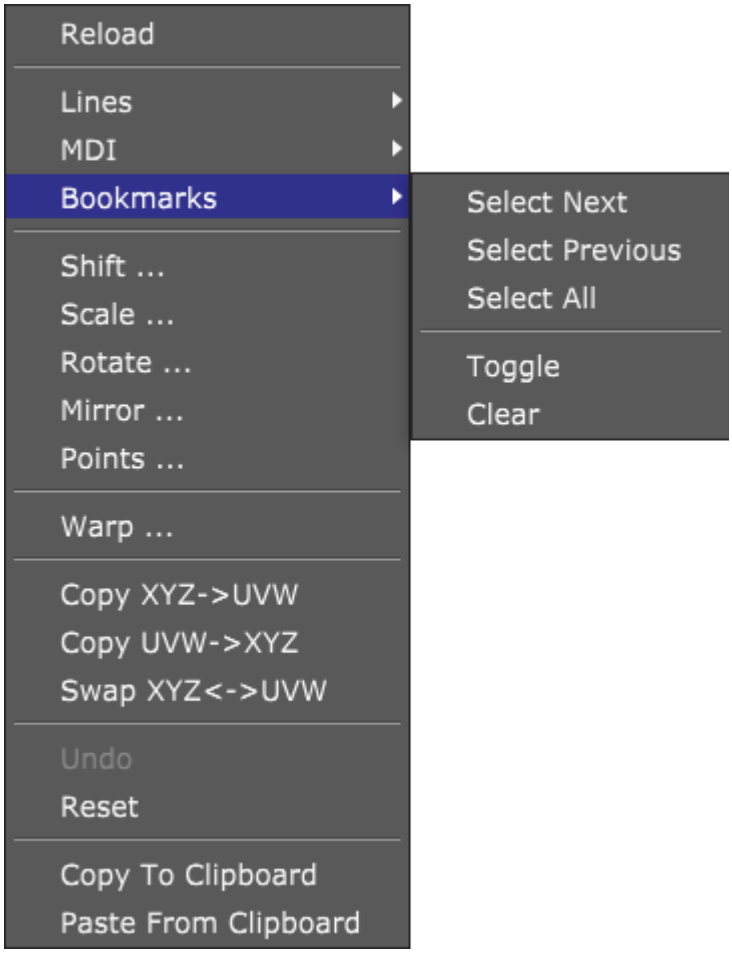
2.6.3.3 Clear history

Clears list of MDI commands previously used.

2.6.4 Bookmarks

G-code lines can be bookmarked. ‘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 shortcut key. Multiple lines can be bookmarked. When a line has been bookmarked it highlights.

Selection of a ‘Bookmark’ and press of the *Toogle* item will clear it. Alternatively all ‘Bookmarks’ can be cleared at once using *Clear*. When bookmarks are cleared highlight are removed.



2.6.4.1 Select Next

Selects next bookmarked G-code line.

2.6.4.2 Select previous

Selects previous bookmarked G-code line.

2.6.4.3 Select All

Selects all bookmarked G-code lines.

2.6.4.4 Toggle

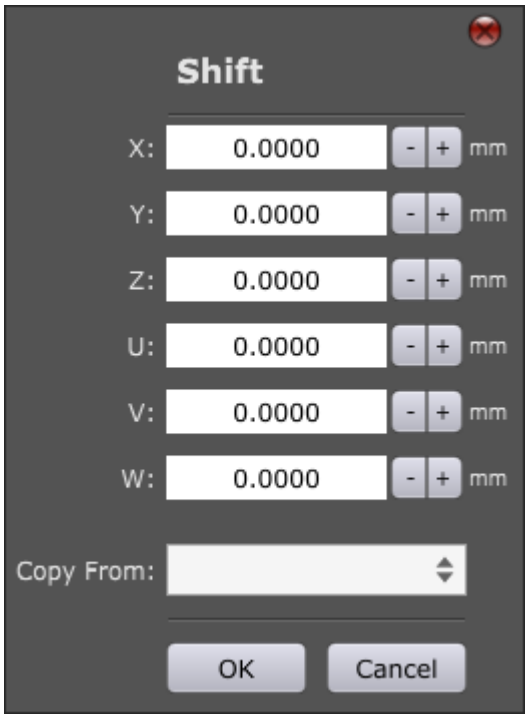
Toggles bookmark on selected G-code line.

2.6.4.5 Clear

Clears all bookmarks from G-code lines.

2.6.5 Shift

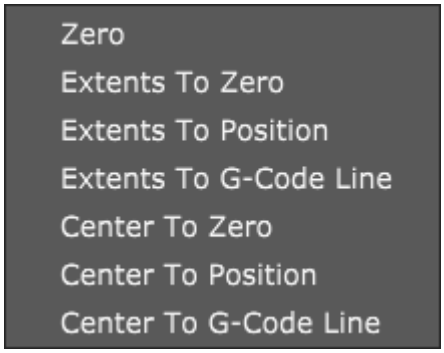
Shift translates or moves the tool-path in 3D space using one of many options.



X,Y,Z,U,V,W:
Shifts toolpath in any axis for inserted value.

2.6.5.1 Copy From

User can shift toolpath using available preset shift options.



2.6.5.1.1 Zero

Sets shift value for all axes to zero.

2.6.5.1.2 Extents To Zero

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

2.6.5.1.3 Extents To Position

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

2.6.5.1.4 Extents To G-Code Line

Aligns the ‘minimum’ edges of the tool-path extents with position of selected g-code line.

2.6.5.1.5 Center To Zero

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

2.6.5.1.6 Center To Position

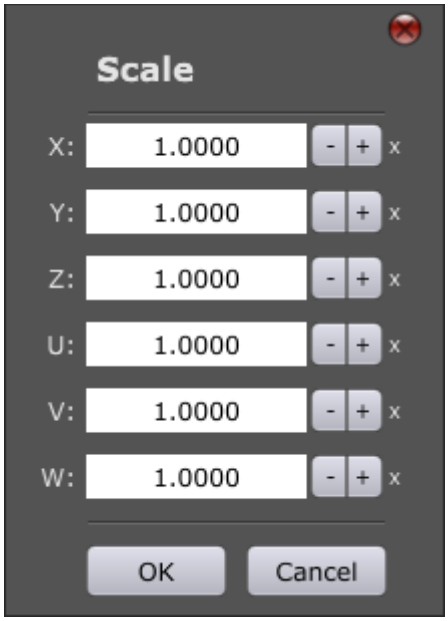
Aligns the center of the tool-path with the current machine position.

2.6.5.1.7 Center To G-Code Line

Aligns the center of the tool-path with position of selected g-code line.

2.6.6 Scale

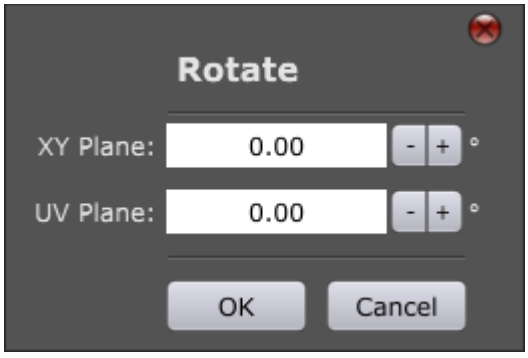
Scale adjusts dimensions of the tool-path using ‘factor’ based scaling. Default value of 1.00 is equivalent to 100% scale or actual size. A scale factor setting of 2.00 increases tool-path dimensions to 200% of the original size in the relevant axis.



X,Y,Z,U,V,W:
Scales toolpath of selected axis for inserted value.

2.6.7 Rotate

Rotates the tool-path XY and UV plane with the origin as the center of rotation. Positive values rotate the tool-path in a clockwise direction. Negative values cause anti-clockwise rotation.

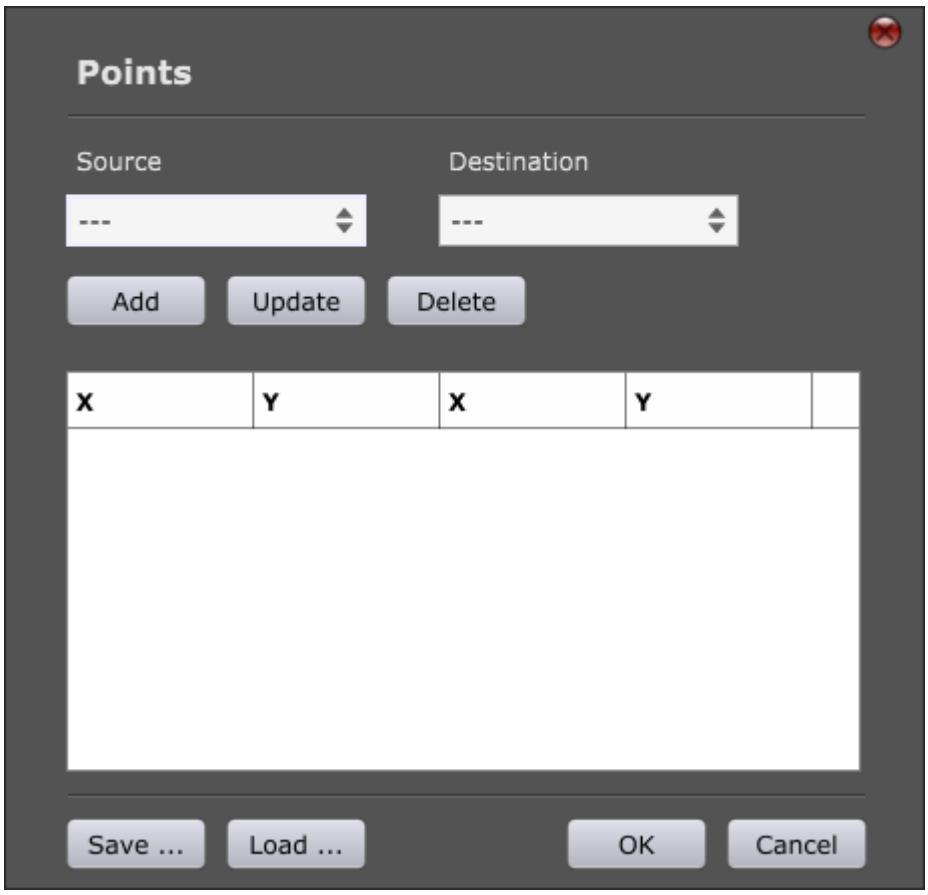


2.6.8 Mirror

Mirror function mirrors the tool-path over X, Y, Z and U,V,W axes. Axis of reflection is selected with enabling radio button.

2.6.9 Points

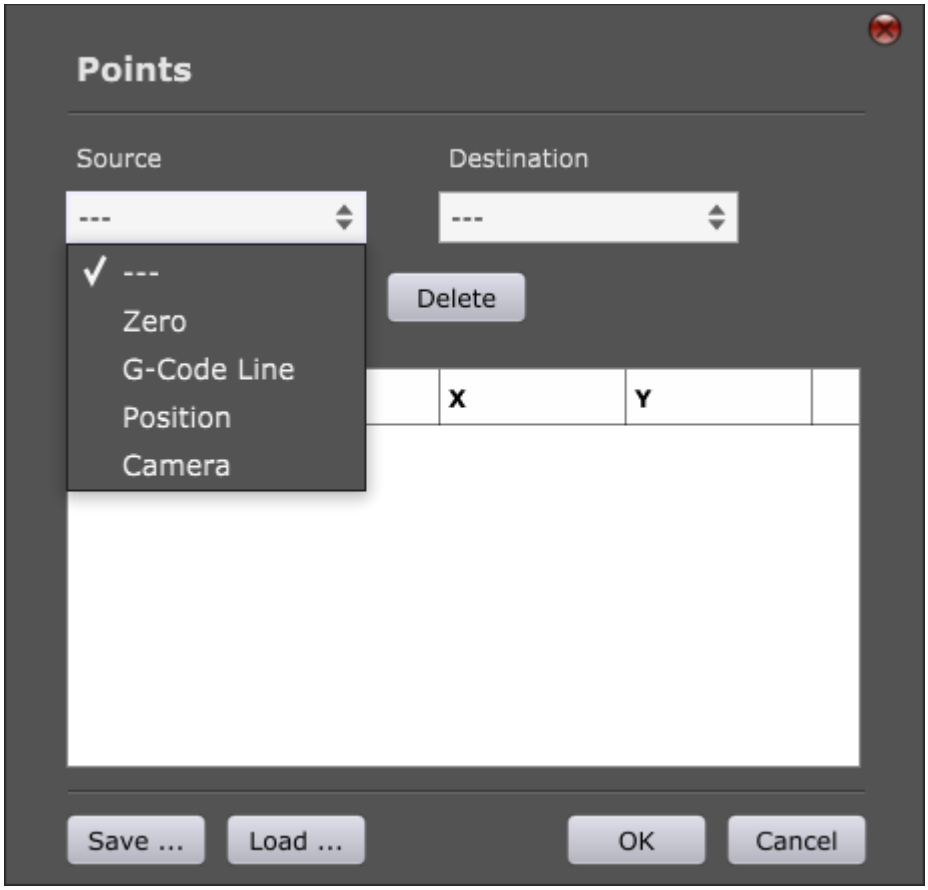
Calculates transformation from first group of points to second group of points.



2.6.9.1 Source

Beside manual input, user can select option from drop down menu for source coordinates of transformation point.

Choose among options: Zero, G-Code Line, Position, Camera:



Zero:

Sets zero coordinate values for source point.

G-code Line:

Sets coordinate values for source point obtained from selected line in g-code panel.

Position:

Sets coordinate values for source point obtained from machine position.

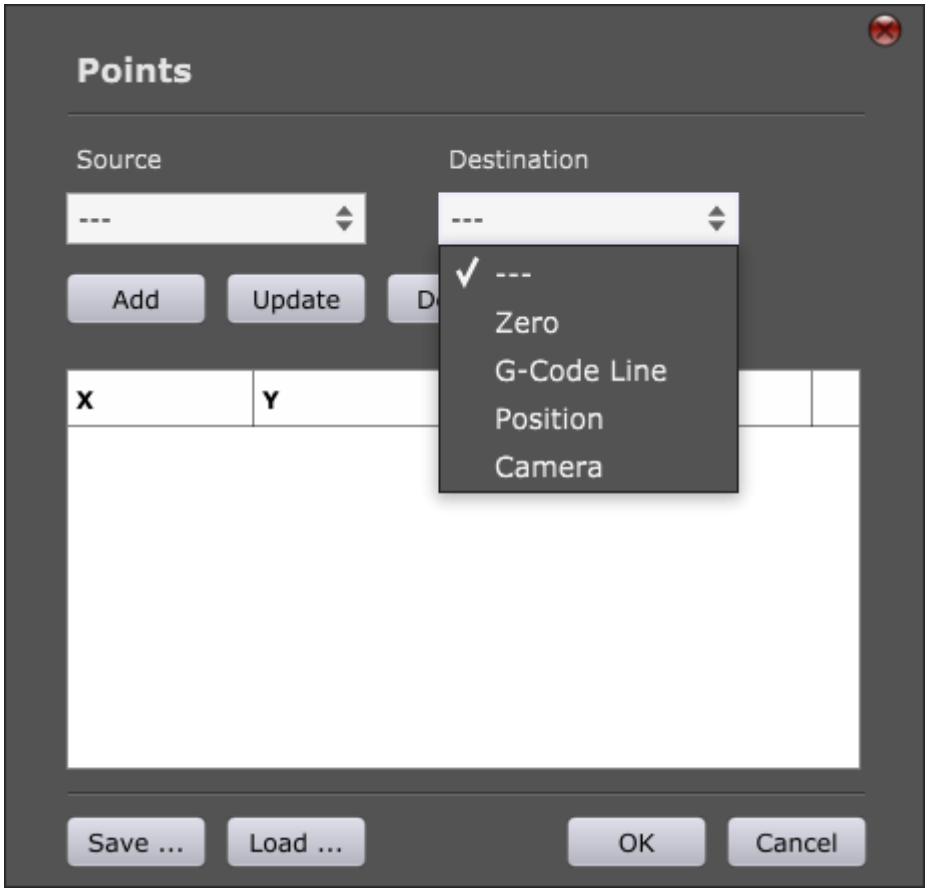
Camera:

Sets coordinate values for source point obtained from camera position.

2.6.9.2 Destination

Beside manual input, user can select option from drop down menu for destination coordinates of transformation point.

Choose among options: Zero, G-Code Line, Position, Camera:



Zero:

Sets zero coordinate values for destination point.

G-code Line:

Sets coordinate values for destination point obtained from selected line in g-code panel.

Position:

Sets coordinate values for destination point obtained from machine position.

Camera:

Sets coordinate values for destination point obtained from camera position.

Add:

Adds new point row.

Update:

Updates highlighted point row with new values depending on 'Source' and/or 'Destination' option selected.

Delete:

Deletes point row.

Save:

Saves points configuration.

Load:

Loads points configuration.

2.6.10 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.

2.6.11 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.

2.6.12 Copy UVW → XYZ

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

2.6.13 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.

2.6.14 Undo

Cancels applied changes from program menu items: *Shift, Scale, Rotate, Mirror, Points, Copy..., Swap...*

2.6.15 RESET

Resets all parameters of Program menu features. Any applied changes from program menu will be reset.

2.6.16 Copy to Clipboard

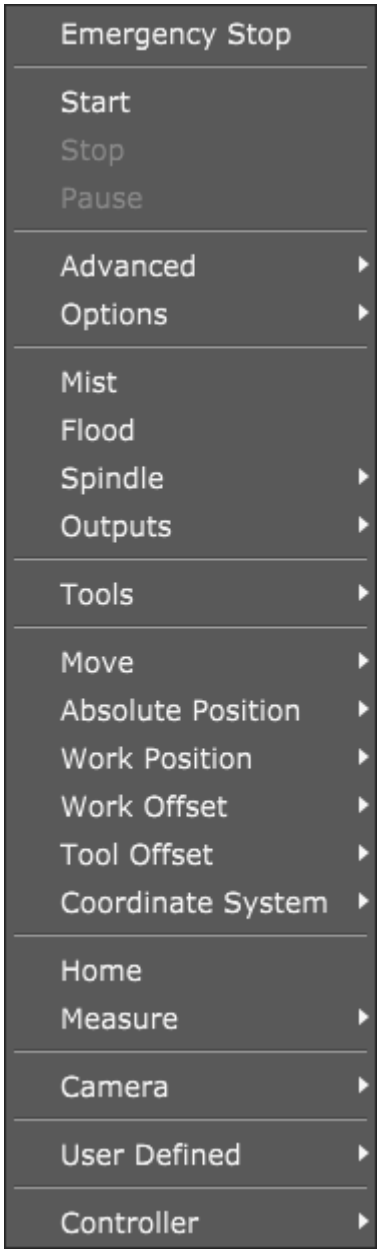
Copies entire contents of the G-Code panel to the windows clipboard.

2.6.17 Paste from Clipboard

Pastes windows clipboard content to the G-Code panel.

2.7 Machine Menu

Provides means to directly interact with controlled machine and attached hardware.



2.7.1 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.

2.7.2 Start

Starts execution of the current G-Code program.

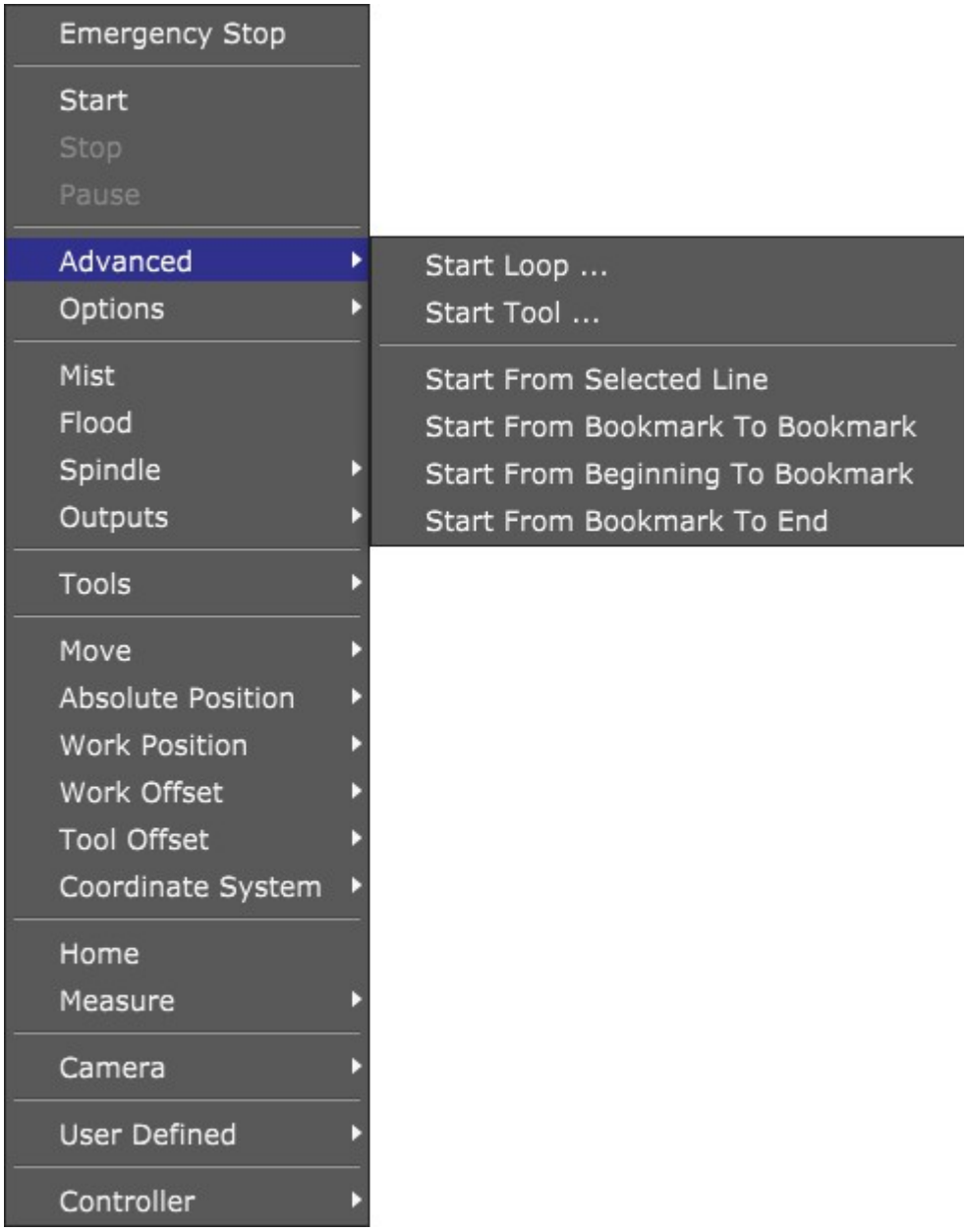
2.7.3 Stop

Stops the machine using a normal stop.

2.7.4 Pause

Pauses / Resumes execution of the current G-Code program.

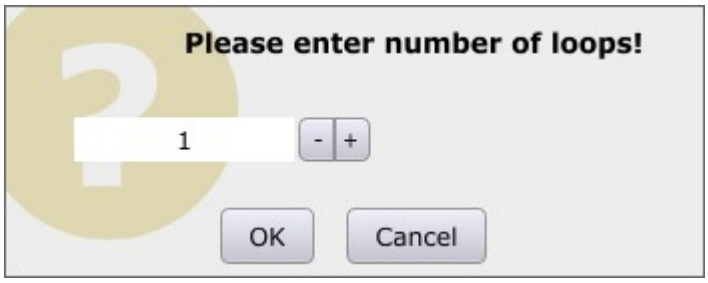
2.7.5 Advanced



2.7.5.1 Start Loop

Starts looped execution of the current G-Code program.

Inserted value represents number of loops of the G-code program.



2.7.5.2 Start Tool

Program will start at location where entered tool number is first used:

2.7.5.3 Start From Selected Line

Program will start at location of selected g-code line in g-code panel.

2.7.5.4 Start From Bookmark To Bookmark

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

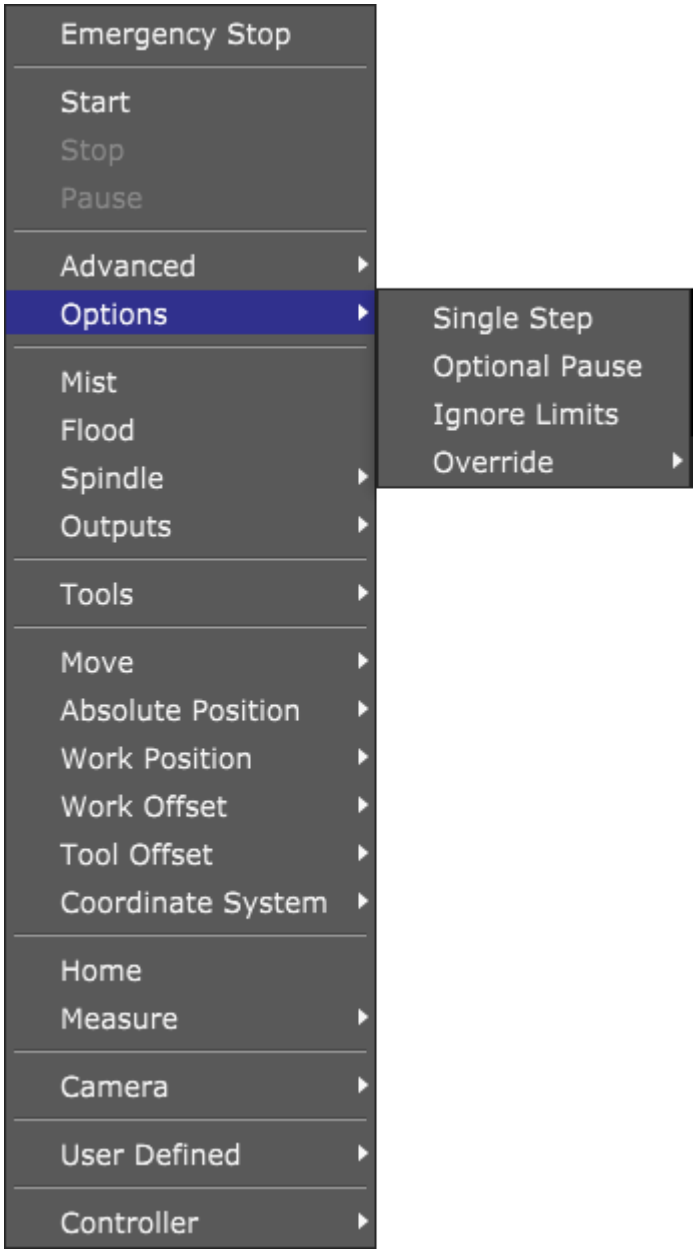
2.7.5.5 Beginning to Bookmark

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

2.7.5.6 Start From Bookmark to End

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

2.7.6 Options



2.7.6.1 Single Step

Enables single step execution of g-code program. Each program line is executed separately, for which user needs to press Start button each time. (Please note that this is not implemented yet)

2.7.6.2 Optional Pause

When *Optional pause* option is enabled, g-code M01 pauses program. If this option is disabled, software continues with program execution.

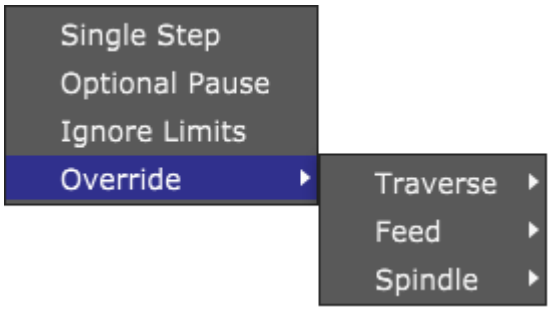
2.7.6.3 Ignore Limits

Ignores soft limits of machine. These limits are set in *File/Settings/Motion/Range*

2.7.6.4 Override

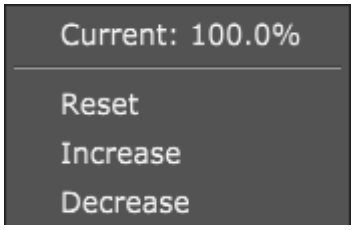
Overrides speed settings for *Traverse*, *Feed*(F-word) and *Spindle*(S-word) speeds.

These speeds are set with F or S words in g-code program or in settings: *File/Setting/Program Options.*



2.7.6.4.1 Traverse

Overrides speed setting for *Traverse* speed. Traverse speed setting is set in settings: *File/Setting/Program Options/Traverse Speed*



Current: Current override speed value. 100% means there is no speed override used. 200% means speed is overridden for 100%.

E.g.: If speed is set to 100mm/min and *Current* value is 200%, overridden traverse speed value is 200mm/min.

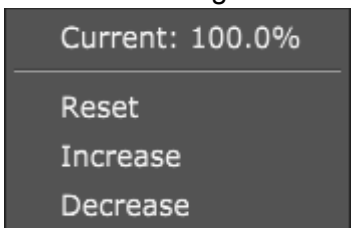
Reset: Resets any speed override.

Increase: Increases speed override setting for 10%.

Decrease: Decreases speed override setting for 10%.

2.7.6.4.2 Feed

Overrides speed setting for *Feed* speed. Feed speed setting is set in in g-code program with F-word or in settings: *File/Setting/Program Options/Feed Speed*



Current: Current override speed value. 100% means there is no speed override used. 200% means speed is overridden for 100%.

E.g.: If speed is set to 100mm/min and *Current* value is 200%, overridden feed speed value is 200mm/min.

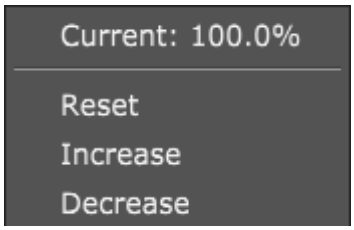
Reset: Resets any speed override.

Increase: Increases speed override setting for 10%.

Decrease: Decreases speed override setting for 10%.

2.7.6.4.3 Spindle

Overrides speed setting for *Spindle* speed. Spindle speed setting is set in in g-code program with S-word or in settings: *File/Setting/Program Options/Spindle Speed*



Current: Current override speed value. 100% means there is no speed override used. 200% means speed is overridden for 100%.

E.g.: If speed is set to 1000RPM and *Current* value is 200%, overridden spindle speed value is 2000RPM.

Reset: Resets any speed override.

Increase: Increases speed override setting for 10%.

Decrease: Decreases speed override setting for 10%.

2.7.7 Mist

Toggles *Mist* output pin. When *Mist* output is active, there is check next to Mist menu item and toolbar button for Mist is active.

M-code M07/M09 status is activated accordingly.

2.7.8 Flood

Toggles *Flood* output pin. When *Flood* output is active, there is check next to Flood menu item and toolbar button for Flood is active.

M-code M08/M09 status is activated accordingly.

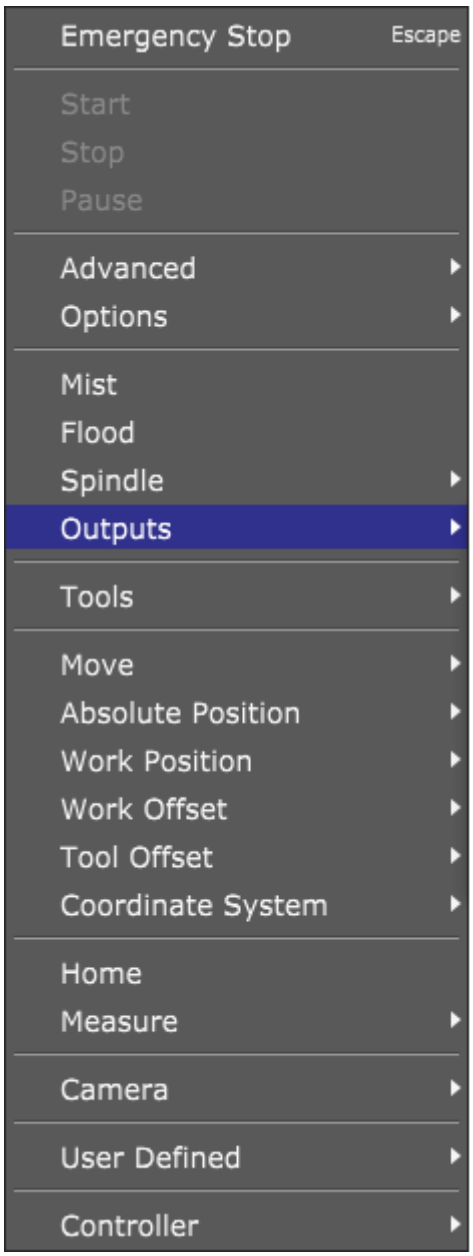
2.7.9 Spindle

Toggles *Spindle* output pin. When *Spindle* output is active, there is check next to Spindle menu item and toolbar button for Spindle is active.

M-code M03/M05 status is activated accordingly.

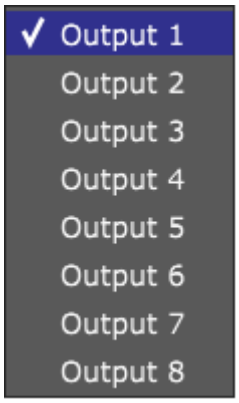
2.7.10 Outputs

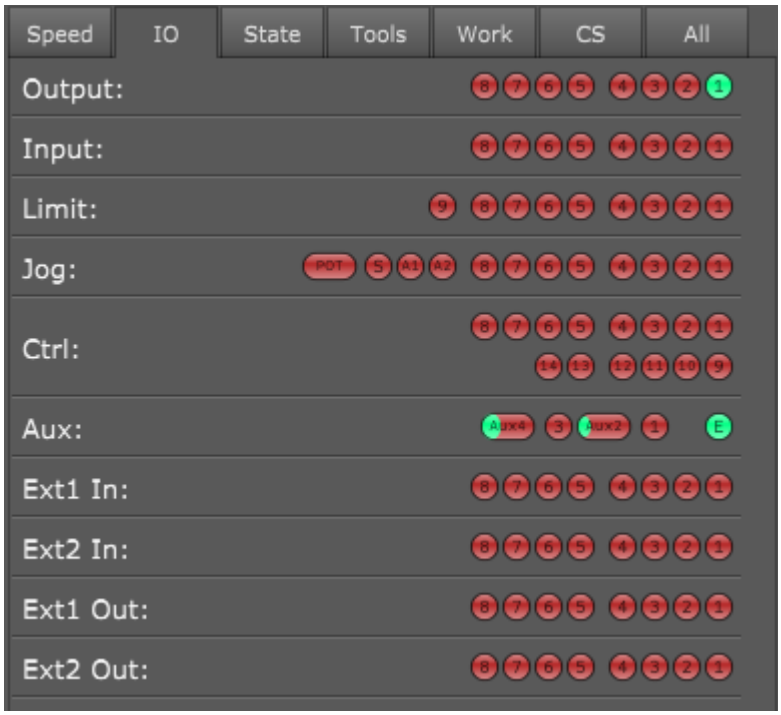
Controls digital outputs of controller via sub-menu.



2.7.10.1 Outputs 1-8:

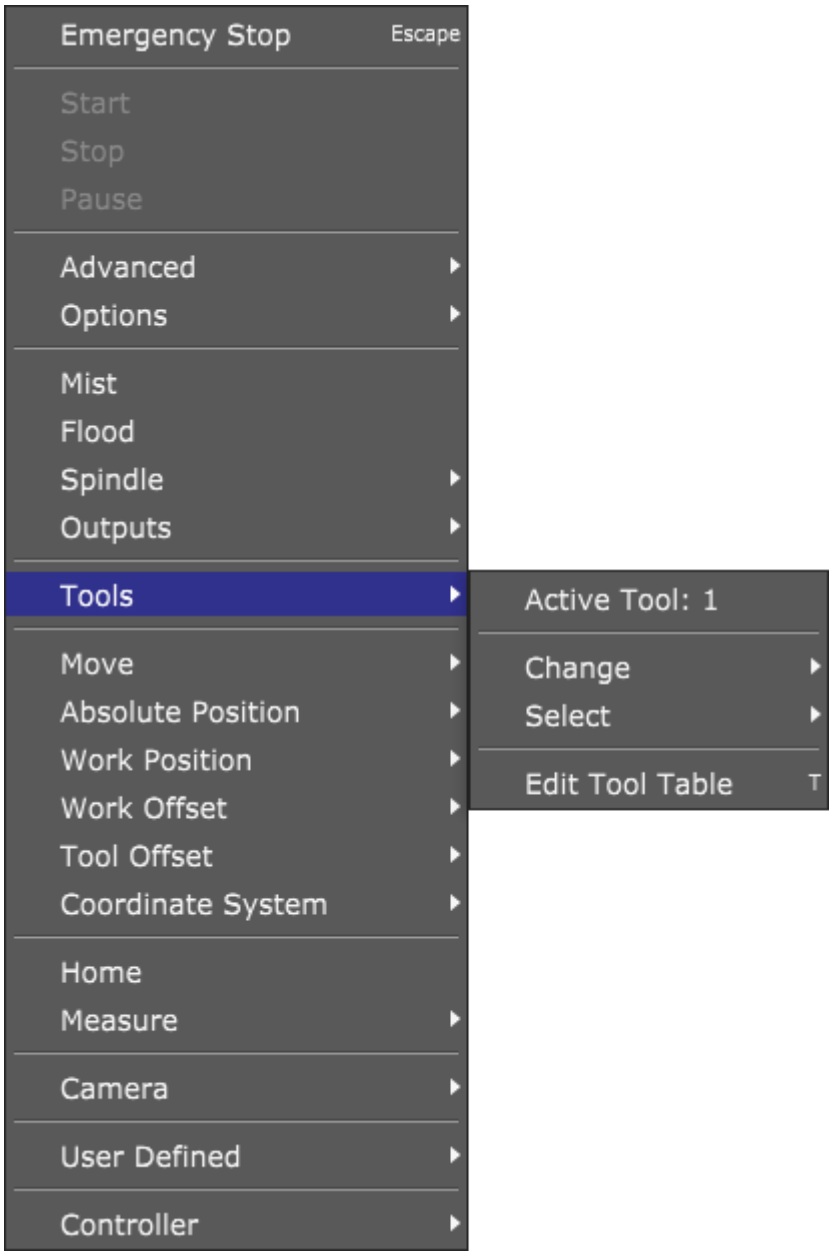
Selected digital output pin can be activated or deactivated. When output is active, there is check next to Output pin menu item. Active output pin will also be indicated under IO tab on the main screen.





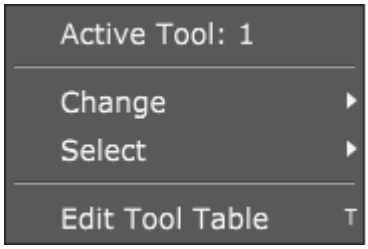
2.7.11 Tools

Provides means for changing and selecting tool as also editing a tool table.



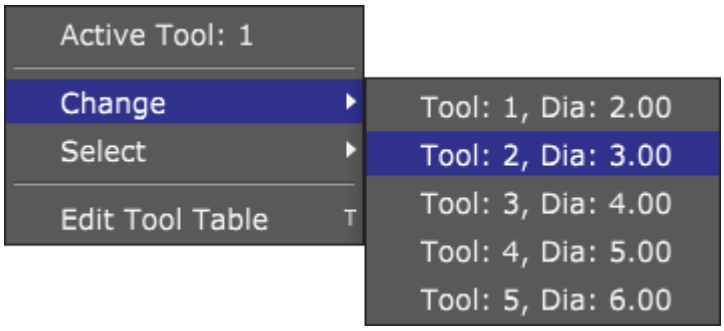
2.7.11.1 Active Tool

Displays current active tool.



2.7.11.2 Change

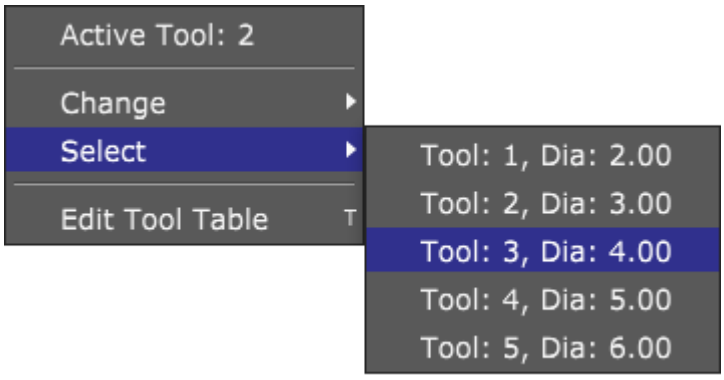
Initiates tool change procedure. Newly changed tool will be set as active tool. If tool table is created, user can select tool from sub-menu. Equivalent to *Change* action is *M6* g-code.



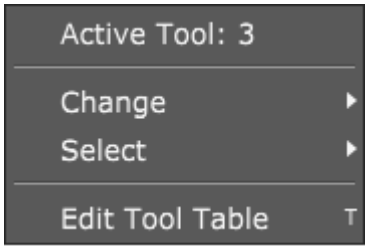
Newly active tool upon tool change to Tool 2:

2.7.11.3 Select

Sets active tool without tool change procedure. If tool table is created, user can select tool from sub-menu. Equivalent to *Select* action is *M61 Qn* g-code, where 'n' is tool number.

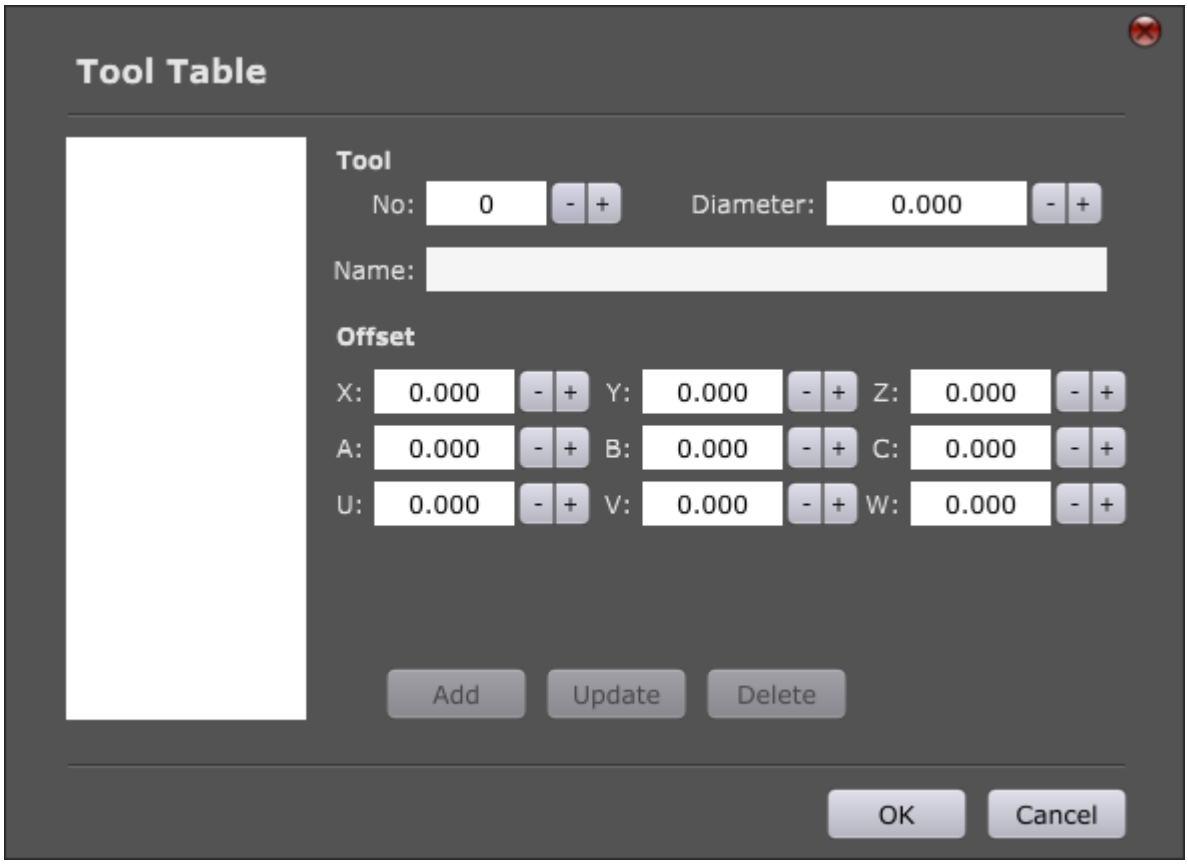


Newly active tool upon selecting Tool 3:



2.7.11.4 Edit Tool Table

Creates tool table with predefined diameters and offsets.



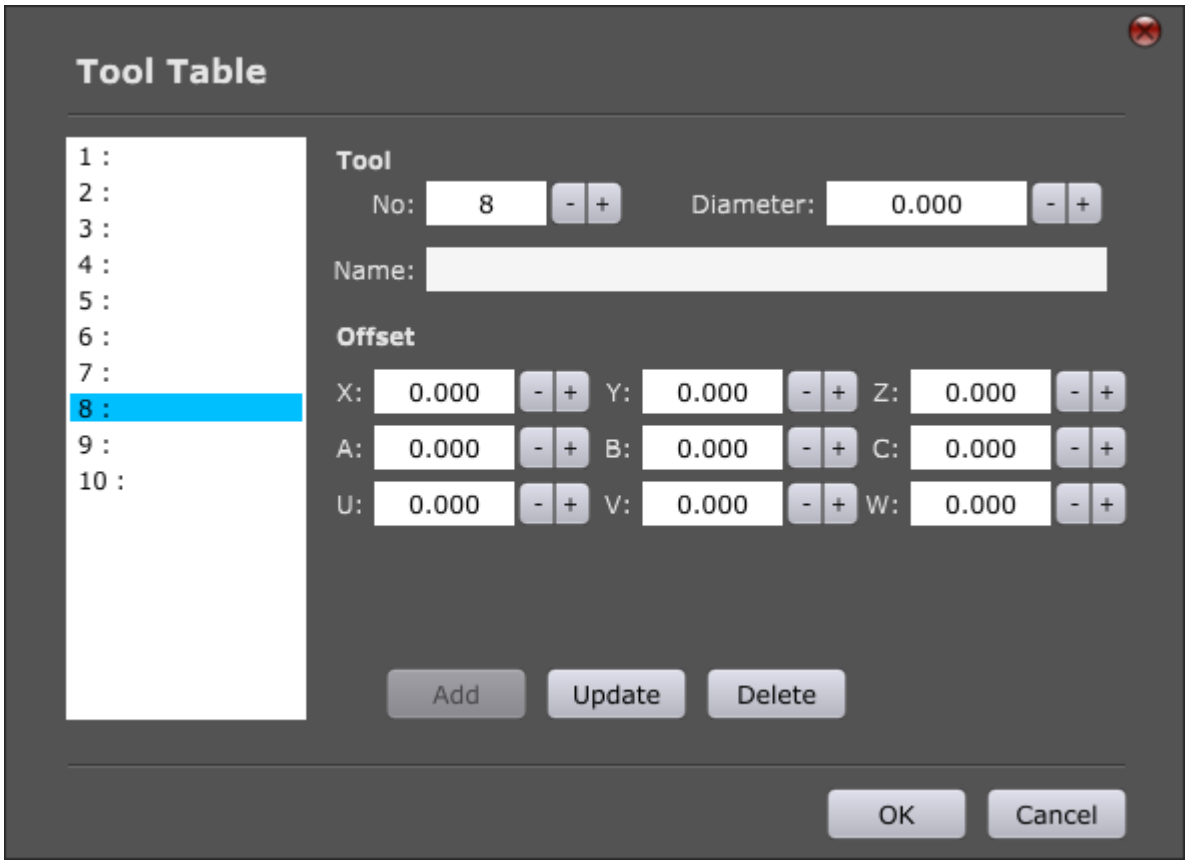
2.7.11.4.1 Tool

No:

Sets number of tool. User can choose from numbers 1-255.

Number can be set directly by writing into insert bar or with '+' and '–' buttons.

If tool number entered is the same as one of the tools that is already populated in tool table, then that tool will be highlighted:

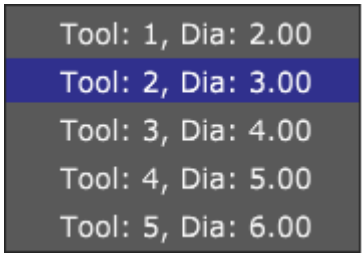


Diameter:

Sets diameter of tool. User can choose from numbers 1-10000.

Number can be set directly by writing into insert bar or with '+' and '–' buttons.

