

CommandCNC 1.1

Release Notes

What's new in 1.1

Auto-Set support for DTHC.....	2
Fixture Offsets.....	7
User Interface Rework.....	11
Output buttons can now be customized.....	12
Support for two more axes.....	14
New THC delay “Auto” setting.....	15
Miscellaneous other fixes and improvements.....	15

Please read this document carefully before contacting CandCNC! There are some new features in this release, and some controls on the screen have been moved in order to make room for new features and to simplify the interface.

Auto-Set support for DTHC

Auto-Set is a new feature in the latest DTHC firmware which allows the DTHC to sample the voltage at the beginning of a cut and make small adjustments to the target cutting volts which the torch height control will try to maintain. This helps to account for things such as consumable wear, which can cause small changes in the cutting volts at the proper torch height over time.

In order for Auto-Set to function there are several things you must have in place:

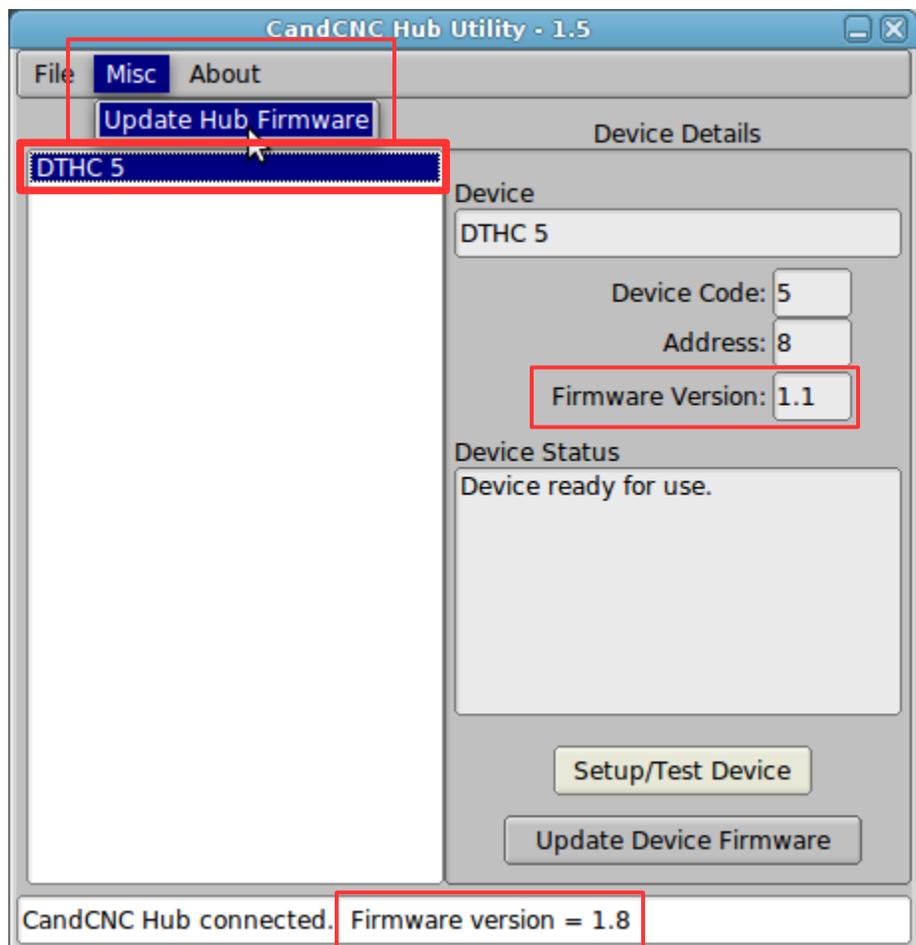
- CommandCNC 1.1 or later
- Hub firmware 1.8 or later (**usb485hub_1.8.ucccx**) *
- DTHC firmware with support for Auto-Set:
 - DTHC 5: version 1.1 or later (**dthc5_1.1.rcccx**) *
 - DTHC-IV: version 2.1 or later (**dthcIV_main_2.1.rcccx**) *
- Correct combination of Sensor PWM card and MIC cable (see next page)

* The CommandCNC 1.1 update will place the required firmware files in your Downloads/firmware folder

To check your firmware versions, make sure the control box is powered on and open the Hub Utility.

The hub firmware version will be shown at the bottom of the window, and the DTHC firmware version is shown in the right side about the middle of the window when the DTHC device is selected in the list on the left.

If your firmware versions are older than the requirements shown above then you can update them using the “Misc → Update Hub Firmware” for the HUB, and the “Update Device Firmware” button for the DTHC.

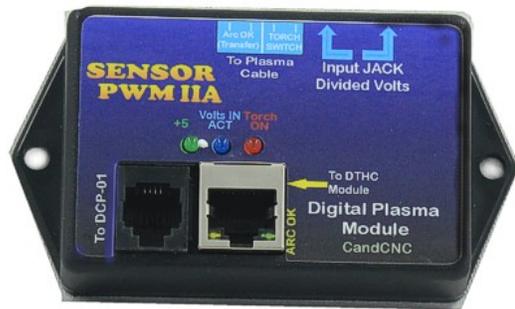
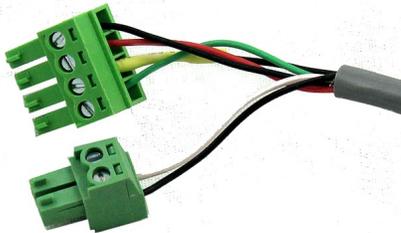


For Auto-Set to be able to sample volts correctly, it is important that you have the correct combination of Sensor PWM card and MIC cable. The following table lists the valid options:

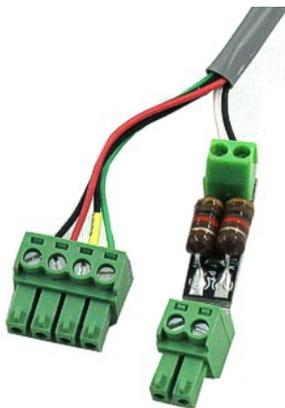
MIC-01 with 3.81mm connector and Sensor PWM IIA



MIC-03 with Sensor PWM IIA



MIC-03A (with filter card) with Sensor PWM II

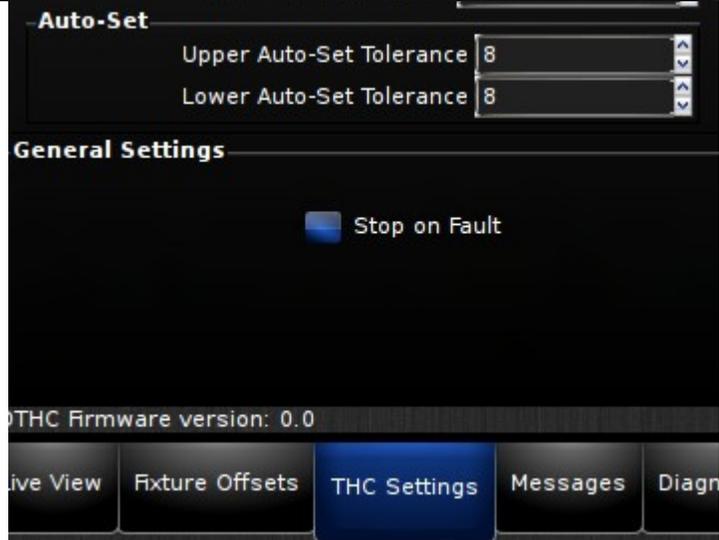


There are a couple of places now with information relevant to Auto-Set in CommandCNC. The first is in the THC box at the bottom right of the CommandCNC window.

There is an Auto-Set section with a check-box to enable Auto-Set, Sample Volts, and Last Sample Status.