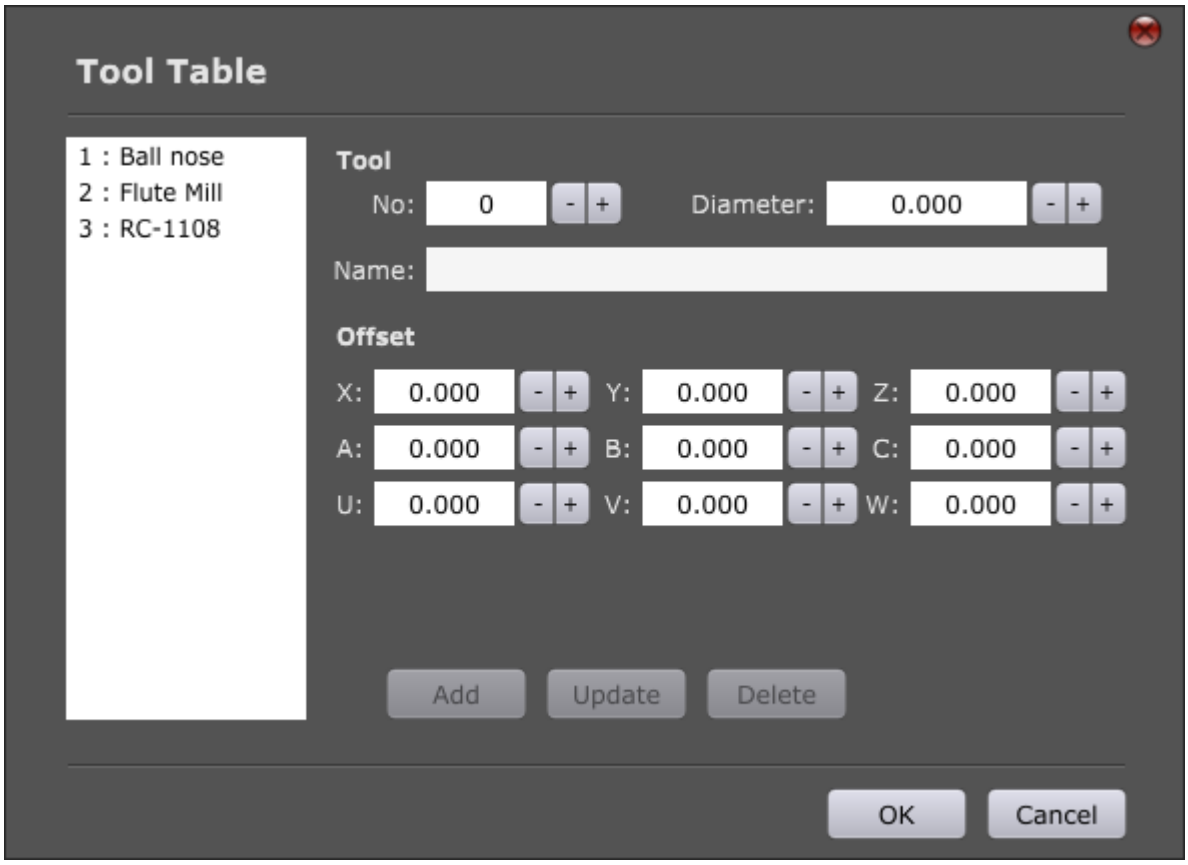
Tools diameter value will be visible in *Change* and *Select* sub-menu:



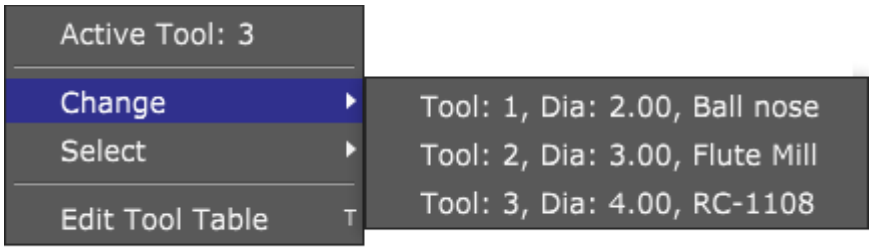
Name:

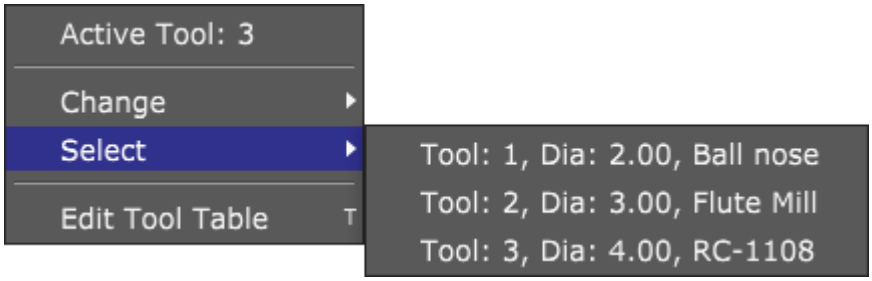
Sets name of tool.

Tool name will be visible in tool table list:



and in *Change* and *Select* sub-menu:





2.7.11.4.2 Offset

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

Sets tool offset of tool. Offset can be set in all 9 axes.

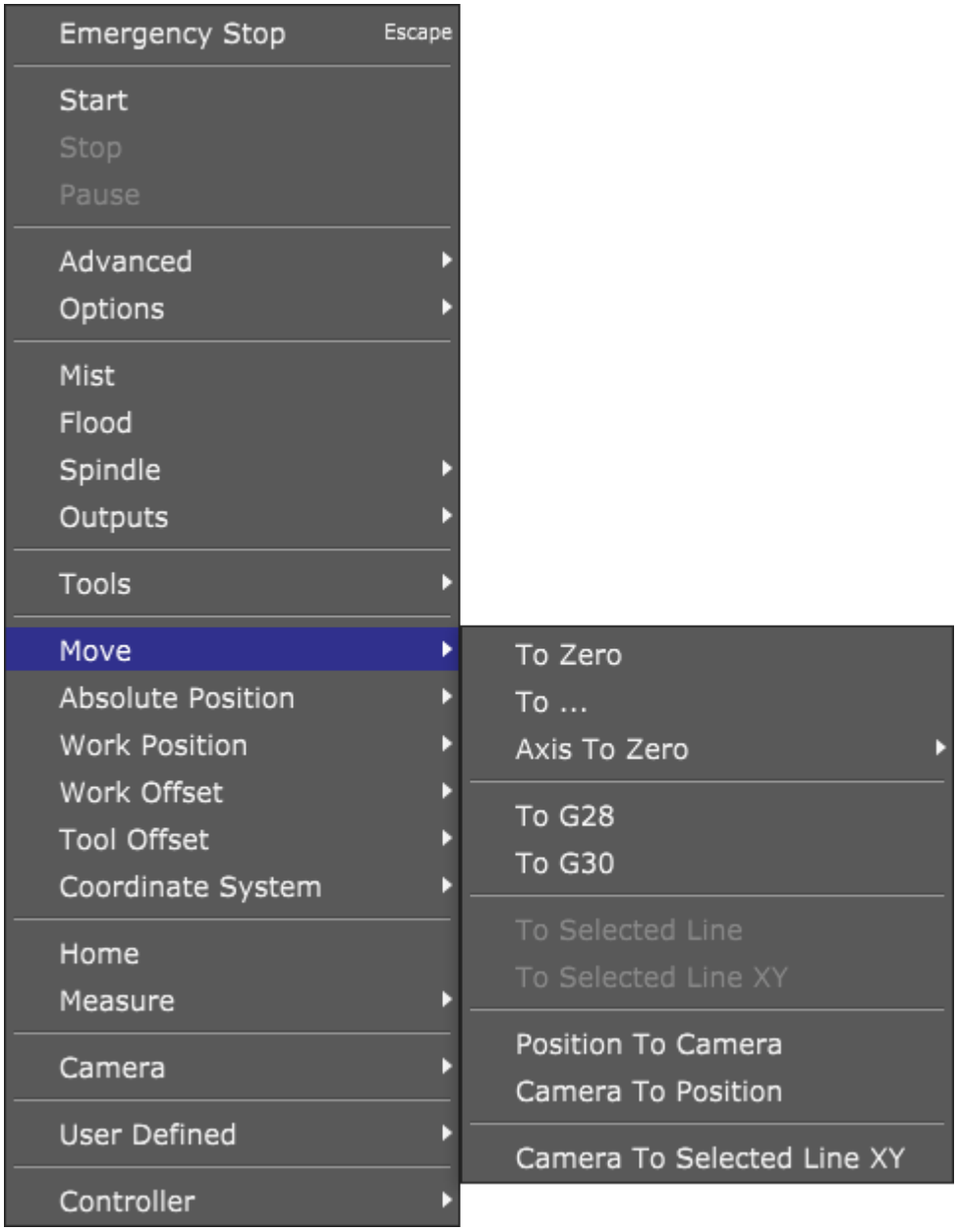
Example: Creating a new tool in tool table

Click *Machine/Tools/Edit Tool Table*. Under Tool No.: insert number for new tool. Insert *Diameter* value and its name. Insert tool offset, usually only in Z axis. Then click *Add*. Tool will appear in the tool list.

To change parameters of already created tool select tool from tool list and change tools parameters. Then click *Update.*

2.7.12 Move

Provides means to move all machine axes or selected axis/axes to zero or predefined position values.

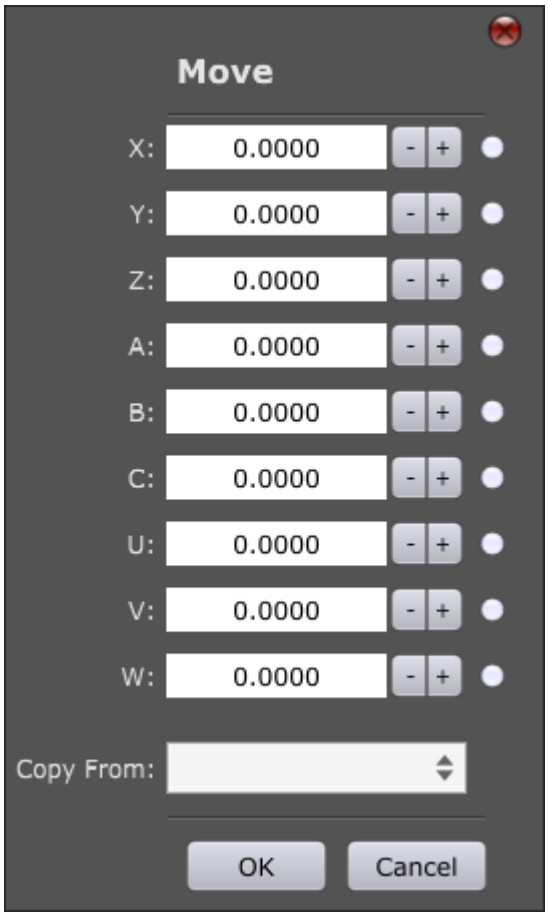


2.7.12.1 To Zero

Moves all machine axes to zero position value. If no *Work Offset* is set machine will move to absolute zero. If *Work Offset* is set, machine will move to zero *Work Position.*

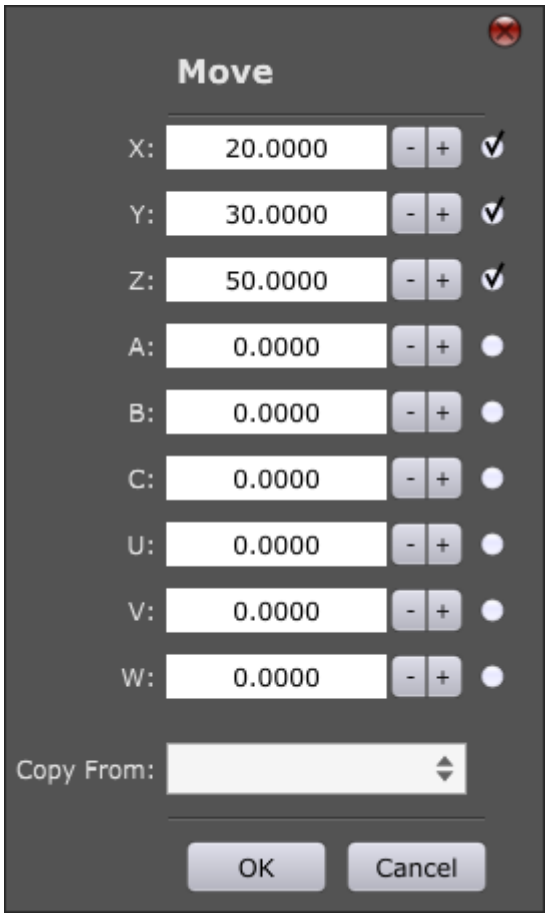
2.7.12.2 To...

Provides means to move selected axis/axes to desired position.



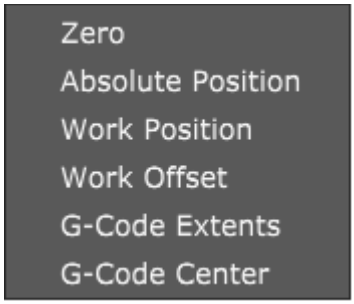
You can move all 9 axes: **X,Y,Z,A,B,C,U,V,W.** To move selected axis to specified position, enable axis radio button, insert axis position and click OK.

E.g.: Move machine axes X,Y and Z to positions 20,30,50:



2.7.12.2.1 Copy From

With *Copy From* user can choose among options provided from drop down menu for faster population of axis position values.



Zero:

Populates all position values of *Move* dialog with zero values.

Absolute Position:

Populates all position values of *Move* dialog with machines current absolute position values.

Work Position:

Populates all position values of *Move* dialog with machines current working position values.

Work Offset:

Populates all position values of *Move* dialog with current working offset values.

G-Code Extents:

Populates all position values of *Move* dialog with g-code program extent values.

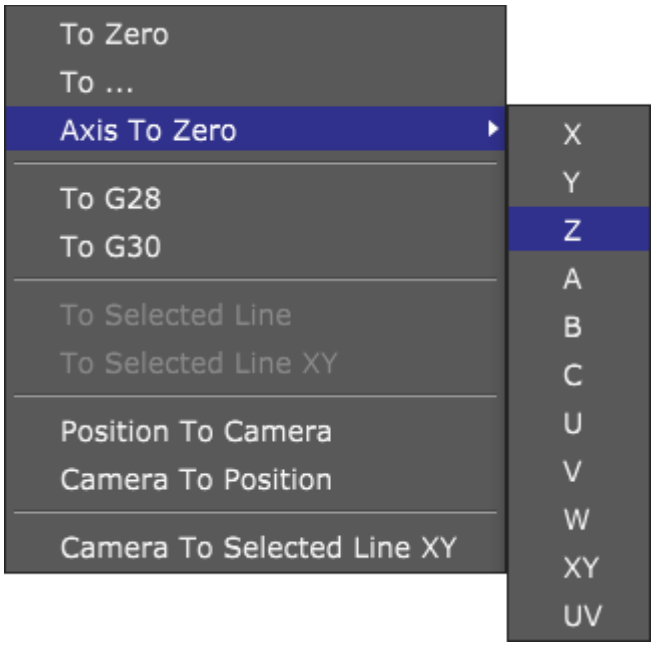
G-Code Center:

Populates all position values of *Move* dialog with g-code program center values.

2.7.12.3 Axis to Zero

Moves selected machine axis to zero position value. If no *Work Offset* is set for selected axis, axis will move to absolute zero position value. If *Work Offset* is set for selected axis, axis will move to axis zero *Work Position*.

User can select axis from sub menu:



2.7.12.4 To G28

Moves machine to *G28* position (preset absolute machine position). To set G28 absolute position value see chapter *2.5.13.6 As G28.*

2.7.12.5 To G30

Moves machine to G30 position (preset absolute machine position). To set G30 absolute position value see chapter *2.5.13.7 As G30.*

2.7.12.6 To Selected Line

Moves machine to position of selected g-code line in g-code window.

2.7.12.7 To Selected Line XY

Moves machine to XY position value of selected g-code line in g-code window.

2.7.12.8 Position to Camera

Moves machine to XY position of camera view.

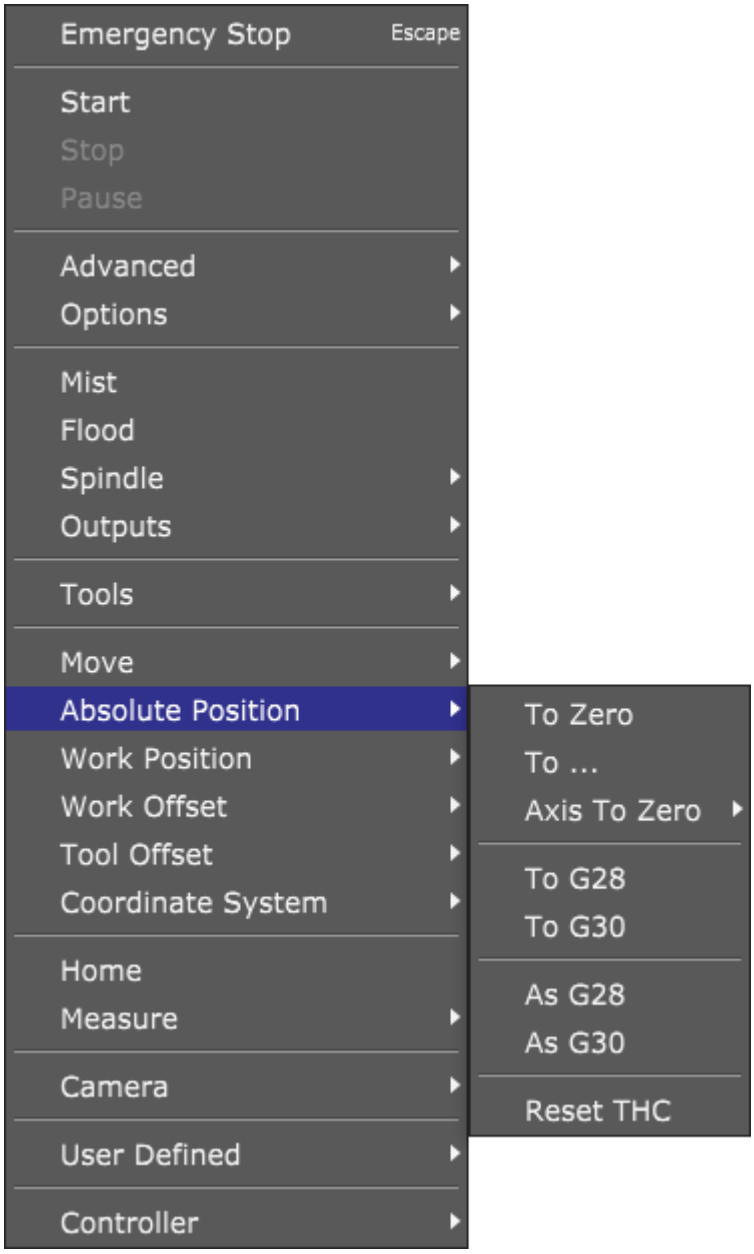
2.7.12.9 Camera To Position

Camera view moves to current machine position.

2.7.12.10 Camera To Selected Line XY

Camera view moves to XY position value of selected g-code line in g-code window.

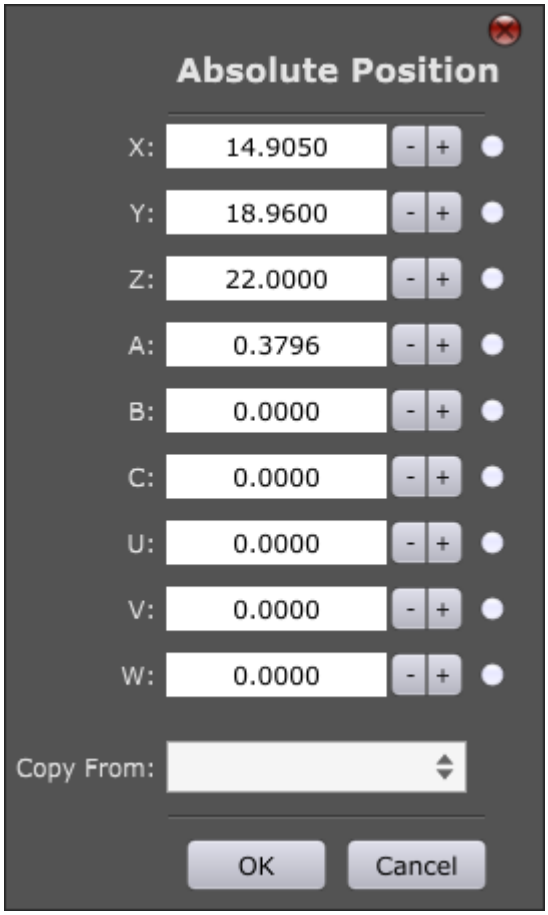
2.7.13 Absolute Position



2.7.13.1 To Zero

Sets all absolute position values of machine axes to zero.

2.7.13.2 To...

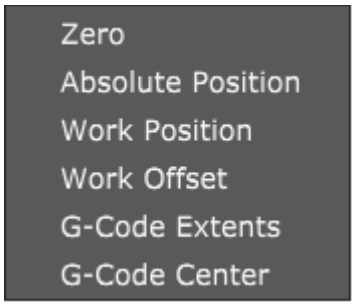


When dialog is displayed, machines current absolute position values are already populated.

You can set absolute position values for all 9 axes: **X,Y,Z,A,B,C,U,V,W.** To set absolute position for selected axis, enable axis radio button, insert axis absolute position and click OK.

2.7.13.2.1 Copy From

With *Copy From* user can choose among options provided from drop down menu for faster population of axis absolute position values.



Zero:

Populates all absolute position values of *Absolute position* dialog with zero values.

Absolute Position:

Populates all absolute position values of *Absolute position* dialog with machines current absolute position values.

Work Position:

Populates all absolute position values of *Absolute position* dialog with machines current working position values.

Work Offset:

Populates all absolute position values of *Absolute position* dialog with current working offset values.

G-Code Extents:

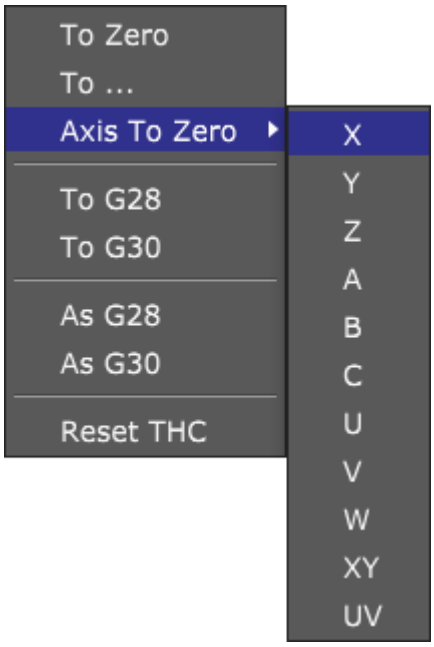
Populates all absolute position values of *Absolute position* dialog with g-code program extent values.

G-Code Center:

Populates all absolute position values of *Absolute position* dialog with g-code program center values.

2.7.13.3 Axis To Zero

Sets selected machine axis to zero absolute position value. User can select axis from sub menu:



2.7.13.4 To G28

Sets machine absolute position values to values set with G28 (preset absolute machine position). To set G28 absolute position value see chapter *2.5.13.6 As G28.*

2.7.13.5 To G30

Sets machine absolute positions values to values set with G30 position(preset absolute machine position). To set G30 absolute position value see chapter *2.5.13.7 As G30.*

2.7.13.6 As G28

Sets current absolute machine position as G28 position.

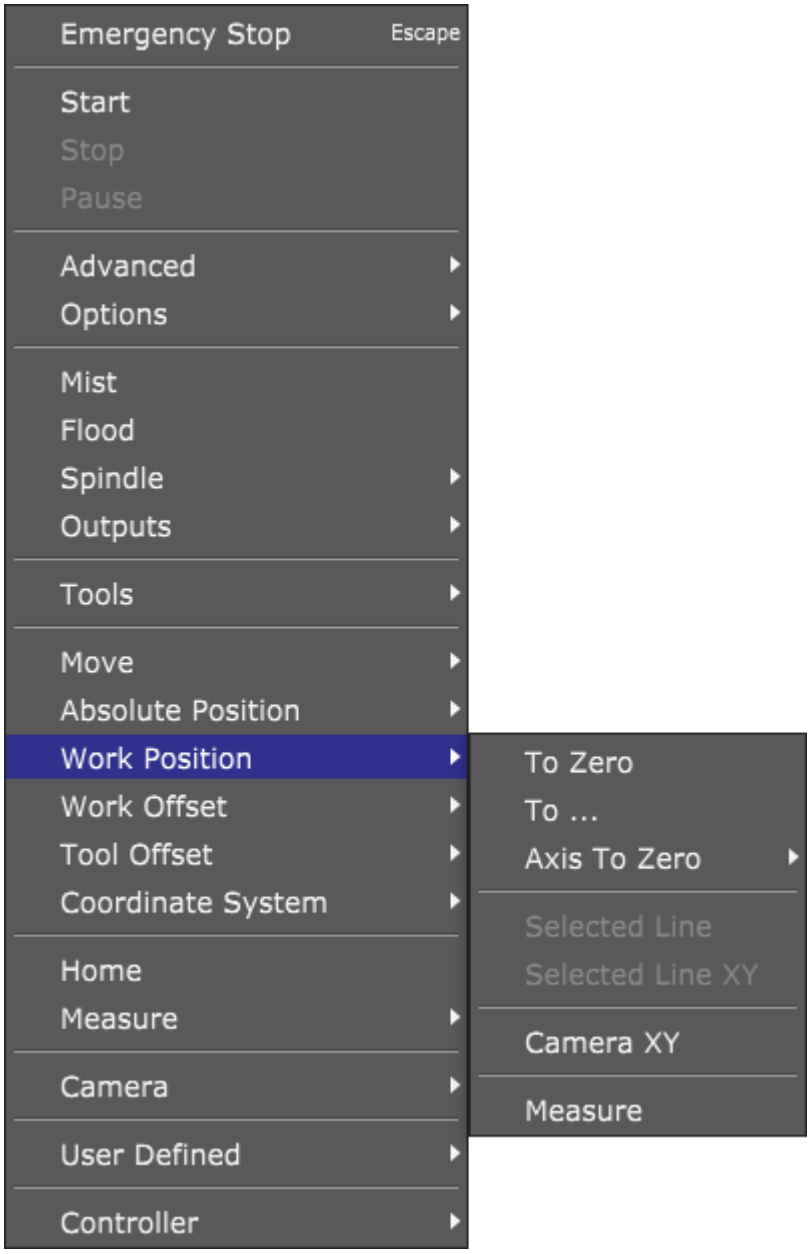
2.7.13.7 As G30

Sets current absolute machine position as G30 position.

2.7.13.7.1 Reset THC

When using THC device, THC sends controller compensation commands. Controller then adds “THC” offset accordingly. *Reset THC* deletes this “THC” offset.

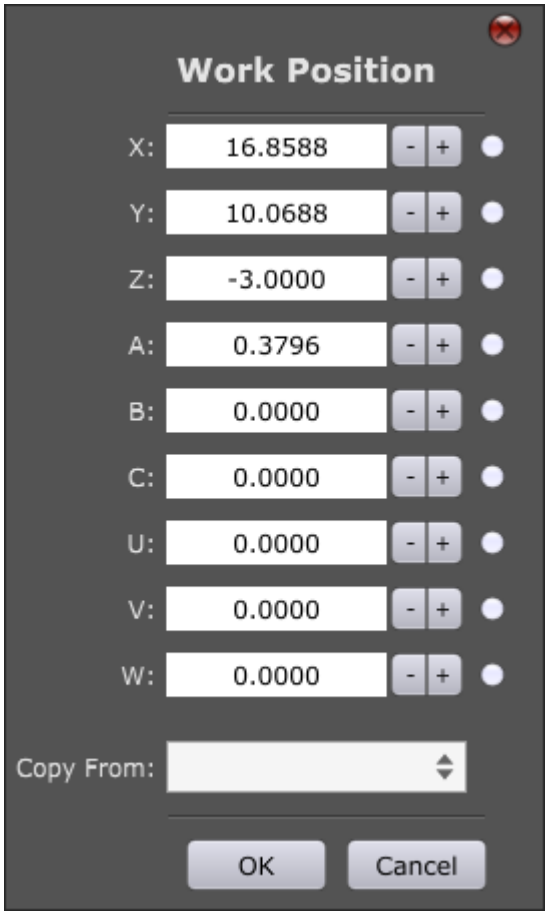
2.7.14 Work Position



2.7.14.1 To Zero

Sets all work position values of machine axes to zero.

2.7.14.2 To...

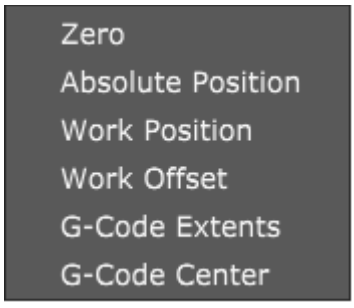


When dialog is displayed, machines current working position values are already populated.

You can set working position values for all 9 axes: **X,Y,Z,A,B,C,U,V,W.**

To set working position for selected axis, enable axis radio button, insert axis working position and click OK.

2.7.14.2.1 Copy From



Zero:

Populates all working position values of *Working position* dialog with zero values.

Absolute Position:

Populates all working position values of *Working position* dialog with machines current absolute position values.

Work Position:

Populates all working position values of *Working position* dialog with machines current working position values.

Work Offset:

Populates all working position values of *Working position* dialog with current working offset values.

G-Code Extents:

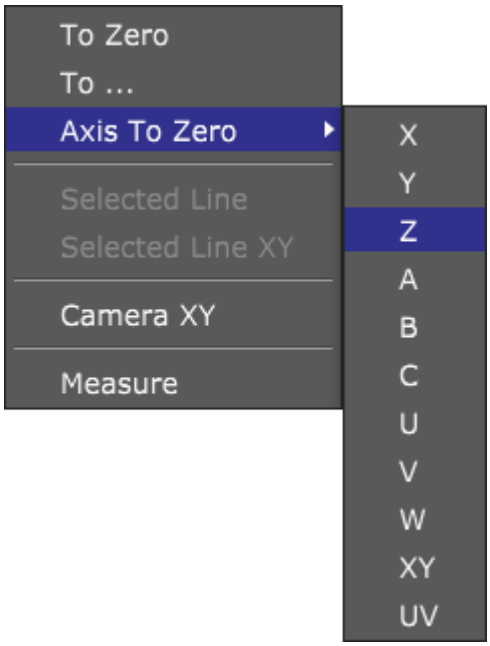
Populates all working position values of *Working position* dialog with g-code program extent values.

G-Code Center:

Populates all working position values of *Working position* dialog with g-code program center values.

2.7.14.3 Axis To Zero

Sets selected machine axis working position value to zero. User can select axis from sub menu:



2.7.14.4 To Selected Line

Work position values of machine axes are set with position values of selected g-code line in g-code window.

2.7.14.5 To Selected Line XY

Work position values of machine XY axes are set with XY position values of selected g-code line in g-code window.

2.7.14.6 Camera XY

Work position values of machine XY axes are set with Camera view.

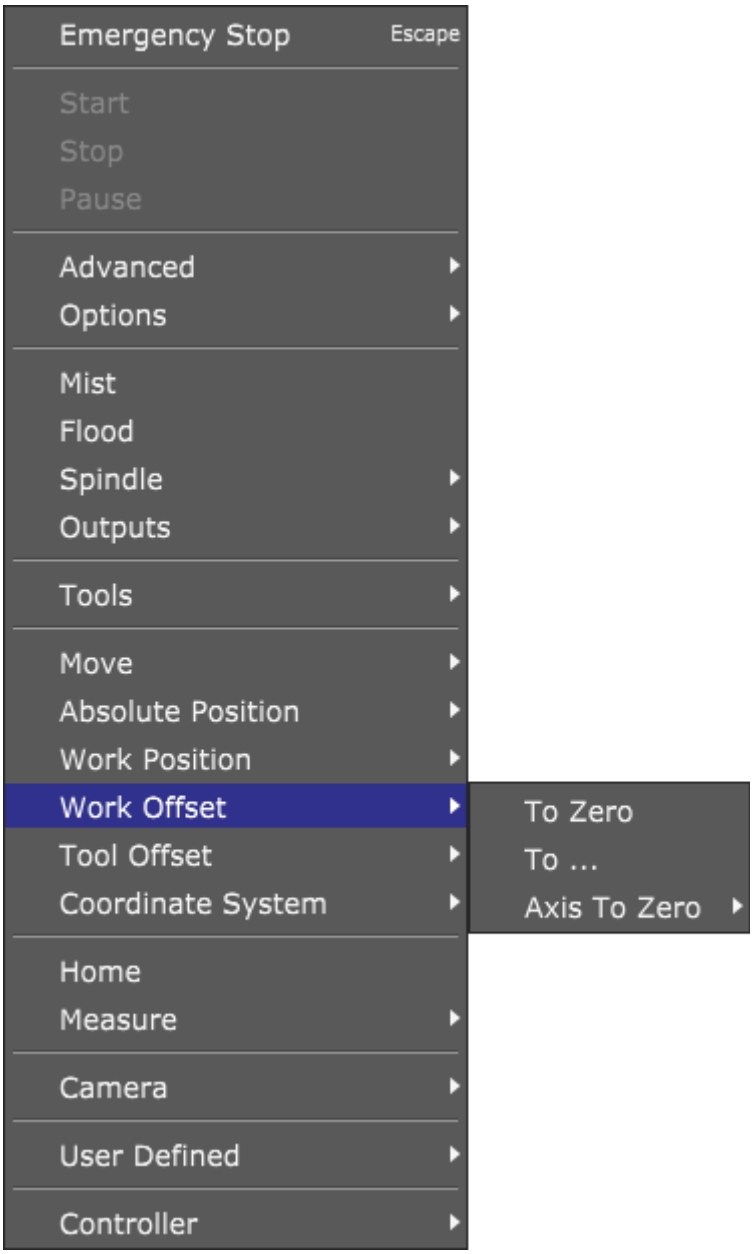
2.7.14.7 Measure

Initiates *Work Position Z - Measure* procedure.

Z axis will start descending at current machine position and when sensor/probe input is activated(surface of material), software sets Work Position Z to zero.

Please note: To avoid any damage, please check if movable sensor/probe is connected to controller input and sensor inputs is configured in settings: *File/Settings/Program Options/Measure/Sensor 1 or Sensor 2*

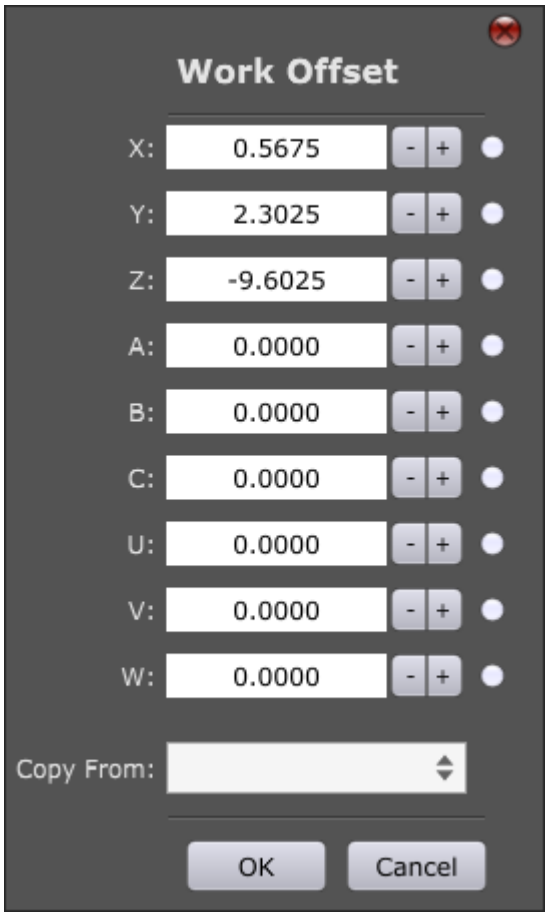
2.7.15 Work Offset



2.7.15.1 To Zero

Sets Work Offset value to zero for all machine axes.

2.7.15.2 To...

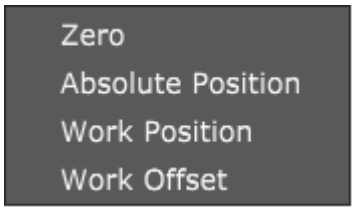


When dialog is displayed, current work offset values are already populated.

You can set work offset values for all 9 axes: **X,Y,Z,A,B,C,U,V,W.**

To set work offset for selected axis, enable axis radio button, insert axis work offset and click OK.

2.7.15.2.1 Copy From



Zero:

Populates all work offset values of *Work Offset* dialog with zero values.

Absolute Position:

Populates all work offset values of *Work Offset* dialog with machines current absolute position values.

Work Position:

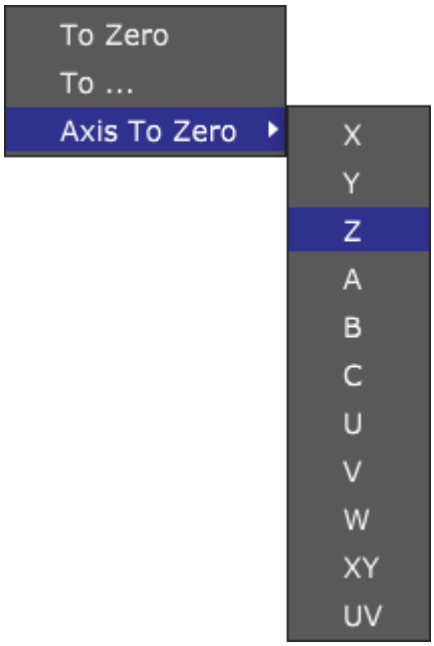
Populates all work offset values of *Work Offset* dialog with machines current working position values.

Work Offset:

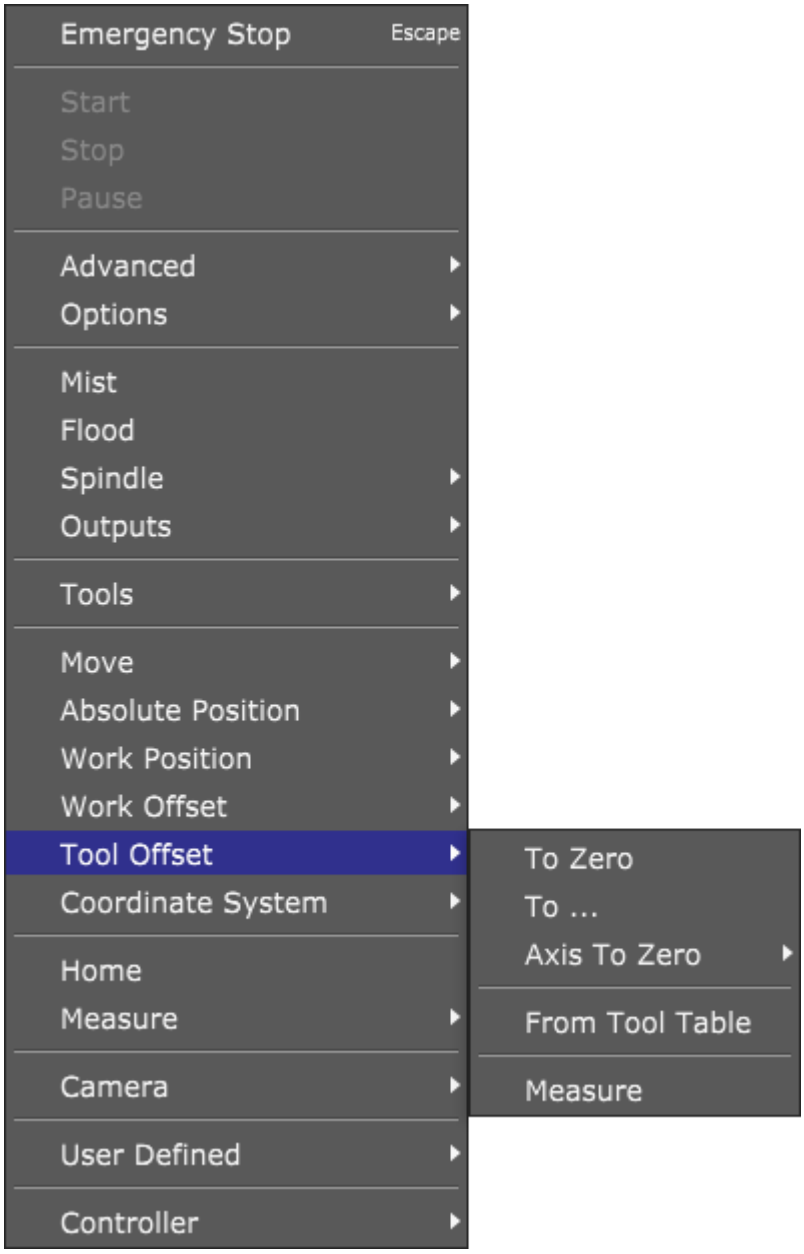
Populates all work offset values of *Work Offset* dialog with current working offset values.

2.7.15.3 Axis To Zero

Sets selected machine axis work offset value to zero. User can select axis from sub menu:



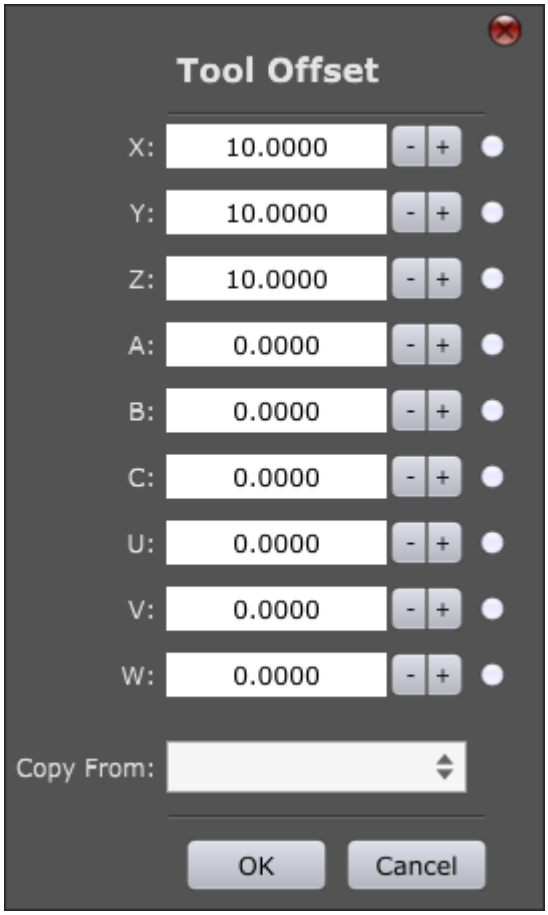
2.7.16 Tool Offset



2.7.16.1 To Zero

Sets Tool Offset value to zero for all machine axes.

2.7.16.2 To...

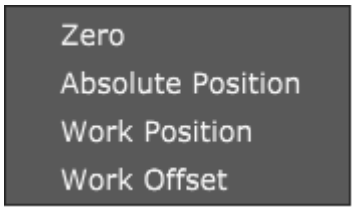


When dialog is displayed, current tool offset values are already populated.

You can set tool offset values for all 9 axes: **X,Y,Z,A,B,C,U,V,W.**

To set tool offset for selected axis, enable axis radio button, insert axis tool offset and click OK.

2.7.16.2.1 Copy From



Zero:

Populates all tool offset values of *Tool Offset* dialog with zero values.

Absolute Position:

Populates all tool offset values of *Tool Offset* dialog with machines current absolute position values.

Work Position:

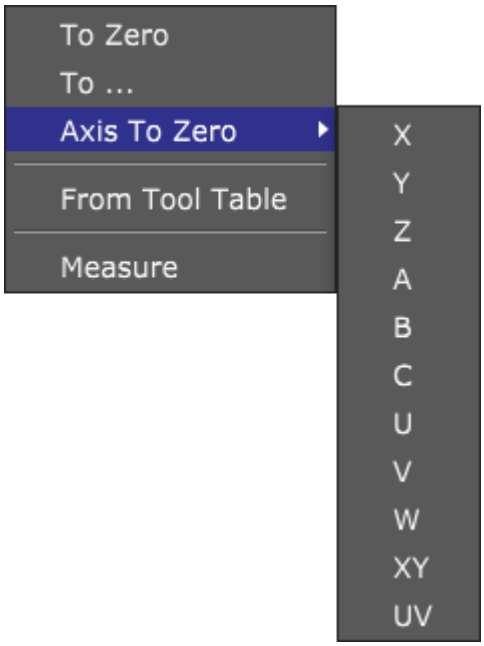
Populates all tool offset values of *Tool Offset t* dialog with machines current working position values.

Work Offset:

Populates all tool offset values of *Tool Offset* dialog with current working offset values.

2.7.16.3 Axis to Zero

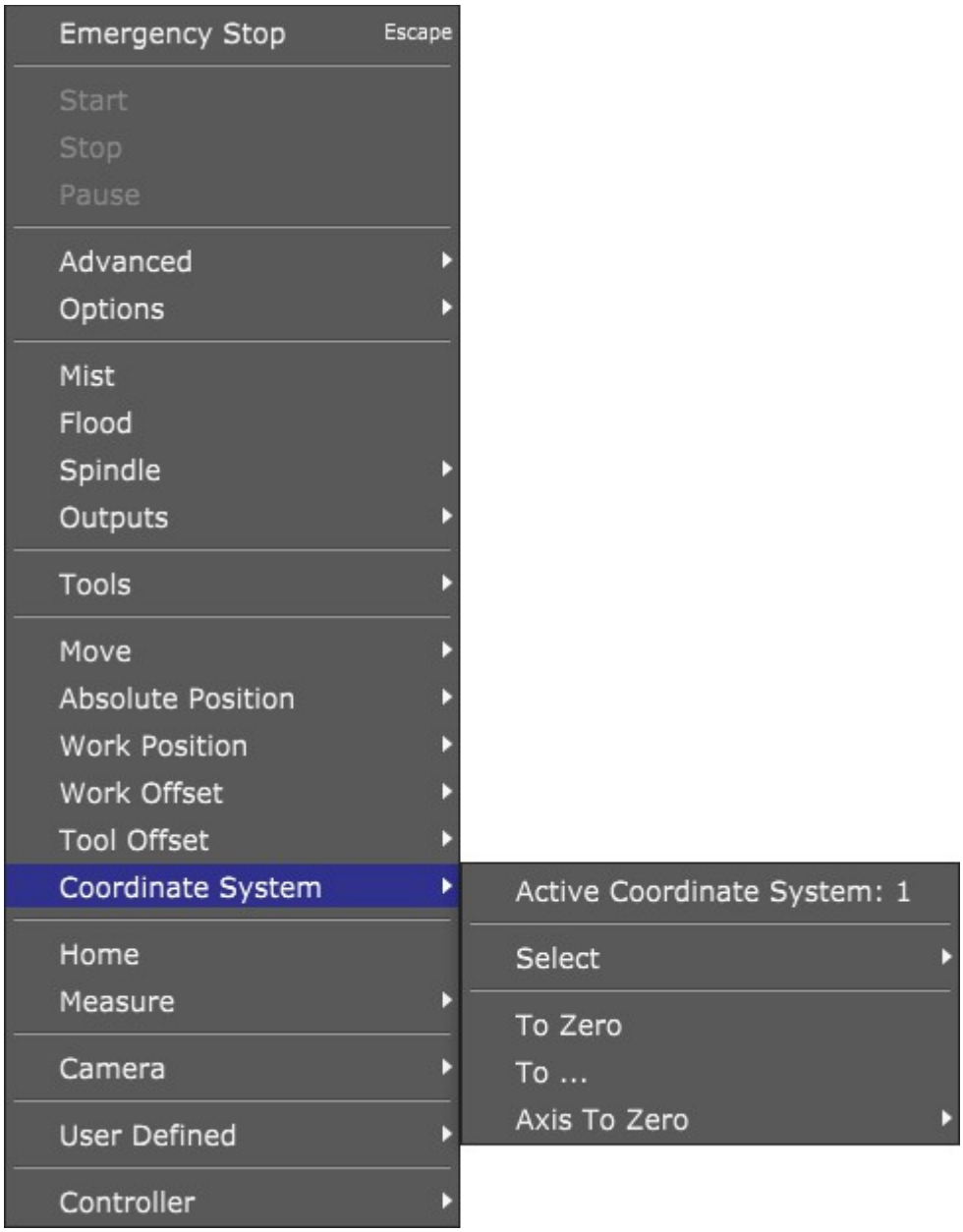
Sets selected machine axis tool offset value to zero. User can select axis from sub menu:



2.7.16.4 From Tool Table

Sets *Tool Offset* value for current from tool table.

2.7.17 Coordinate system

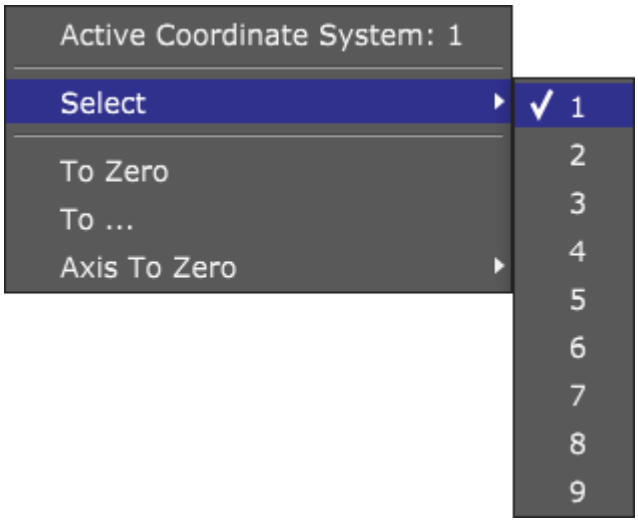


2.7.17.1 Active Coordinate system

Displays currently active coordinate system.

2.7.17.2 Select

Sets active coordinate system. User can choose among 9 coordinate systems. Active coordinate system is selected from sub menu:

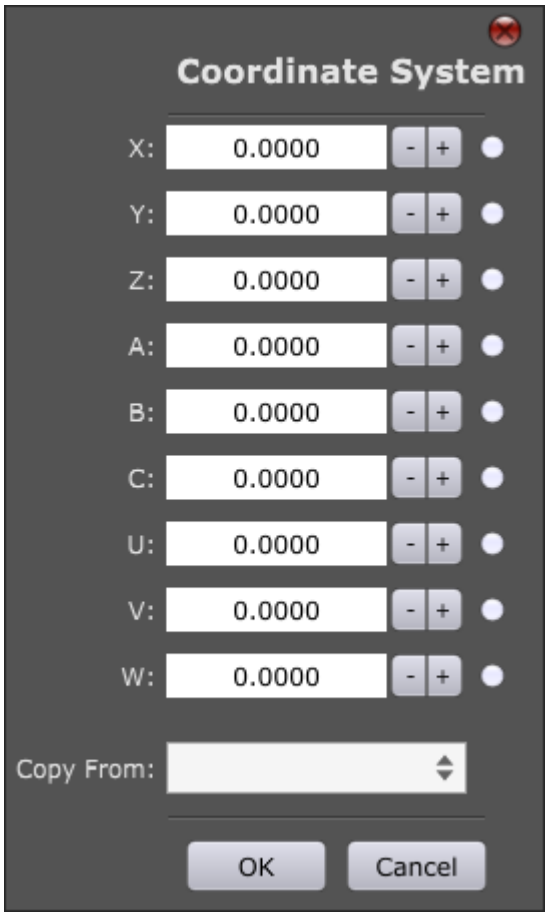


2.7.17.3 To Zero

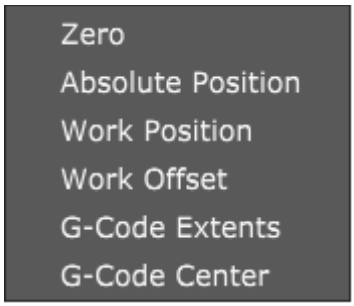
Sets currently active coordinate system's offset to zero.

2.7.17.4 To...

Opens a dialog where user can set offset value for each axis of currently active coordinate system.



2.7.17.4.1 Copy From



Zero:

Populates all axis offset values of *Coordinate System* dialog with zero values.

Absolute Position:

Populates all axis offset values of *Coordinate System* dialog with machines current absolute position values.

Work Position:

Populates all axis offset values of *Coordinate System* dialog with machines current working position values.

Work Offset:

Populates all axis offset values of *Coordinate System* dialog with current working offset values.

G-Code Extents:

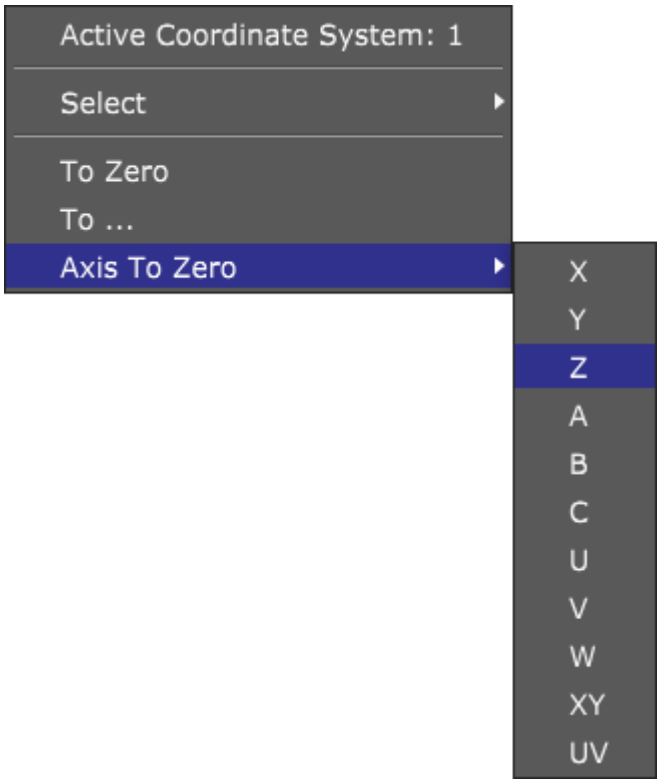
Populates all axis offset values of *Coordinate System* dialog with g-code program extent values.

G-Code Center:

Populates all axis offset values of *Coordinate System* dialog with g-code program center values.

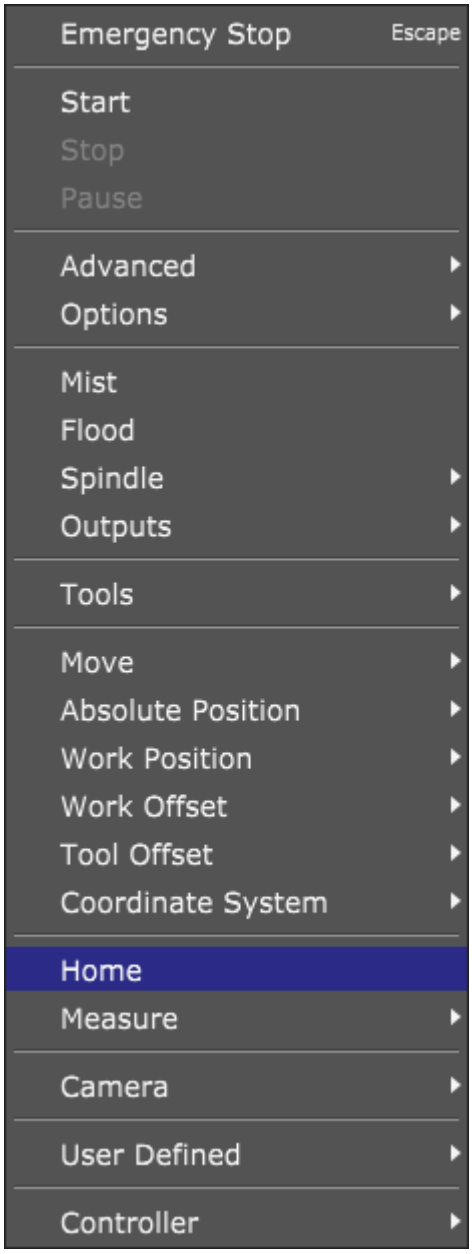
2.7.17.5 Axis To Zero

Sets offset value to zero for selected axis of active coordinate system. User can select axis from sub menu:

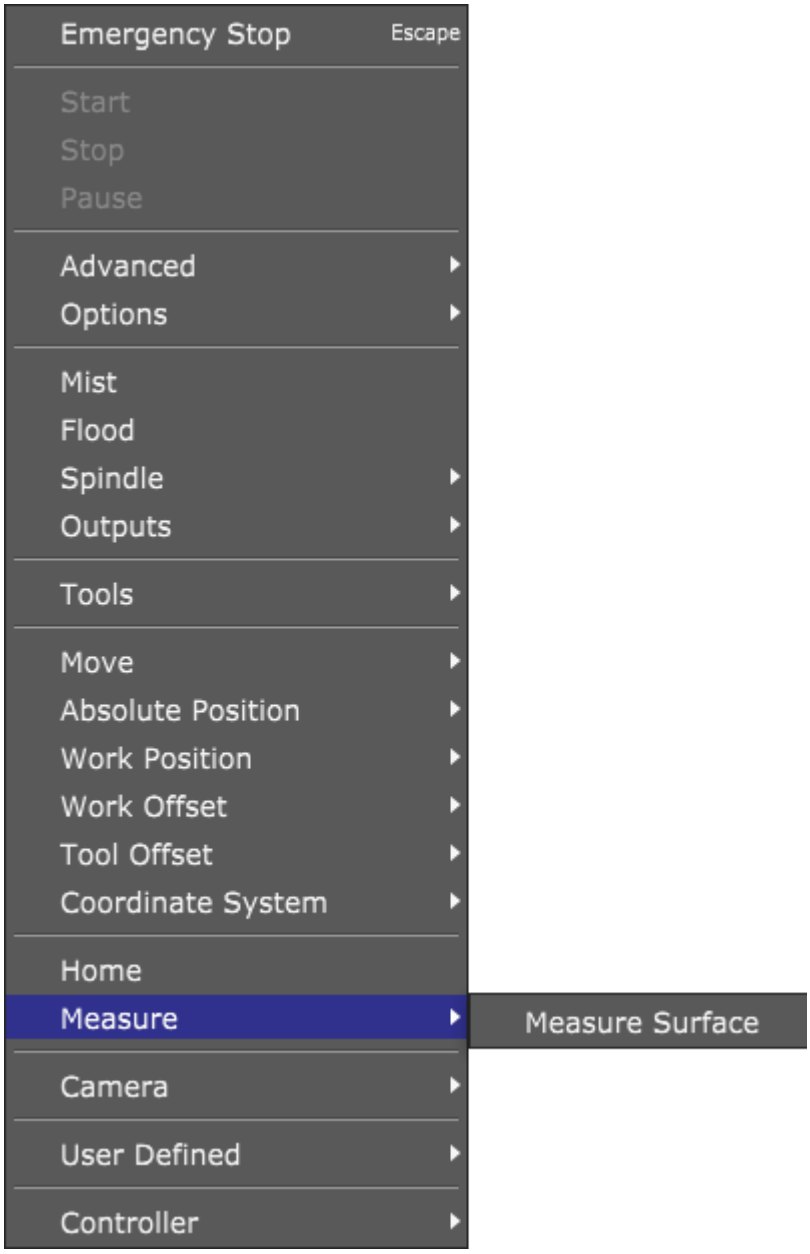


2.7.18 Home

Initiates homing procedure. On how to configure homing procedure please refer to chapter: *2.7.8.8. Homing*

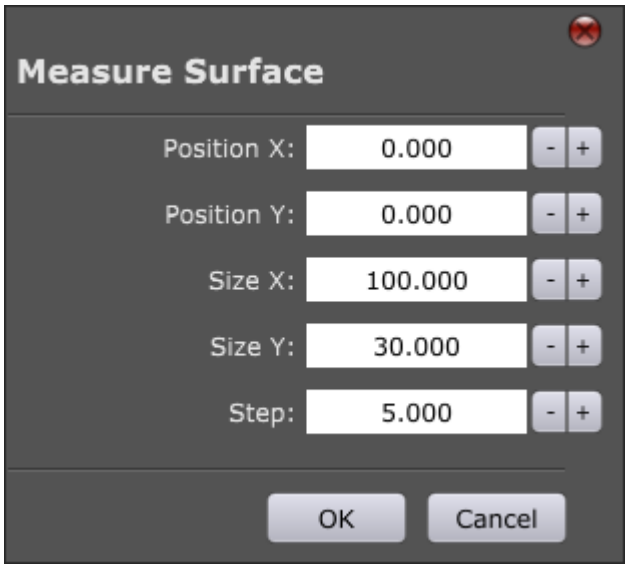


2.7.19 Measure



2.7.19.1 Measure surface

Opens up a dialog for *Measure Surface* procedure.



2.7.19.1.1 Position X:

X axis start position of *Measure Surface* procedure. If work offset is set, then this is relative coordinate value, if work offset is not set, then this is absolute coordinate value.

2.7.19.1.2 Position Y:

Y axis start position of *Measure Surface* procedure. If work offset is set, then this is relative coordinate value, if work offset is not set, then this is absolute coordinate value.

2.7.19.1.3 Size X:

X axis max. dimension of measure surface.

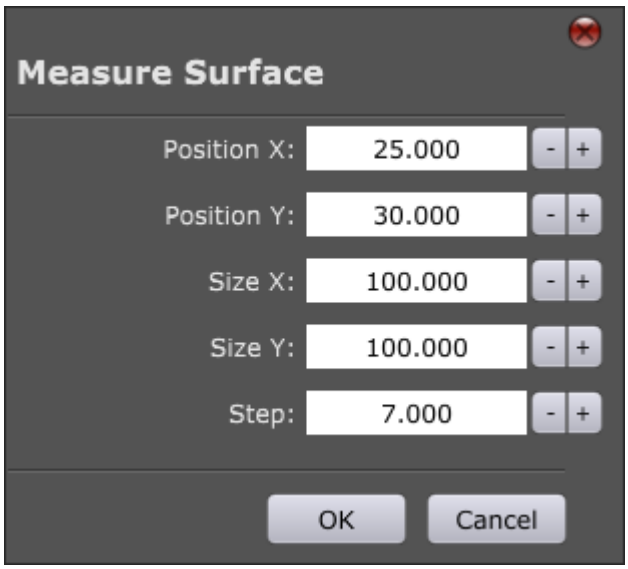
2.7.19.1.4 Size Y:

Y axis max. dimension of measure surface.

2.7.19.1.5 Step:

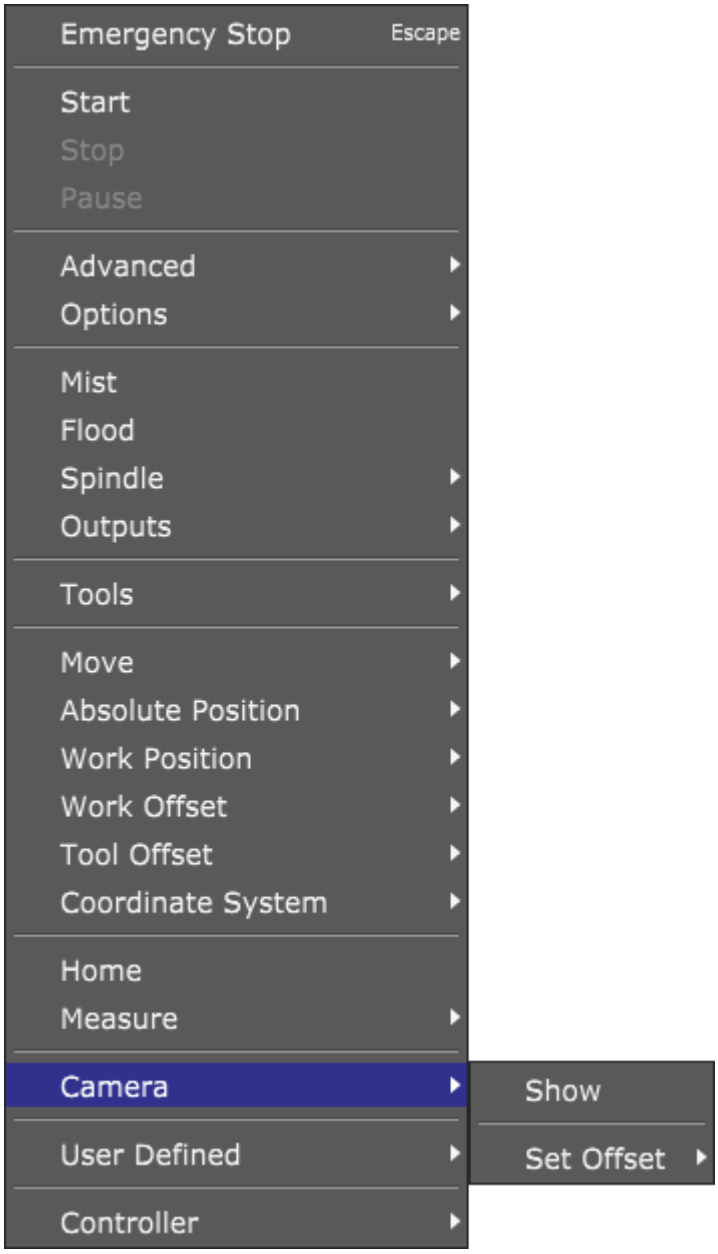
Distance increment between two neighboring measuring points.

Example: We need to measure surface of dimension 100mm x 100mm. There should be 7 mm between each measuring points. We want to start measuring at position X25, X30.



Please note: To avoid any damage, please check if movable sensor/probe is connected to controller input and sensor inputs is configured in settings: *File/Settings/Program Options/Measure/Sensor 1 or Sensor 2*

2.7.20 Camera

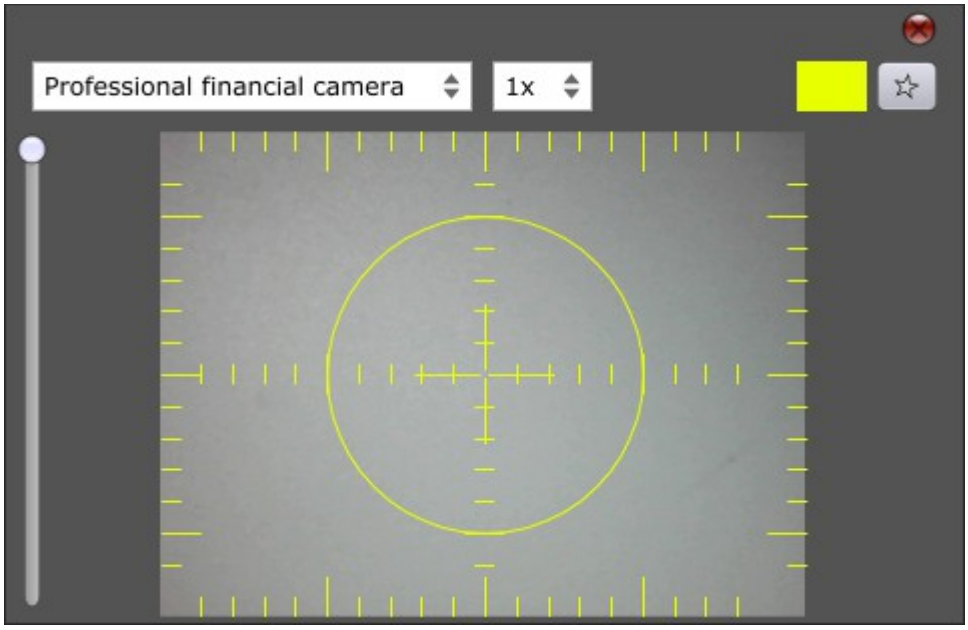


2.7.20.1 Show

Camera Show 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.

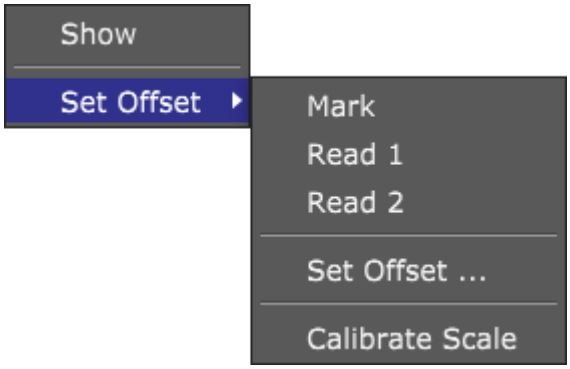
Available USB web-cams are listed in a drop-down in the center of the panel. Depending on attached hardware, options to use camera filters may also be shown. PlanetCNC TNG software provides adjustment, so the basic web-cam option is selected. Basic web-cam hardware is suitable. It is much less costly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

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.



2.7.20.2 Set Offset

Camera Offset is set here.



2.7.20.2.1 Mark

Marks camera reference point on machine table which will be used for setting camera offset.

2.7.20.2.2 Read 1

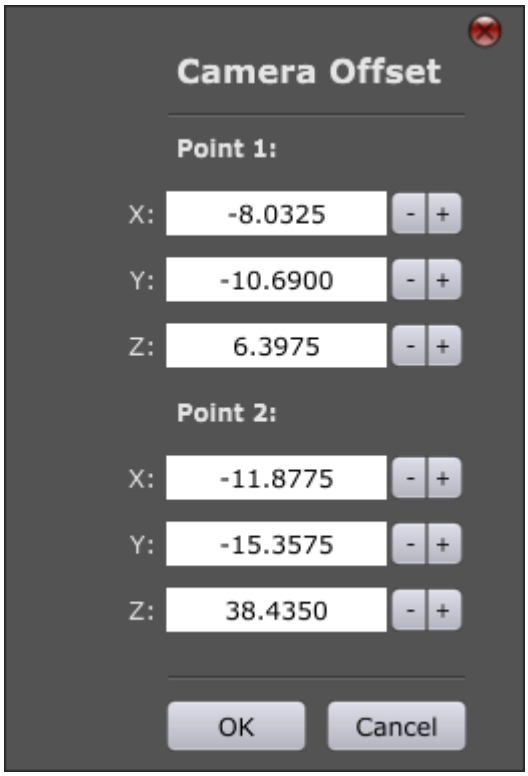
Saves camera reference point position coordinates (X,Y,Z) captured at the lowest Z axis height possible, using camera view. Saved coordinates refer to *Point 1* of *Set Offset* dialog.

2.7.20.2.3 Read 2

Saves camera reference point coordinates (X,Y,Z) captured at the highest Z axis height possible using, camera view. Saved coordinates refer to *Point 2* of *Set Offset* dialog.

2.7.20.2.4 Set Offset

Opens a dialog with current *Point 1 and Point 2* values already populated (if *Read1* and *Read 2* actions were already performed). Click OK to set *Camera Offset*.



How to set Camera two point offset:

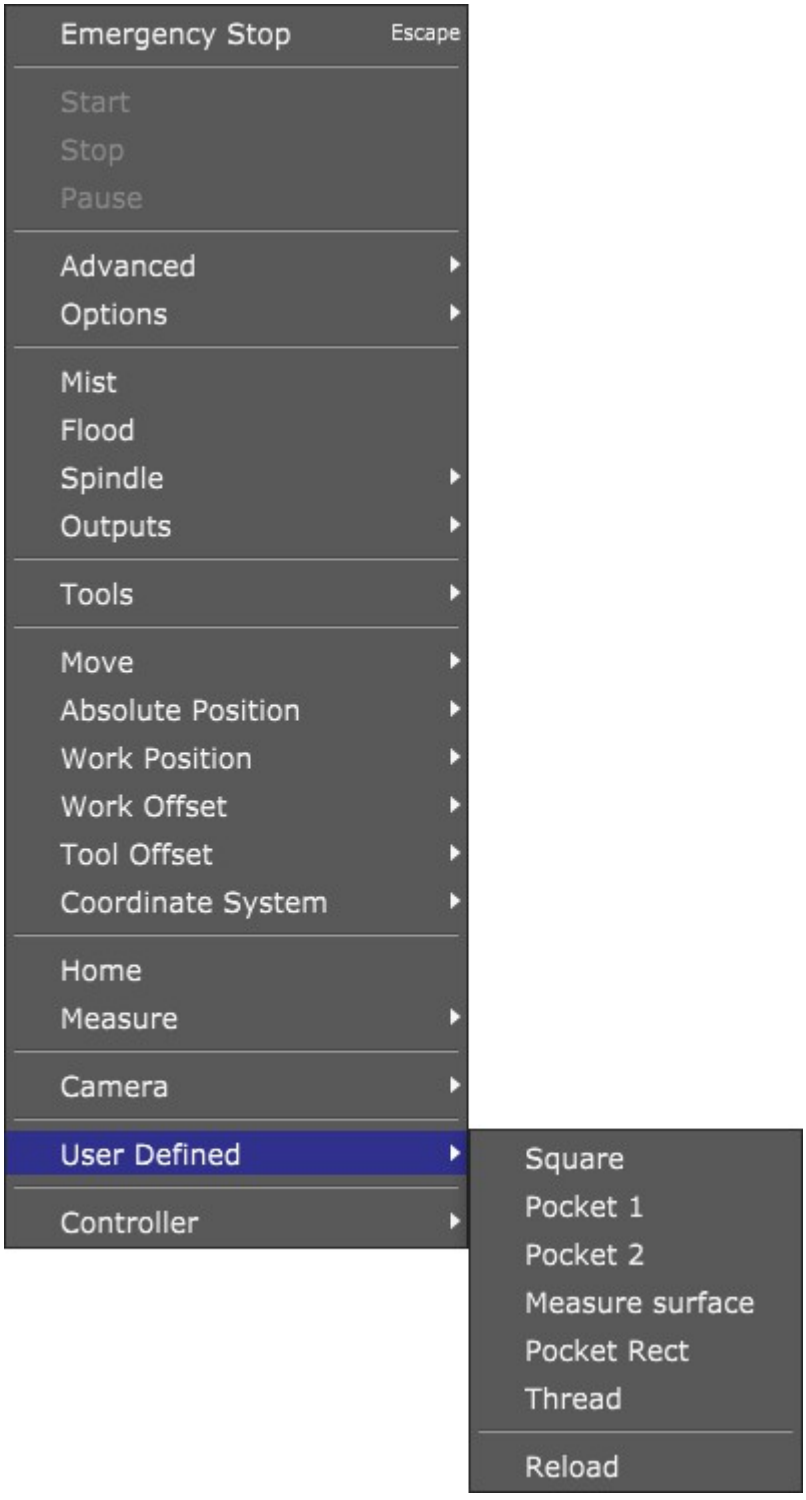
1. Open camera window with *Machine/Camera/Show...*
2. Mark camera reference point on machine table. You can use marker, small drill or tool, ... Whatever you find useful.
3. Position tool exactly over this camera reference point and click: "Machine/Camera/Set Offset/Mark"
4. Move machine and locate camera reference point with camera view. Use lowest Z level possible.
5. When camera reference point is in center of camera cross-hair marker click: "Machine/Camera/Set Offset/Read 1"
6. Now move Z to highest possible level where marker is still visible on camera view and center it.
7. When camera reference point is in center of camera cross-hair marker click: "Machine/Camera/Set Offset/Read 2"
8. To set camera offset click: *Machine/Camera/Set Offset → OK*

2.7.20.2.5 Calibrate Scale

Calibrates camera viewfinder scale/grid for measuring purposes.

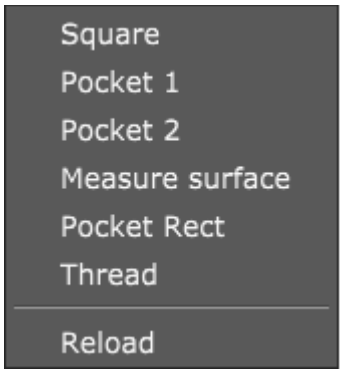
2.7.21 User Defined

User can write and configure his own programs which are available in *User Defined* sub-menu. PlanetCNC TNG already offers some pre-existing user defined programs.



User defined programs should be located in *PlanetCNC64.Scripts* or *PlanetCNC32.Scripts* folder in main install directory.

User defined program file should be named: *Machine.UserDefined_UDn.gcode,* where 'n' stands for number of program. Numbers 1-6 are already reserved for pre-existing user defined programs that are available in *User Defined* sub-menu:



-Square (Machine.UserDefined_UD1.gcode)

-Pocket 1 (Machine.UserDefined_UD2.gcode)

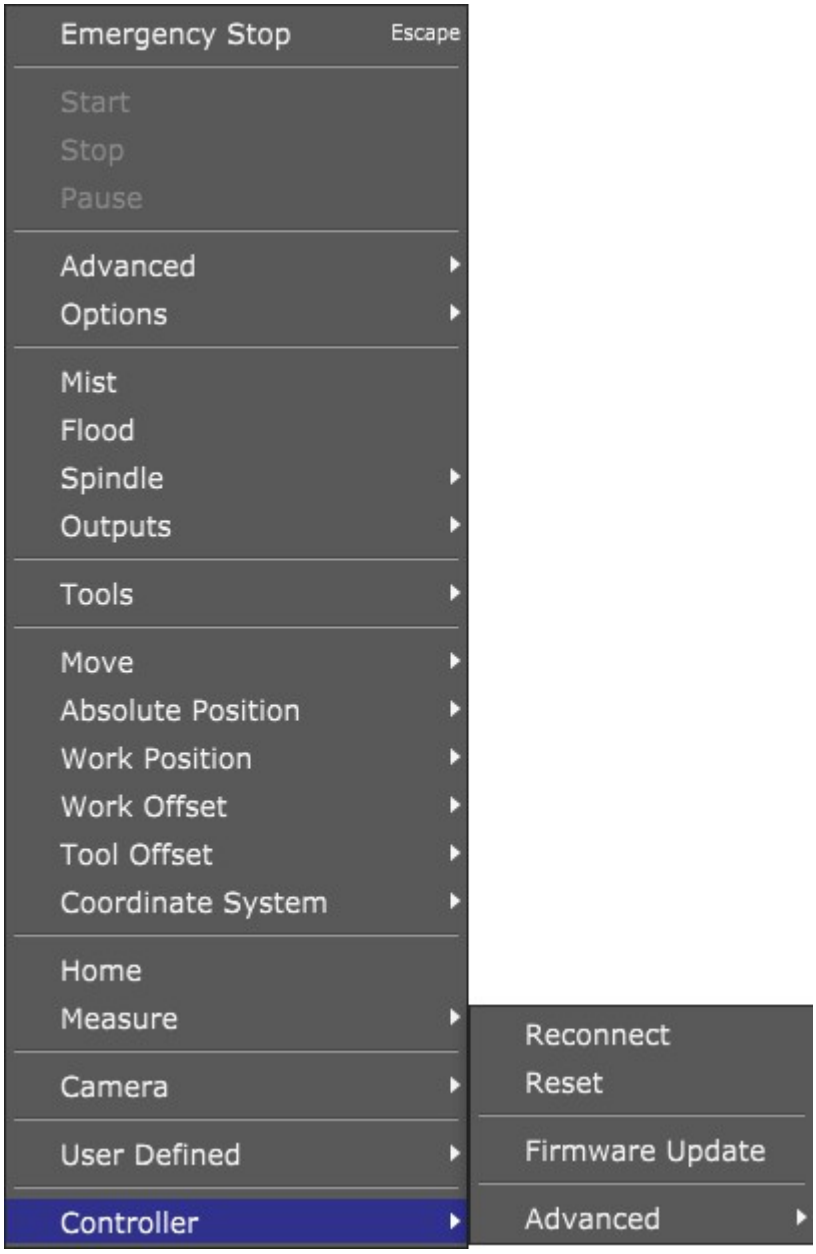
-Pocket 2 (Machine.UserDefined_UD3.gcode)

-Measure surface (Machine.UserDefined_UD4.gcode)

-Pocket Rectangle (Machine.UserDefined_UD5.gcode)

-Thread (Machine.UserDefined_UD6.gcode)

2.7.22 Controller



2.7.22.1 Reconnect

Reconnects controller with software.

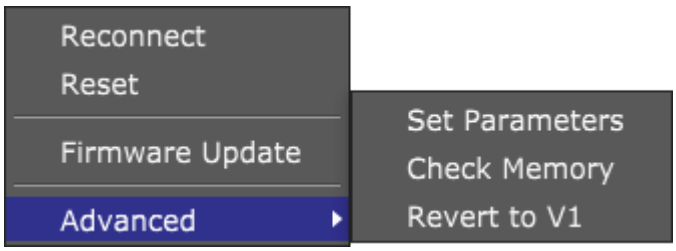
2.7.22.2 Reset

Resets controller.

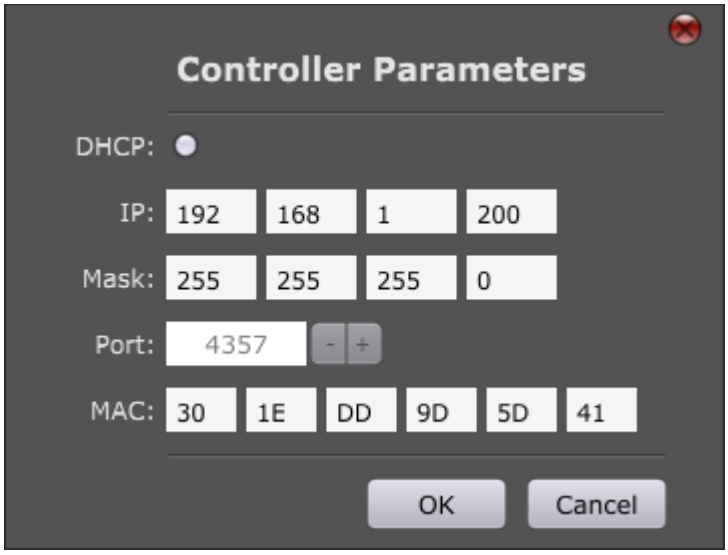
2.7.22.3 Firmware Update

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.

2.7.22.4 Advanced



2.7.22.4.1 Set Parameters



DHCP:

DHCP status of controller

IP:

IP address of controller

Mask:

Mask of controller

Port:

MAC:

MAC address of controller

2.7.22.4.2 Check Memory

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

2.7.22.4.3 Revert to V1

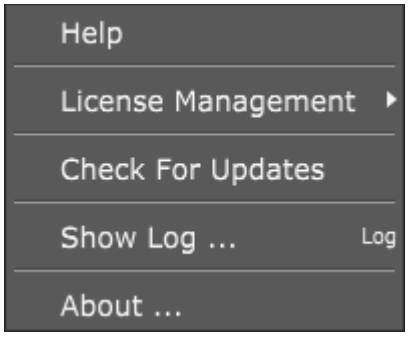
Controller can be used with both motion software's from PlanetCNC, *PlanetCNC TNG* software and *CNC USB controller* software.

Controller's firmware needs to be updated with software version that will be used with: PlanetCNC TNG software or CNC USB controller software.

Controller updated with TNG firmware cannot be used with CNC USB sw and controller updated with CNC USB sw firmware cannot be used with TNG software.

If user wants to use controller with CNC USB software once controllers firmware is updated with TNG software, *Revert to V1 reverts* controller firmware so that it can be used with CNC USB software.

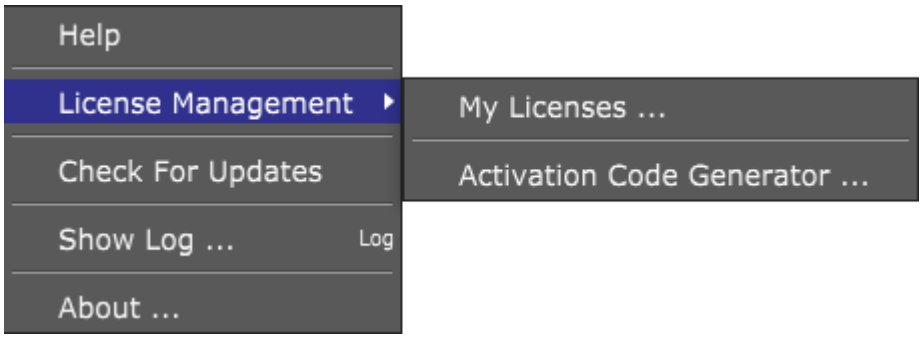
2.8 Help



2.8.1 Help

Opens help file.

2.8.2 License management



2.8.2.1 My Licenses

Opens up a dialog where user can import license activation code and imports and exports license files.

2.8.2.1.1 License list:

Displays license file imported in PlanetCNC TNG software. License files have *.lic* file extension. They are located in main installation folder of PlanetCNC TNG software.

2.8.2.1.2 Import...

Opens up dialog where user can enter license activation code. User can paste license activation code using c/p keyboard keys or *Paste From Clipboard* button. Clicking *OK* button confirms license code.

2.8.2.1.3 Import File

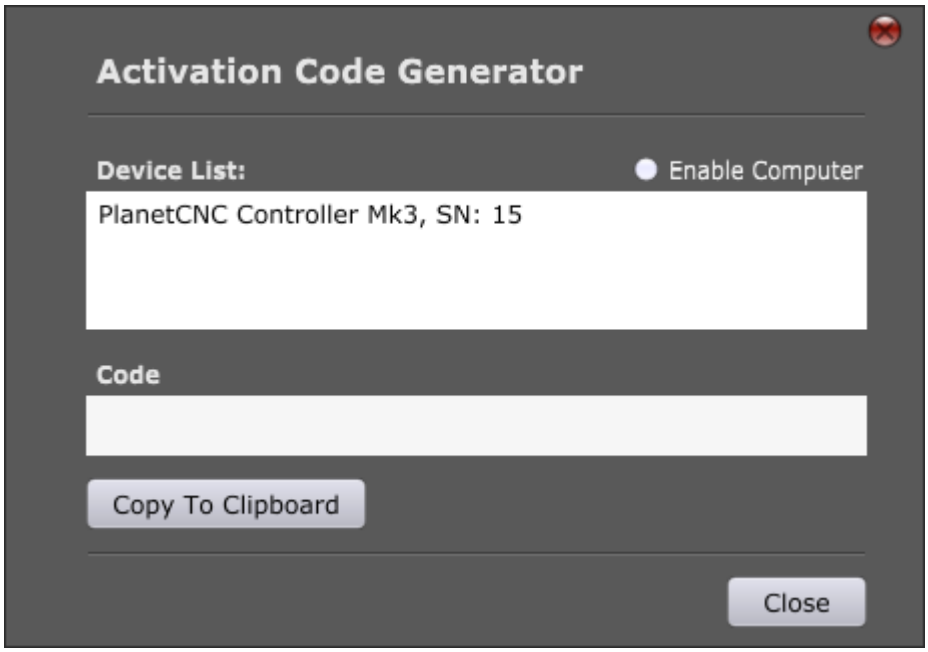
Imports license *.lic* file.

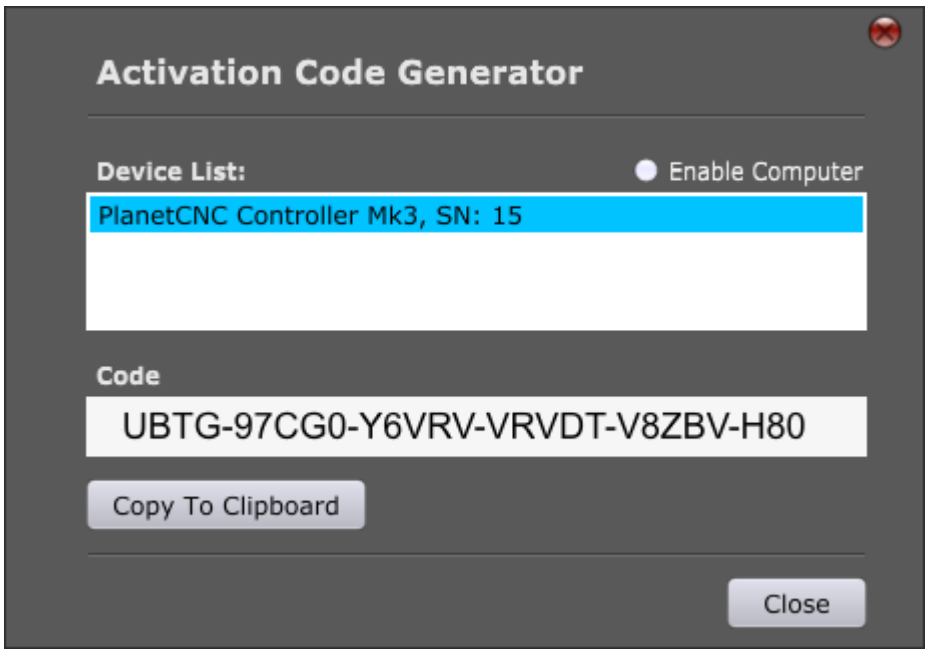
2.8.2.1.4 Export File

Exports license *.lic* file.

2.8.2.1.5 Activation Code Generator

Activation code generator displays controllers unique code. In order that code is displayed, controller needs to be selected under *Device List* so that becomes highlighted:



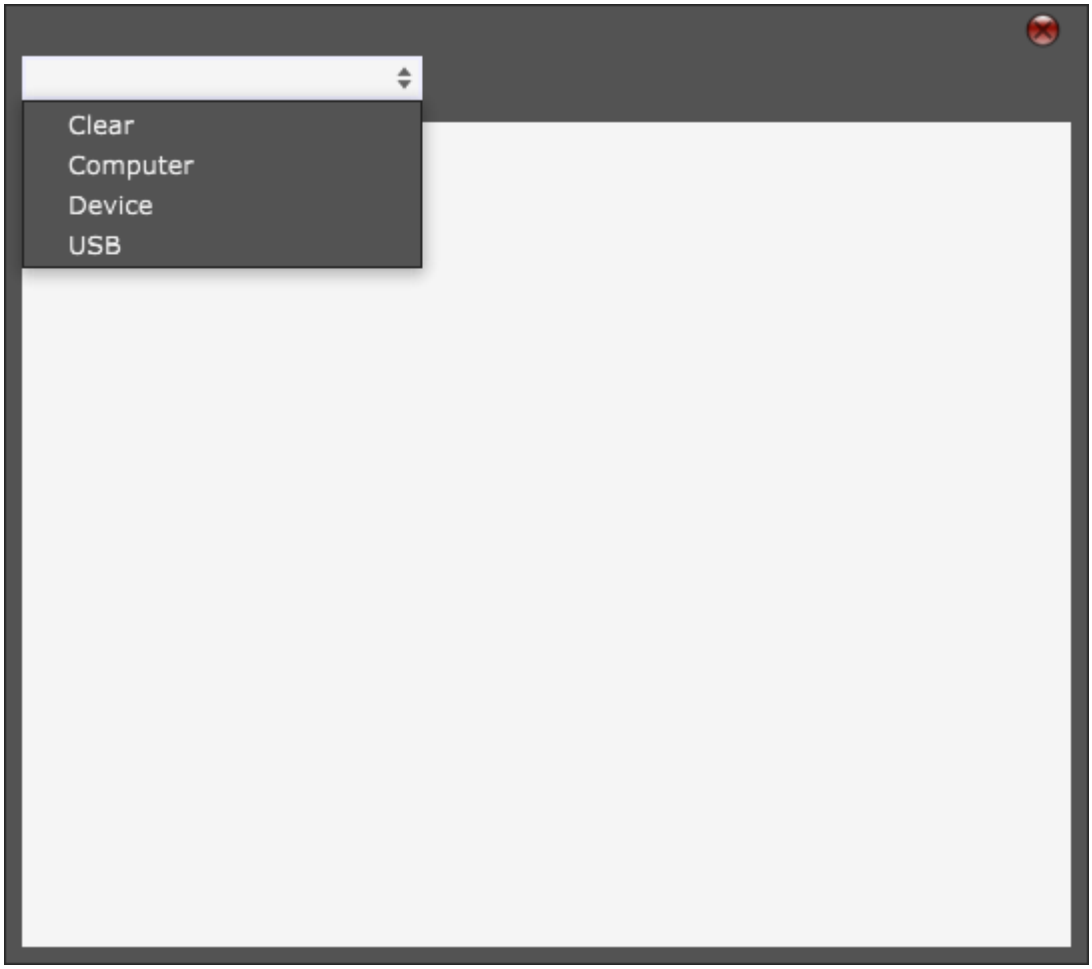


With *Copy To Clipboard button,* code is copied to clipboard and can be directly pasted to e.g. e-mail body text.

2.8.3 Check For Updates

Checks for software updates at PlanetCNC download page: *http://planet-cnc.com/software*

2.8.4 Show Log



2.8.5 About

Displays software version and firmware version of connected controller.

2.9 Settings

Settings panel allows setup and configuration 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.

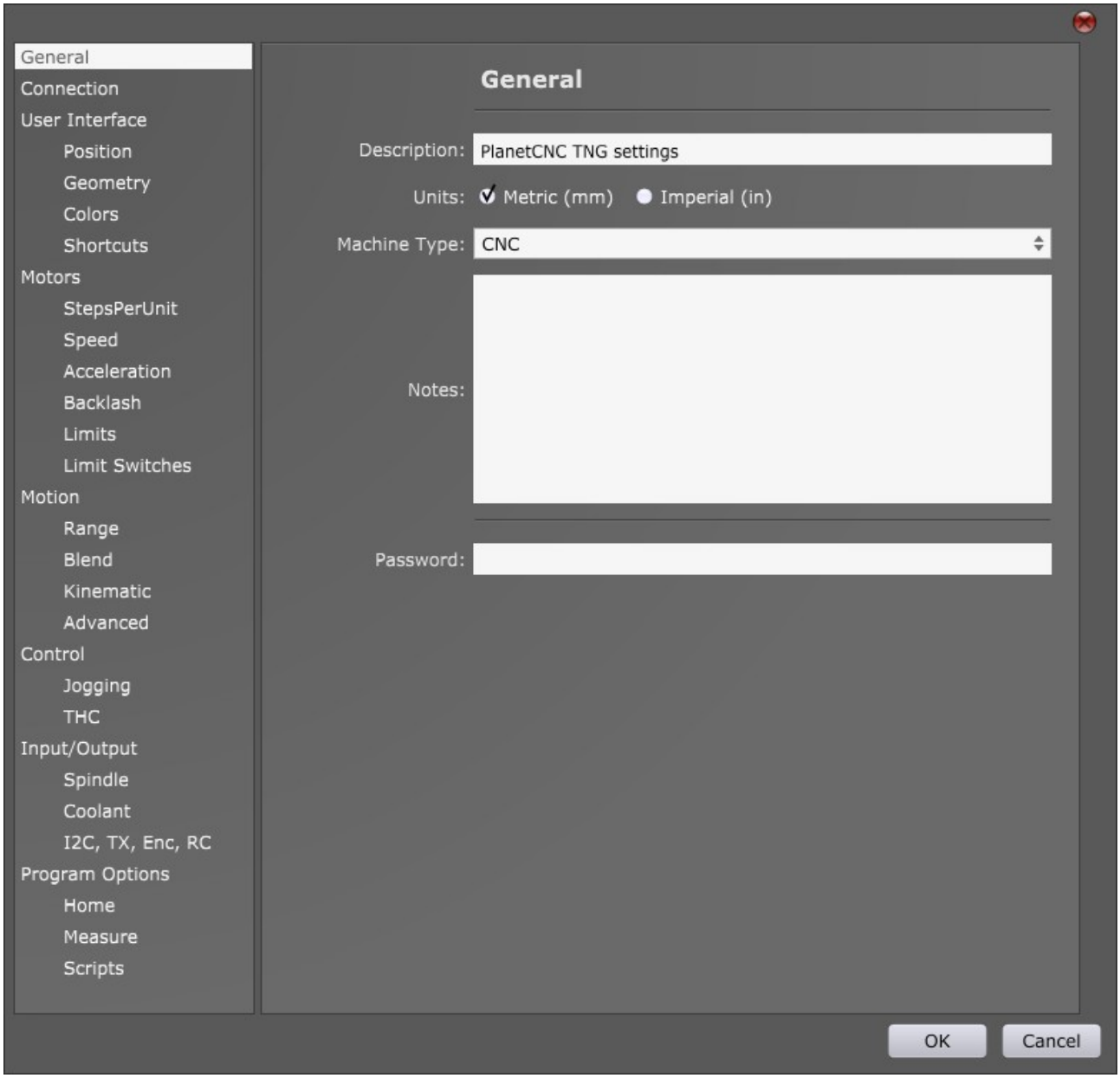
It's MOST IMPORTANT that software is appropriately configured before any attempt is made to control CNC machinery. Failure to do so, may result in serious injury or damage to machinery. With unknown or untested machinery, it's safer to keep initial motion rates below anticipated maximums. Emergency stop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

2.9.1 General

Under ‘General’ tab you can insert settings description, measurement units, machine type, notes and settings password.



2.9.1.1 Description

Anything written in the description bar will be displayed at the top right corner of PlanetCNC TNG software window. This way you can easily indicate settings configuration(settings file) currently in use.

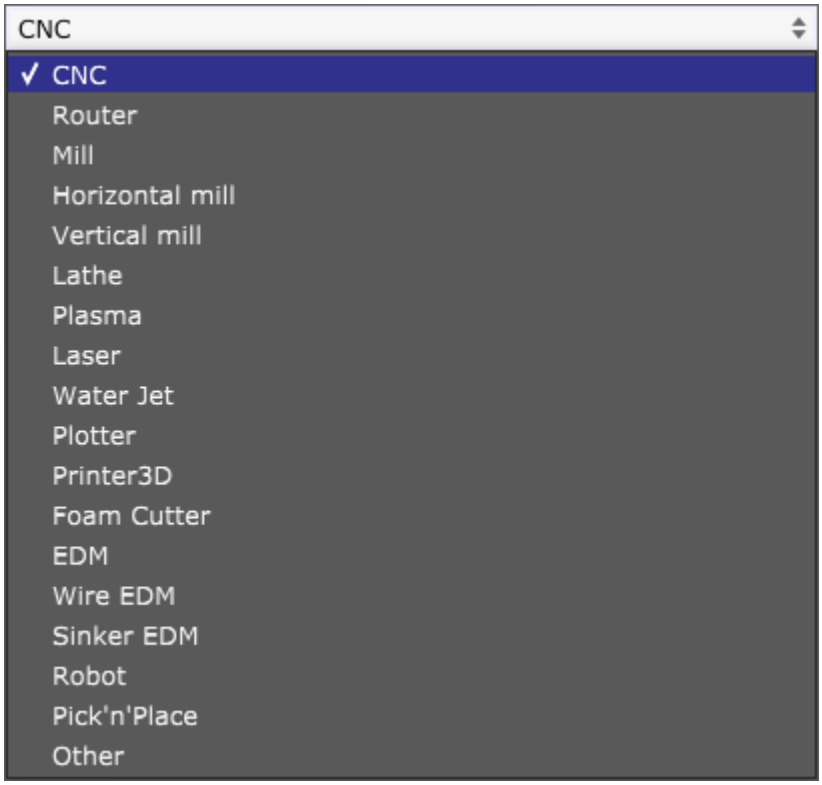
Description text is displayed at top right corner of PlanetCNC TNG window:

2.9.1.2 Units

Selected option sets measurement units which will be used in software. Metric (millimeters-mm) or Imperial (inches-in) units can be used. Depending on measurement unit set, all values in settings will be recalculated to measurement unit selected here.

2.9.1.3 Machine type

Select type of you machine. Option selected does not have an affect on any other setting(only for description purposes).



2.9.1.4 Notes

Here you can write notes describing your machine setup, setting configuration and any info that might be relevant to machine operator.

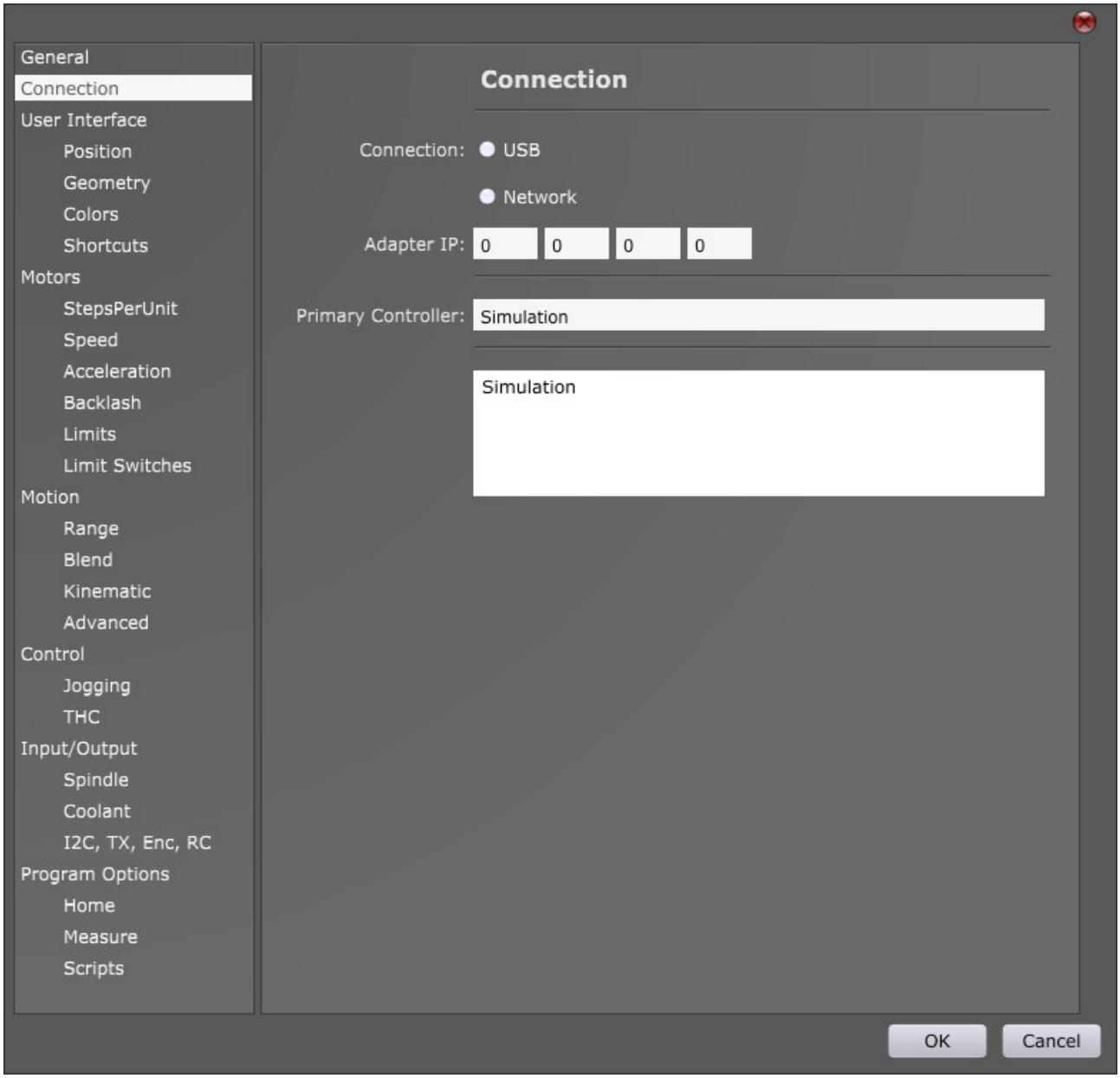
2.9.1.5 Password

If you wish to protect your settings configuration from being changed by another person, you can use password and protect settings from being modified.