There are also a couple of settings in the THC Settings → DTHC Settings page:



The way Auto-Set works in a nutshell is this: if DTHC is enabled and Auto-Set is enabled, on the first cut after a touch-off the cutting voltage is sampled for a short period, and if the voltage falls within the specified tolerance of the Preset Volts, then that sampled voltage will be shown in “Sample Volts” and will be used instead of the Preset Volts.

After a valid sample the “Last Sample Status” → Good LED will light, but if the sampled voltage falls outside the specified tolerance, the sample will be rejected and the “Last Sample Status” → Bad LED will light.

The Upper and Lower Auto-Set Tolerance settings are to set the number of volts above and below the Preset Volts which will be considered a “good” sample. So using the example in the images above, a valid sample is anything up to 8 volts above or below the preset of 120 volts – or any voltage from 112 to 128 volts.

Since the Sample Volts are actually used instead of the Preset Volts, and they must be within the given tolerance of Preset Volts, you need a way to reset the Sample Volts if, for example, you start cutting a different gauge of metal. This is mostly automatic. If a new Preset Volts is sent to the THC (different than the one it is already set to), it will reset Sample Volts to the new Preset Volts. So simply loading and running gcode for the new gauge of metal will send the new preset and reset the Sample Volts. Another way to reset Sample Volts to Preset is to simply un-check the “Enable Auto-Set” box, and then check it again.

SheetCAM post

There is also support for Auto-Set in the SheetCAM post processor rev26 (CandCNCPlasmaLLCNC-rev26.scpst) and later. Auto-Set can be enabled or disabled for each operation with a check-box in the operation’s settings window.

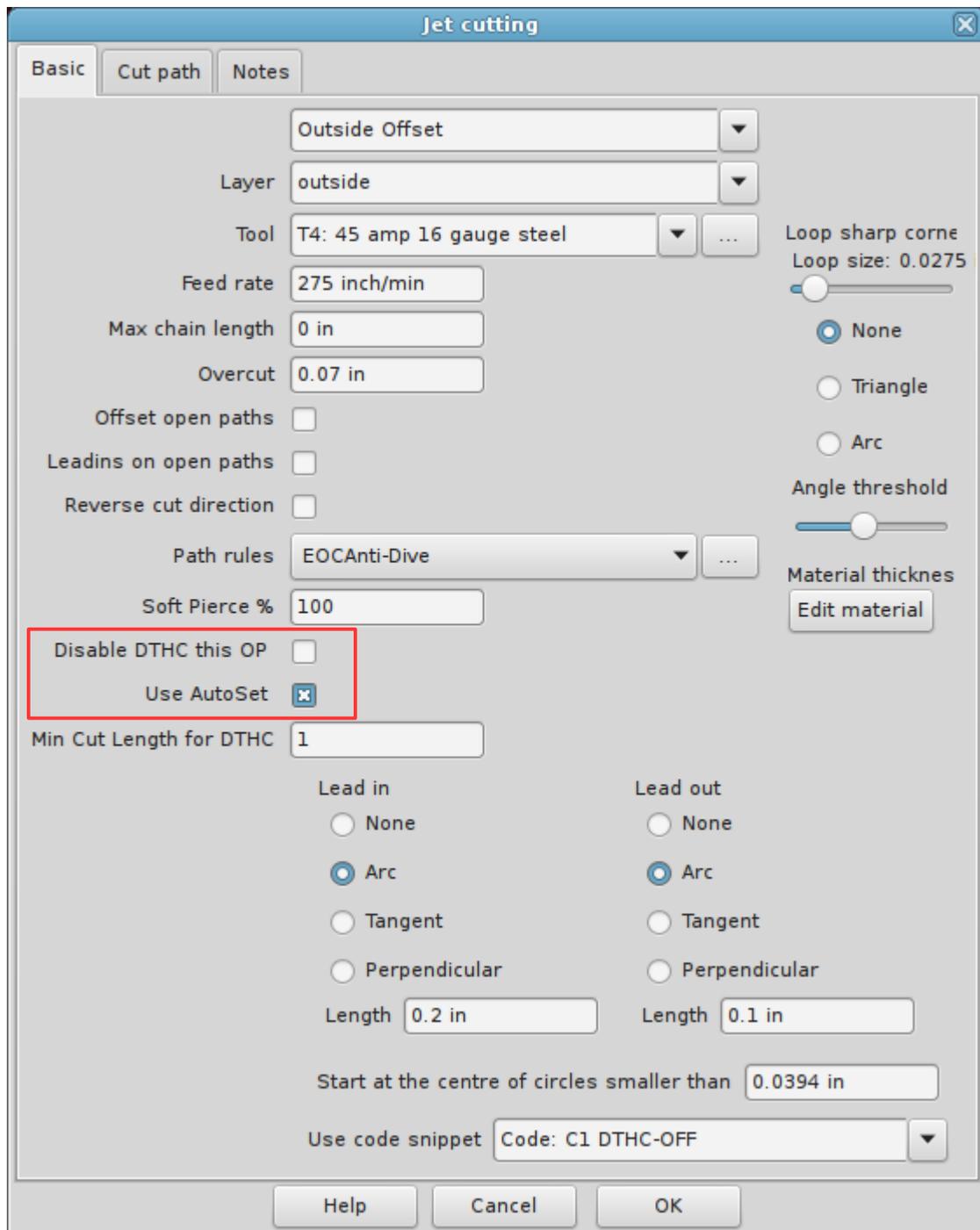
This post is included in the CommandCNC 1.1 Update and will be found in the SheetCAM-CandCNC/Posts directory after installing the update.