Next time you open settings, software will require password:

2.9.2 Connection

Under 'Connection' tab you can set type of connection used for controller communication, computers network adapter IP number, primary controller selection and other motion controller based information.



2.9.2.1 Connection

PlanetCNC motion controllers can use USB or Ethernet type of communication, depending on controller.

Mk3 supports both USB and Ethernet, while Mk3/4 and Mk3ECO controllers support only USB communication.

2.9.2.1.1 USB:

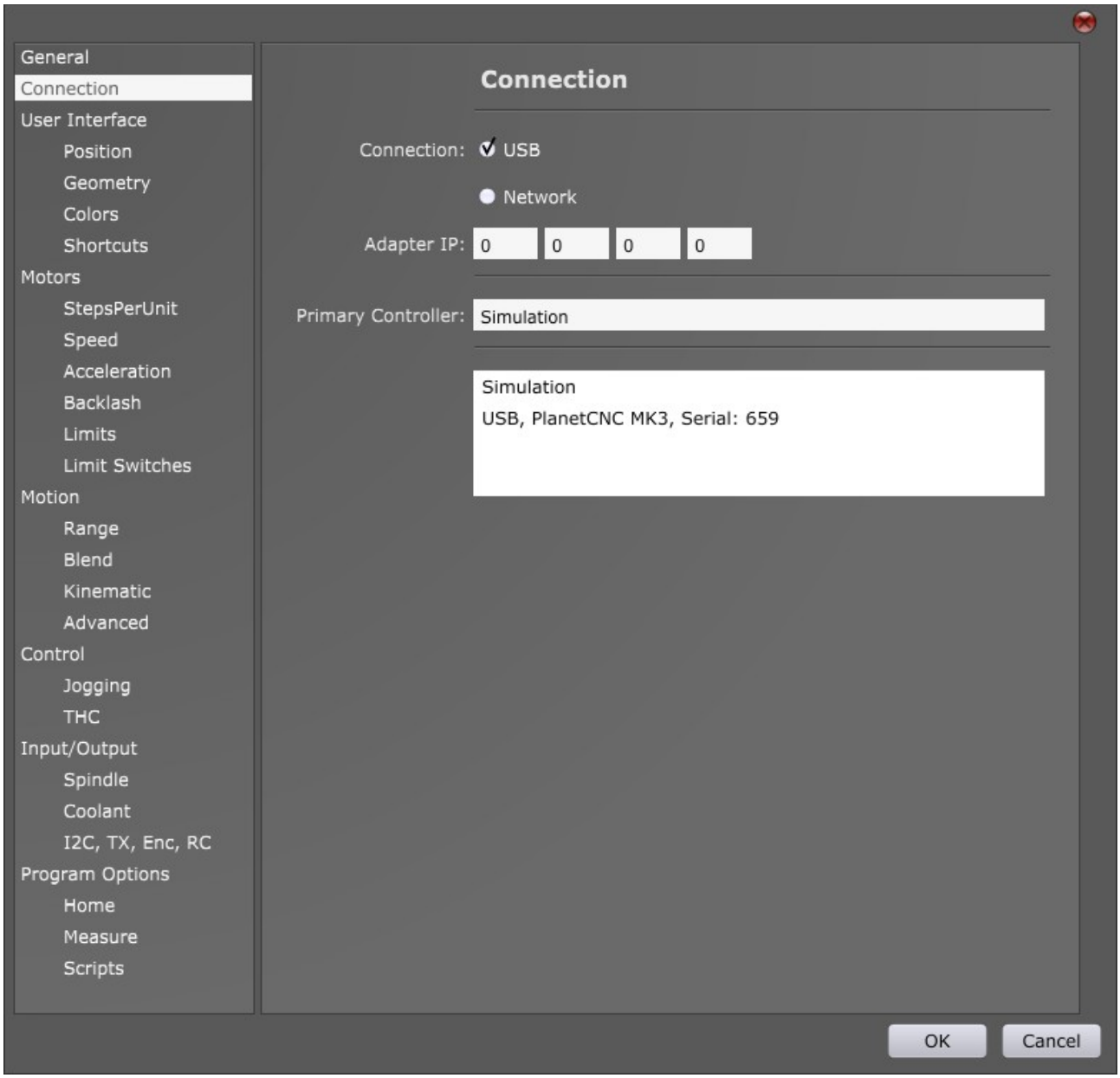
When USB option is enabled, controller list will be populated only with controller(s) that are connected to PC via USB port.

Controller list provides user with basic controller info: USB, PlanetCNC Mk3, Serial: 659

USB: Type of connection

PlanetCNC Mk3, Mk34: Version of PlanetCNC controller

Serial: Serial number of controller

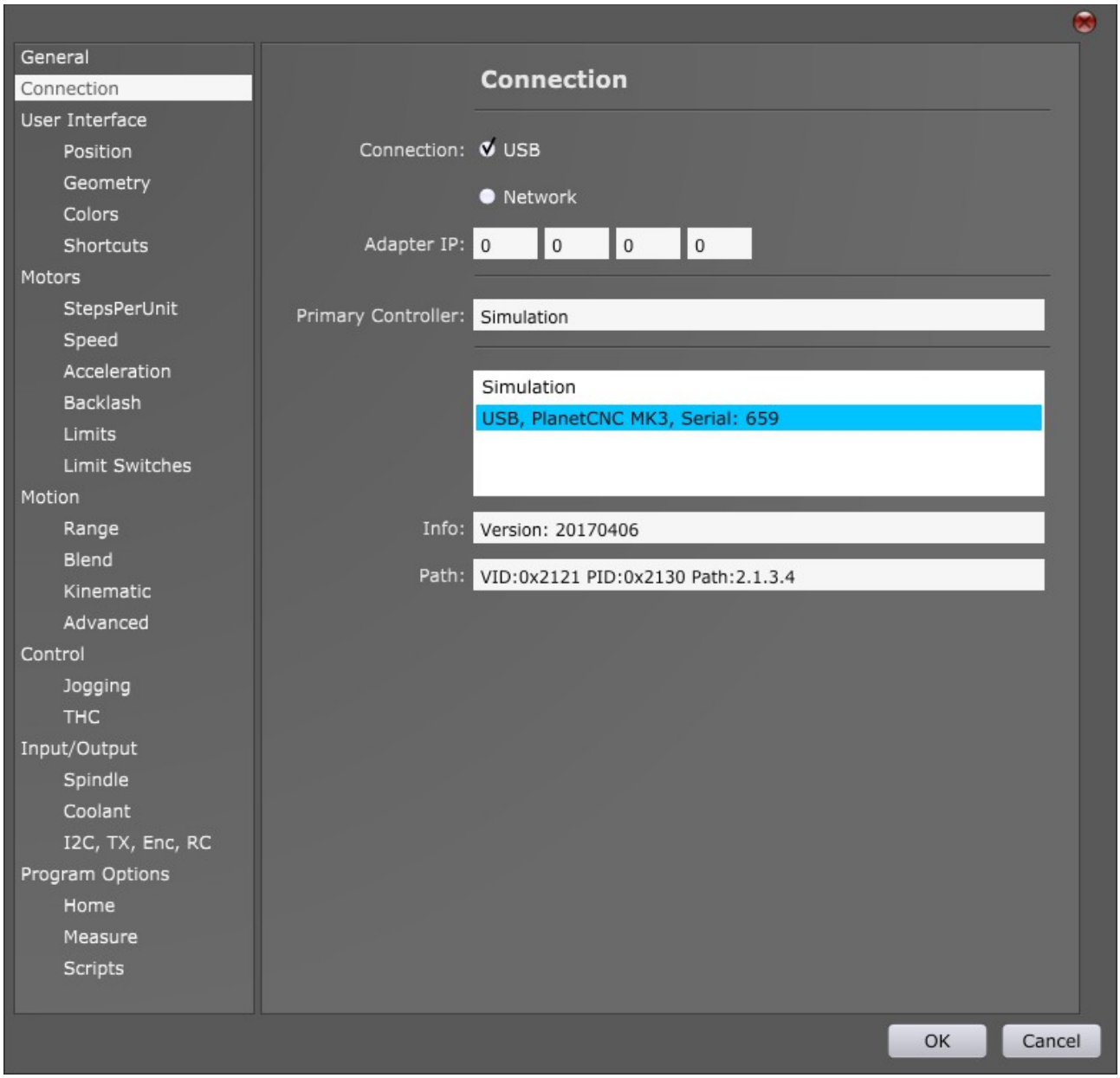


If you select controller from device list you will be able to see additional info of controller (Info and Path bars will appear):

Info: Controllers firmware version(Version:).

Path: VID(Vendor ID) and PID(Product ID) number.

Path numbers 2.1.3.4 represent USB connection port hierarchy.



2.9.2.1.2 Network:

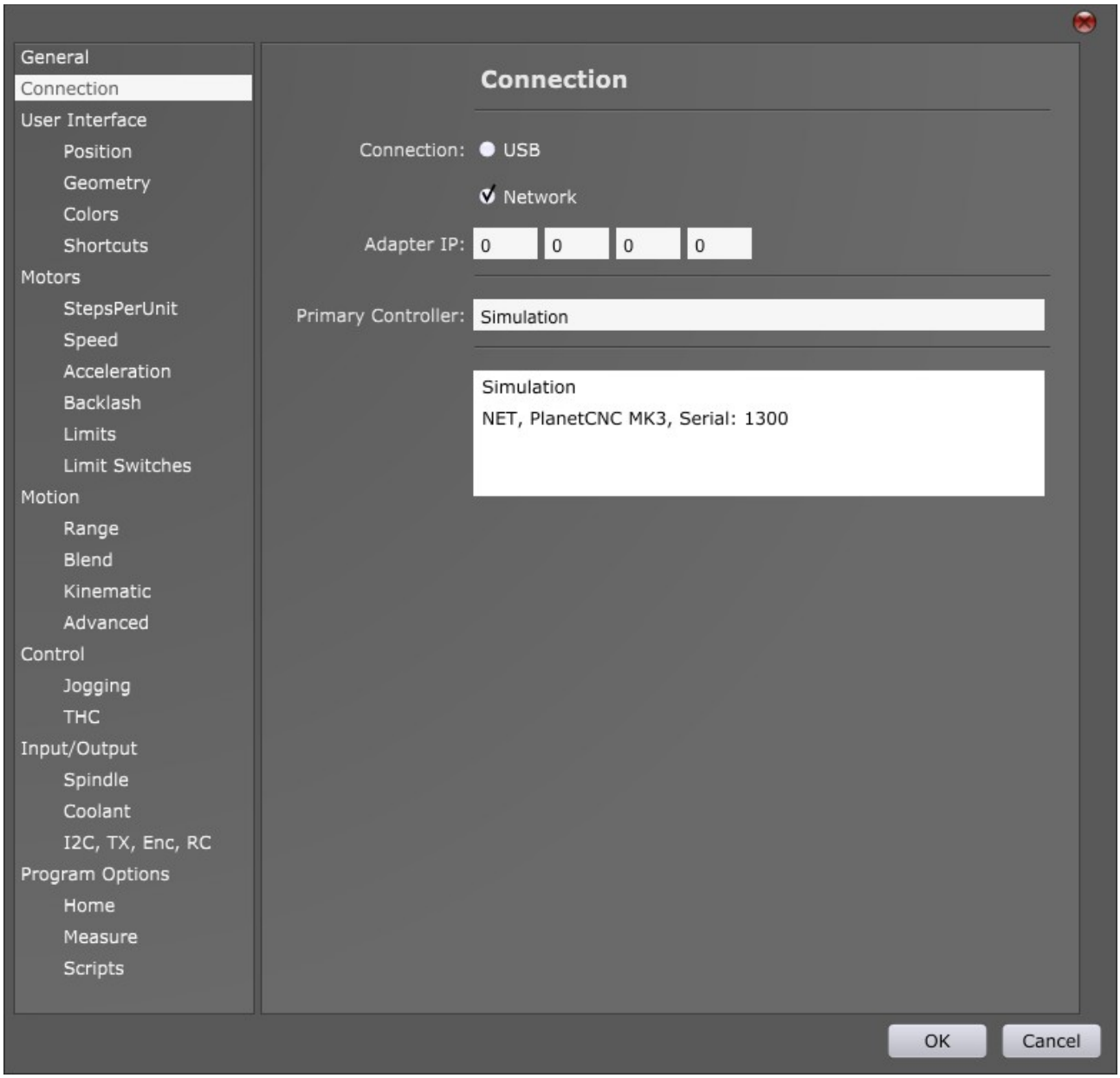
When Network option is enabled, controller list will be populated only with controller(s) that are connected with PC via network cable or are connected to network.

Controller list provides user with basic controller info: NET, PlanetCNC MK3, Serial: 1300

NET: Type of connection

PlanetCNC Mk3, Mk34: Version of PlanetCNC controller

Serial: Serial number of controller



When you select displayed controller from device list you will be able to see additional info of controller(Info, IP, Mask, MAC and DHCP bars will appear):

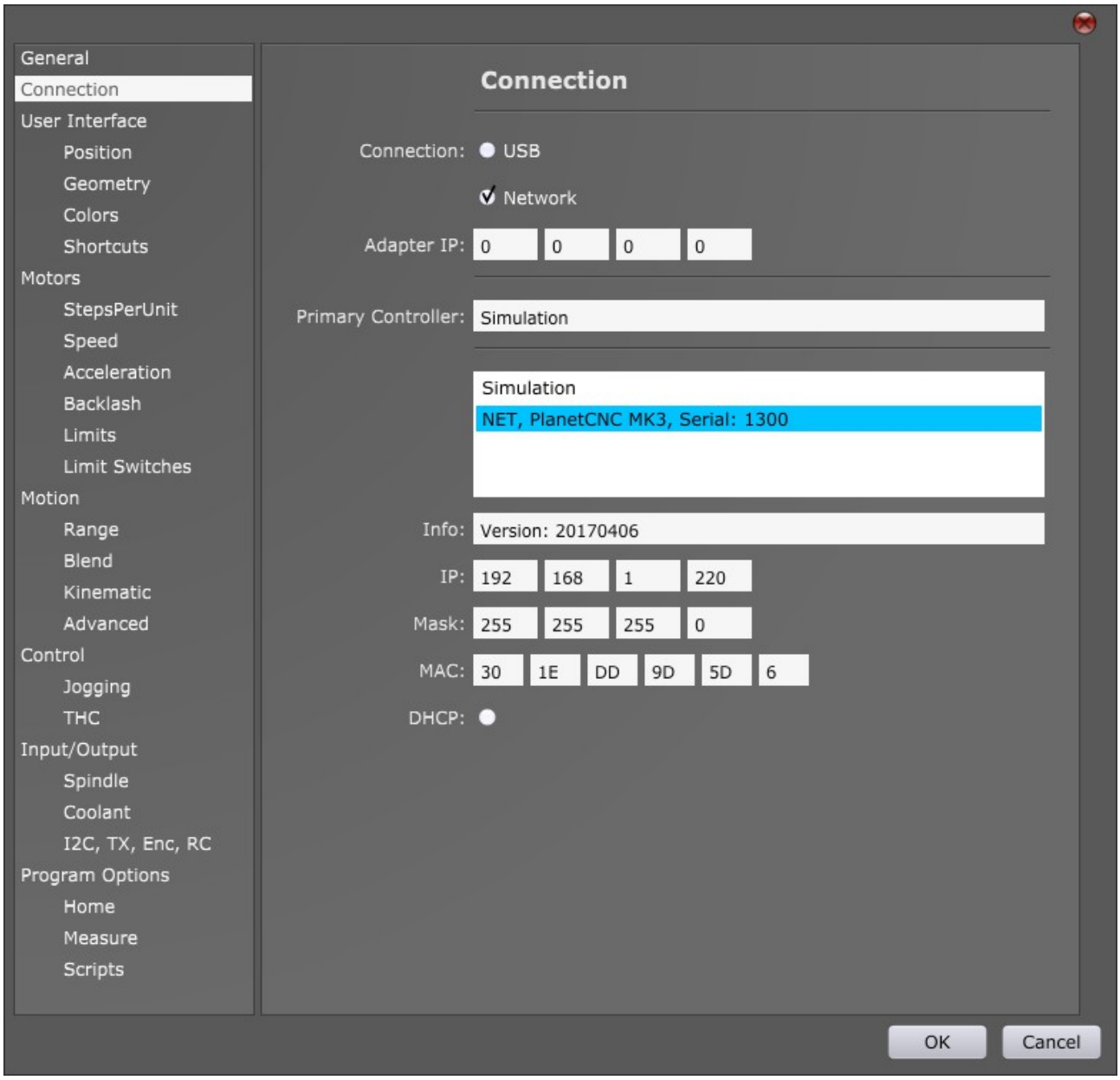
Info: Controllers firmware version(Version: 20170406).

IP: IP number of motion controller.

Mask: Network mask

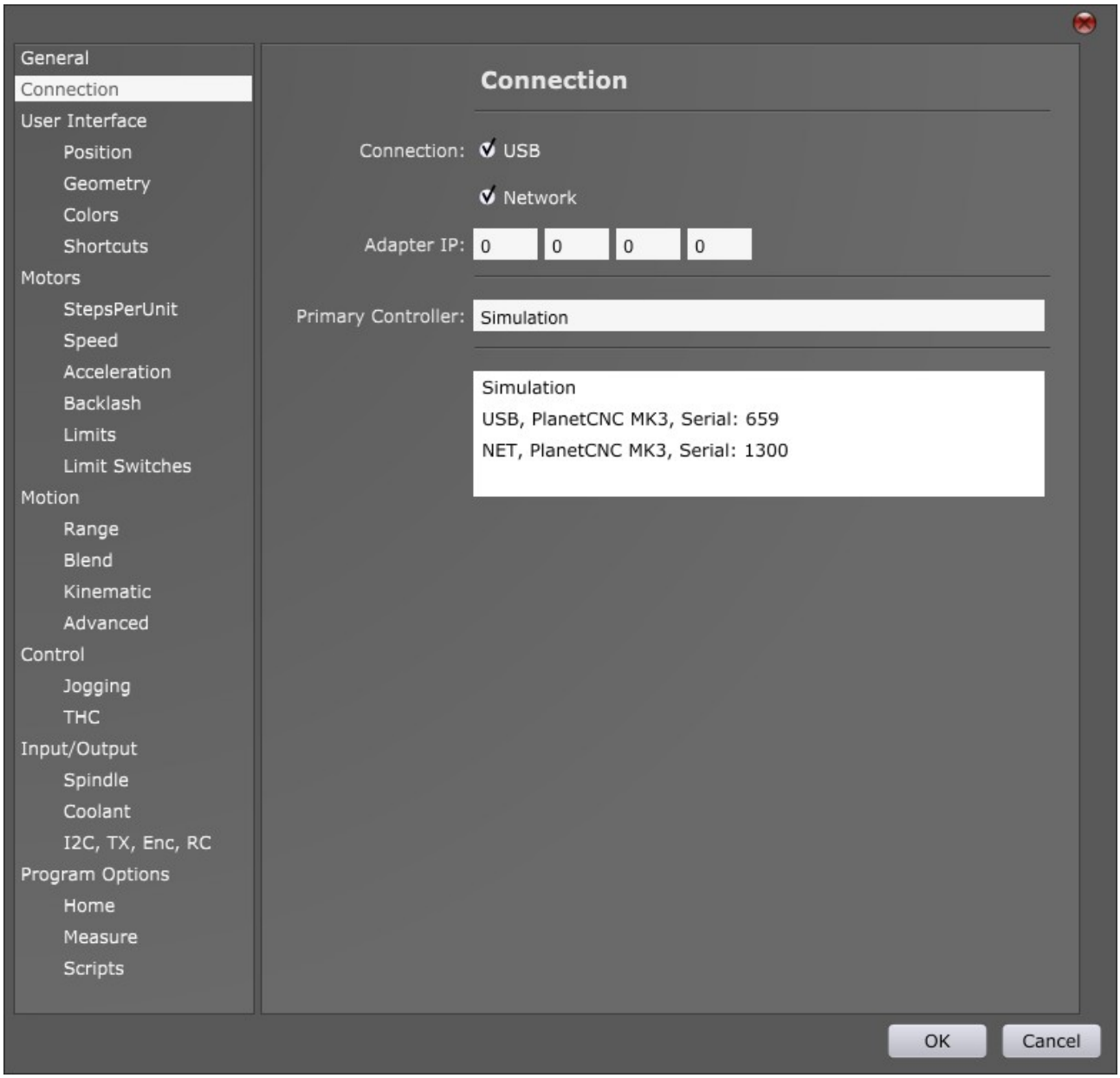
MAC: MAC address of controller

DHCP option: Enables DHCP



2.9.2.1.3 USB & Network:

If both, USB and Network options are enabled, controller list will be populated with controllers that are connected with PC via USB and Ethernet.



2.9.2.2 Adapter IP

Your computer is able to use numerous network adapters, which means your controller can be connected with computer trough any of them. Here you can insert your computers network adapter IP number.

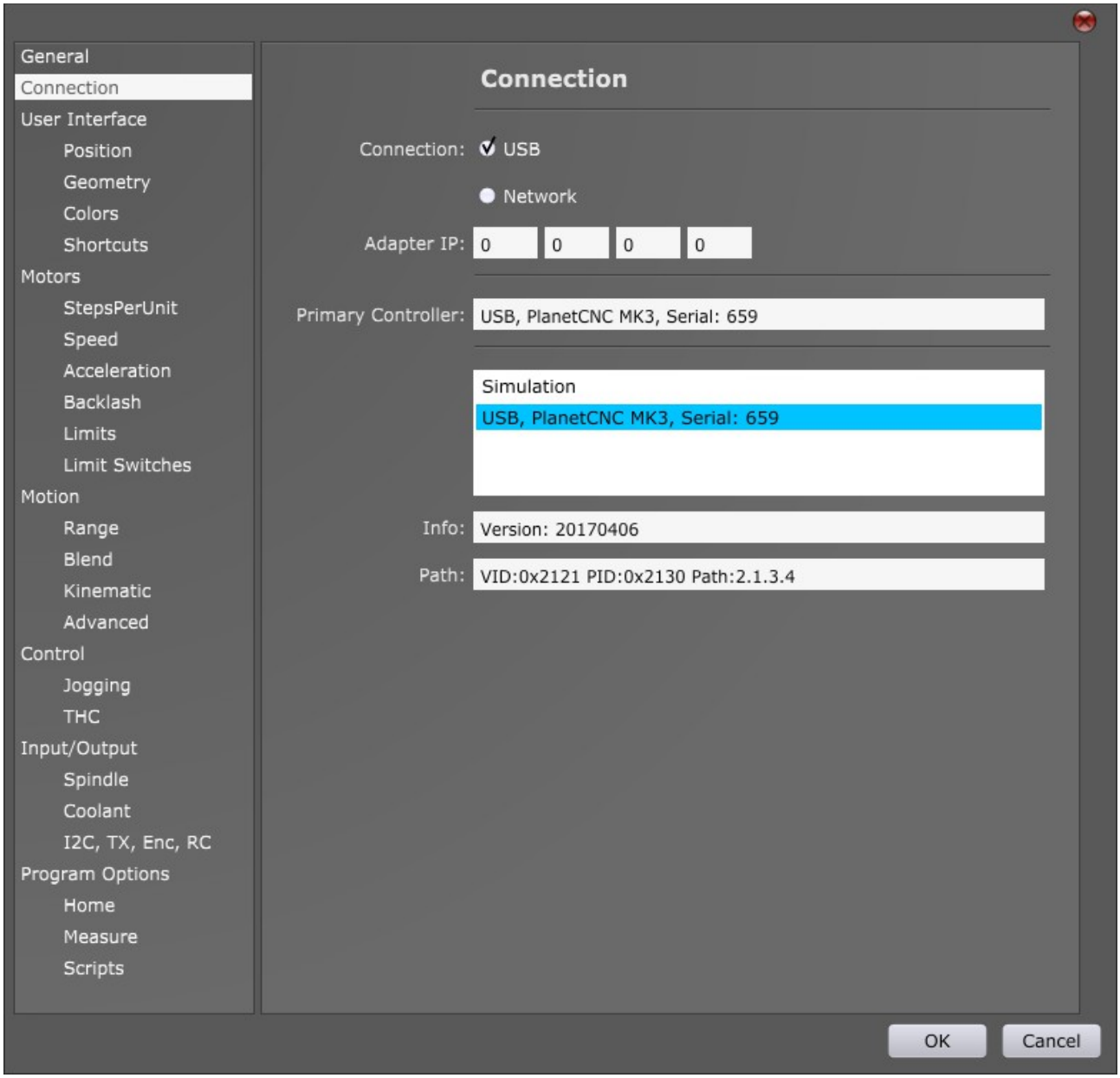
2.9.2.3 Primary Controller:

Primary controller is controller which we select and assign as our main controller device. Primary controller is controller that will control CNC machine.

To assign controller as your primary controller you need to double click on it from the controller list.

You will see that Primary Controller bar now displays your selected controller.

IMPORTANT: Selecting primary controller is necessary in order to control your CNC machine.



2.9.3 User Interface:

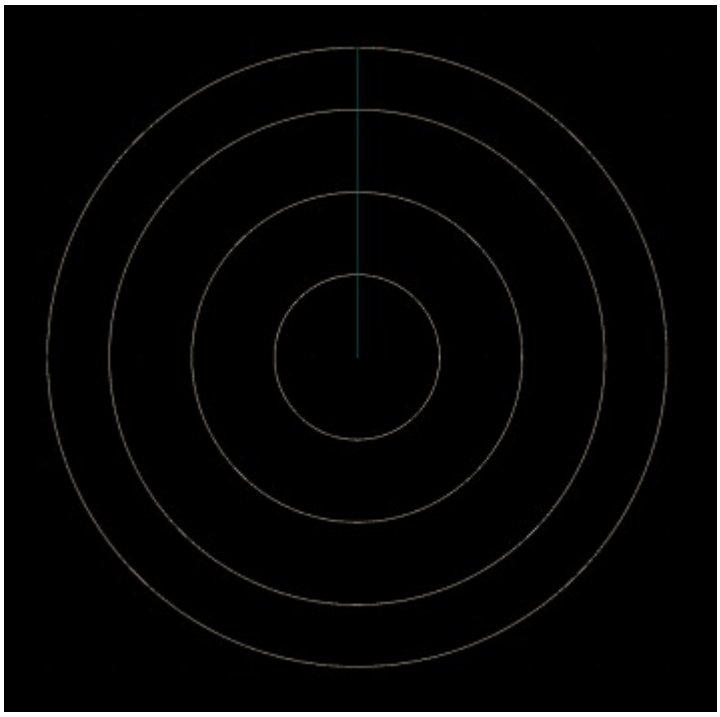
Under 'User Interface' tab you can configure all parameters and features related to PlanetCNC TNG software user interface.

2.9.3.1 Display Resolution:

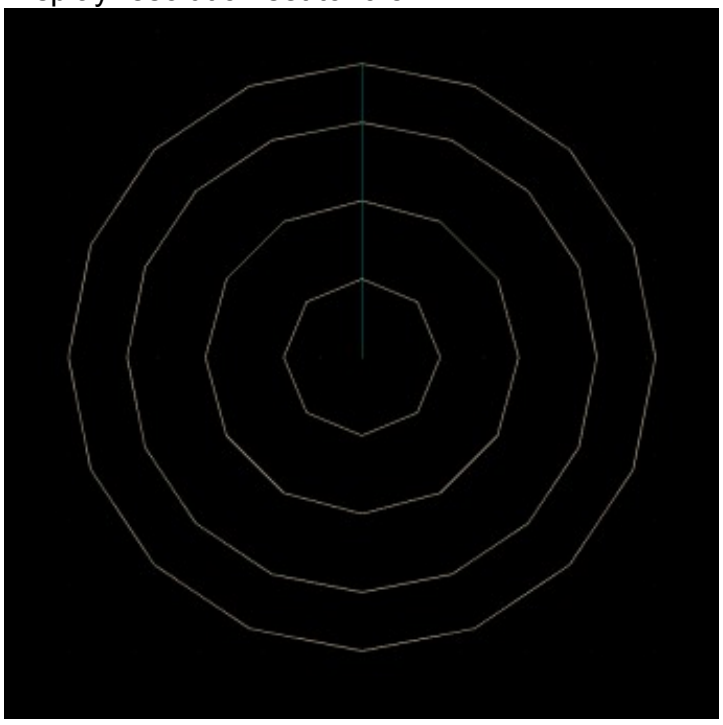
This parameter sets tolerance for which displayed toolpath is allowed to deviate from actual toolpath. Units for this parameters are millimeters.

Please note that this parameter will only affect the displayed toolpath.

Display resolution set to: 0.01mm



Display resolution set to: 0.8mm



2.9.3.2 Display Refresh:

This parameter sets user interface display refresh time. Value inserted is time in milliseconds, time after displayed PlanetCNC TNG interface will refresh.

2.9.3.3 3D Display Refresh

This parameter sets 3D display refresh time. Value inserted is time in milliseconds, after which 3D display of PlanetCNC TNG will refresh.

2.9.3.4 Line Limit:

This parameter sets max number of program g-code lines that can be loaded into PlanetCNC TNG software. Toolpath will be displayed only to the point of last g-code line loaded.

Note that even if g-code program is not loaded and tool path is not displayed completely, software will still execute it in its entirety.

2.9.3.5 Time Limit:

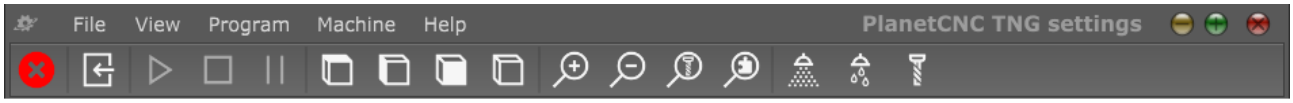
Value inserted is time in seconds after which PlanetCNC TNG software stops loading of g-code program.

Note that even if g-code program is not loaded and tool path is not displayed completely, software will still execute it in its entirety.

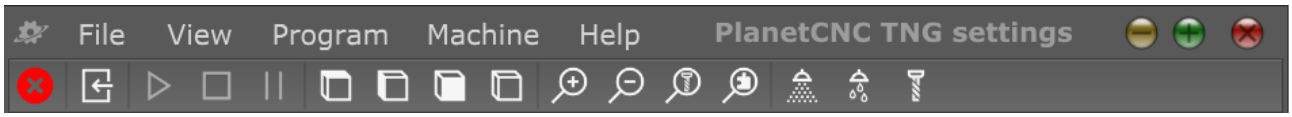
2.9.3.6 Titlebar Size:

Titlebar is located horizontally at the top of the user interface. It populates menu titles: *File, View, Program, Machine, Help* and *Settings Description*. With this parameter you set the size of the titlebar.

Titlebar size at value 25:



Titlebar size at value 35:



2.9.3.7 Toolbar Size

Toolbars are located vertically left of the user interface and horizontally at the top of the user interface(below of the titlebar) . With this parameter you set the size of the toolbar icons.

Toolbar(vertical) size at value 40:

Toolbar(vertical) size at value 60:

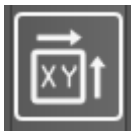
2.9.3.8 Toolbar Hints:

Toolbar hints are pop-up clouds with names of toolbar buttons. Toolbar hints appear whenever you hover with your mouse over specific toolbar button.

Hide: Toolbar hints will not appear.

Show: Toolbar hints will appear.

Hide enabled:



Show enabled:

2.9.3.9 Spindle RPM:

Spindle RPM display is located under Speed tab of user interface:

Options below allow you to set the type of displayed RPM value:

As Set: RPM value displayed is as sent to controller. Value reflects current modal state.

From Index: RPM value displayed is RPM value from index signal measurement.

From Encoder: RPM value displayed is RPM value from encoder signals measurement.

Only one option selected is allowed.

2.9.3.10 Active:

Speed tab can also provide user with helpful information such as active G-codes, M-codes and other codes.

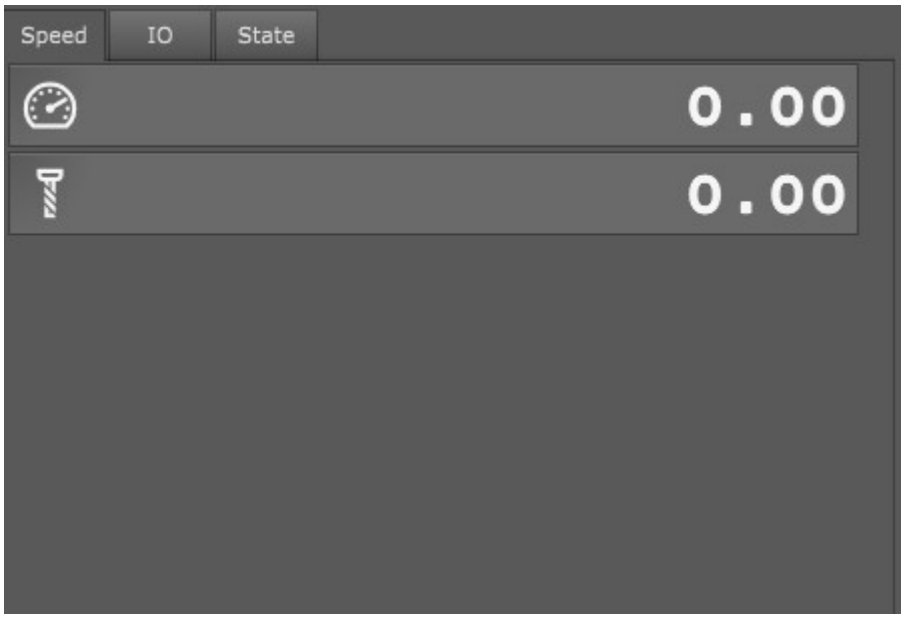
Options below allow you to set the type of active g-codes being displayed under Speed tab:

G-codes: Active G-codes are displayed

M-codes: Active M-codes are displayed

Other: Other codes displayed are Feed speed(F word) and S speed(S-word).

All thee options disabled:

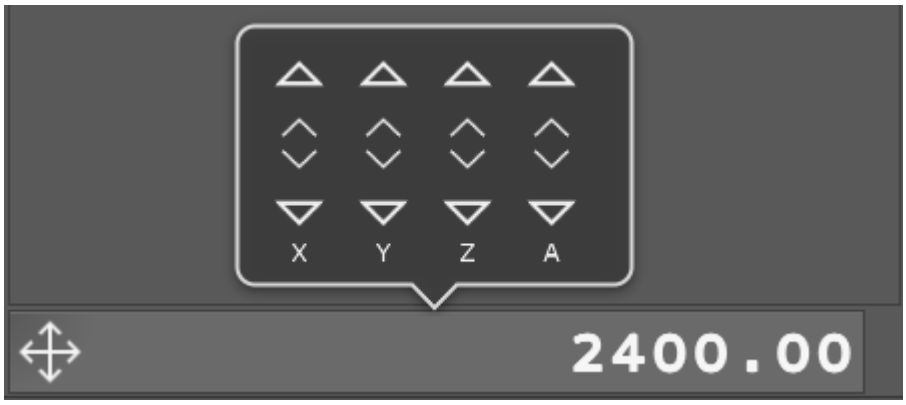


All three options enabled:

2.9.3.11 Jog Panel:

You can set fixed mode of jog panel and three types of jogging keys formation: normal, *Cross XY* and *Cross UV*.

Default state of jogging panel is hidden. For jog panel to appear, click jog panel button  :



Fixed: Jog panel is fixed and it will not hide in case of in-active state.

XY Cross: Cross formation of XY axis jogging keys.

UV Cross: Cross formation of UV axis jogging keys.

Fixed jog panel with none of cross options enabled:

Fixed jog panel with *UV Cross* option enabled:

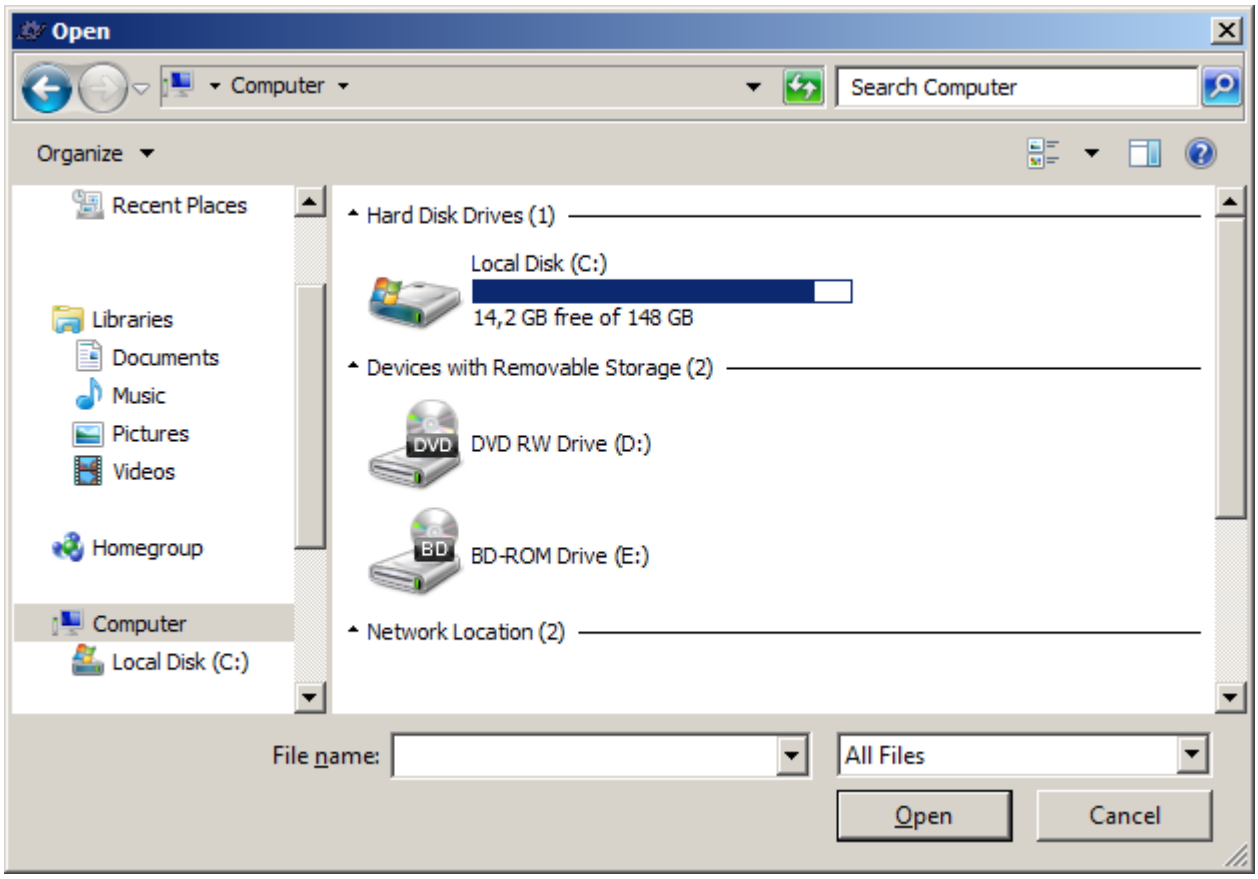
Fixed jog panel with *XY Cross* option enabled:

2.9.3.12 *Native File Dialogs:*

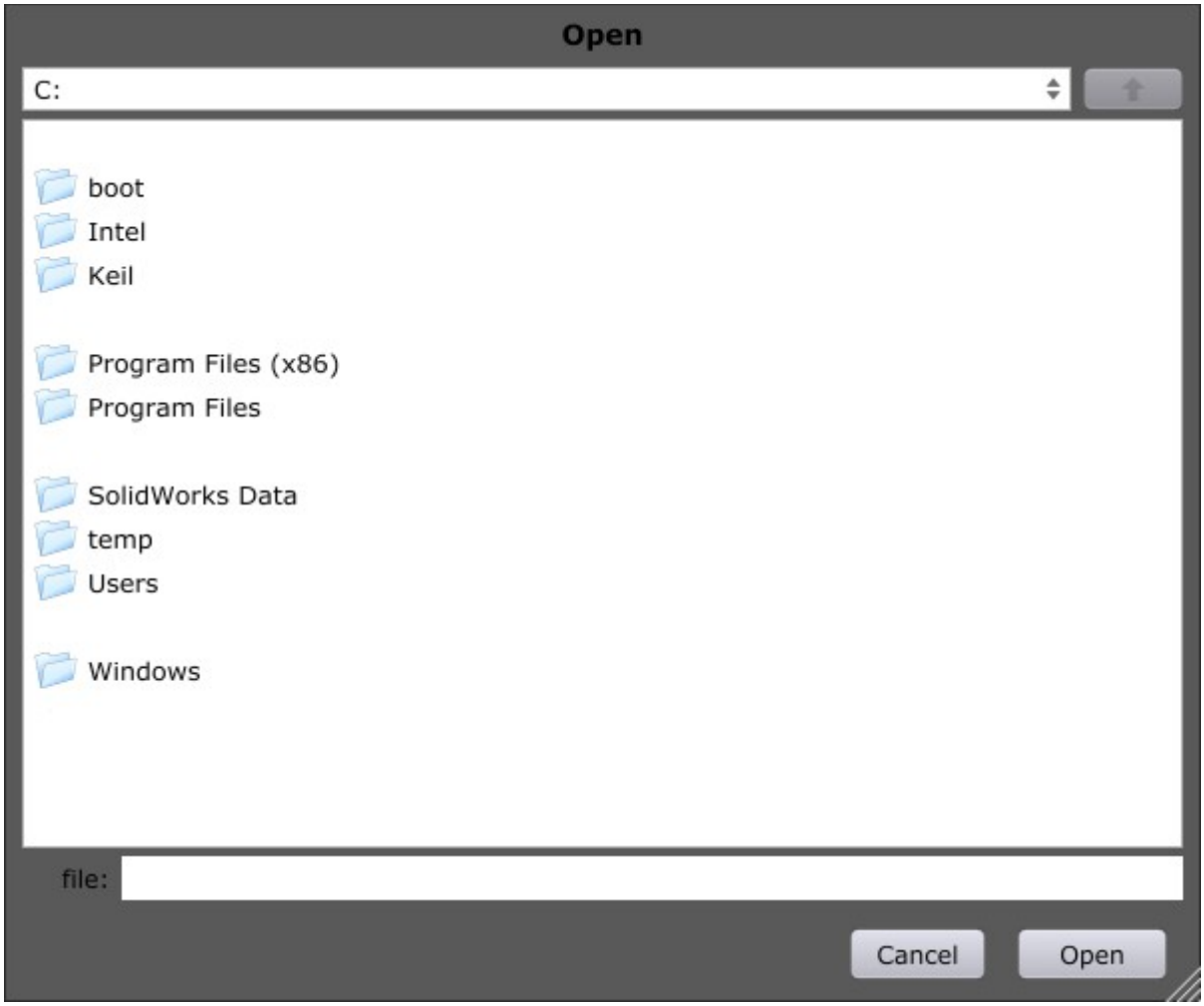
Feature dialogs located under 'File' menu can be displayed in your computers native OS form or in PlanetCNC TNG software form.

Click *Enable* button to enable use of *Native File Dialogs*.

Native File Dialogs enabled:



Native File Dialogs disabled:

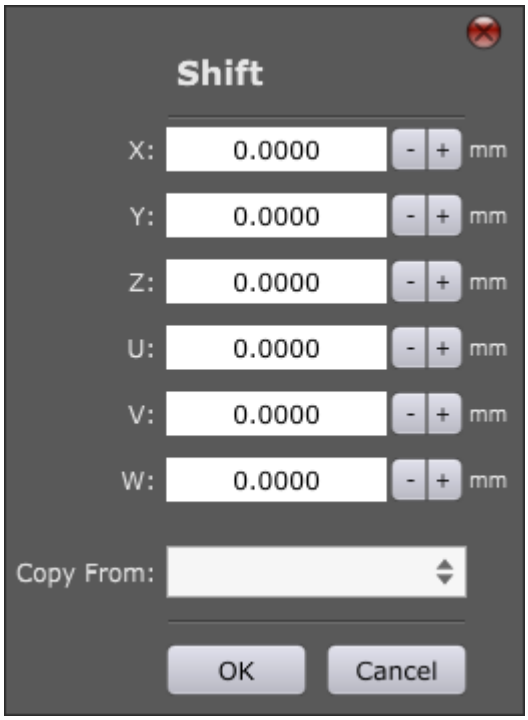


2.9.3.13 Dialog Options:

OK/Cancel:

Program feature dialog will use OK and Cancel buttons. OK button confirms parameter configuration and closes the dialog. Changes to the program will be made after you click Reload from Program menu.

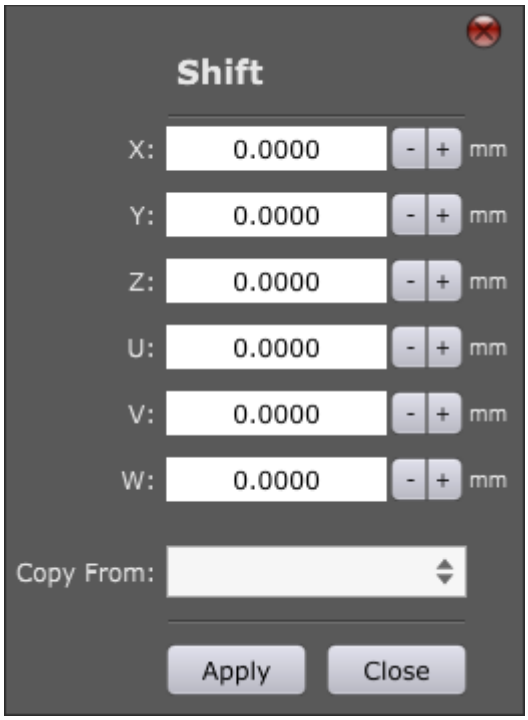
Cancel button closes dialog. No changes will be made.



Apply/Close:

Program feature dialog will use Apply and Close buttons. Apply button confirms parameter configuration. Changes to the program will be made after you close the dialog and click Reload from Program menu. If Auto Reload(Chapter 2.7.3.17) is enabled, changes to the program will be made instantly after Apply button clicked.

Cancel button closes dialog. No changes will be made.



Non Modal:

When selected, even if program feature dialog is opened, you can use use main user interface in the background to some extent, such as moving, zoom in, zoom out of displayed toolpath or navigating trough menus.

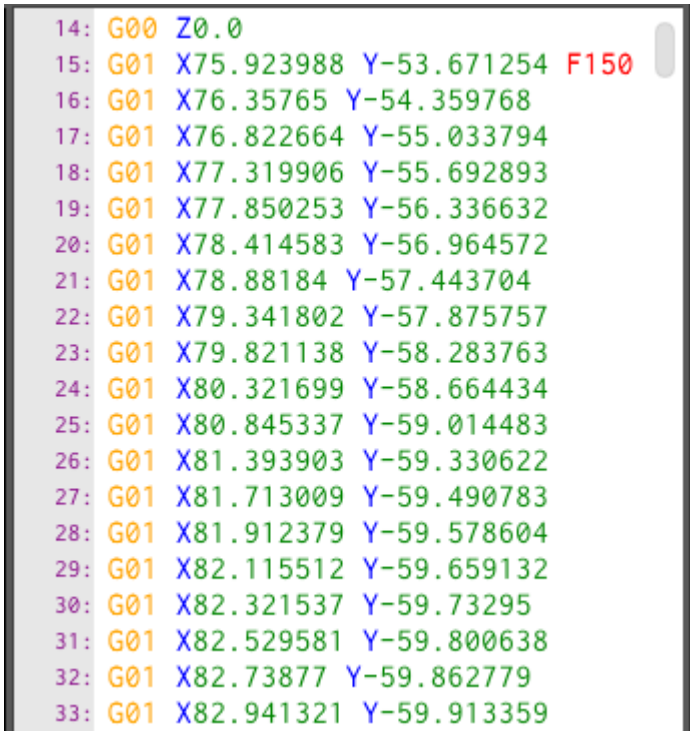
Modal:

When selected, this feature locks any use of main user interface in the background when program feature dialog is opened.

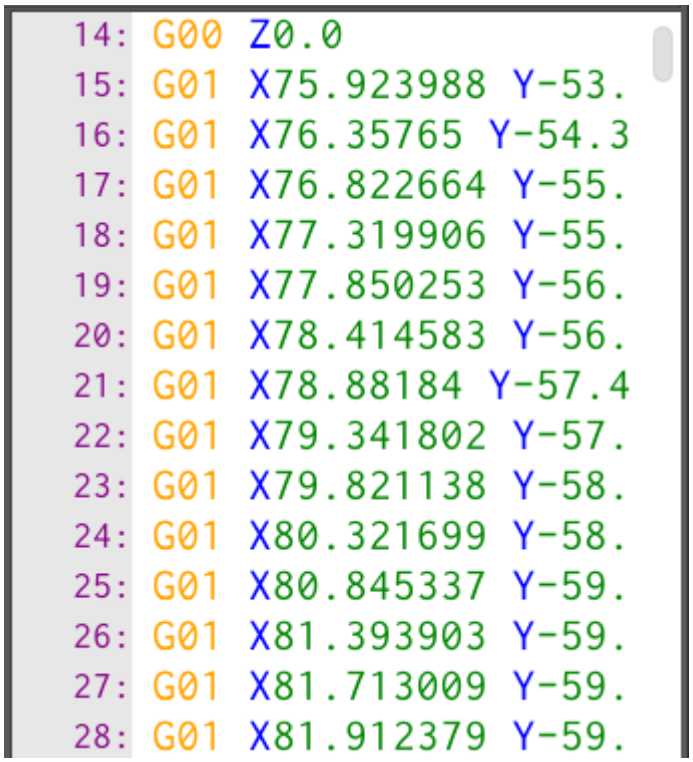
2.9.3.14 G-Code Text size:

This parameter sets g-code text size. G-code window is located at the left side of user interface.