Auto-Set is enabled by default.

For most operations you can leave Auto-Set enabled, but material that is not flat (such as corrugated) needs Auto-Set disabled.

Some things to be aware of:

- The Auto-Set enable in SheetCAM will override the setting in CommandCNC each time the gcode is run. This means that you can override the Auto-Set during a cut, **but each new operation or re-run of the code will apply the setting from the code.**
- Since Auto-Set only samples voltage if DTHC is enabled, Auto-Set will also be disabled when DTHC is disabled for any reason – such as the “Disable DTHC this OP” setting in the operation (see image below).
- Since Auto-Set only samples voltage if DTHC is enabled, Auto-Set will also not sample on cuts shorter than the “Min Cut Length for DTHC” setting in the operation (see image below).



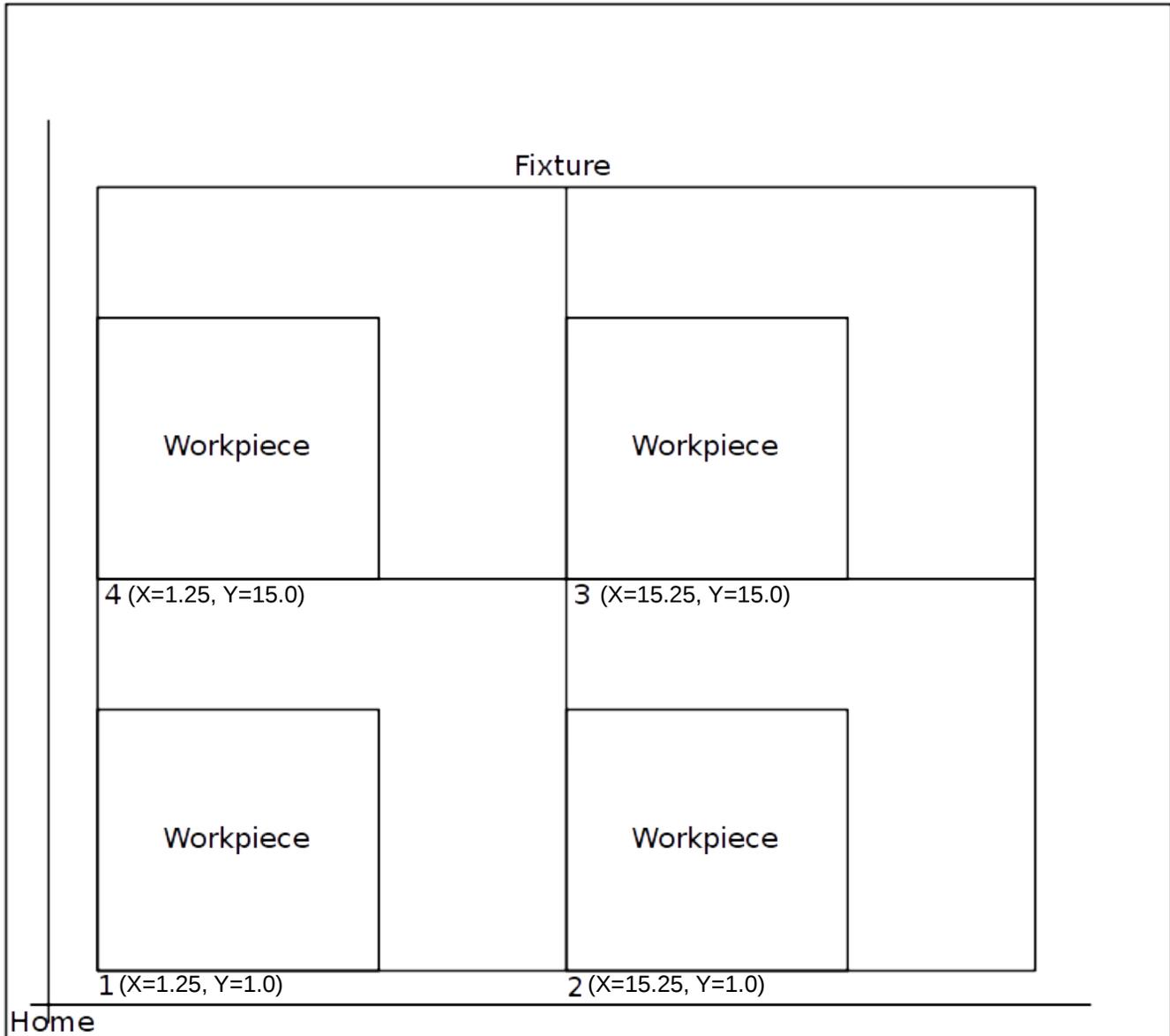
If you are still unsure about the use of Auto-Set you can simply disable it in SheetCAM and on screen in CommandCNC and the DTHC will operate exactly as it did before.

Auto-Set is also not a solution for a DTHC that is not working correctly!

Fixture Offsets

There is now an easy way to enter offsets for up to 9 fixture positions in CommandCNC. A fixture is a fixed location that will hold material to cut – usually with some means of holding the material in place. To hopefully help in understanding fixture offsets, an example fixture setup is illustrated and described below in more detail.

Table



In the illustration above; the large outer square represents the table of a machine, and the point where two lines cross in the lower left corner represents the “Home” location of the X and Y axes. This is the location the machine considers X=0, Y=0 or (0, 0) after homing (referencing) to the home switches.

We will call this “Machine Zero”

The inner square is a fixture plate, having four locations where material to be machined or cut (workpieces) can be placed. The lower left corner of each fixture position in this example are the reference points labeled 1, 2, 3, and 4. Each of these are at the positions shown in relation to machine zero. So the lower left corner of fixture 1 is (X=1.25", Y=1.0"), fixture 2 is (X=15.25", Y=1.0"), and so on. These numbers (1,2,...) do not have to be in that order – that is just how I numbered the positions in this example. The important part is the actual position or offset of the fixture from machine zero.