G-code text size at value 18:



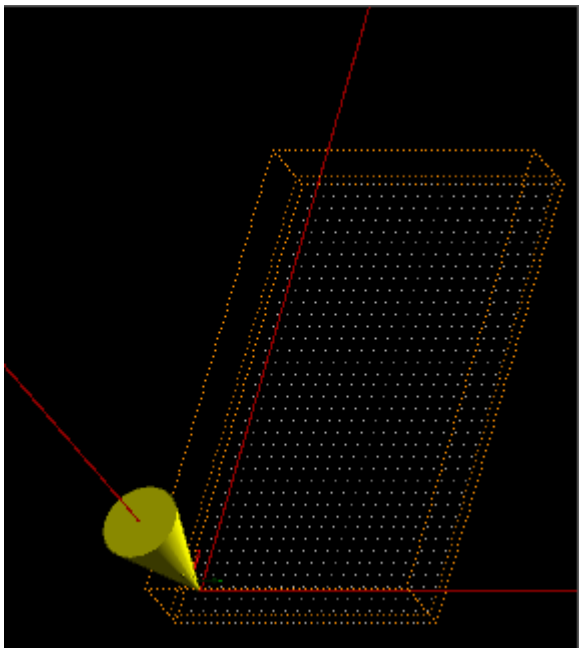
G-code text size at value 25:



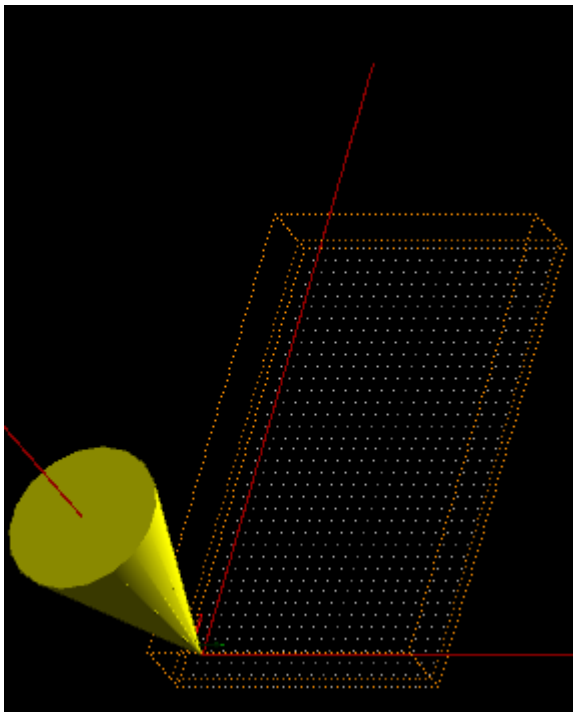
2.9.3.15 Handle Size:

Handle (yellow cone) represents the tool position. You can change its size with Handle Size parameter.

Handle size at value 10:



Handle size at value 15:



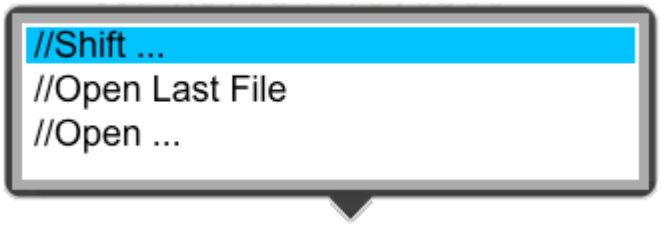
2.9.3.16 Command History

When enabled, software tracks history of previously used commands.

You can see list of commands used in chronological order if you focus MDI bar and then press Up keyboard arrow key.

You are able to navigate through the command history list with Up or Down keyboard arrow keys.

With Enter keyboard key you select command from the history list and again with Enter keyboard key you execute it.



2.9.3.17 Auto Reload

If any program changes were made using Program menu dialogs, this parameter when enabled, will automatically reload modified program after program dialog is closed.

Enable:

If any program changes were made using Machine menu dialogs, this parameter when enabled, will automatically reload changes after Machine dialog is closed.

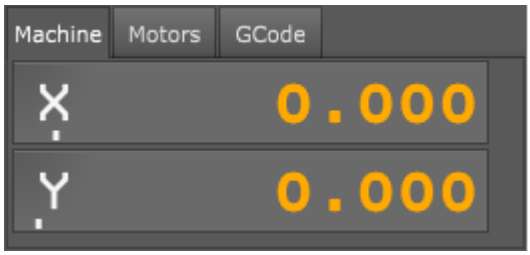
2.9.3.18 User Interface/Position

Under *User Interface/Position* tab you can set number of displayed axis rows, number of decimals displayed and you can assign axis coordinate values to desired row.

2.9.3.18.1 Number of Rows

Sets number of displayed axis rows under Machine, Motor and Gcode tabs.

Number of Rows set to 2:



Number of Rows set to 4:

2.9.3.18.2 Number of Decimals

Sets number of displayed decimals for axis rows.

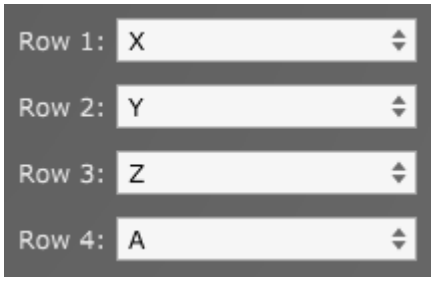
Number of decimals set to 3:

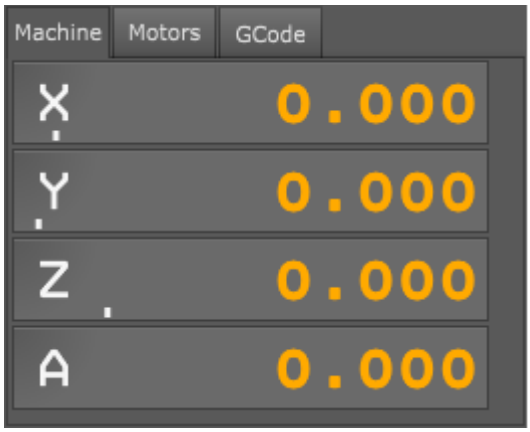
Number of decimals set to 1:

2.9.3.18.3 Rows

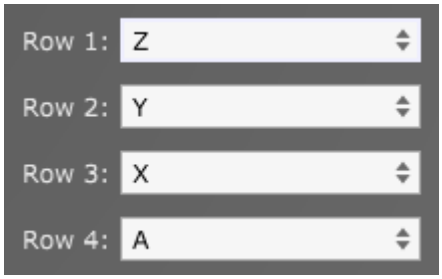
Assigns axis to desired row.

Axis X assigned to row 1, Axis Y assigned to row 2, Axis Z assigned to row 3:





Axis X assigned to row 3, Axis Y assigned to row 2, Axis Z assigned to row 1:



2.9.3.19 User Interface/Geometry

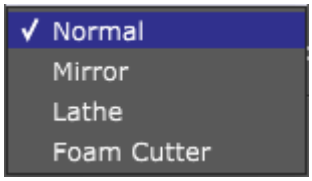
Parameters under 'Geometry' tab allow customization of 3D axis and tool display to fit most types of machines.

Presets for most common machine types are available for your convenience.

NOTE: These parameters do not have any effect on axis motion itself, only on axis 3D display.

2.9.3.19.1 Select Preset:

You can choose between 4 presets: *Normal, Mirror, Lathe* and *Foam Cutter*.



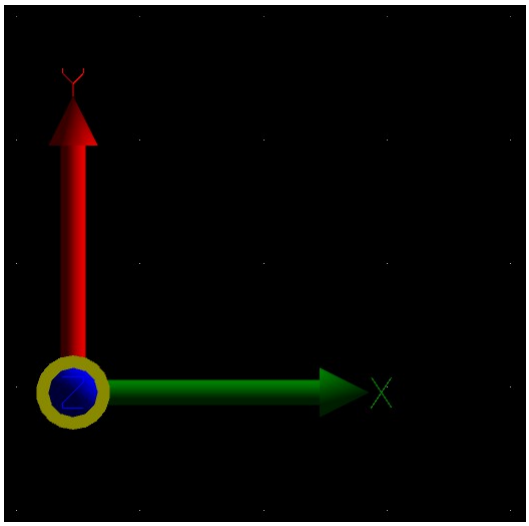
Each preset automatically adjusts *Rotate&Mirror, Axis* and *Orientation* parameters so that 3D axis and tool display suit to corresponded machine type(E.g.: Lathe and Foam Cutter).

2.9.3.19.2 Rotate & Mirror:

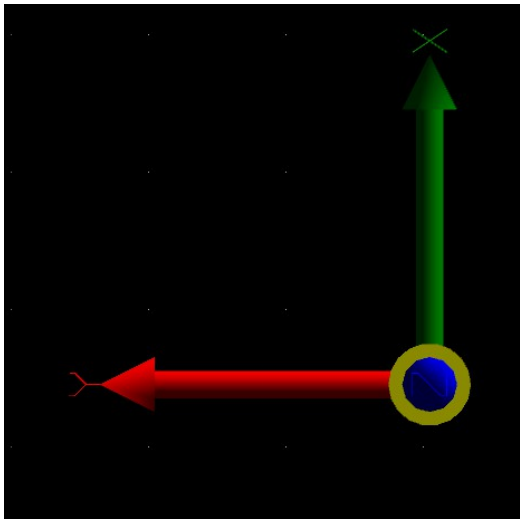
Rotate:

Rotate parameter will rotate 3D display of coordinate system. Rotate parameter units are degrees. Rotation will be applied over selected axis in clockwise direction.

Rotation in all axes is set to 0:



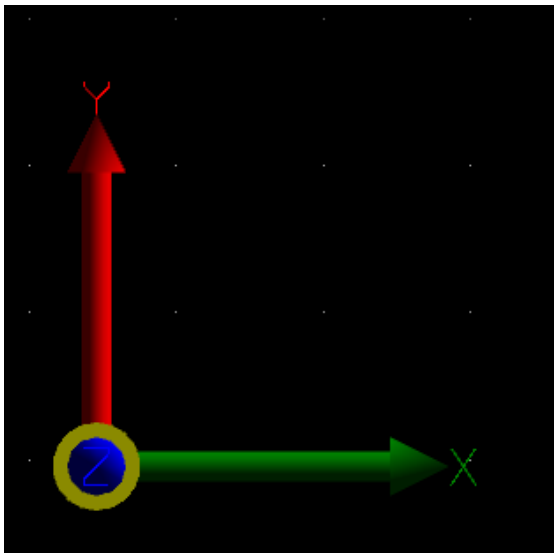
Rotation over Z axis set to 90 degrees:



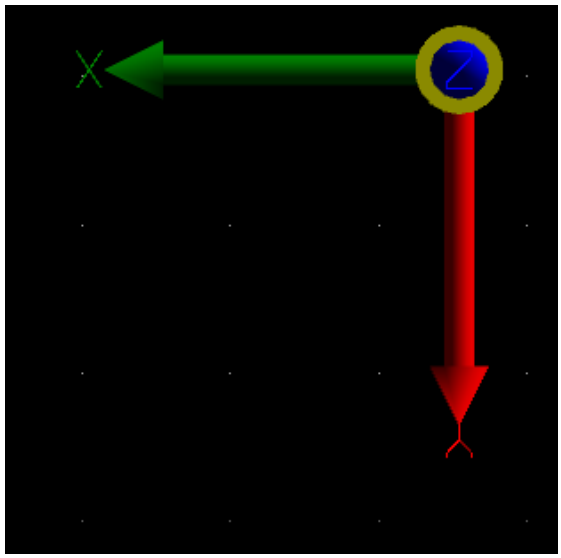
Mirror:

Enabled radio button enables mirror option for selected axis. Displayed axis of 3D coordinate system will be mirrored.

Mirror option for all axes is disabled:

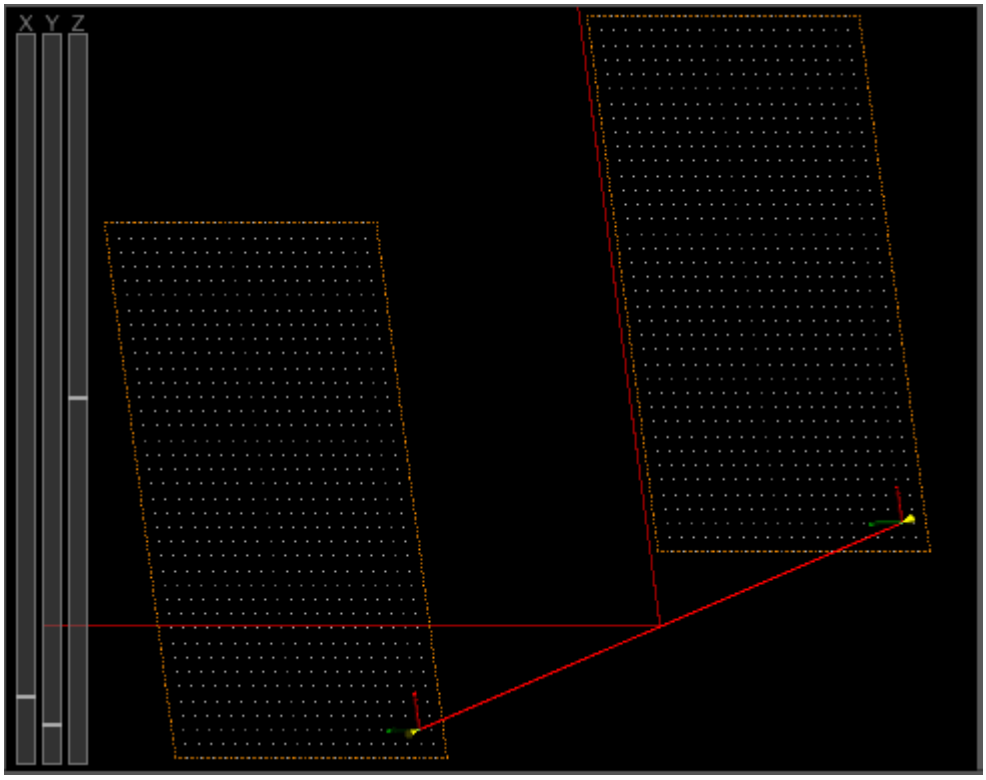


Mirror options for X and Y axes enabled:



2.9.3.19.3 Axis System:

When enabled, axis system tools will be “connected” with displayed line (E.g.: hot wire of foam cutter machine XY → UV).



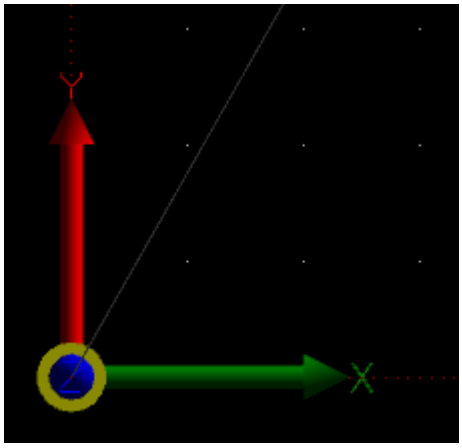
2.9.3.19.4 XYZ:

Display settings for XYZ axes of 3D coordinate system. You can enable or disable 3D display of selected axis. Set origin of axes and tool orientation.

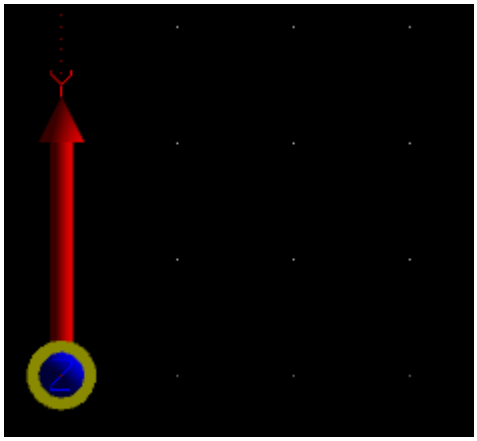
Axis:

Radio button enables or disables display of selected axis of 3D coordinate system.

All axes display is enabled:



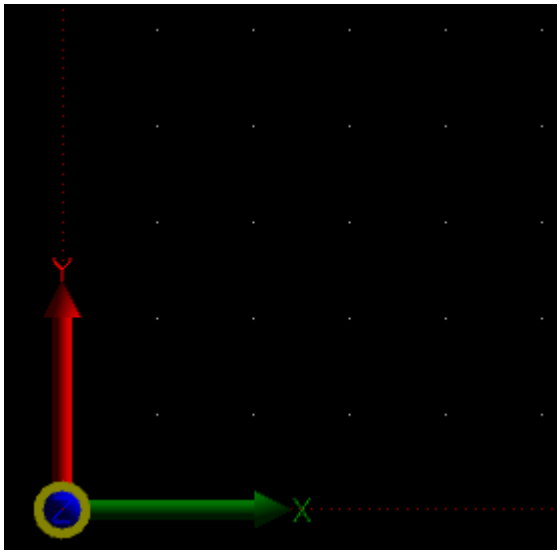
X axis display is disabled:



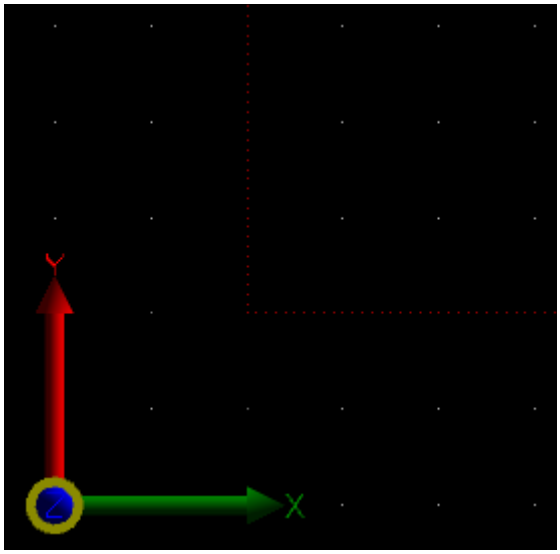
2.9.3.19.5 Origin

Sets offset of displayed axis origin for selected axis.

Origin parameter set to 0,0:



Origin parameter set to -20,-20:

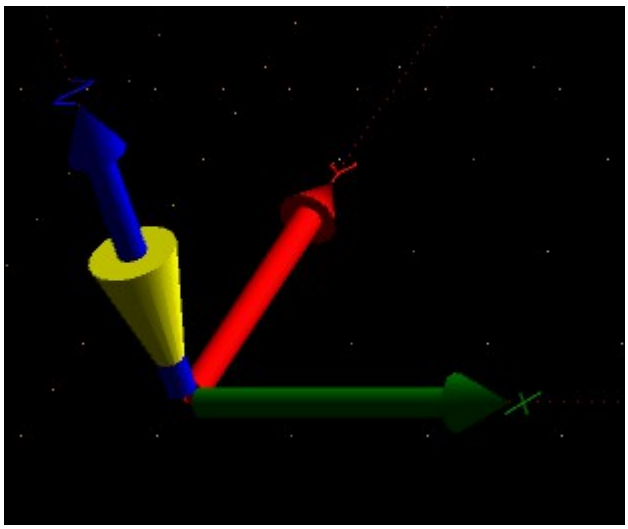


2.9.3.19.6 Orientation

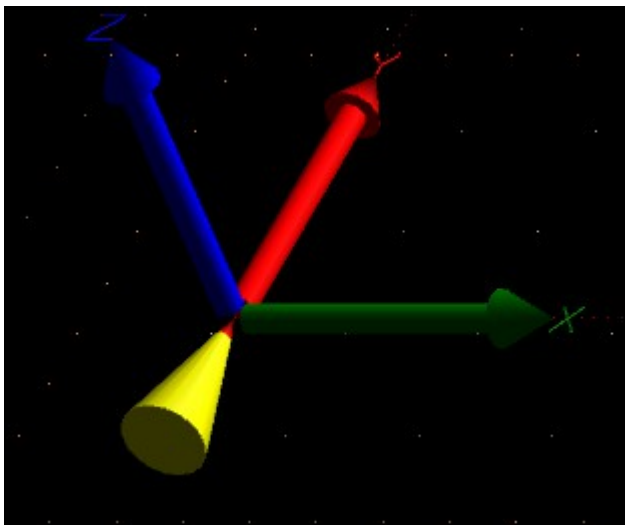
Sets orientation of 3D tool. Orientation parameter unit are degrees. Orientation of 3D tool display will be applied for selected axis in counter clockwise direction.

You can change the 3D tool orientation so that it suits your machine, E.g.: Lathe, where tool is mounted in horizontal direction.

Orientation X set to: 0,0,0



Orientation X set to: 90,0,0



2.9.3.20 ABC:

Display settings for ABC axes of 3D coordinate system. You can enable or disable 3D display of selected axis. Set origin of axes and tool orientation.

2.9.3.20.1 Axis

Radio button enables or disables display of selected axis of 3D coordinate system.

2.9.3.20.2 Origin

Sets offset of displayed axis origin for selected axis.

2.9.3.20.3 Orientation

Sets display orientation of 3D tool. Orientation parameter units are degrees. Orientation of 3D tool display will be applied for selected axis in counter clockwise direction.

2.9.3.21 UVW:

Display settings for UVW axes of 3D coordinate system. You can enable or disable 3D display of selected axis. Set origin of axes and tool orientation.

2.9.3.21.1 Axis

Radio button enables or disables display of selected axis of 3D coordinate system.

2.9.3.21.2 Origin

Sets offset of displayed axis origin for selected axis.

2.9.3.21.3 Orientation

Sets display orientation of 3D tool. Orientation parameter units are degrees. Orientation of 3D tool display will be applied for selected axis in counter clockwise direction.

2.9.3.22 *User Interface/Colors*

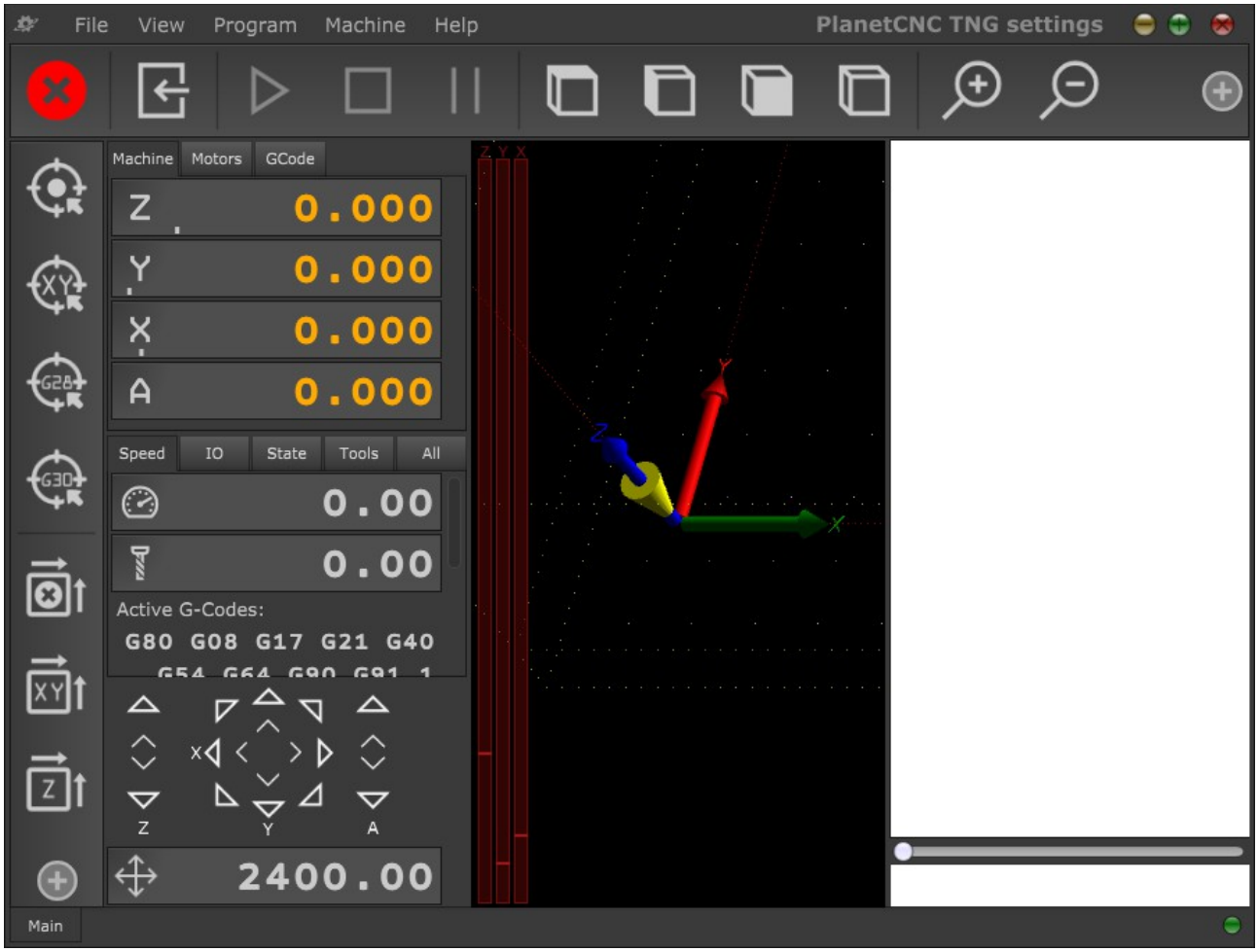
Under *Colors* tab you can change, configure and set colors of user interface display.

2.9.3.22.1 Color set

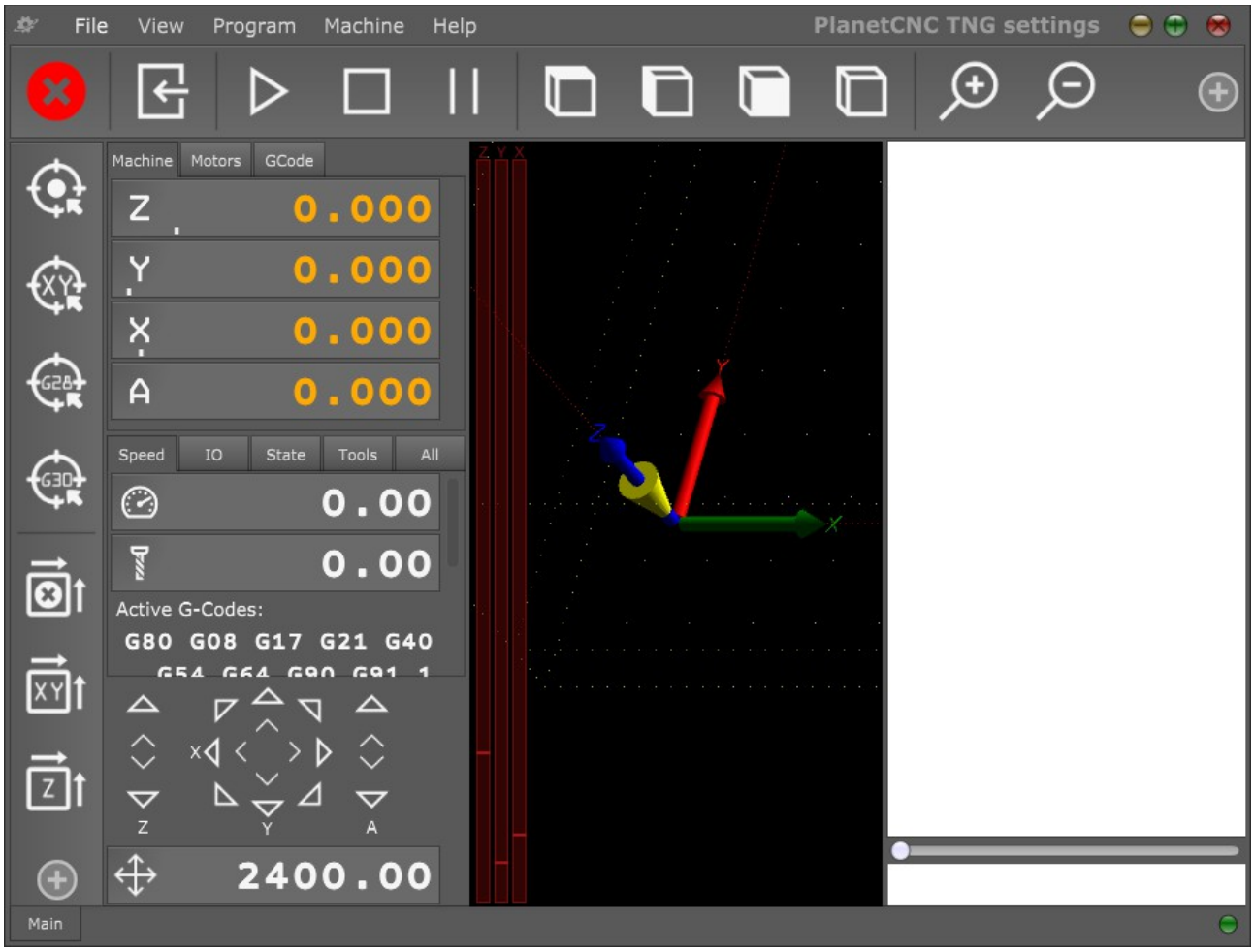
You can select color set of PlanetCNC TNG interface from drop down menu:

You can choose between four color sets of user interface display:

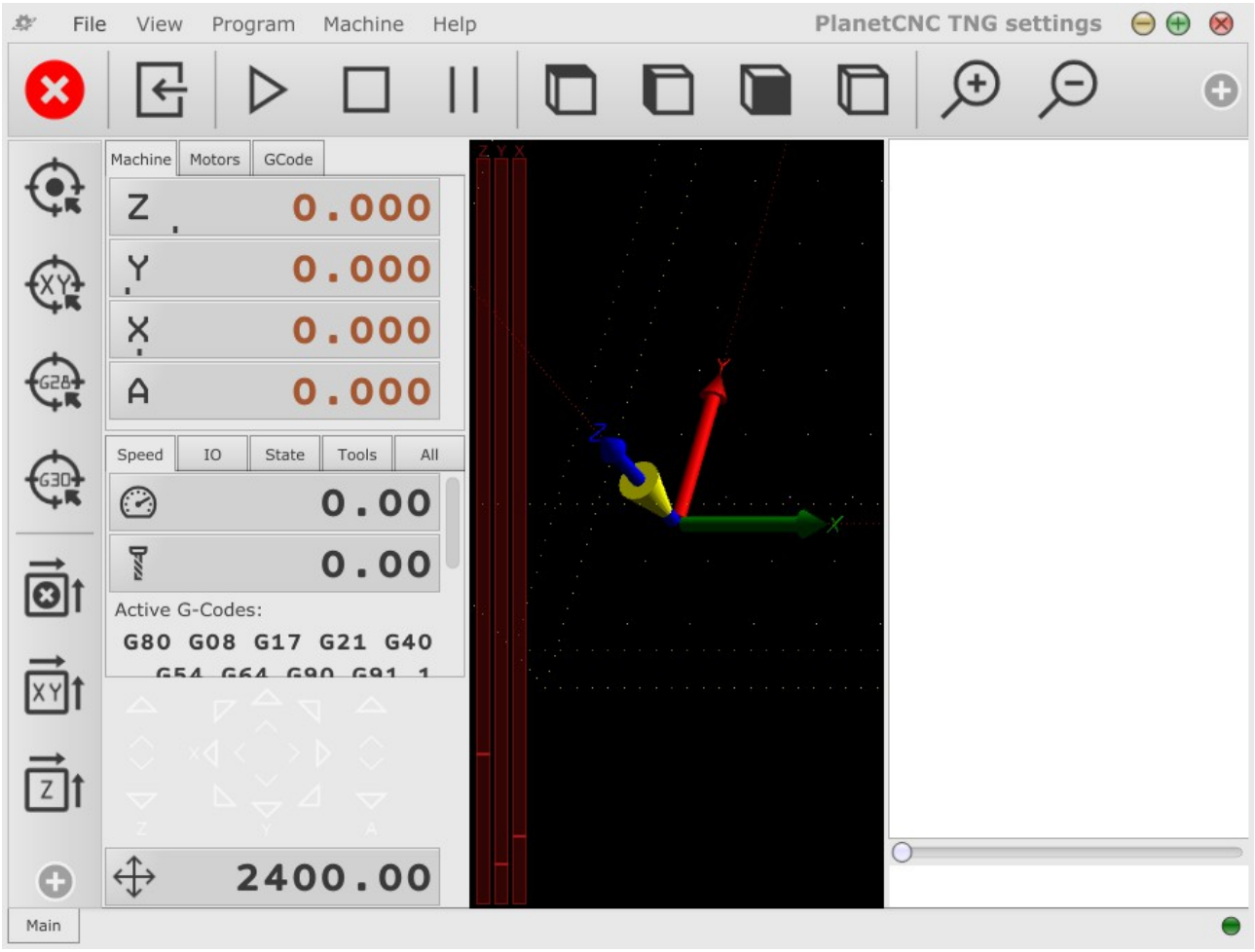
Dark:



Medium:

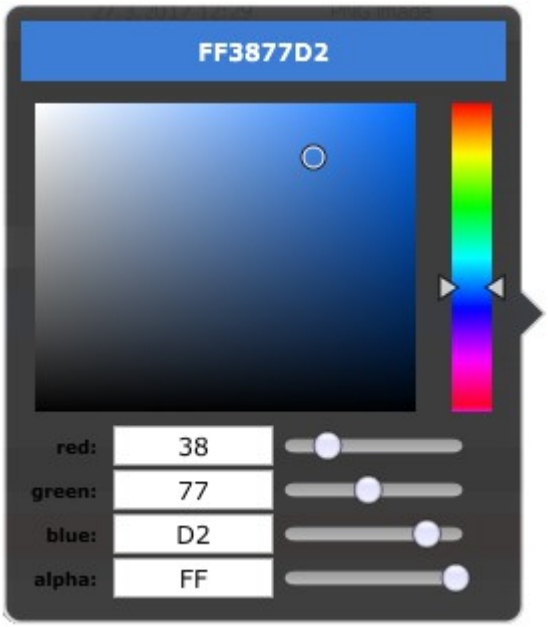

File
View
Program
Machine
Help
PlanetCNC TNG settings
Machine
Motors
GCode
Z 0.000
Y 0.000
X 0.000
A 0.000
Speed
IO
State
Tools
All
0.00
0.00
Active G-Codes:
G80 G08 G17 G21 G40
Z
Y
A
2400.00
Main

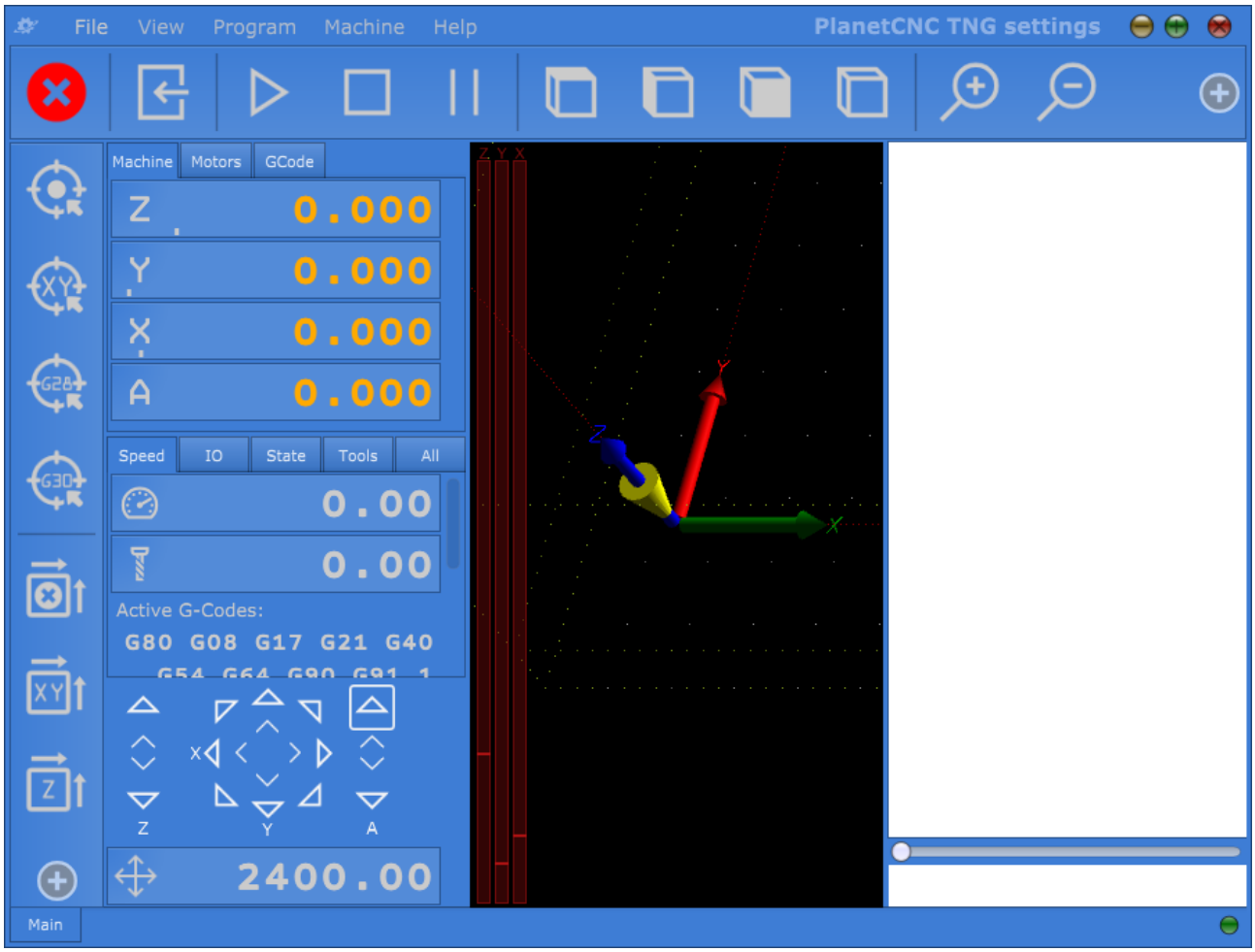
Light:


File
View
Program
Machine
Help
PlanetCNC TNG settings
Machine
Motors
GCode
Z 0.000
Y 0.000
X 0.000
A 0.000
Speed
IO
State
Tools
All
0.00
0.00
Active G-Codes:
G80 G08 G17 G21 G40
2400.00
Main

Custom:

To set custom color set from dialog you click on the star button next to the color bar:





2.9.3.22.2 3D colors

You can customize colors of user interface 3D display such as Background color, Tool position color, Limits etc...

You can set colors for:

Background

Axes

Grid

Limits

Extents

Extents Feed

Gauges

Traverse

Line

Arc

Sync

Probe

Selected

History

Position

Simulation

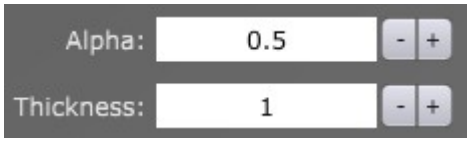
G28

G30

Wire

Wire Skin

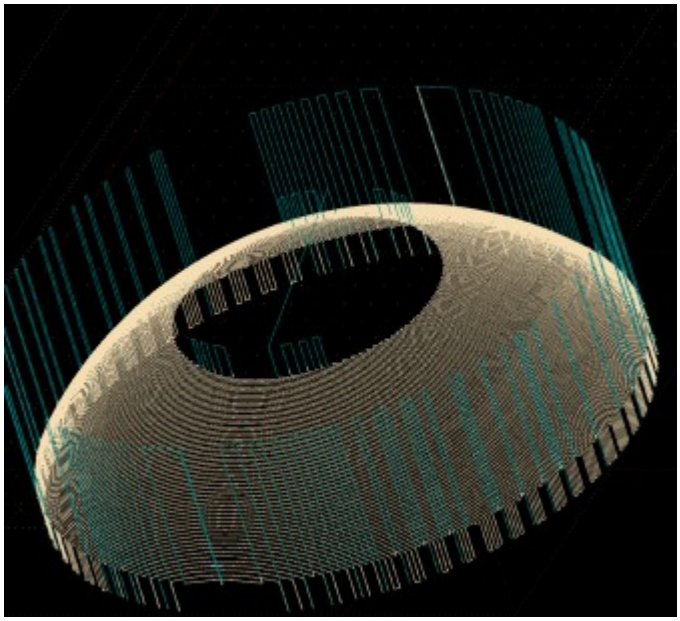
Alpha & Thickness:



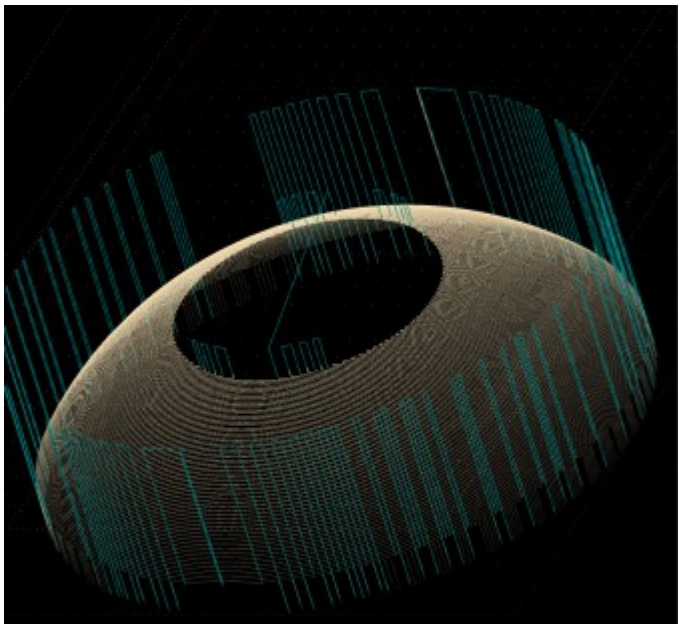
Alpha:

This parameter will affect displayed toolpath in such way, that toolpath at highest Z level will be less transparent than the one at the lowest Z level. This way user gets better sense of depth of displayed toolpath. This also helps with better overall presentation of displayed toolpath.

Displayed toolpath with *Alpha* parameter set at 1:



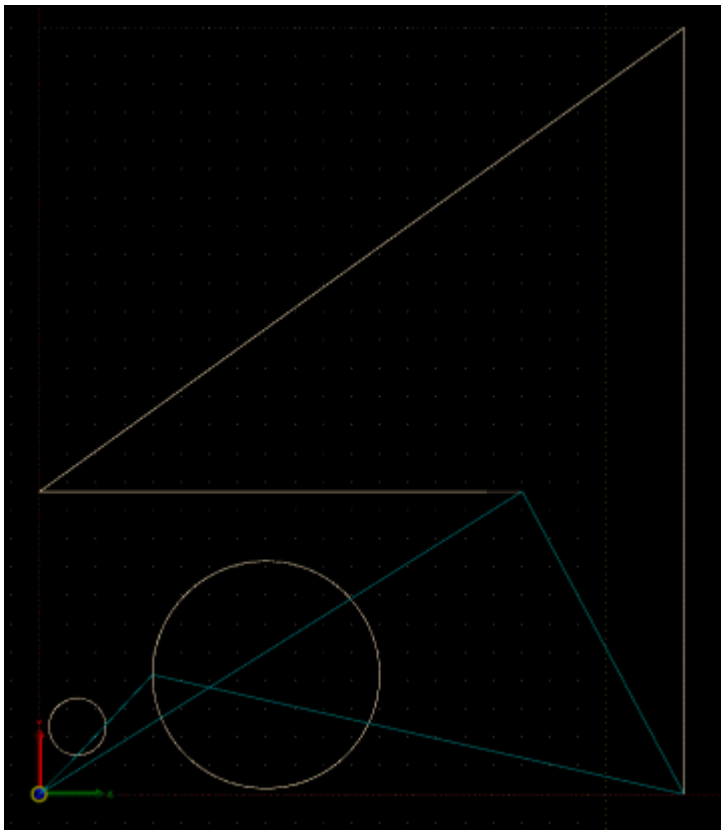
Displayed toolpath with *Alpha* parameter set at 0.1:



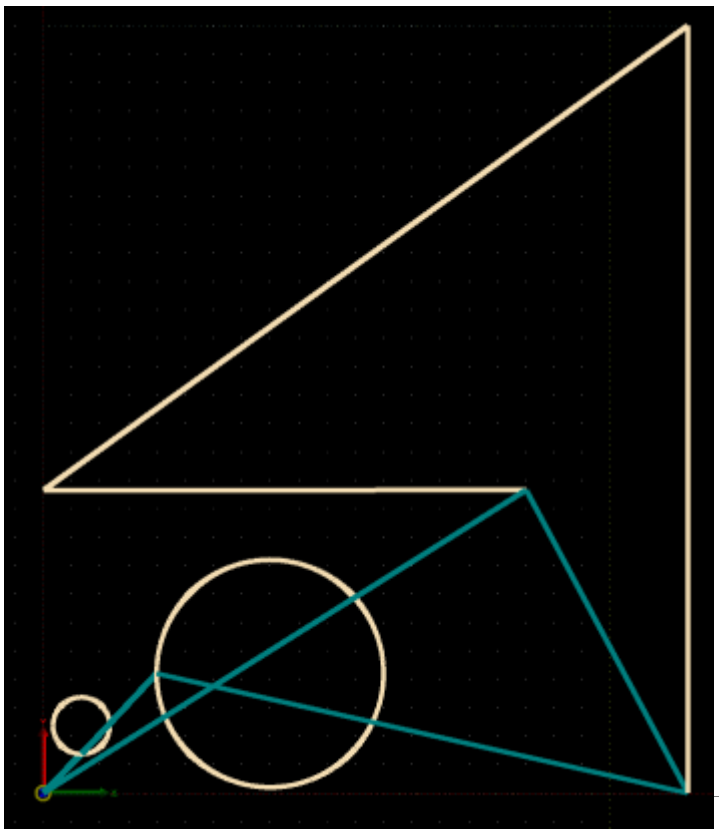
Thickness:

With this parameter you can set thickness of displayed toolpath.

Displayed toolpath with *Thickness* parameter set at 1:



Displayed toolpath with *Thickness* parameter set at 5:



2.9.3.22.3 G-Code colors

You can customize colors of G-Code program lines displayed in G-Code window.

You can set colors of g-code words such as G words, M words etc...

You can set colors for:

Line No

G Word

M word

Axis Words

IJKR Words

N Word

O Word

S Word

T Word

F Word

Numbers

Comment

Other

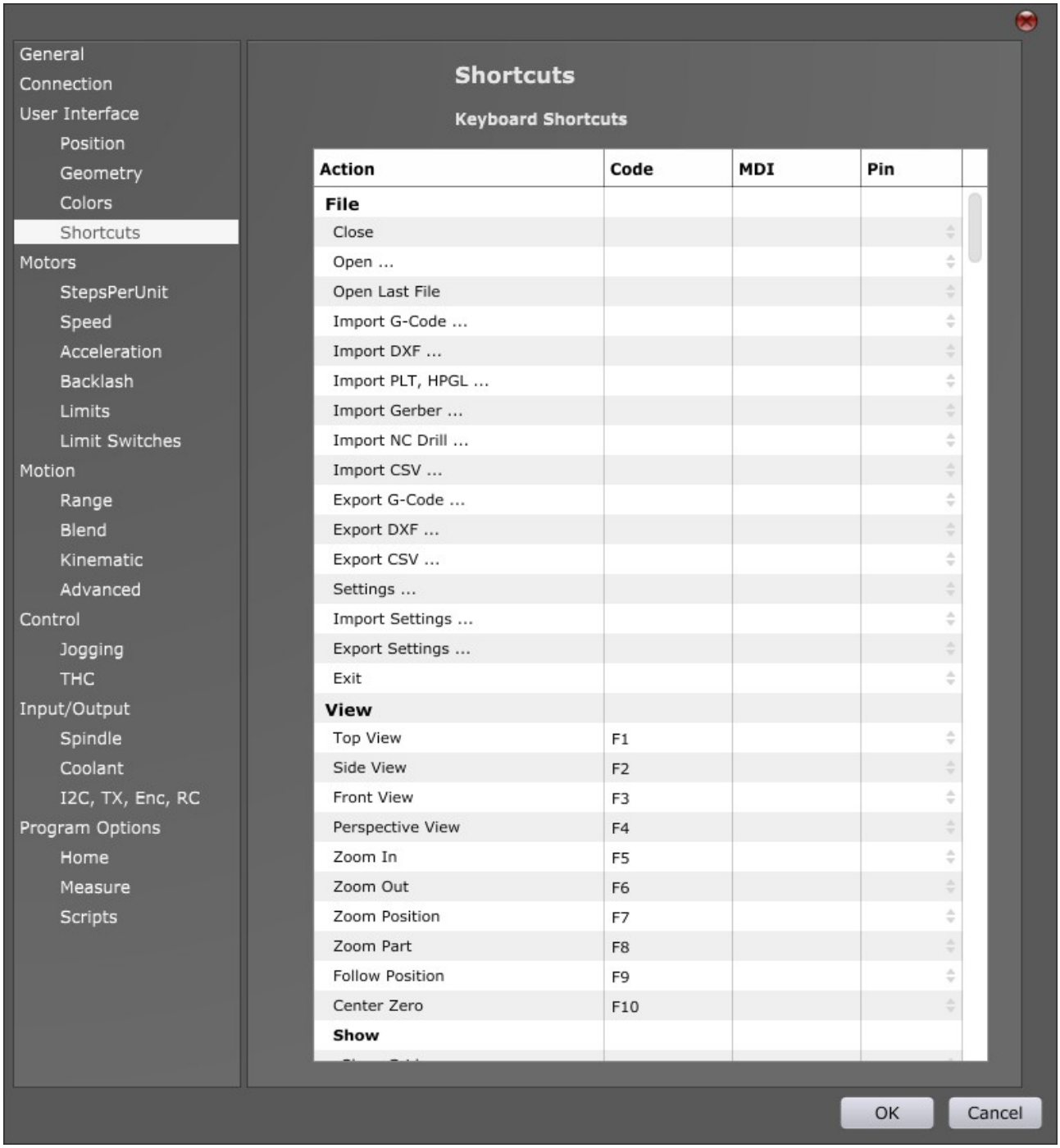
2.9.3.23 Shortcuts

Use of shortcuts can greatly reduce time when configuring machine for certain job and overall helps with more fluent work with PlanetCNC TNG.

You can set keyboard shortcuts for program features included in *File, View, Program, Machine* and *Help* menus. *Jogging* and *Overrides* set of shortcuts is also available.

With PlanetCNC TNG you can map your controller digital inputs to manipulate desired features, such as work positions etc..

You can set **Code**, **MDI** and **Pin** shortcuts for specific **Action**.



2.9.3.23.1 Action:

Here are listed all features that are included in *File*, *View*, *Program*, *Machine* and *Help* menus. Actions are organized in chapters: ***File***, ***View***, ***Program***, ***Machine*** and ***Help***

2.9.3.23.2 Code:

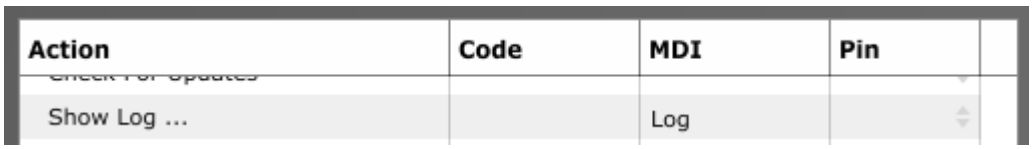
Here you can set unique code for each selected Action.

This can be either one single code (for example keyboard key for letter S as shortcut code for Action: Settings; settings dialogue will open when we press keyboard key S) or combination of multiple keyboard keys (for example combinations of keyboard keys Shift +Ctrl+Down; machine will step jog in Y- direction with keyboard keys Shift +Ctrl+Down) .

2.9.3.23.3 MDI:

Sets unique MDI code for specific Action.

Let say we want to create MDI shortcode for showing *Log* dialog. Find *Show Log* in *Help* section of *Action* chapter and enter desired MDI shortcode: E.g.: *Log*



When you try to execute desired MDI code from MDI window you need to insert symbol “/” before MDI code and click enter.

Log dialog will be displayed.

2.9.3.23.4 Pin:

You can map your controllers digital input to activate specific Action.

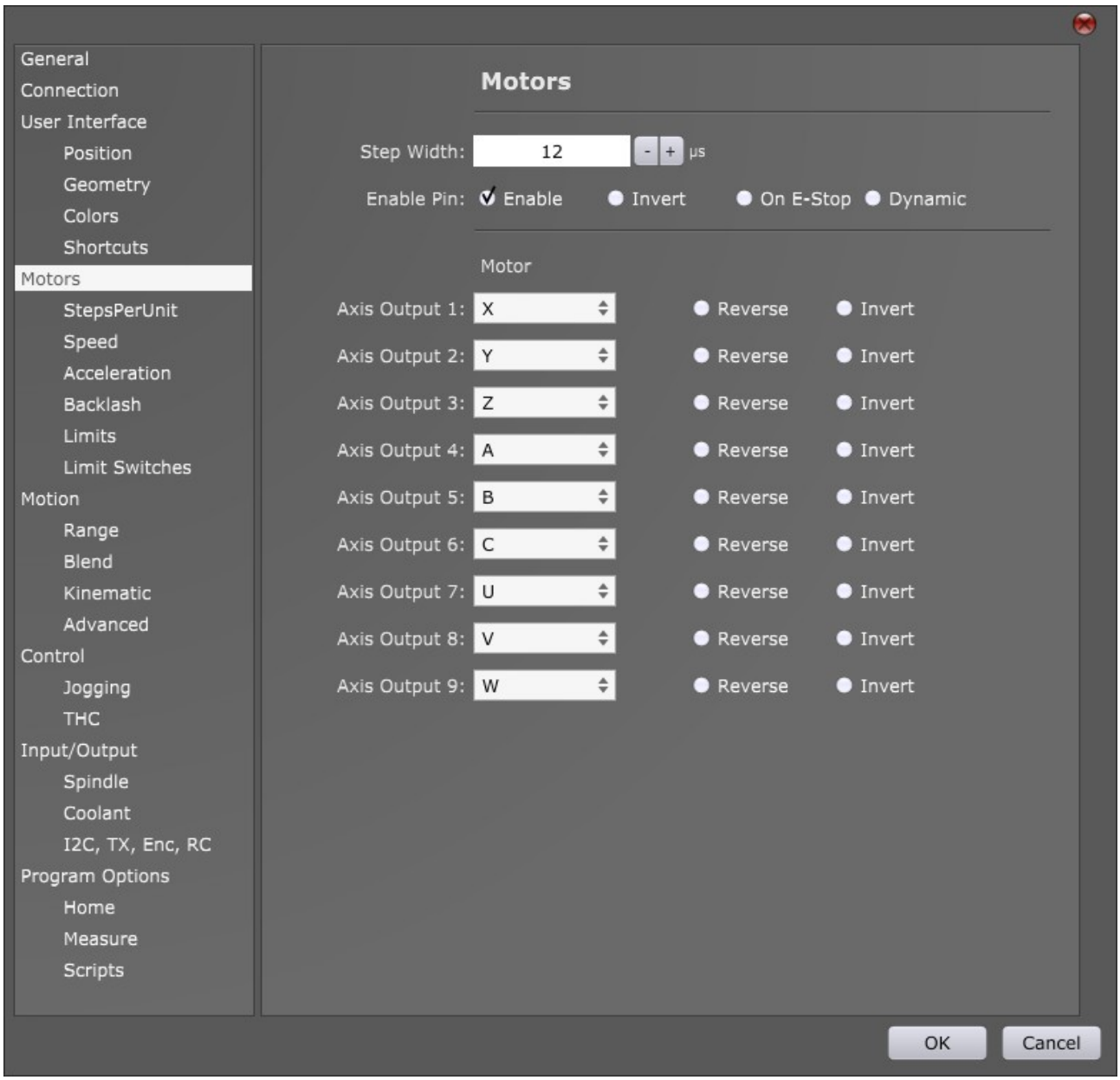
Example:

From drop down menu select digital input for specific *Action*.

When digital input is active, action mapped to that input will execute:

2.9.4 Motors

Under this group of settings you can configure settings important for compatibility with stepper/servo drives, enable signal configuration, controller axis output configuration, motor speeds and accelerations, axis backlash value, limitations of motors as also hardware limit switch configuration.



2.9.4.1 Step Width

Width of STEP pulse. Units of parameter are micro seconds.

If step frequency is too high for inserted *Step Width* value, *Step Width* will be set to 50% of duty cycle.

2.9.4.2 Enable Pin

PlanetCNC controllers support *Enable* signal via enable pin located on axis output header.

Enable:

Enables use of Enable signal.

Invert:

Inverts enable signal(pulse) from 0V to 5V or from 5V to 0V to match your stepper driver requirements.

On E-Stop:

Enable signal is disabled when E-stop is active.

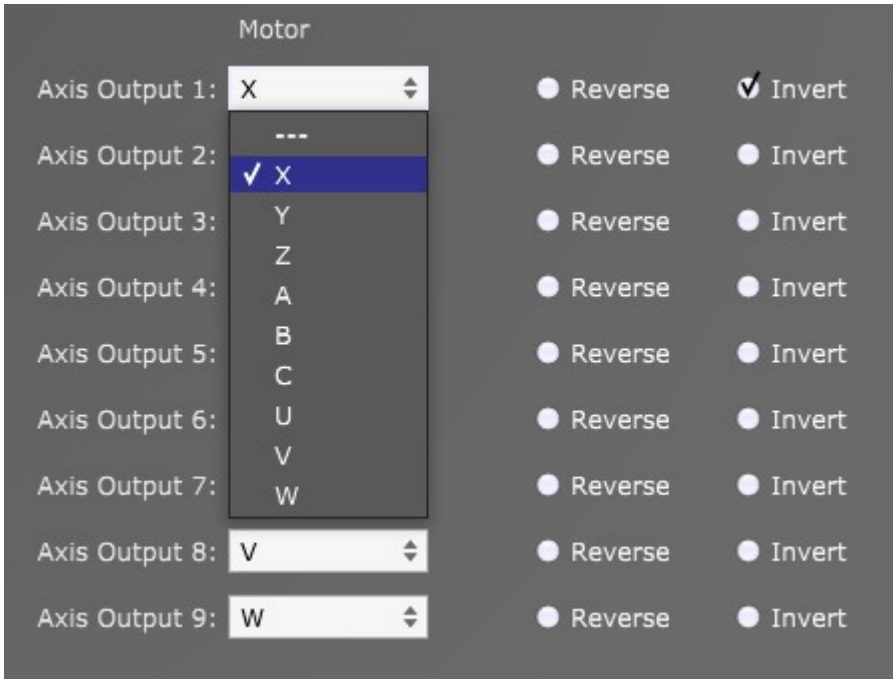
Dynamic:

Enable signal is enabled only when motion is applied to motors.

2.9.4.3 Motor

PlanetCNC controllers support up to 9* interpolated axis outputs, depending on controller used.

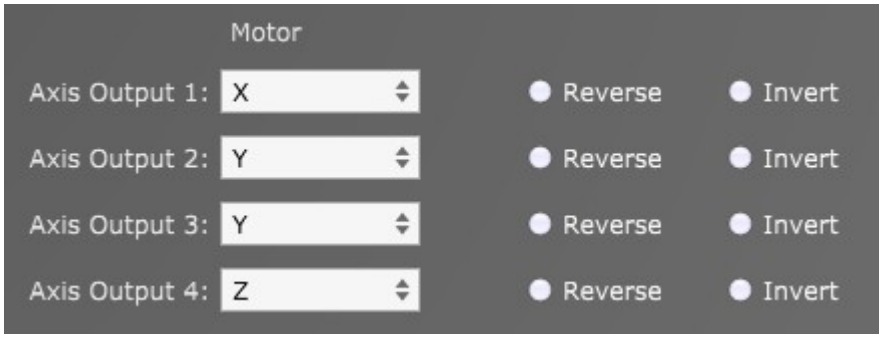
Each axis output can generate steps for any of 9 linear or rotational axes.



Axis Output 1-9*:

For each of 1-9* axis outputs you can select axis motor of your CNC.

If you need two axis outputs to generate synchronized steps (two axis outputs used for same machine axis -> slave axes) you just set same axis motor for both axis outputs:



Reverse:

Reverses direction of motor rotation.

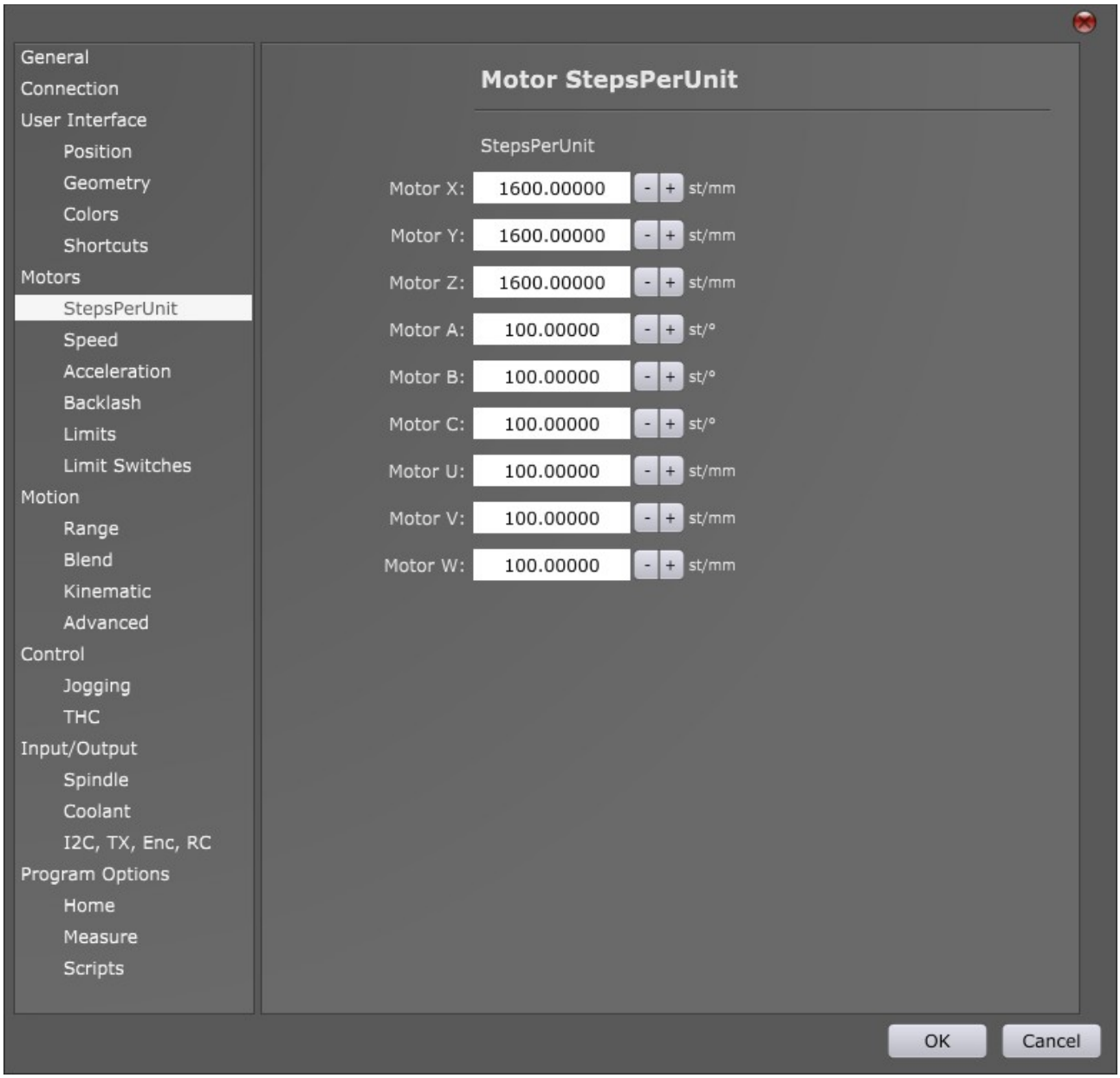
Invert:

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

*4 for Mk3/4 and Mk3ECO

2.9.4.4 Steps Per Unit

Steps per unit value (in further text SPU) defines how many step pulses controller needs to generate in order that machine axis moves for distance of one unit. Units can be in millimeters or in inches.

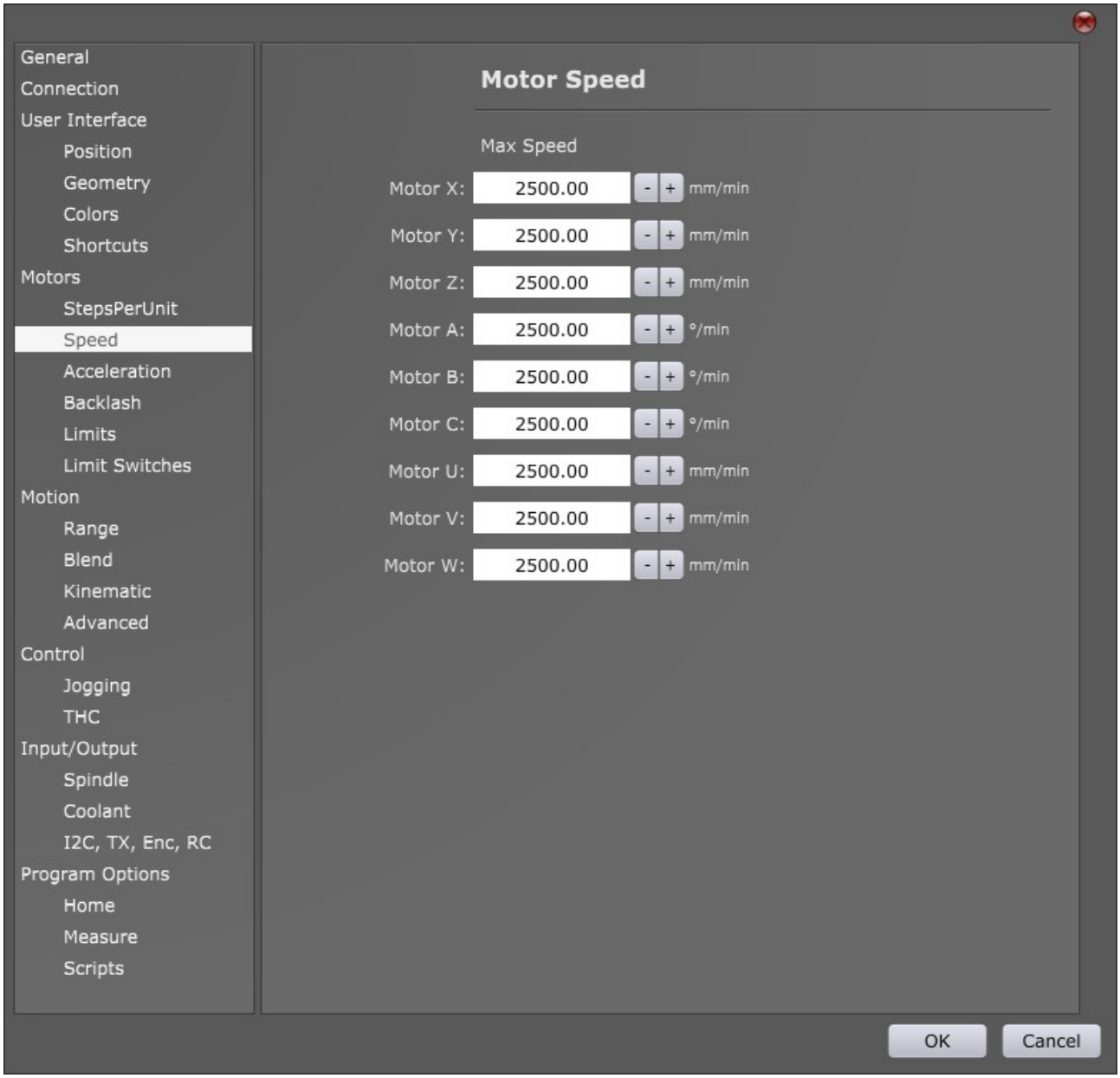


Motor settings for linear axes X,Y,Z, U ,V, W use steps per millimeter units (st/mm) while motor settings for rotational axes A, B, C use steps per degree units (st/).

2.9.4.5 Speed

You can set maximum speed of each axis motor.

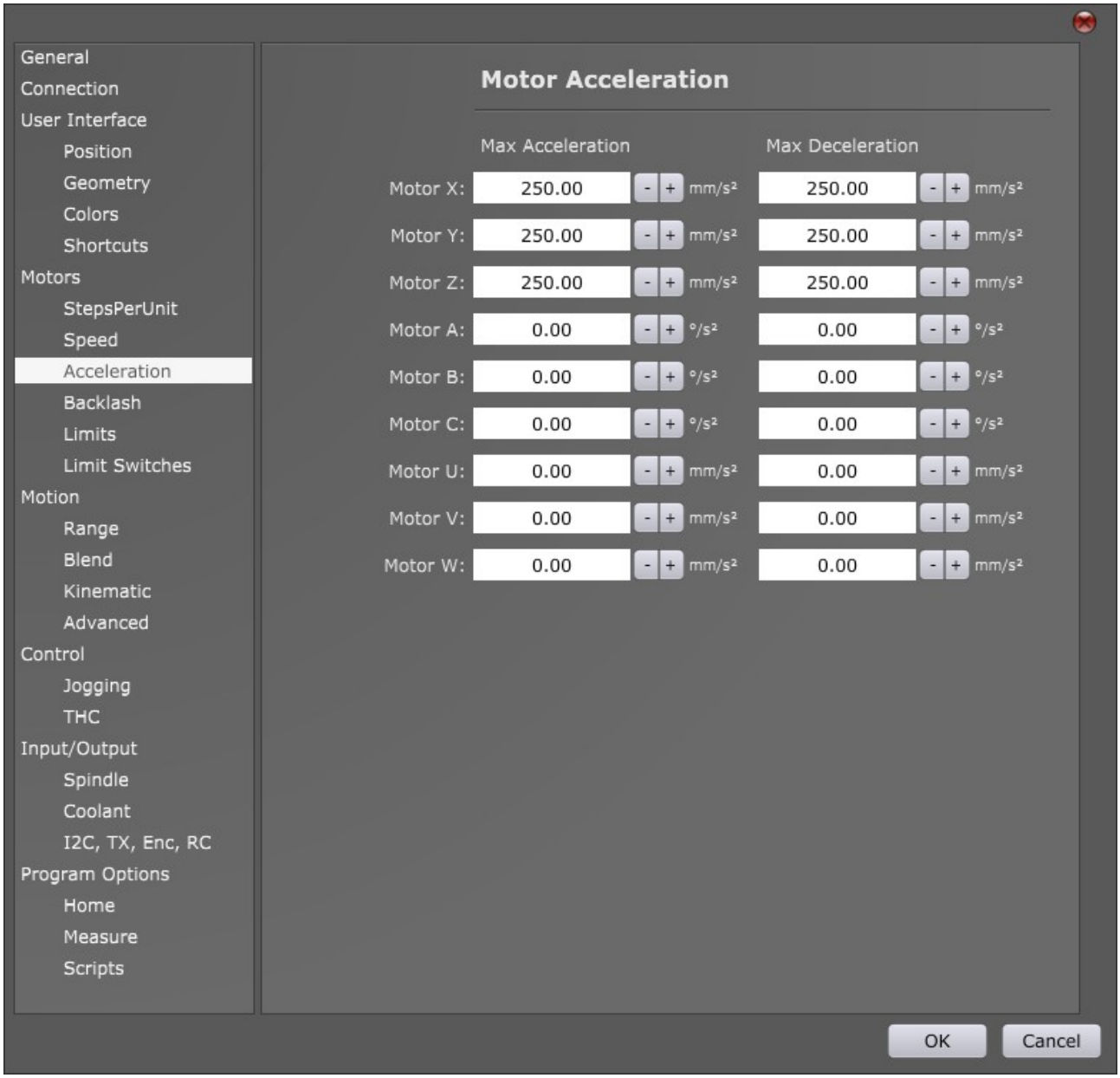
Motors will not exceed speed values set with these parameters.



2.9.4.6 Acceleration

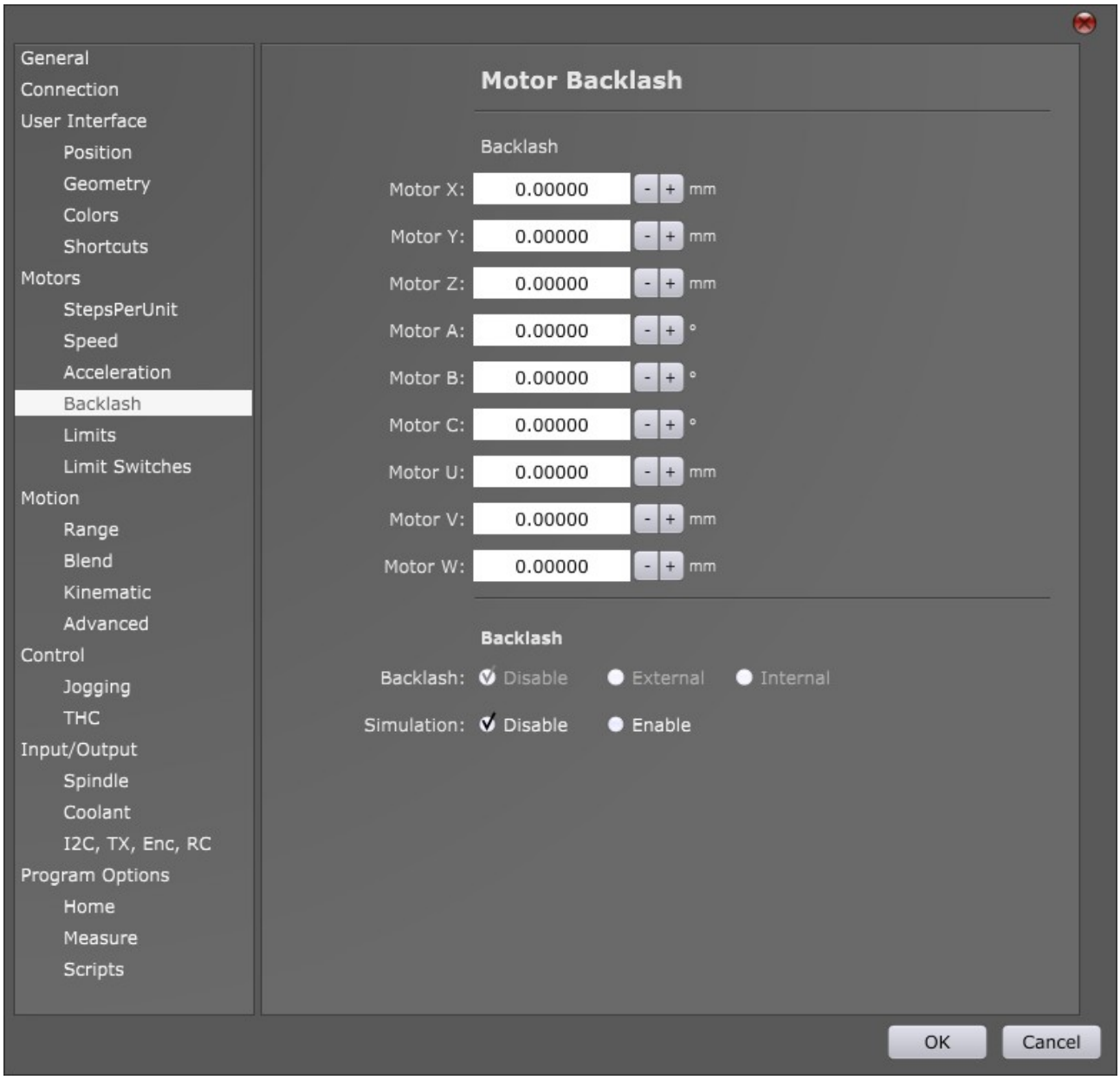
You can set maximum acceleration and deceleration values of each axis motor.

Motors will not exceed acceleration and deceleration values set with these parameters.



2.9.4.7 Motor Backlash

Set backlash compensation value if you machine has backlash.



Units of backlash compensation values for linear axes are in mm, for rotational axes are in degrees.

2.9.4.7.1 Backlash

Disable: Disables backlash compensation.

External: Controller receives external command for backlash compensation. **Currently not supported.**

Internal: Enables backlash compensation. Software will generate additional steps necessary for backlash compensation.

2.9.4.7.2 Simulation

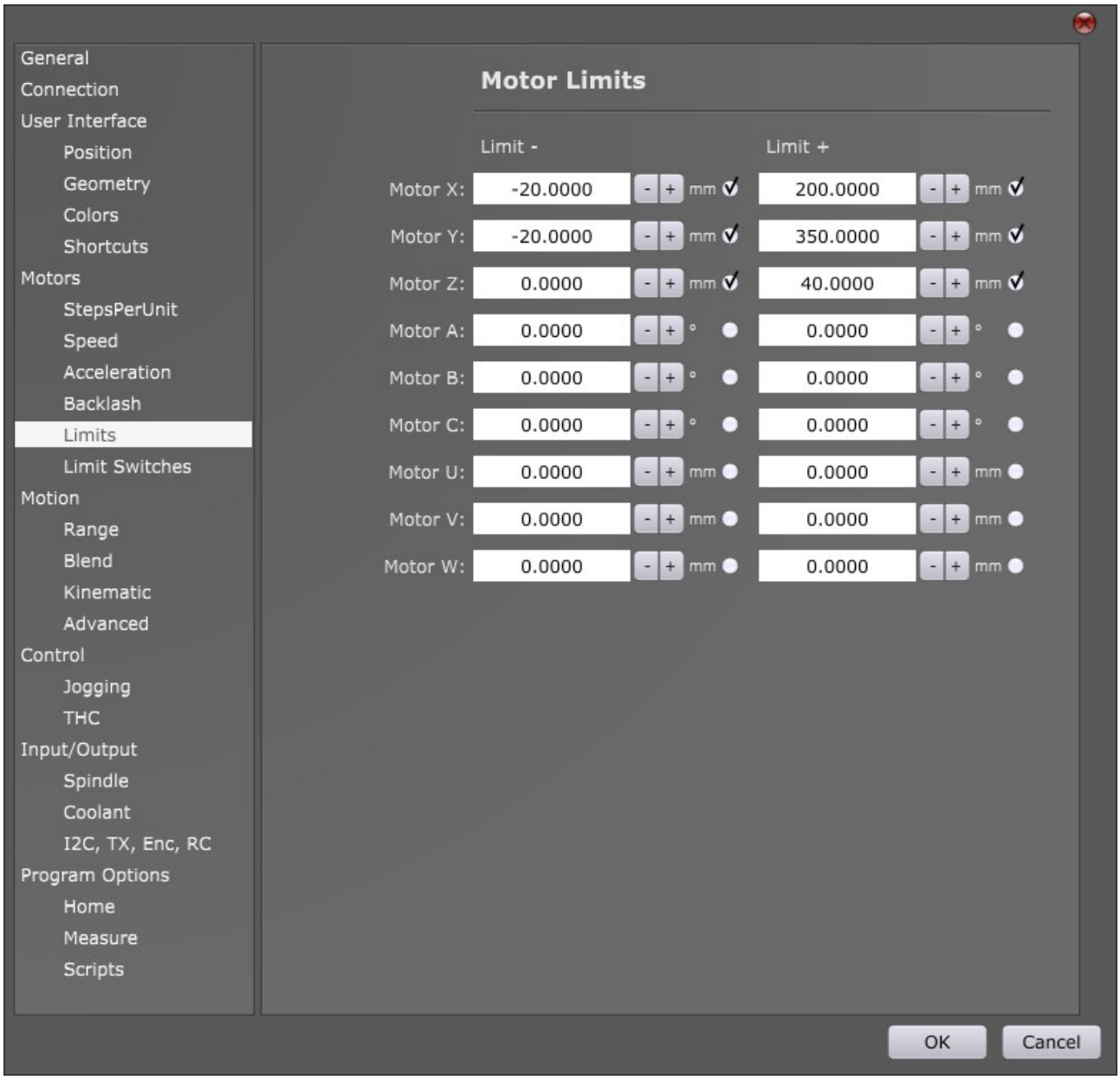
If *Simulation* is enabled in settings under chapter *Connection (*Chapter 2.7.2*)*, you can enable or disable the backlash compensation in simulation mode.

Disable: Disables the backlash compensation in simulation mode.

Enable: Enable the backlash compensation in simulation mode.

2.9.4.8 Motor Limits

You can set movement limitations for each axis motor.



Limit- value sets limitation of motor movement in negative direction.

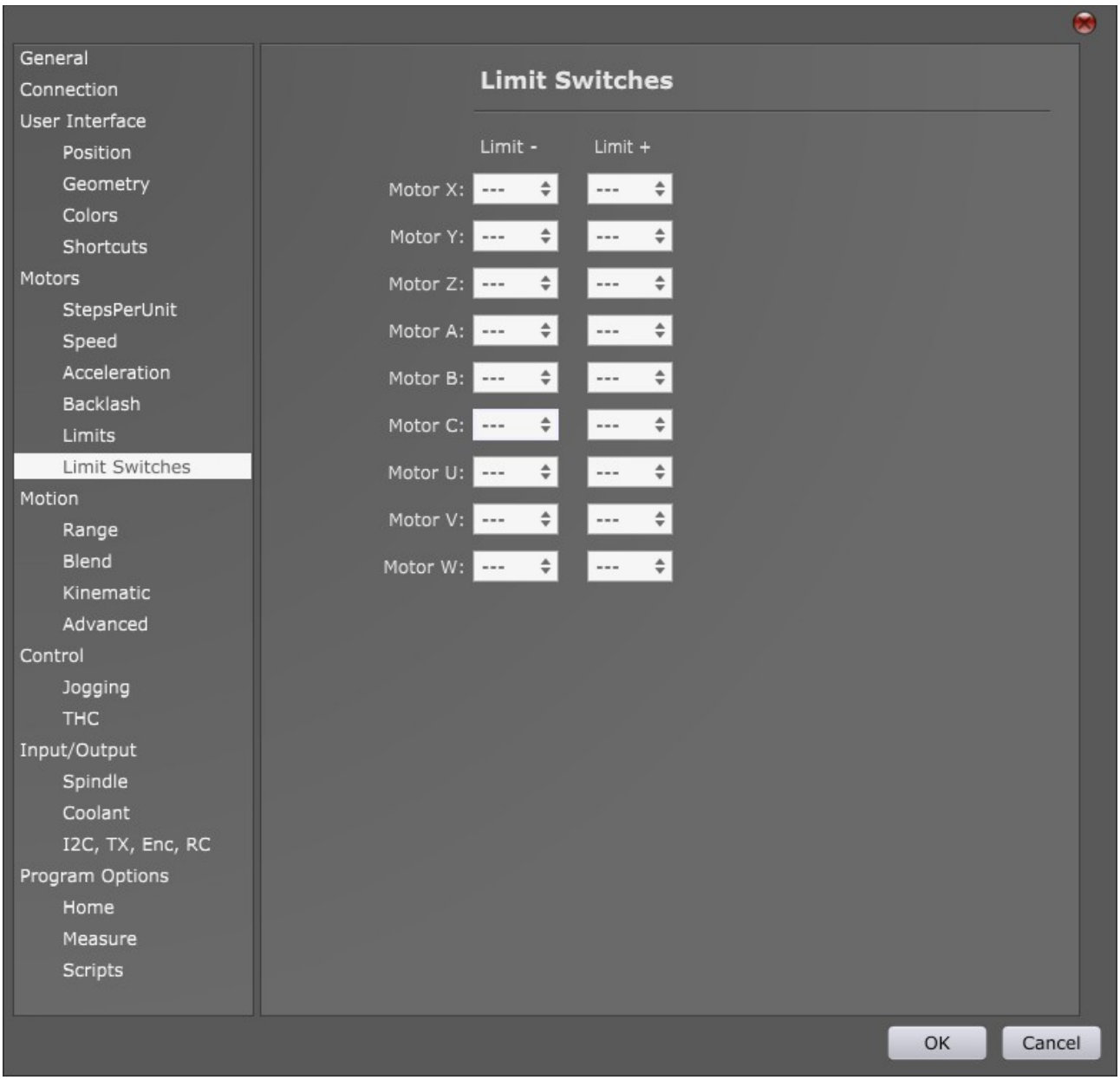
Limit+ value sets limitation of motor movement in positive direction.

If we want motors to stop when limits are reached, we must enable motor limitation. To enable Motor limits for specified axis, enable the radio button.

2.9.4.9 Limit Switches

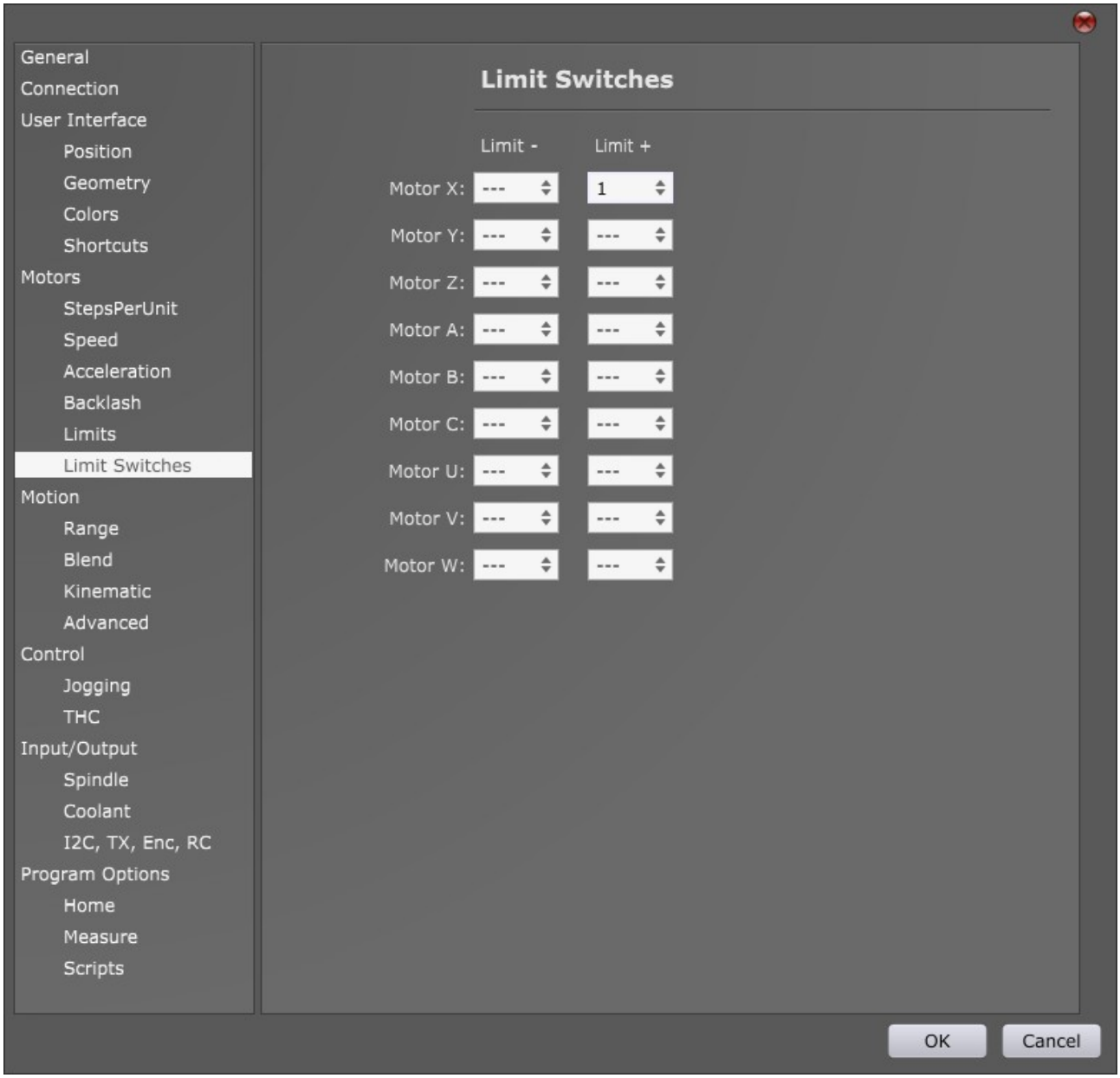
Using these parameters we can dedicate each axis motor its limit switch input.

We can assign limit switch input for each direction of motor movement.

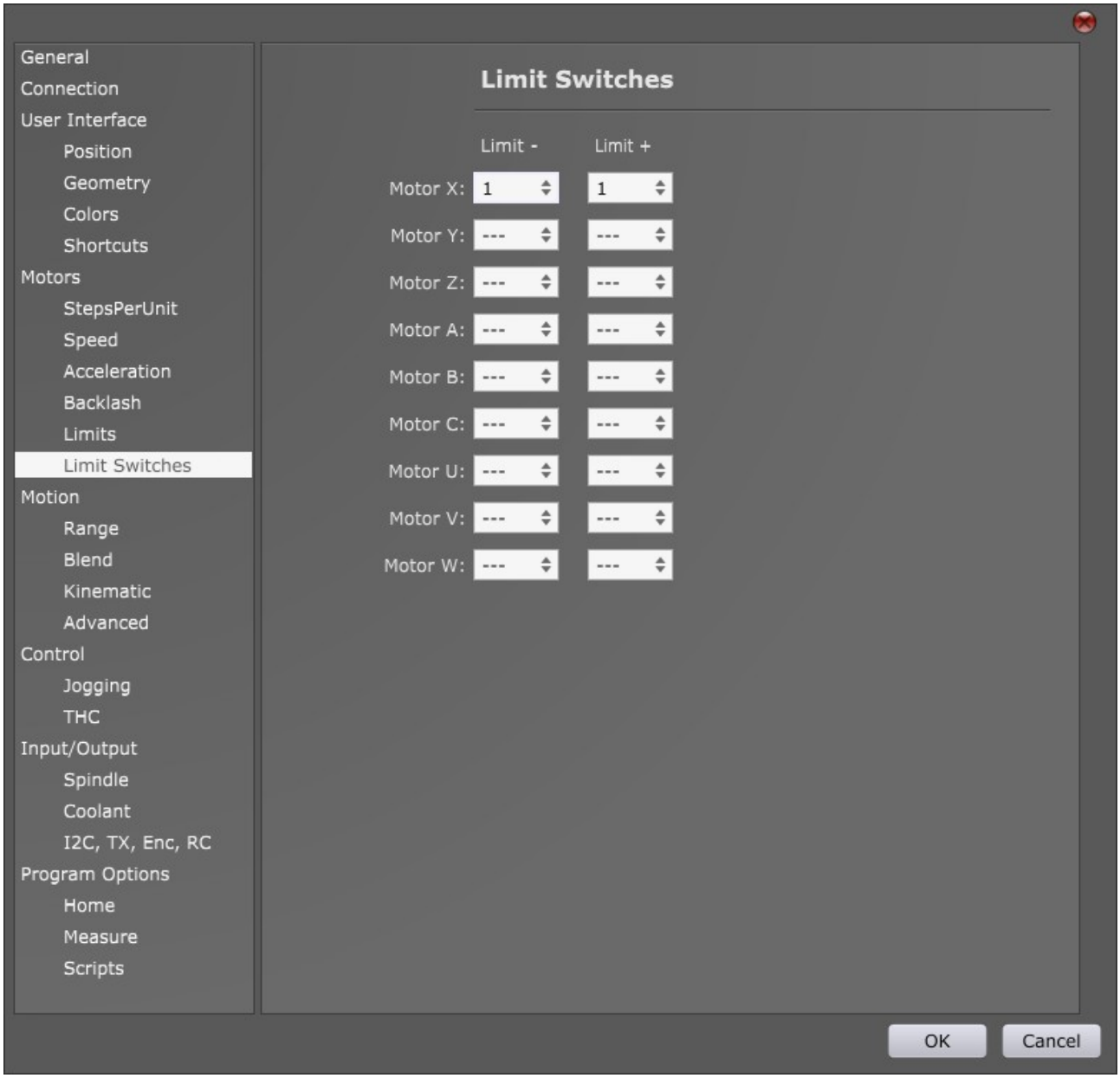


There are many methods of limit switch connection, here are few examples.:

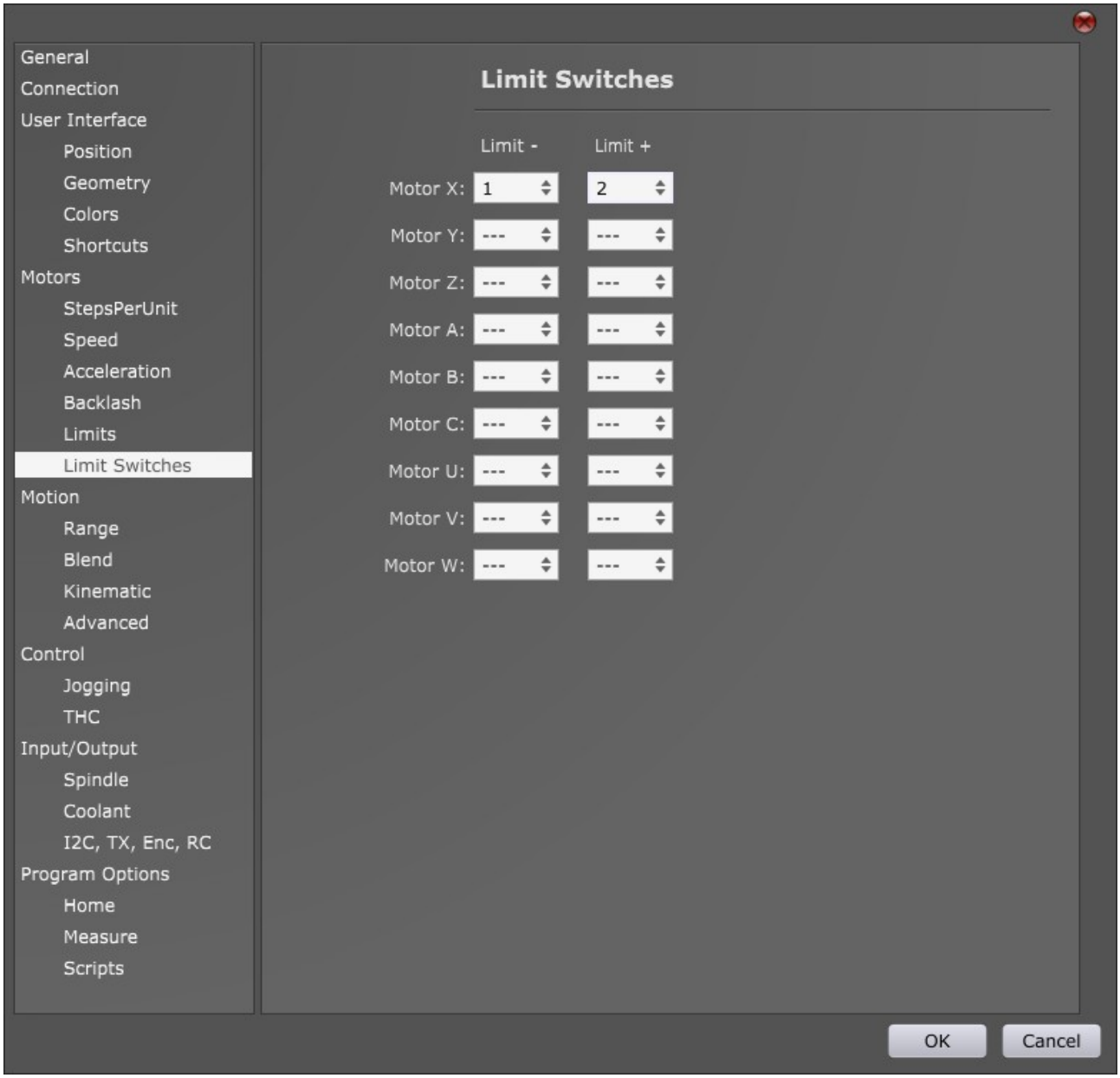
-One limit switch for axis X motor (positive direction), connected to controllers limit input 1:



-Two limit switches for axis X motor (negative and positive direction), connected to same limit input of controller.



-Two limit switches for X axis(negative and positive direction), connected to separate limit inputs of controller, 1 and 2:



Same principles that were shown above for X axis can be used for all other axes.

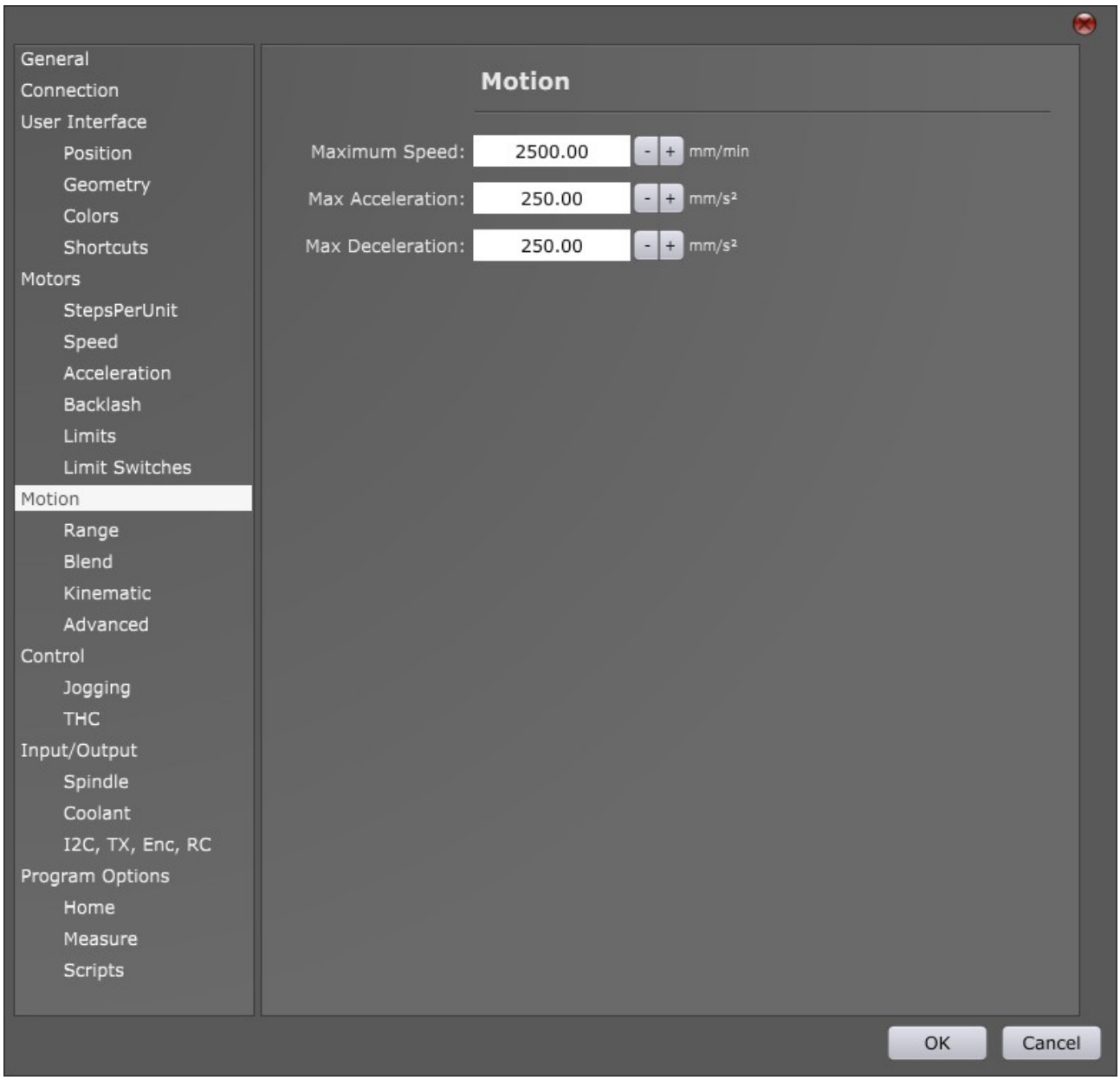
2.9.5 Motion

This settings tab deals with machine motion in 3D space (from effector (tool) point of view).

Short explanation:

With use of combined motors per axis and special mechanics you can achieve movement which doesn't follow the same kinematic rules as linear motion CNC's. At that moment we do not perceive each motor as independent axis but as a system, which movement is result of simultaneous motion of multiple motors. Examples of such machines are H-bot, Delta printer or multiple axis machines.

You can configure motion related parameters such as maximum machine/effector speeds, acceleration, deceleration. Machine motion range, blend and advanced settings.