Fixture Offsets in CommandCNC are an easy way to set and use the fixture positions without having to enter them in your part program. Your part program can then be written so it is in reference to (X=0, Y=0), but it can be cut or machined in any of the fixtures by changing to that fixture offset before cutting it. Fixture offsets can be selected in g-code with the code G54 through G59.3 – giving you up to 9 fixture positions.

The default fixture offset is G54. This is what is selected when CommandCNC starts, and is selected each time a program ends. This means that if you select fixture offset G55 and run a program, when the program finished CommandCNC will go back to G54. If you want to run another program in the G55 position, you will have to select the G55 offset again!

Offset	X	Y	Z	Offset Name
G54	1.2500	1.0000	0.0000	Fixture 1
G55	15.2500	1.0000	0.0000	Fixture 2
G56	15.2500	15.0000	0.0000	Fixture 3
G57	1.2500	15.0000	0.0000	Fixture 4
G58	0.0000	0.0000	0.0000	G58
G59	0.0000	0.0000	0.0000	G59
G59.1	0.0000	0.0000	0.0000	G59.1
G59.2	0.0000	0.0000	0.0000	G59.2
G59.3	0.0000	0.0000	0.0000	G59.3

Change to Selected Offset
Edit

Live View
Fixture Offsets
Messages
Diagnostics



There is now a page in CommandCNC called Fixture Offsets which allows you to edit and select offsets. To edit the values in offsets, click the Edit button, then select the value you wish to change, and after typing in the new value press the Enter or Tab key to accept it. Click the Edit button again when you are finished to exit edit mode.

In the image above is shown the Fixture Offsets with the offsets entered for our example in the G54, G55, G56, and G57 Offsets. Notice that a name can be assigned to each offset as desired.

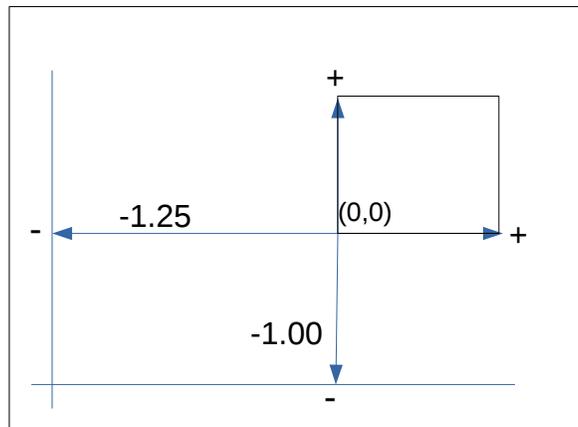
The offset in red lettering is the offset currently in use. If you select an offset and click the button labeled "Change to Selected Offset", it will change the system to use that offset.

In the DRO box there is also a label which shows the Fixture Offset that is currently in use. This label shows the name that was entered for the offset and in parentheses it shows the Gxx code for that offset.



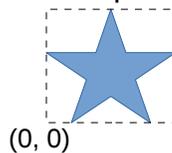
In the images above the tool is actually in the machine home position, which is machine zero (0, 0). When Fixture 1 is selected, the position in the DROs now shows (-1.25, -1.0), and when Fixture 2 is selected the position in the DROs changes to (-15.25, -1.0) – even though the machine has not moved.

This is because the new working zero (0, 0) position of the machine becomes the corner of the fixture – which is set by the offsets entered for that fixture in the table as discussed above.