Maximum Speed:

Sets maximum speed of tool/effector. Tool/effector will not exceed speed value set with this parameter.

Max Acceleration:

Sets maximum acceleration of tool/effector. Tool/effector will not exceed acceleration value set with this parameter.

Max Deceleration:

Sets maximum deceleration of your tool/effector. Tool/effector will not exceed deceleration value set with this parameter.

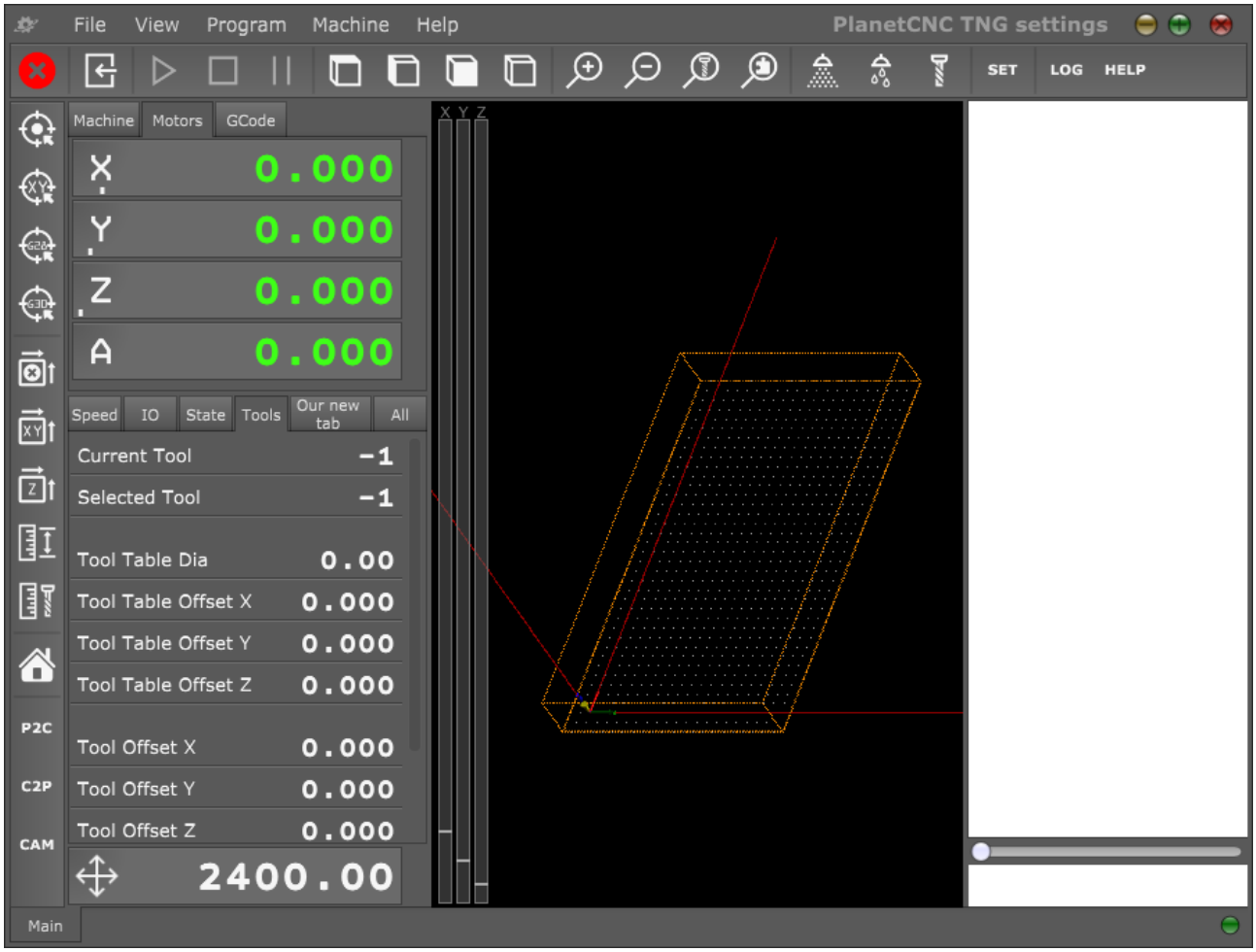
2.9.5.1 Range

With *Range* parameters you set software limits of machine.

Range- value sets limitation of machine/effector (tool) movement in negative direction, **Range+** value sets limitation of machine/effector (tool) movement in positive direction for specific axis.

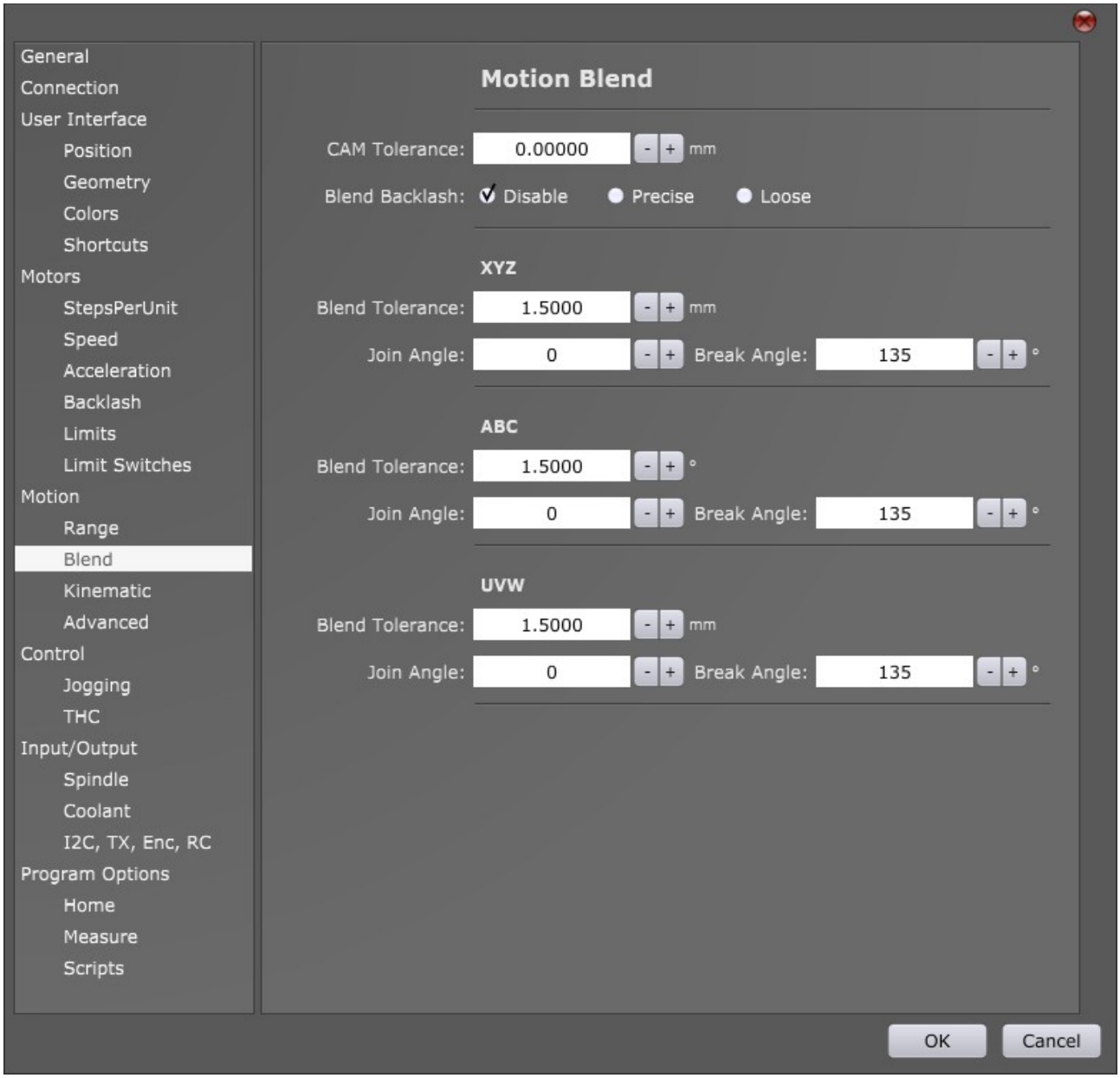
With *Range* values inserted, we set motion limits of machine. If we want machine to stop when limits are reached we must enable them. To enable Range limits for specified axis click the radio button:

When you set and enable your machines motion range values, you will notice that 3D representation of motion range limits will be displayed accordingly on the main screen:



2.9.5.2 Blend

This group of settings helps with tool-path optimization.



2.9.5.2.1 CAM tolerance:

Optimizes toolpath program performance by removing short and un-efficient g-code lines in g-code program.

If your program includes linear moves that are shorter than inserted value of this parameter, then software will remove them.

Recommended value: 0.001mm

2.9.5.2.2 Blend Backlash/Disable

When selected, blending of Backlash compensation moves is disabled.

2.9.5.2.3 Blend Backlash/Precise

When selected, blending of backlash moves is enabled, motion is well inside the CAM tolerance values.

2.9.5.2.4 Blend Backlash/Loose

When selected, blending of backlash moves is enabled, motion is not restricted by CAM tolerance values.

2.9.5.2.5 XYZ/Blend Tolerance

XYZ axis tolerance(maximum distance) that blended toolpath can deviate from toolpath.

2.9.5.2.6 XYZ/Join Angle

Angle value of changed direction between machines current and next move. If angle is greater than this value, motion between these two moves will be blended.

Set at 0 by default.

2.9.5.2.7 XYZ/Break Angle

Angle value of changed direction between machines current and next move. If angle is greater than this value, motion between these two moves will not be blended.

Set at 135 by default.

2.9.5.2.8 ABC/Blend Tolerance

ABC axes tolerance (maximum distance) that blended toolpath can deviate from toolpath.

2.9.5.2.9 ABC/Join Angle

Angle value of changed direction between machines current and next move. If angle is greater than this value, motion between these two moves will be blended.

Set at 0 by default.

2.9.5.2.10 ABC/Break Angle

Angle value of changed direction between machines current and next move. If angle is greater than this value, motion between these two moves will not be blended.

Set at 135 by default.

2.9.5.2.11 UVW/Blend Tolerance

UVW axes tolerance (maximum distance) that blended toolpath can deviate from toolpath.

2.9.5.2.12 UVW/Join Angle

Angle value of changed direction between machines current and next move. If angle is greater than this value, motion between these two moves will be blended.

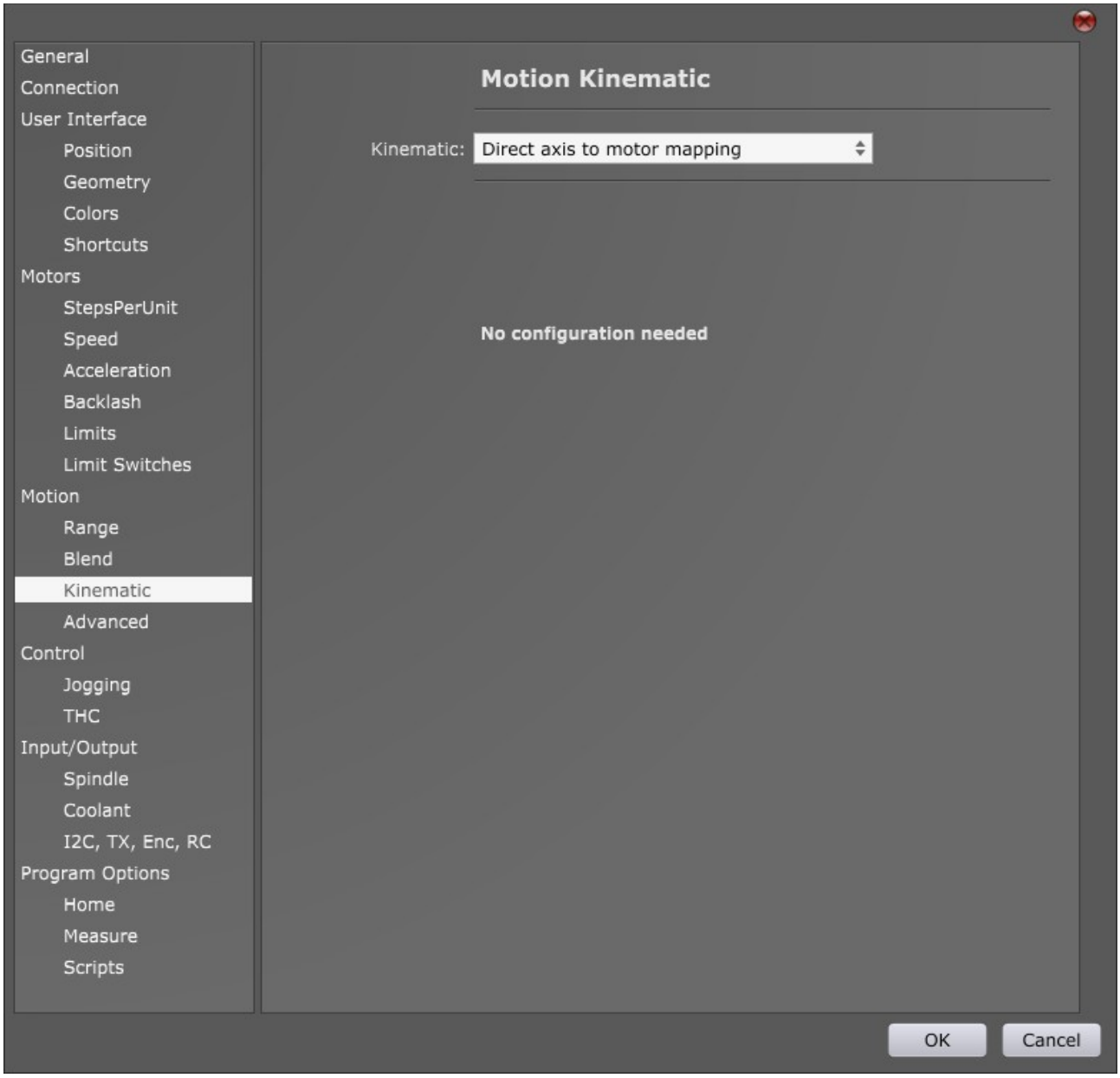
Set at 0 by default.

2.9.5.2.13 UVW/Break Angle

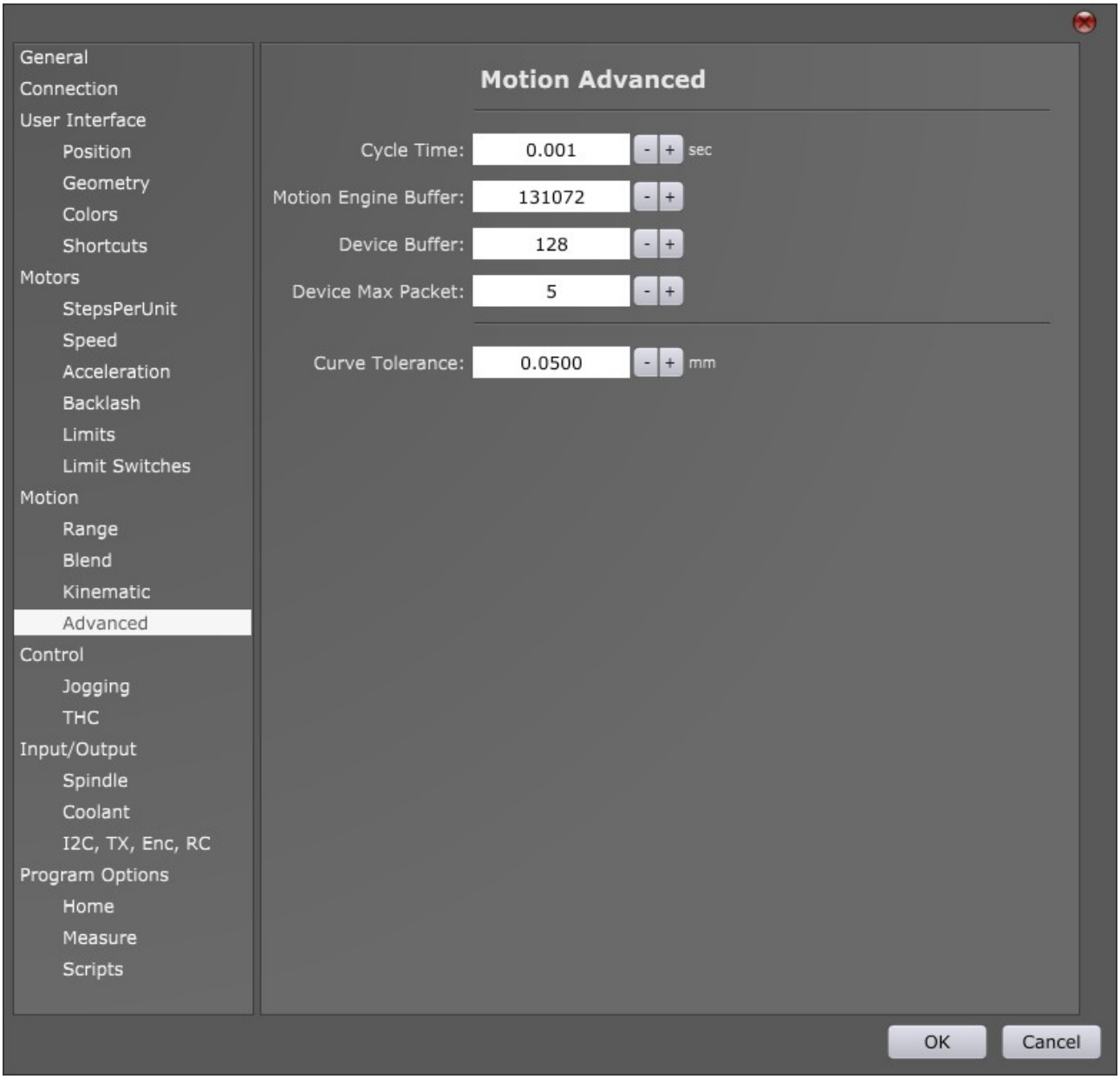
Angle value of changed direction between machines current and next move. If angle is greater than this value, motion between these two moves will not be blended.

Set at 135 by default.

2.9.5.3 Kinematic



2.9.5.4 Advanced



2.9.5.4.1 Cycle Time

Motion interpolator cycle time.

Leave this at default value, otherwise controller will not function correctly.

2.9.5.4.2 Motion Engine buffer

Motion interpolator buffer size.

Leave this value at default value, otherwise controller will not work correctly.

2.9.5.4.3 Device Buffer

Controller buffer size. Lowering this value increases controller response time but also reduces fault tolerance. Using default value 128 is highly recommended.

2.9.5.4.4 Device Max Packets

Max size of data packets sent to controller.

Lowering this value reduces response time but also reduces fault tolerance.

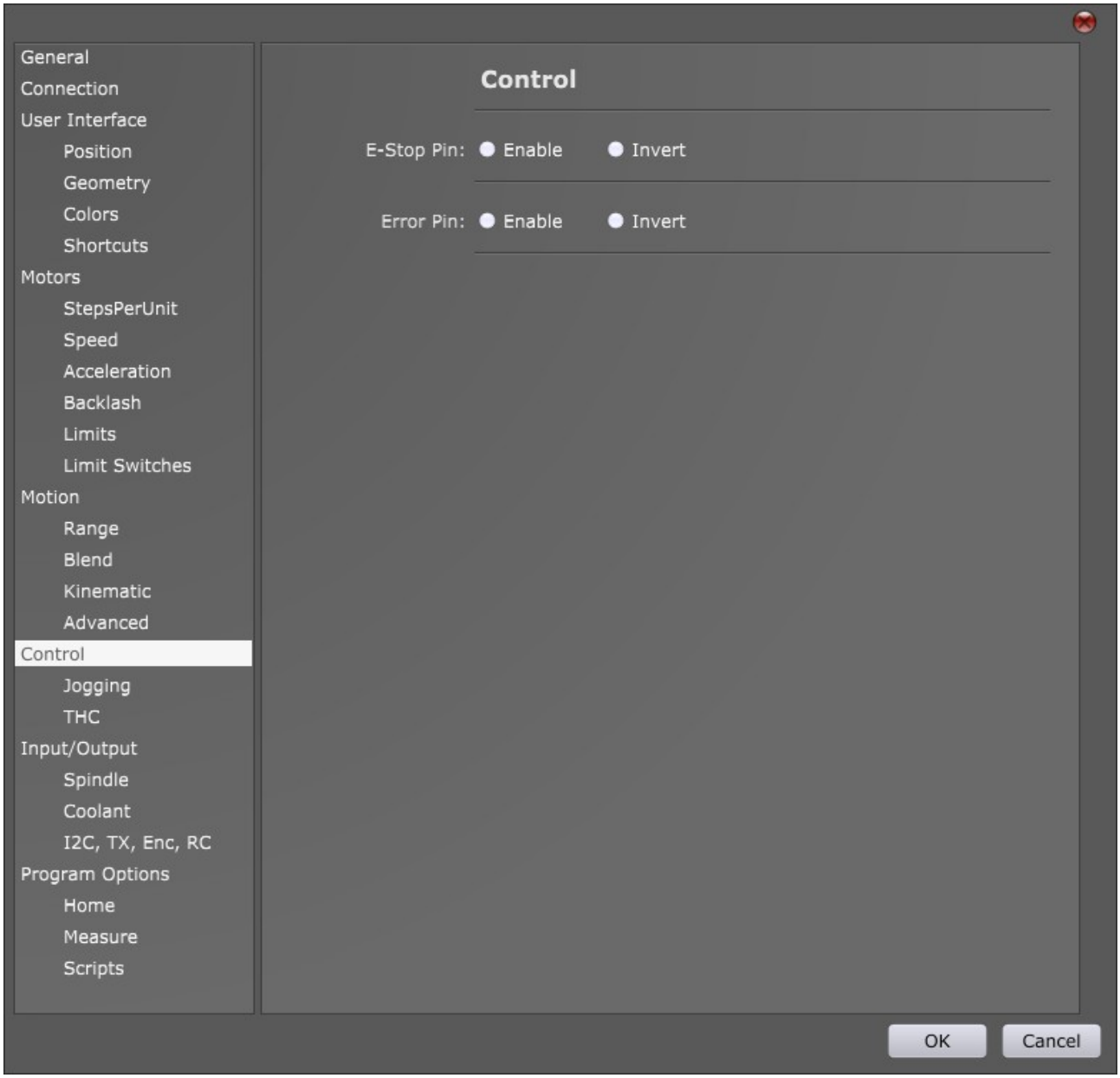
Default value 5 is optimal balance between the response time and fault tolerance.

2.9.5.4.5 Curve tolerance

Motion interpolator curve interpolation parameter.

2.9.6 Control

You can set E-Stop, Error Pin configuration, Jogging keyboard and hand-wheel configuration and THC relates parameters.



2.9.6.1 E-Stop Pin

Enable: Enables E-stop for input pin configured under *Settings/User Interface/Shortcuts/Machine/E-Stop* → Pin

Invert: Inverts E-stop for input pin configured under *Settings/User Interface/Shortcuts/Machine/E-Stop* → Pin.

2.9.6.2 Error Pin

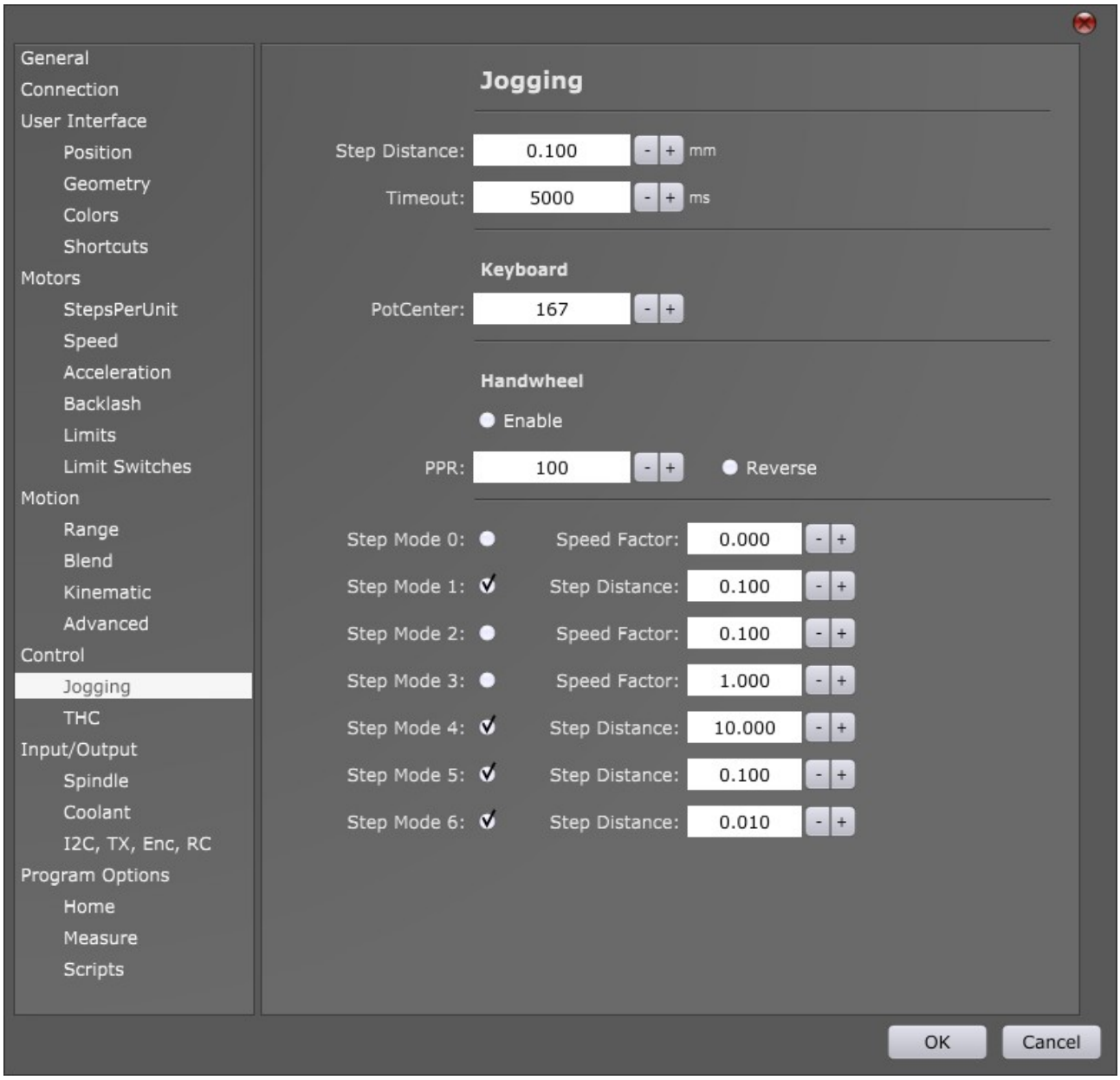
Enable: Enables Error input pin of controller(supported only with Mk3).

Invert: Inverts Error input pin of controller.

For more info on Error pin, please read Mk3 controller user manual.

2.9.6.3 Jogging

With PlanetCNC TNG software you can use Jogging keyboard and/or MPG pendant to jog your machine. Here you can set jogging parameters for either jogging keyboard or MPG pendant.



Step distance:

Distance of step when you jog your machine in “Step” mode. Units are millimeters.

Example:

Step jogging comes very useful when you need short distance increments of motion. Usually when you need to move tool tip towards the material surface to set working position Z=0.

Jog inputs of controller are configured in Settings/User Interface/Shortcuts/Jogging -> Pin.

Timeout:

Keyboard timeout if no keyboard change event is detected. After this time jogging stops.

2.9.6.3.1 Keyboard

PotCenter:

This parameter sets logarithmic/exponential behavior of potentiometer response.

NOTE:To obtain this value, connect jogging keyboard to PlanetCNC controller and rotate jogging potentiometer. You can observe this value with *JogPot* parameter.

2.9.6.3.2 Handwheel

Enable:

Enables use of encoder for purposes of jogging when using MPG pendant or standalone encoder.

For more info regarding controllers encoder inputs please read Mk3 and Mk3/4 controller user manual.

Note: These settings apply to encoder connected to CTRL pins: EN1A and EN1B

PPR:

Enter PPR value (parts per revolution) of your encoder. Usually around 100PPR.

Reverse:

Reverses direction of jogging when encoder is used for jogging.

Step Mode 0-6:

You can configure seven modes of jogging when using MPG pendant with PlanetCNC TNG. To select between modes you use MPG hardware switch(x1, x10, x100).

Step Mode 0:

Step Mode 1:

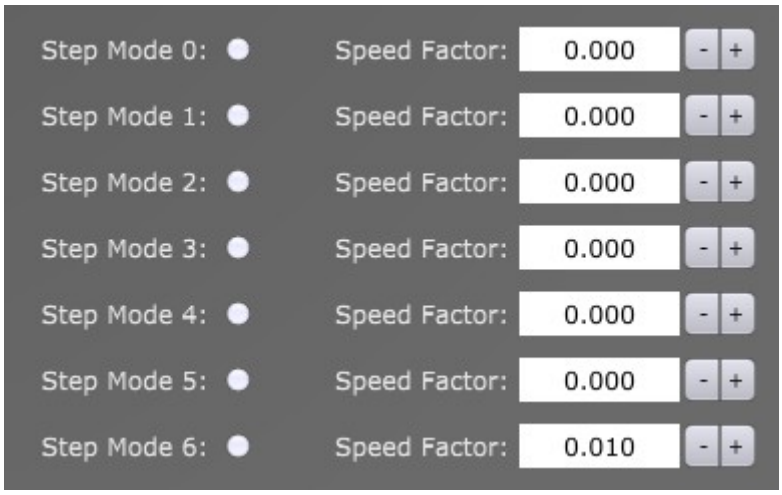
Step Mode 2:

Step Mode 3:

Step Mode 4:

Step Mode 5:

Step Mode 6:



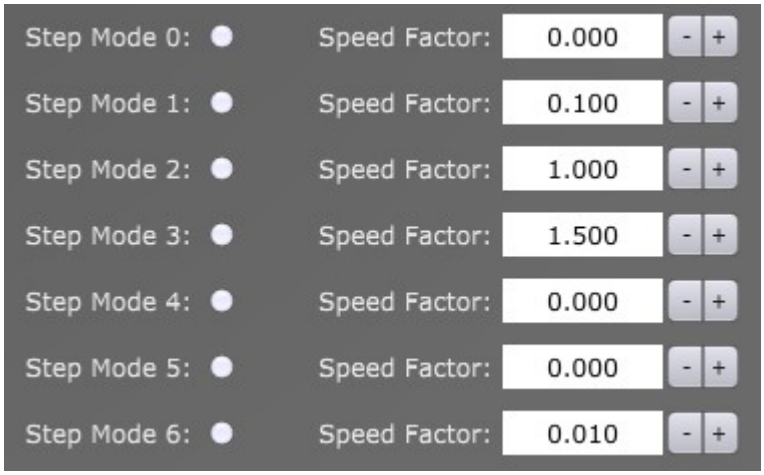
Each step mode can be configured either as *Speed Factor* or *Step Distance*.

If radio button is left unchecked Speed Factor mode will be used, if you enable the round button Step Distance mode will be used.

Speed Factor:

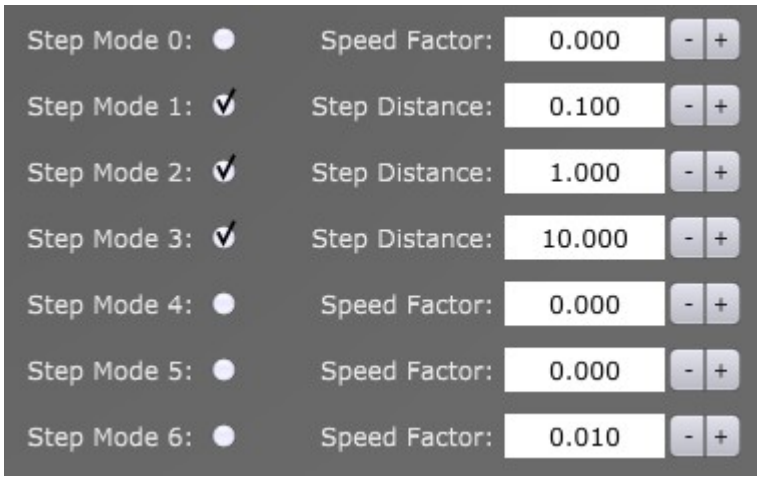
Rotation of encoder wheel will be translated into motion using Speed Factor value.

Value is speed factor.



Step Distance:

Rotation of encoder will jog machine for distance that is set with Step Distance.



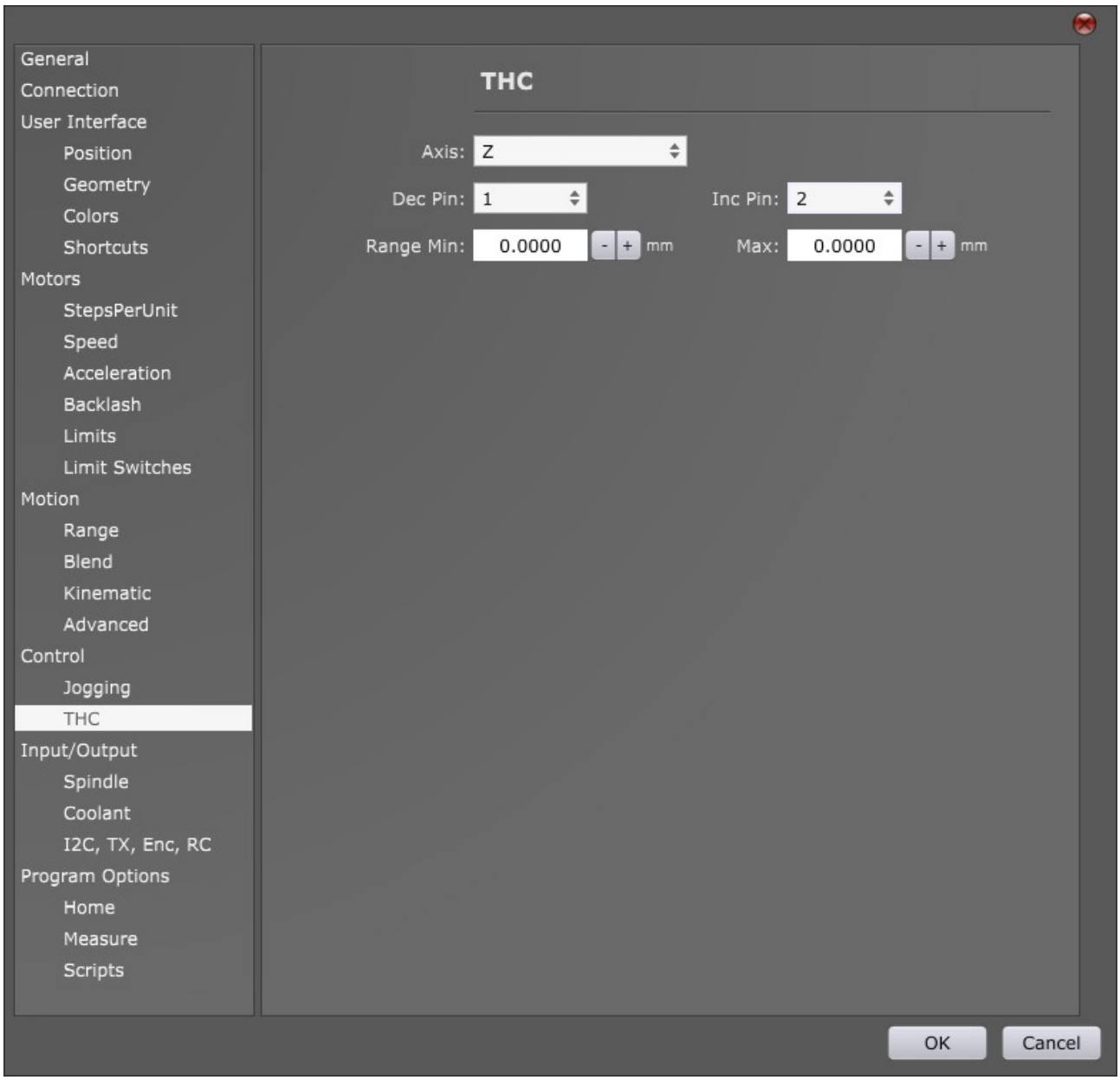
Please note:

With MPG pendant hardware switch you can select only between Step Mode 1, Step Mode 2 and Step Mode 3. So when you are using using MPG pendant, configure Step Modes 1,2 and 3.

2.9.6.4 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 device sends control signals, which controller interprets as control signals for selected axis in order to compensate for curvature of material surface .



Axis:

From drop down menu select machine axis which will be controlled via THC device.

Dec Pin:

From drop down menu select input pin of controller to which THC device will send signal for axis height decrease. Input pins are located at Input connector of controller.

Inc Pin:

From drop down menu select input pin of controller to which THC device will send signal for axis height increase. Input pins are located at Input connector of controller.

Range Min:

Minimal value of controlled axis decrease. This value is in machine coordinates.

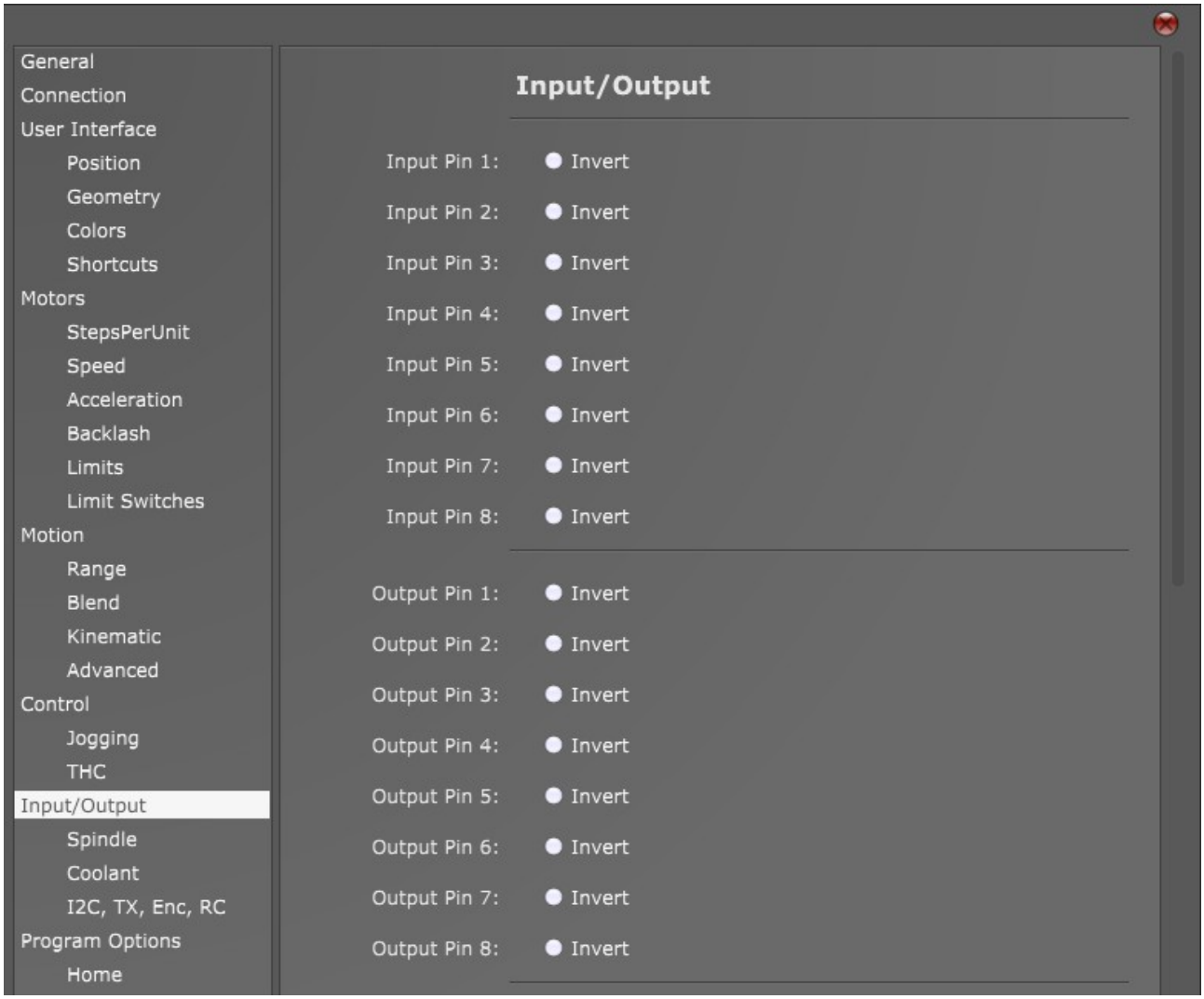
Range Max:

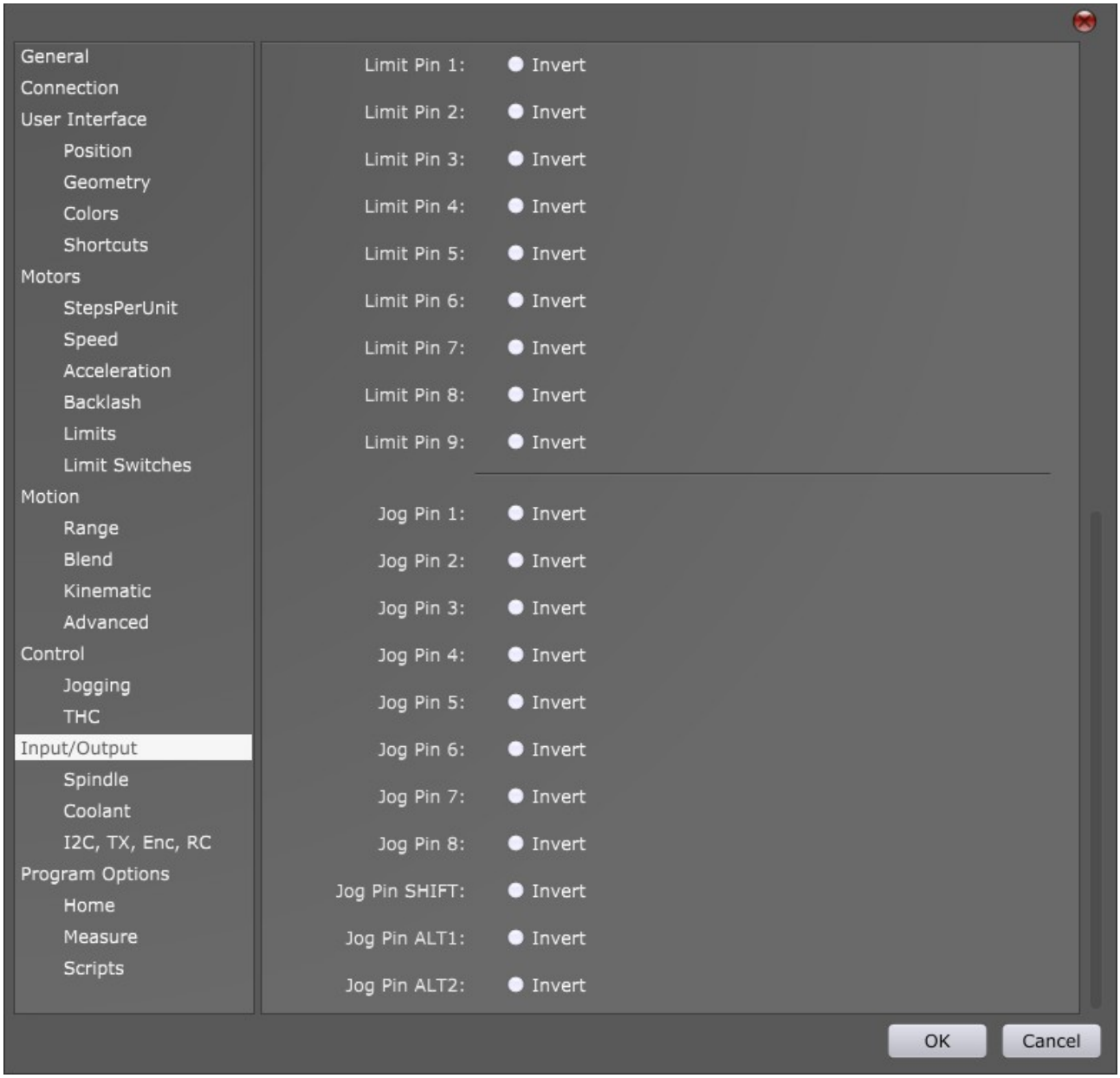
Maximal value of controlled axis decrease. This value is in machine coordinates.

2.9.7 Input/Output

You can invert input and output pins of controller located at *Input*, *Output*, *Limit* and *Jog* header.

Click the radio button to invert desired input or output pin.




General
Connection
User Interface
Position
Geometry
Colors
Shortcuts
Motors
StepsPerUnit
Speed
Acceleration
Backlash
Limits
Limit Switches
Motion
Range
Blend
Kinematic
Advanced
Control
Jogging
THC
Input/Output
Spindle
Coolant
I2C, TX, Enc, RC
Program Options
Home
Measure
Scripts
Limit Pin 1: Invert
Limit Pin 2: Invert
Limit Pin 3: Invert
Limit Pin 4: Invert
Limit Pin 5: Invert
Limit Pin 6: Invert
Limit Pin 7: Invert
Limit Pin 8: Invert
Limit Pin 9: Invert
Jog Pin 1: Invert
Jog Pin 2: Invert
Jog Pin 3: Invert
Jog Pin 4: Invert
Jog Pin 5: Invert
Jog Pin 6: Invert
Jog Pin 7: Invert
Jog Pin 8: Invert
Jog Pin SHIFT: Invert
Jog Pin ALT1: Invert
Jog Pin ALT2: Invert
OK
Cancel

2.9.7.1 Spindle

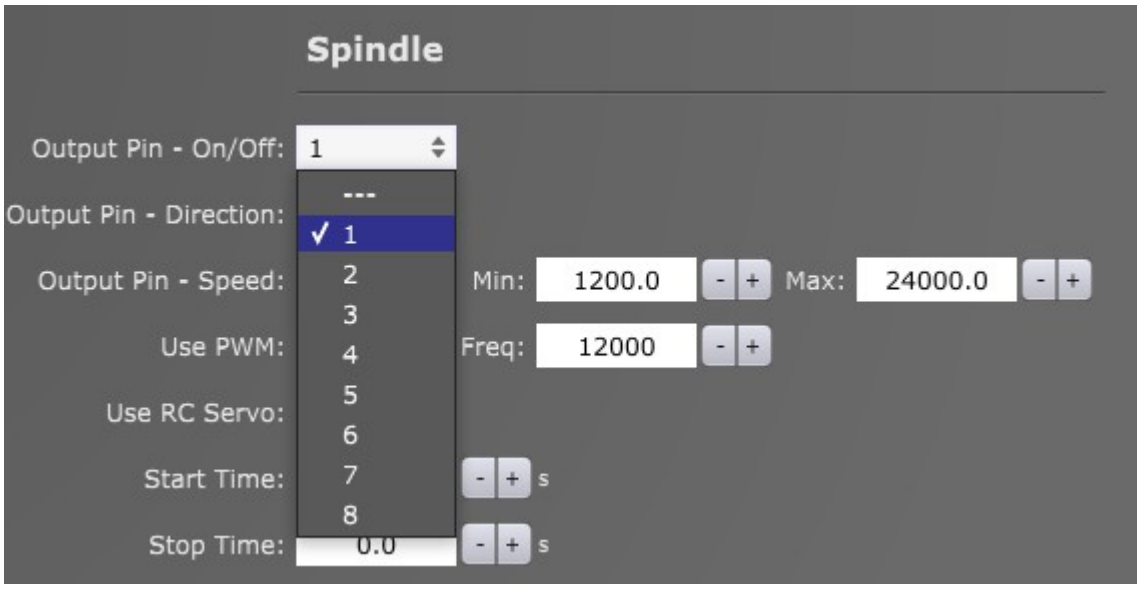
Here you configure output pins for *Spindle* control and input pins for encoder used for spindle synchronization.

NOTE: 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:

2.9.7.1.1 Output Pin – On/Off

Assigns output pin for spindle ON/OFF control (g-code commands M3/M5).

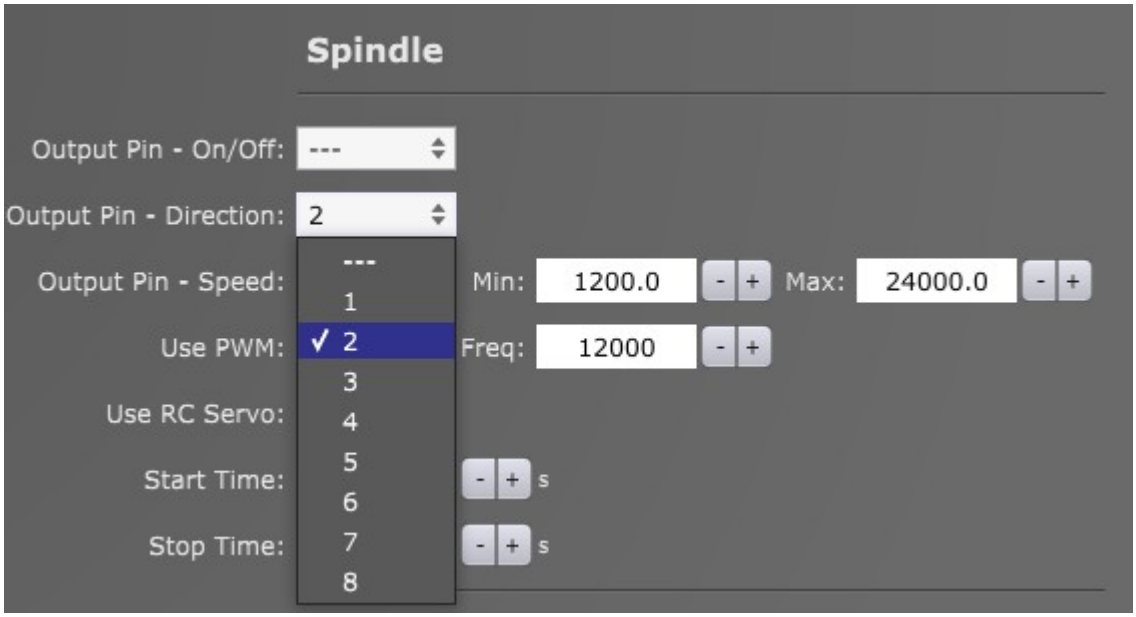
Digital output pin is selected trough drop down menu:



2.9.7.1.2 Output Pin – Direction

Assigns output pin for spindle direction control (g-code command M4).

Digital output pin is selected trough drop down menu:

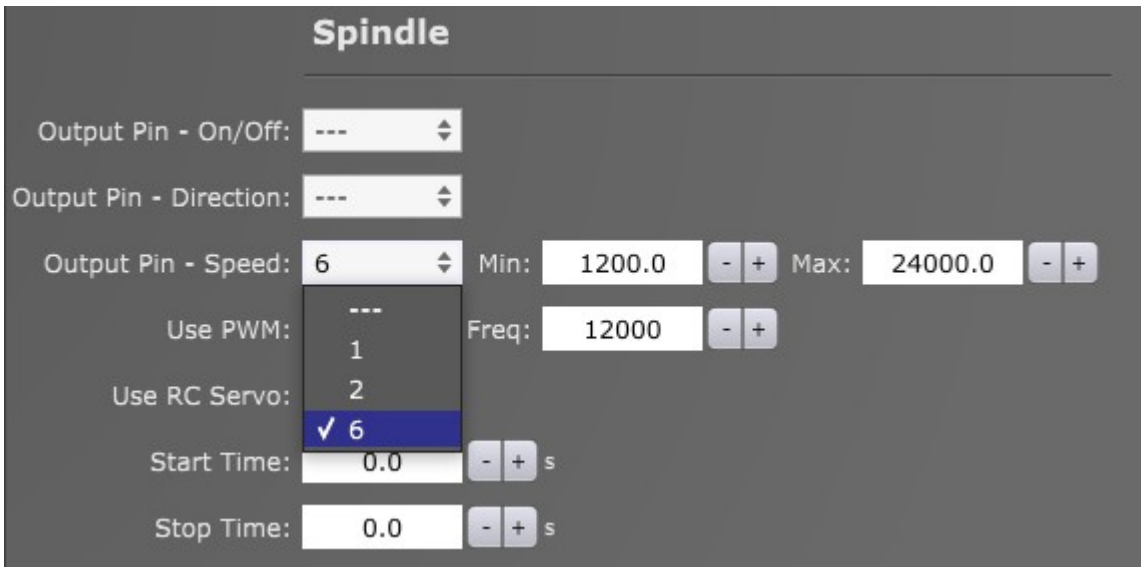


2.9.7.1.3 Output Pin – Speed

Controller output pins 1, 2 and 6 can generate PWM signal(see chapter “Use PWM” below), RC servo signal(see “Use RC Servo” chapter below) or frequency modulated signal.

If neither of the latter two is selected, spindle speed pin will generate frequency modulated signal.

Digital output pin for spindle speed control is selected trough drop down menu:



Min:

Inserted value is min value of spindle RPM. Meaning, when g-code e.g. S1200 will be executed in program, controller speed pin will be generating min value of frequency, PWM modulated signal or RC Servo signal.

Max:

Inserted value is max value of spindle RPM. Meaning, when g-code e.g. S24000 will be executed in program, controller speed pin will be generating max value of frequency, PWM modulated signal or RC Servo signal.

2.9.7.1.4 Use PWM

When this option is enabled, Output – Speed pin will generate PWM signal.

Freq:

Value of PWM base frequency.

2.9.7.1.5 Use RC Servo

When this option is enabled, Output – Speed pin will generate RC Servo signal.

2.9.7.1.6 Start Time

Delay period allowing the spindle to ramp up to its RPM value. This value is in seconds.

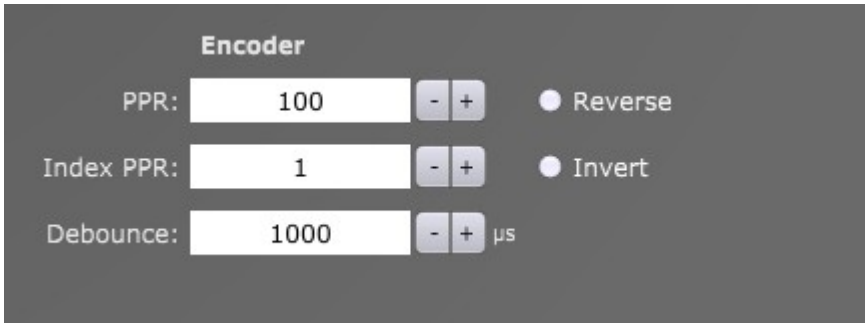
2.9.7.1.7 Stop Time

Delay period allowing the spindle to stop rotating. This value is in seconds.

2.9.7.2 Encoder

Settings group for spindle synchronization.

Note: These settings apply to encoder connected to CTRL pins: EN4A and EN4B



2.9.7.2.1 PPR:

PPR value of your encoder.

Reverse:

2.9.7.2.2 Index PPR

Number of index signals per one revolution of encoder.

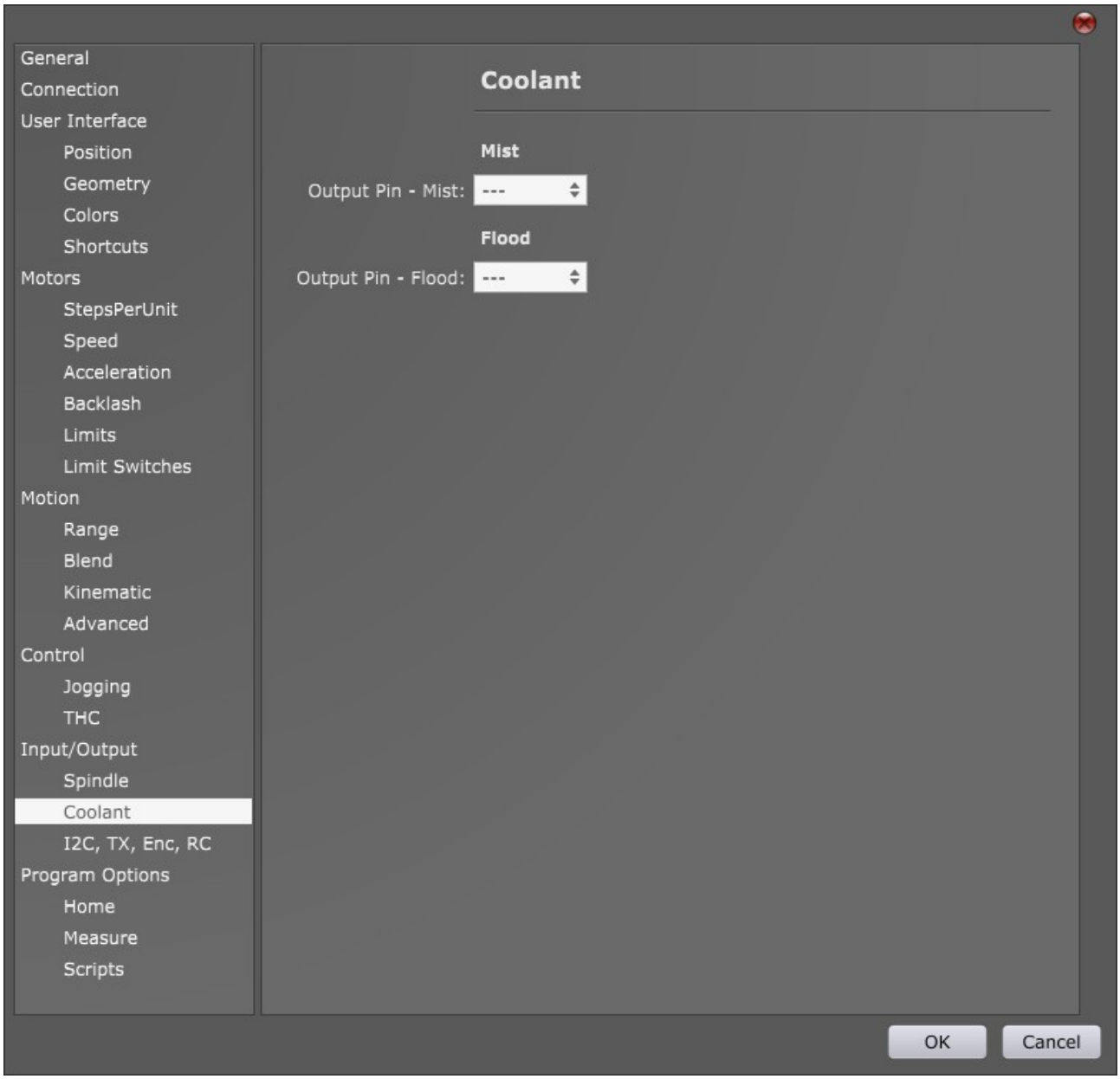
Invert: Inverts Index signal.

2.9.7.2.3 Debounce

Debounce time value for encoder signals.

2.9.7.3 Coolant

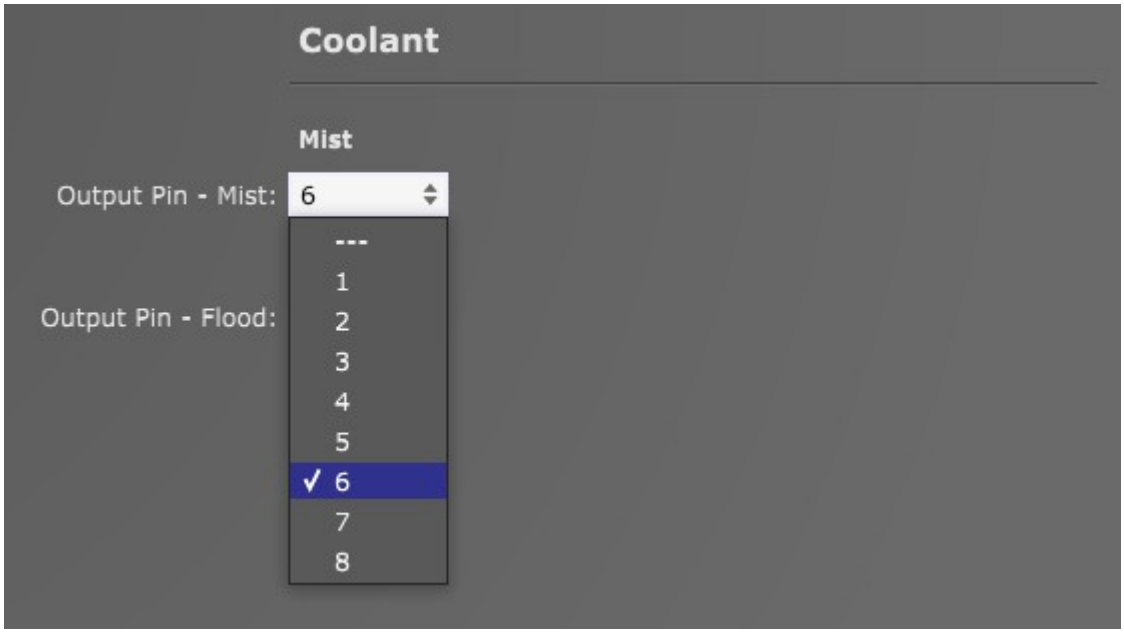
Here you can set digital output pins of controller for coolant control such as Mist and Flood.



2.9.7.3.1 Mist

Assigns output pin for coolant 'Mist' ON/OFF control. (g-code commands M8/M9).

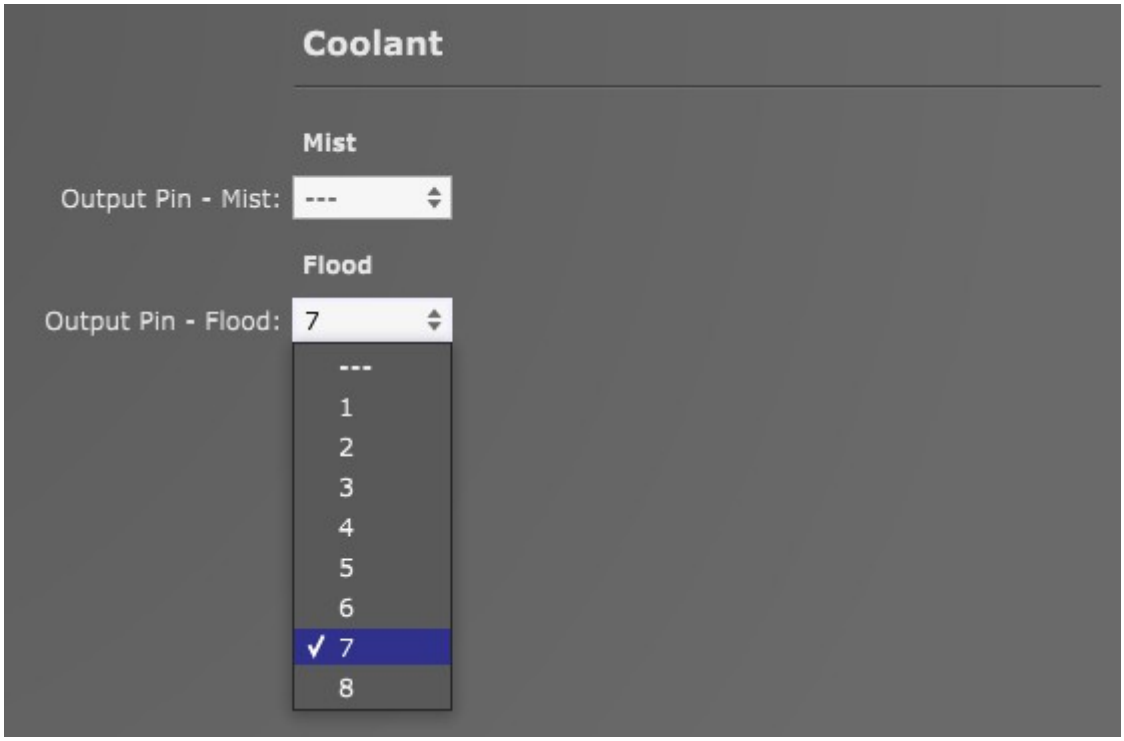
Digital output pin is selected trough drop down menu:



2.9.7.3.2 Flood

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

Digital output pin is selected trough drop down menu:



2.9.7.4 I2C, TX, Enc, RC

You can connect and control external devices with PlanetCNC TNG software and hardware. I2C, serial or SPI communication protocols are supported.

2.9.7.4.1 I2C:

Frequency: Frequency value of I2C communication protocol.

2.9.7.4.2 TX:

Baudrate: Baudrate value of serial communication.

2.9.7.4.3 Encoder:

PPR: PPR value of encoder.

Note: These settings apply to encoder connected to CTRL pins: EN2A and EN2B

2.9.7.4.4 RCServo:

RC Servo Lo: Minimum value of RC servo speed range.

RC Servo Hi: Maximum value of RC servo speed range

2.9.7.4.5 Ext:

Frequency: Frequency value of SPI communication protocol.

SEL1 Pin:

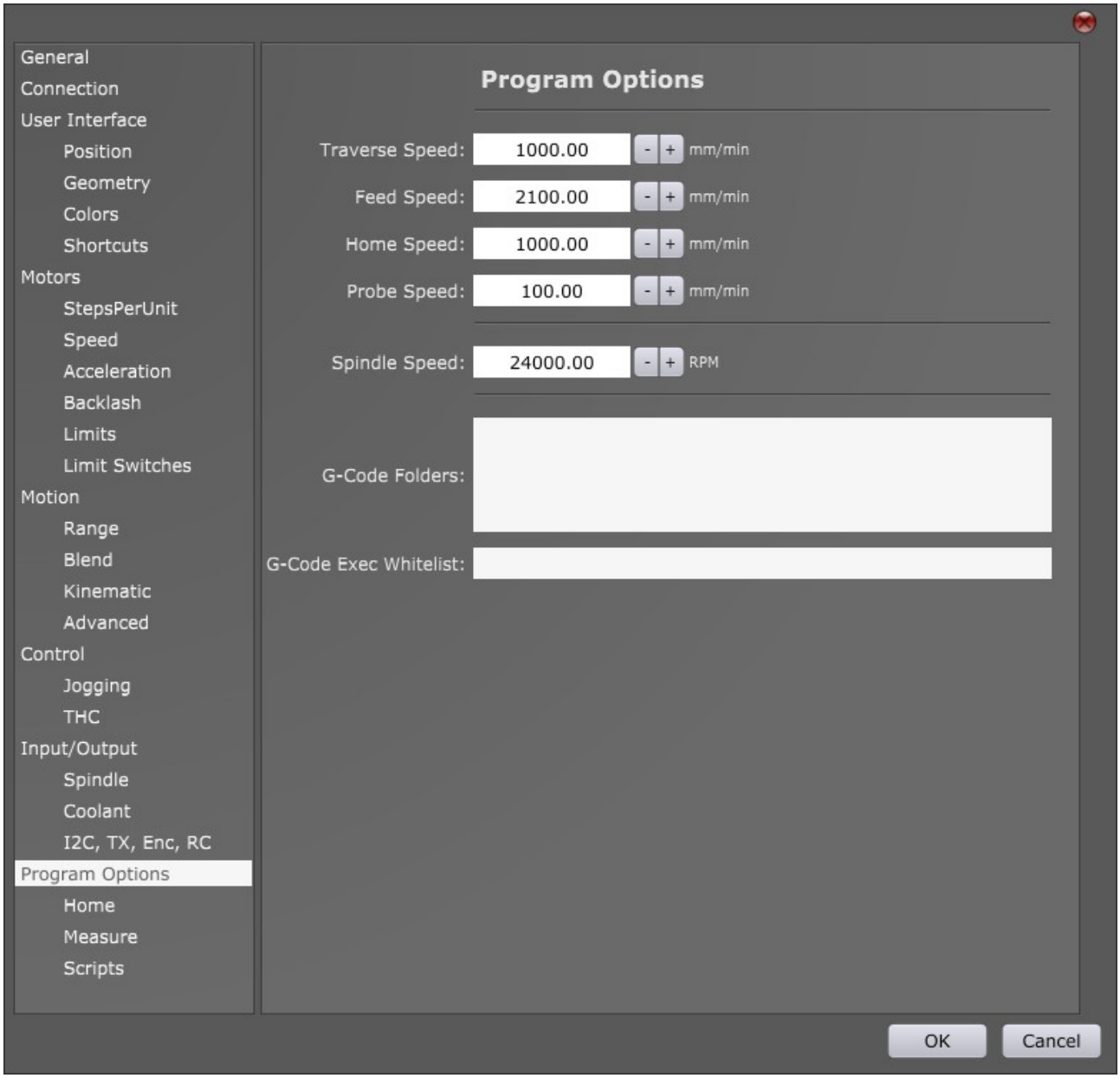
Invert: Inverts SEL1 Pin from 0V to 5V or from 5V to 0V.

SEL2 Pin:

Invert: Inverts SEL2 Pin from 0V to 5V or from 5V to 0V.

2.9.8 Program Options

You can set basic program parameters regarding speed. You can also set g-code folders and set program “Whitelist”.



2.9.8.1 Traverse Speed

G-code programs include traverse(aka rapid) machine moves. During these moves, machine is not cutting the material, instead it is moving above the material usually moving to next cutting position.

These moves are executed with G00 g-code command.

You cannot set speed value of traverse moves with corresponding g-code command (unlike feed speed with F-word).

With this parameter you can set default speed value of traverse machine moves when G00 command is executed in program.

2.9.8.2 Feed Speed

G-code programs include feed machine moves. These moves are executed with G01 g-code command. During these moves machine is cutting the material.

Speed of feed moves is set with F-word g-code command. If g-code program does not include F-word g-code, then value of this parameter will be used in program.

2.9.8.3 Home Speed

With this parameter you can set at which speed machine will reference its axes when homing procedure will be executed.

E.g.: We set Home Speed at 1000mm/min

When machine axes will be referenced, homing of each axis will be performed at 1000mm/min.

2.9.8.4 Probe speed

Probe speed is speed at which machine measures work position or tool offset.

E.g.: We set Probe Speed at 100mm/min

When we measure tool offset using tool sensor, Z axis will descend towards sensor at speed 100mm/min.

2.9.8.5 Spindle Speed

Spindle speed is value of RPM(Rotations Per Minute) of Spindle. In g-code program, this is set by S-word g-code command. If your g-code program does not include S-word command, then value of this parameter will be taken into the account.

2.9.8.6 G-Code folders

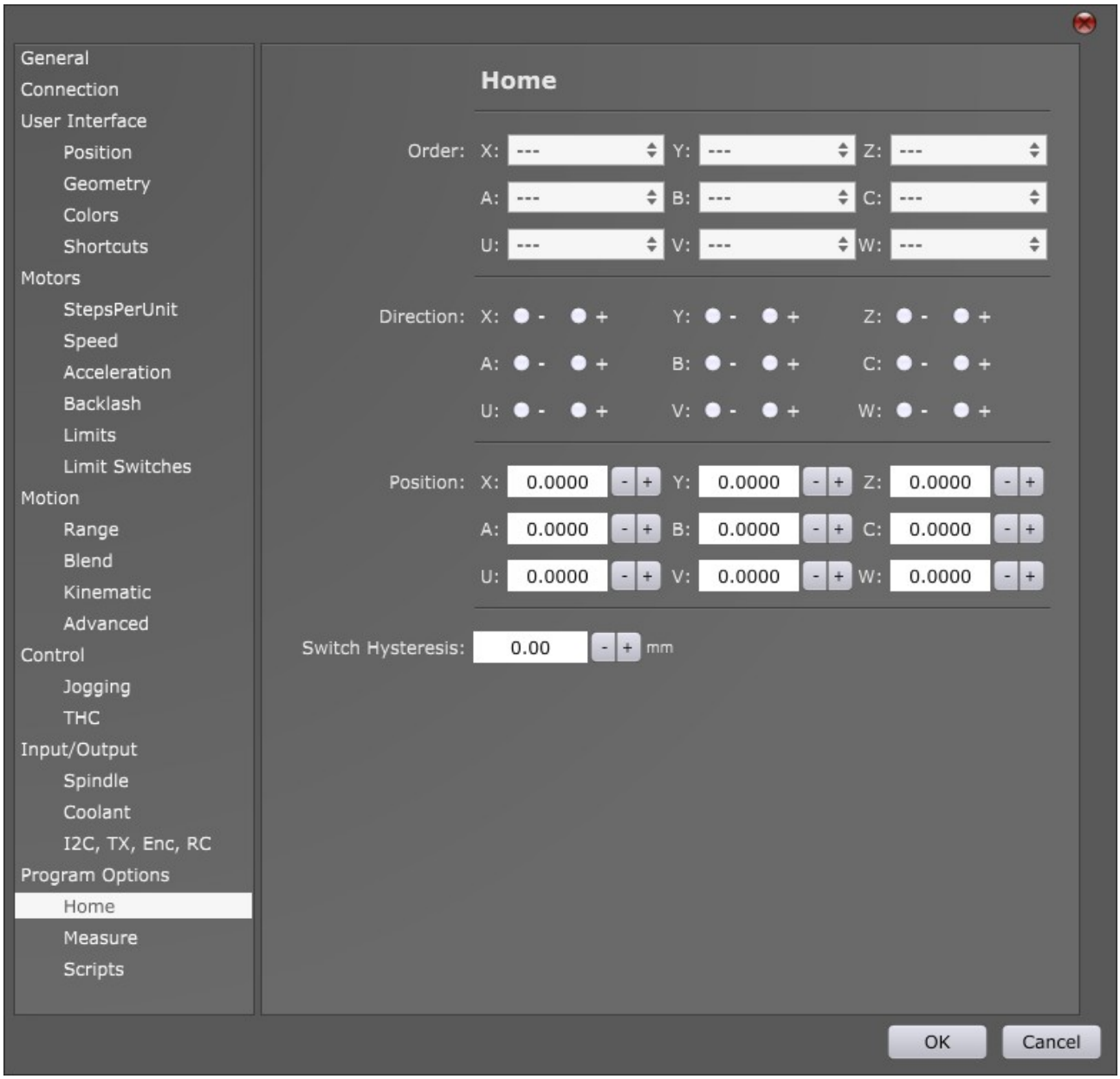
Insert paths of folders, which include your subprograms.

2.9.8.7 G-code Exec Whitelist

PlanetCNC TNG software supports calling and execution of programs within the g-code program. With *G-code Exec Whitelist* you state which programs are allowed to be called and executed.

2.9.8.8 Home

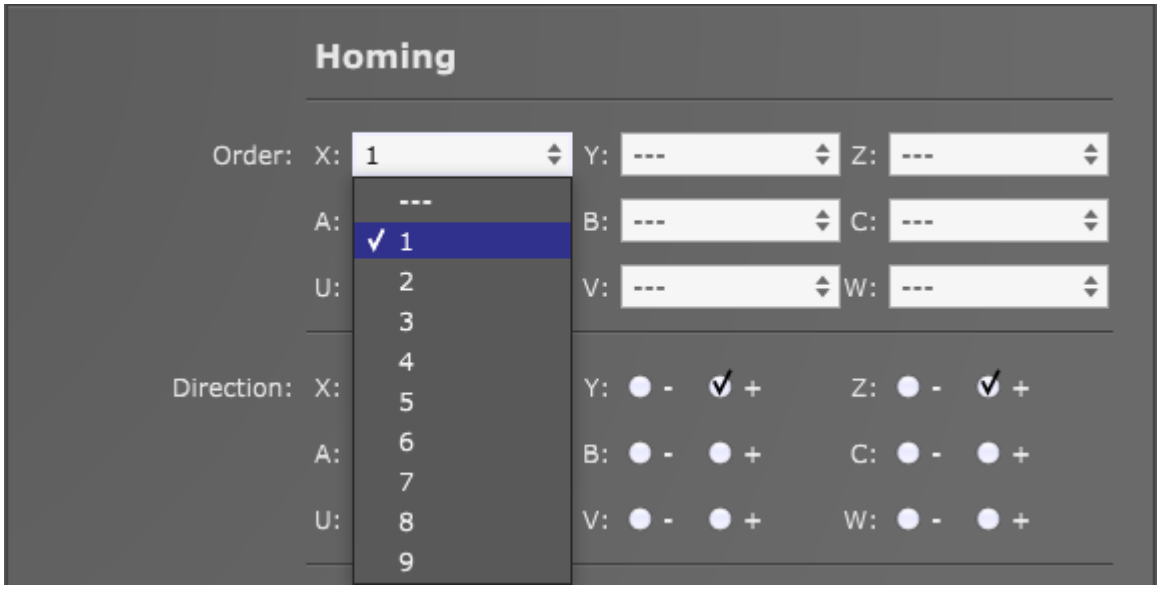
Under this tab user can configure parameters for Homing procedure.



2.9.8.8.1 Order

User can set homing order of machine axes. Select homing order number for each axis from drop down menu.

When number is 1 selected for desired axis, then that axis will be homed first. When number 2 is selected for desired axis, then that axis will be homed second and so on.



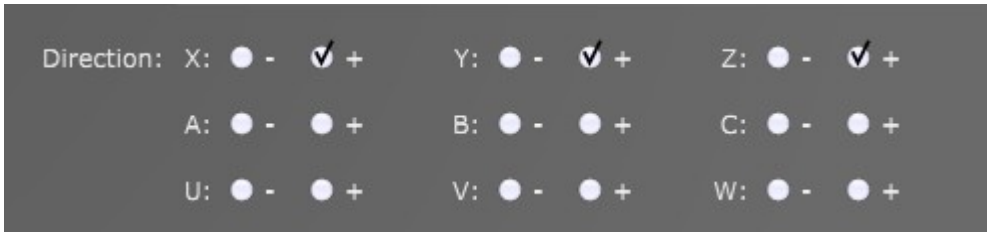
2.9.8.8.2 Direction

User can set homing direction of desired axis.

Round button “-” references machine axis in negative direction.

Round button “+” references machine axis in positive direction.

E.g.: Homing direction configuration at picture below will move all machine axes in positive direction to perform homing.

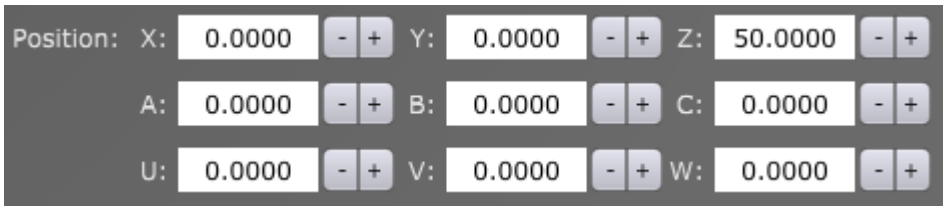


2.9.8.8.3 Position

User can set home position of axes after homing procedure is completed. These are absolute position coordinates.

E.g.: Position values are X=0 ; Y=0 and Z=50

After homing procedure of each axis is completed , X and Y axes will be positioned at 0 and Z axis at height 50mm.



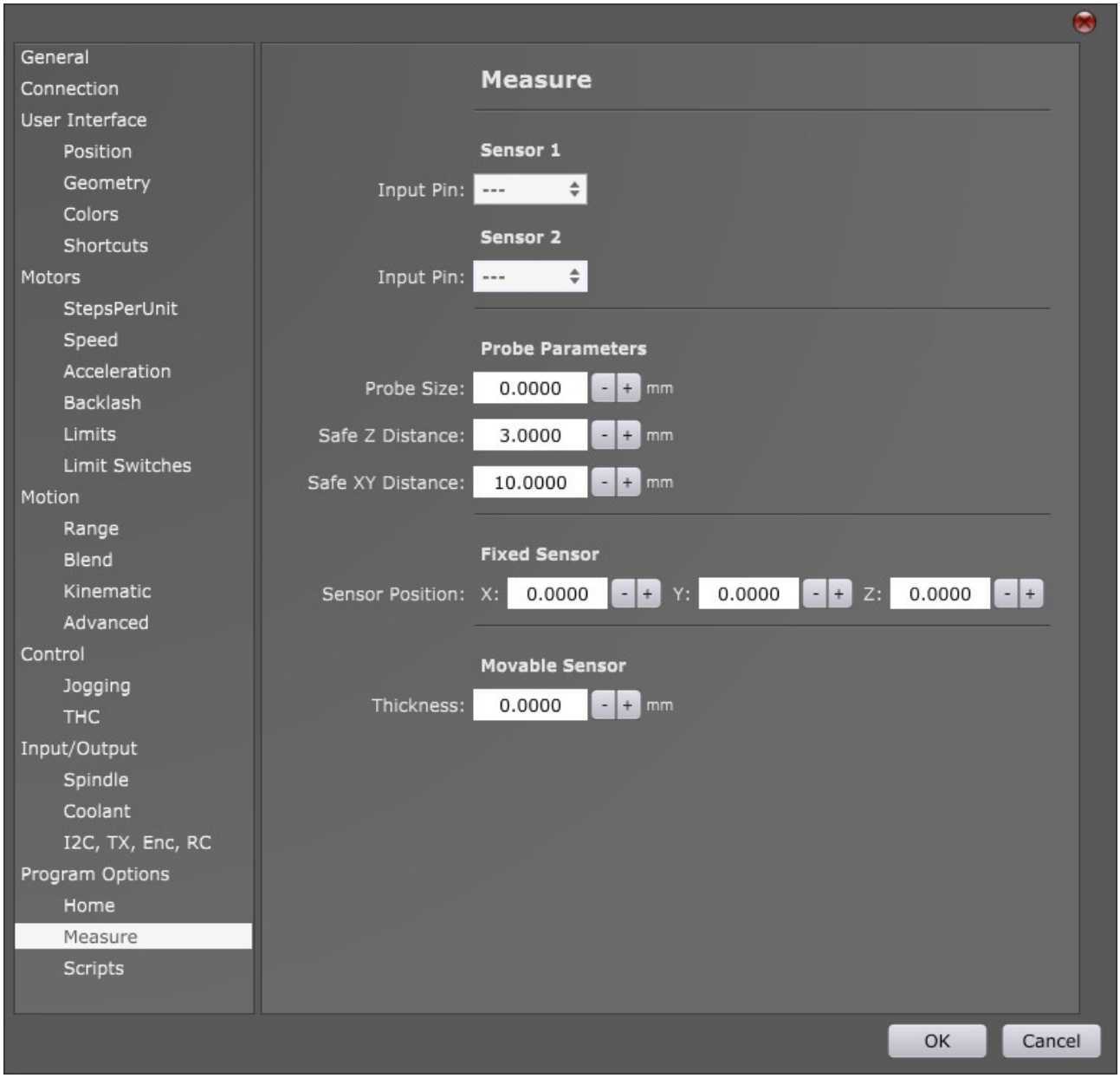
2.9.8.8.4 Switch Hysteresis

Switch hysteresis is distance between the moment when limit switch is activated and when limit switch is released. Value is usually around 2-5mm.

User tests switch hysteresis value on his machine prior setting it here.

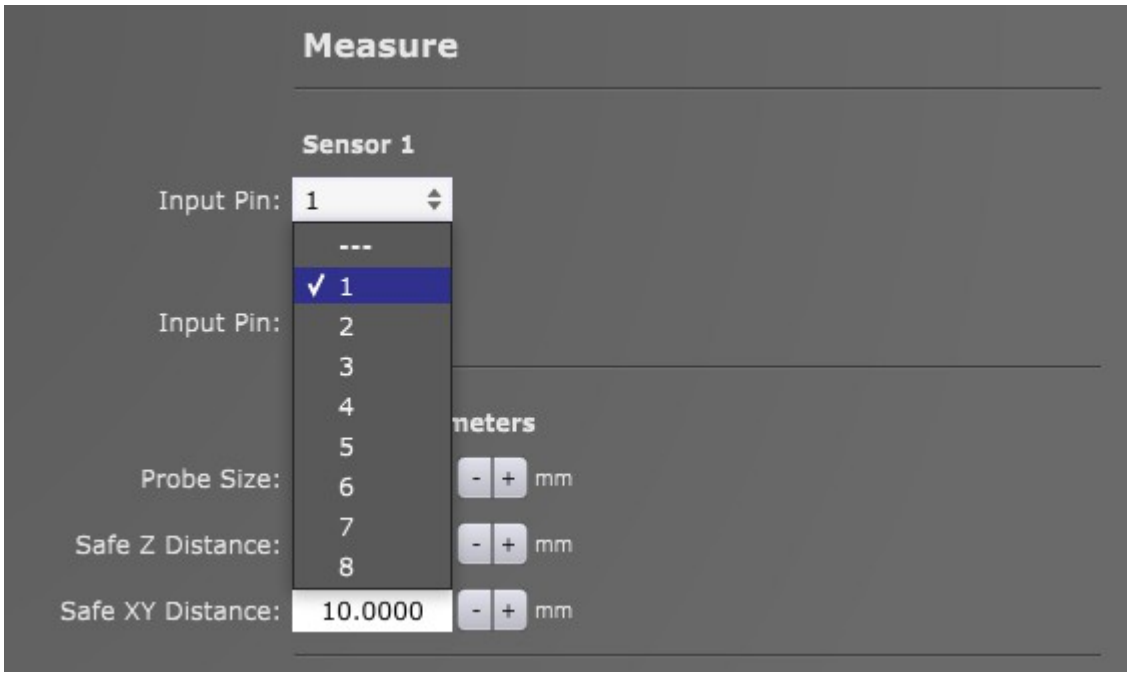
2.9.8.9 Measure

Here you can set sensor inputs, probe, fixed and movable sensor parameters.



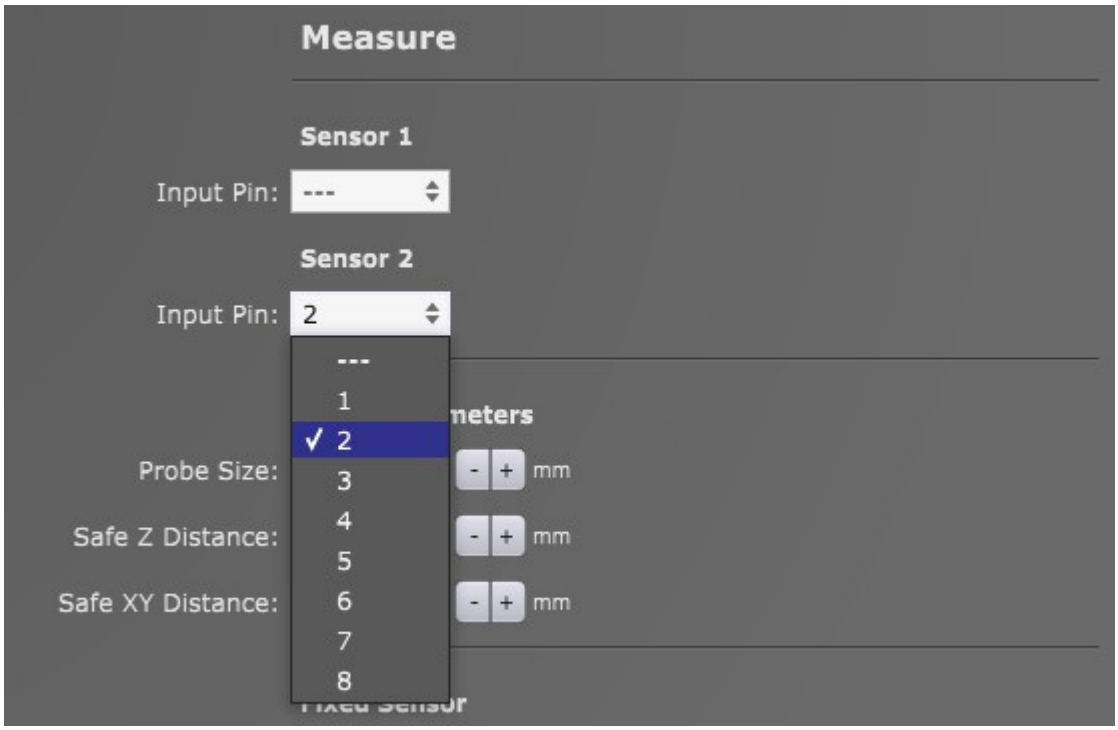
2.9.8.9.1 Sensor 1

Input pin: Assigns digital input pin as Sensor 1. Input pin is selected from drop down menu:

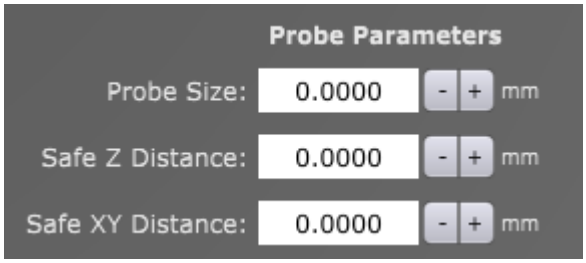


2.9.8.9.2 Sensor 2

Input pin: Assigns digital input pin as Sensor 1. Input pin is selected from drop down menu:



2.9.8.9.3 Probe Parameters



Probe Size:

Radius of the stylus ball of touching probe.

Safe Z Distance:

Distance for which machine Z axis ascends after each Z axis probe measuring.

Safe XY Distance:

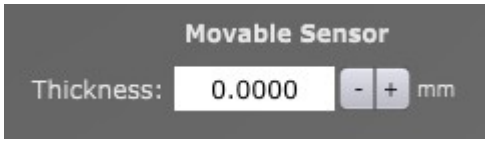
Distance for which machine XY axis retract after each XY axis probe measuring.

2.9.8.9.4 Fixed Sensor

Sensor Position:

Machine X,Y and Z coordinates of the fixed sensor position.

2.9.8.9.5 Movable Sensor



Thickness:

Thickness of movable sensor. If the tool sensor is a simple piece of copper board of thickness 1.6mm, then this value = 1.6mm

2.9.8.10 Scripts

PlanetCNC TNG software supports use of scripts. Scripts are essentially g-code programs that run in the background of PlanetCNC TNG procedures(such as homing, tool change etc...).

You select procedure script file from drop down menu. To open script file in text editor click button Edit Script.

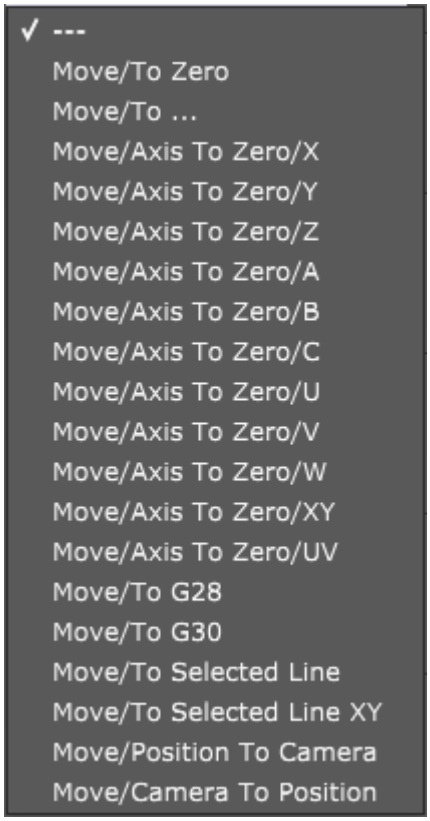
You can edit scripts for:

2.9.8.10.1 Tools:

Tools/Change

Tools/Select

2.9.8.10.2 Move:



Move/To Zero

Move/To...

Move/Axis to Zero/X

Move/Axis to Zero/Y

Move/Axis to Zero/Z

Move/Axis to Zero/A

Move/Axis to Zero/B

Move/Axis to Zero/C

Move/Axis to Zero/U

Move/Axis to Zero/V

Move/Axis to Zero/W

Move/Axis to Zero/XY

Move/Axis to Zero/UV

Move/To G28

Move/To G30

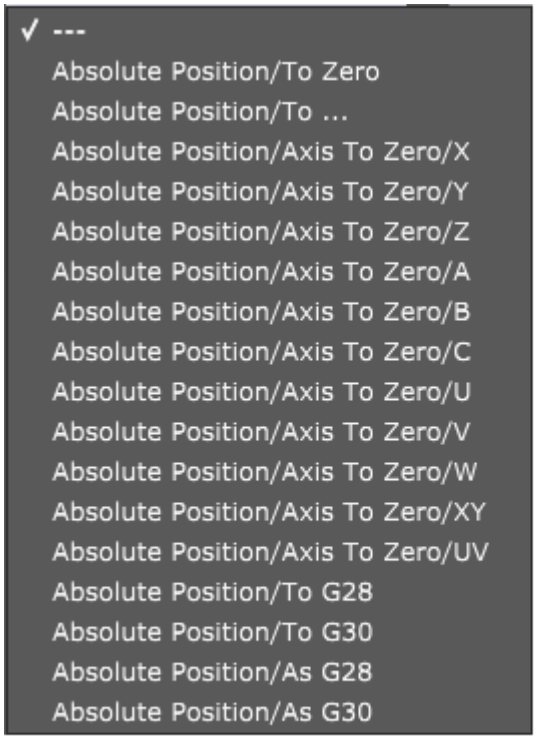
Move/To Selected Line

Move/To Selected Line XY

Move/Position to Camera

Move/Camera To Position

2.9.8.10.3 Absolute Position



Absolute Position/To Zero

Absolute Position/To...

Absolute Position/Axis To Zero/X

Absolute Position/Axis To Zero/Y

Absolute Position/Axis To Zero/Z

Absolute Position/Axis To Zero/A

Absolute Position/Axis To Zero/B

Absolute Position/Axis To Zero/C

Absolute Position/Axis To Zero/U

Absolute Position/Axis To Zero/V

Absolute Position/Axis To Zero/W

Absolute Position/Axis To Zero/W

Absolute Position/Axis To Zero/XY

Absolute Position/Axis To Zero/UV

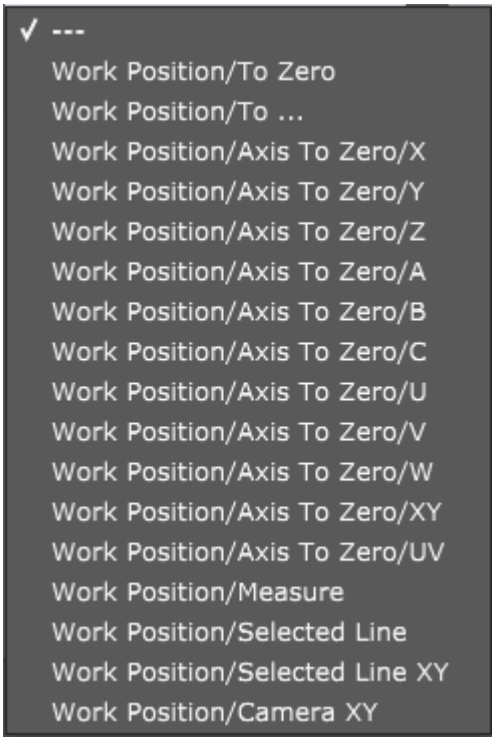
Absolute Position/To G28

Absolute Position/To G30

Absolute Position/As G28

Absolute Position/As G30

2.9.8.10.4 Work Position



Work Position/To Zero

Work Position/To...

Work Position/Axis to Zero/X

Work Position/Axis to Zero/Y

Work Position/Axis to Zero/Z

Work Position/Axis to Zero/A

Work Position/Axis to Zero/B

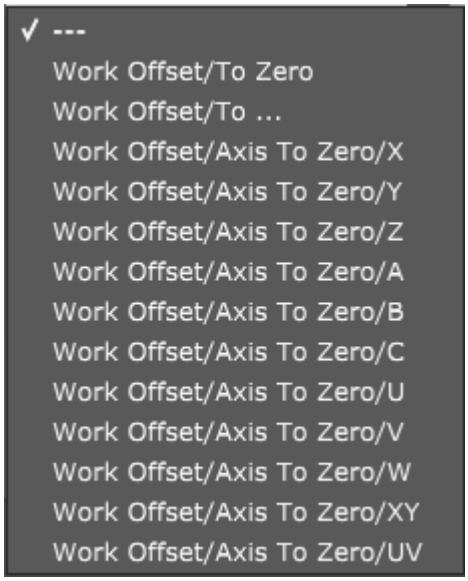
Work Position/Axis to Zero/C

Work Position/Axis to Zero/U

Work Position/Axis to Zero/V

Work Position/Axis to Zero/W
Work Position/Axis to Zero/XY
Work Position/Axis to Zero/UV
Work Position/Measure
Work Position/Selected Line
Work Position/Selected Line XY
Work Position/Camera XY

2.9.8.10.5 Work Offset



Work Offset/To Zero
Work Offset/To
Work Offset/Axis To Zero/X
Work Offset/Axis To Zero/Y
Work Offset/Axis To Zero/Z
Work Offset/Axis To Zero/A
Work Offset/Axis To Zero/B
Work Offset/Axis To Zero/C
Work Offset/Axis To Zero/U
Work Offset/Axis To Zero/V
Work Offset/Axis To Zero/W
Work Offset/Axis To Zero/XY
Work Offset/Axis To Zero/UW

2.9.8.10.6 Tool Offset

Tool Offset/To Zero

Tool Offset/To...

Tool Offset/Axis To Zero/X

Tool Offset/Axis To Zero/Y

Tool Offset/Axis To Zero/Z

Tool Offset/Axis To Zero/A

Tool Offset/Axis To Zero/B

Tool Offset/Axis To Zero/C

Tool Offset/Axis To Zero/U

Tool Offset/Axis To Zero/V

Tool Offset/Axis To Zero/W

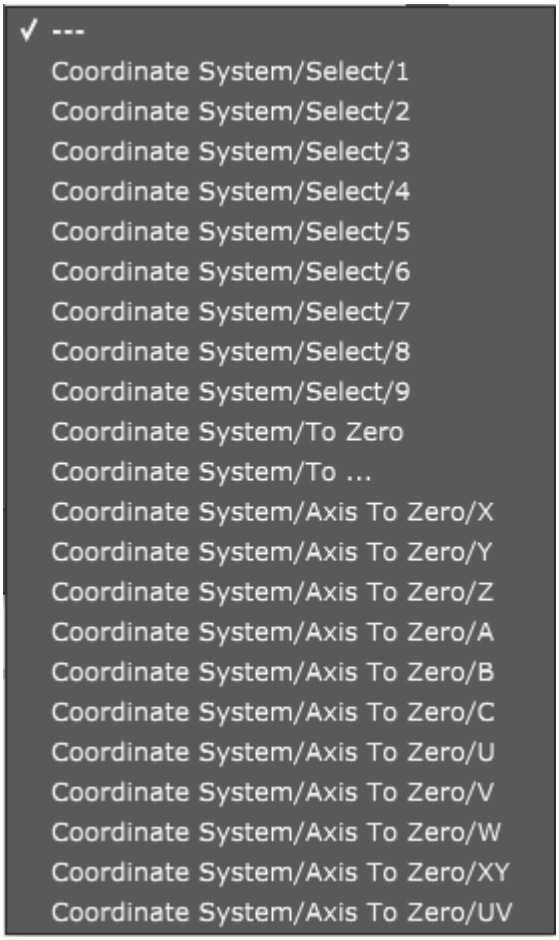
Tool Offset/Axis To Zero/XY

Tool Offset/Axis To Zero/UV

Tool Offset/From Tool Table

Tool Offset/Measure

2.9.8.10.7 Coordinate System



Coordinate System/Select/1

Coordinate System/Select/2

Coordinate System/Select/3

Coordinate System/Select/4

Coordinate System/Select/5

Coordinate System/Select/6

Coordinate System/Select/7

Coordinate System/Select/8

Coordinate System/Select/9

Coordinate System/To Zero

Coordinate System/To...

Coordinate System/Axis To Zero/X

Coordinate System/Axis To Zero/Y

Coordinate System/Axis To Zero/Z

Coordinate System/Axis To Zero/A

Coordinate System/Axis To Zero/B

Coordinate System/Axis To Zero/C

Coordinate System/Axis To Zero/U

Coordinate System/Axis To Zero/V

Coordinate System/Axis To Zero/W

Coordinate System/Axis To Zero/W

Coordinate System/Axis To Zero/XY

Coordinate System/Axis To Zero/UV

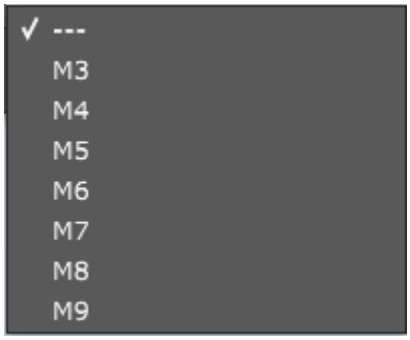
2.9.8.10.8 Home

Home

2.9.8.10.9 Measure

Measure/Measure Surface

2.9.8.10.10 M-Code



M3

M4

M5

M6

M7

M8

M9

Table of Contents