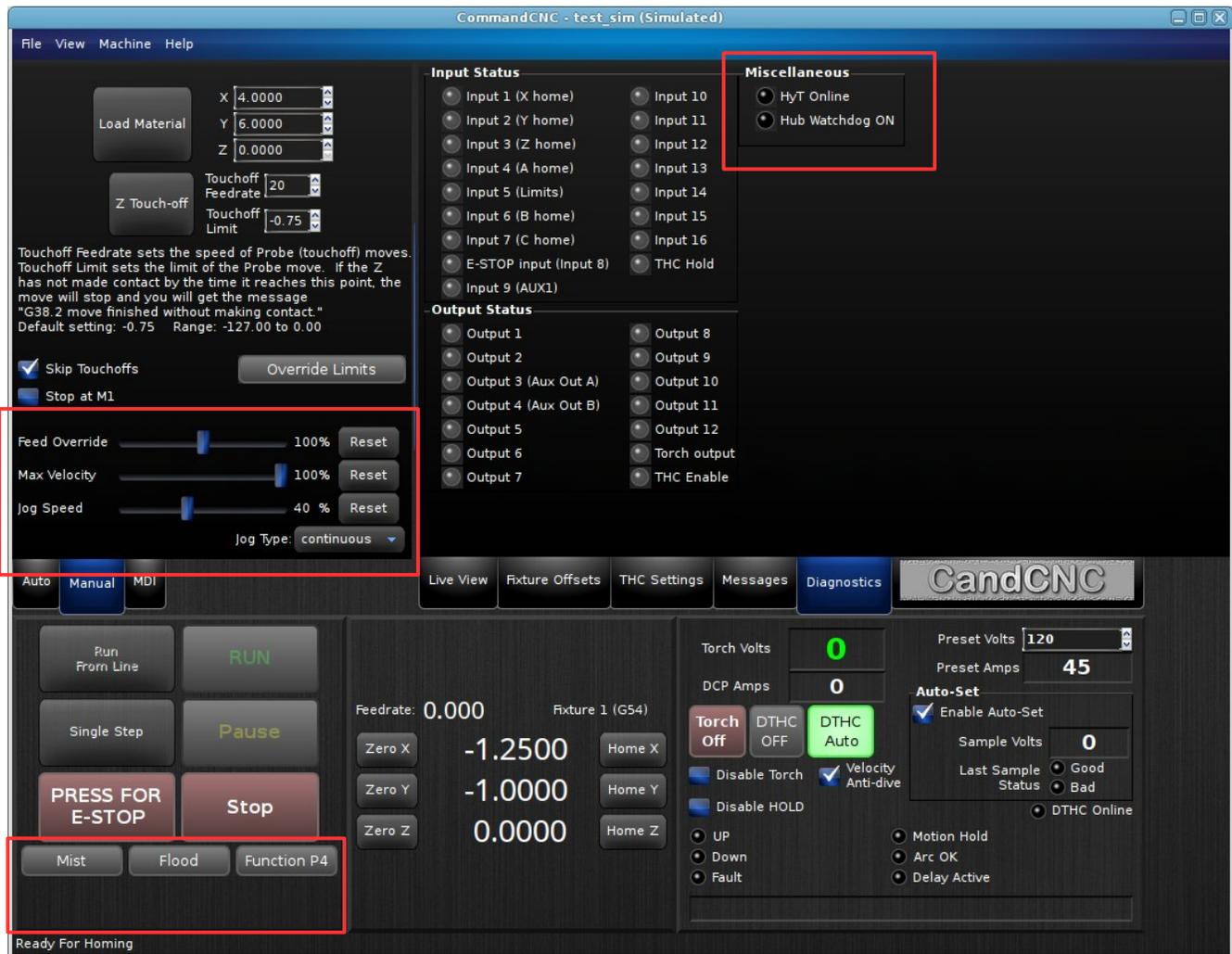
The illustration above shows that if the working zero position is at the lower left corner of the small square, then anything to the left of or below that position is a negative number. So if Fixture 1 is selected from our example and the tool is at the machine zero position, the DROs will read (-1.25, -1.0) because that is where the tool is in relation to the new working zero which is the corner of the fixture.

What all this means is that you can make a part such as this star with the lower left part of



the bounding box at $X=0, Y=0$. Then you can put material in all four fixtures, then select Fixture 1 and cut the part, then select Fixture 2 and it will cut the part in that fixture, then Fixture 3 and so on – without having to change the part itself or its location in the CAM software.

User Interface Rework

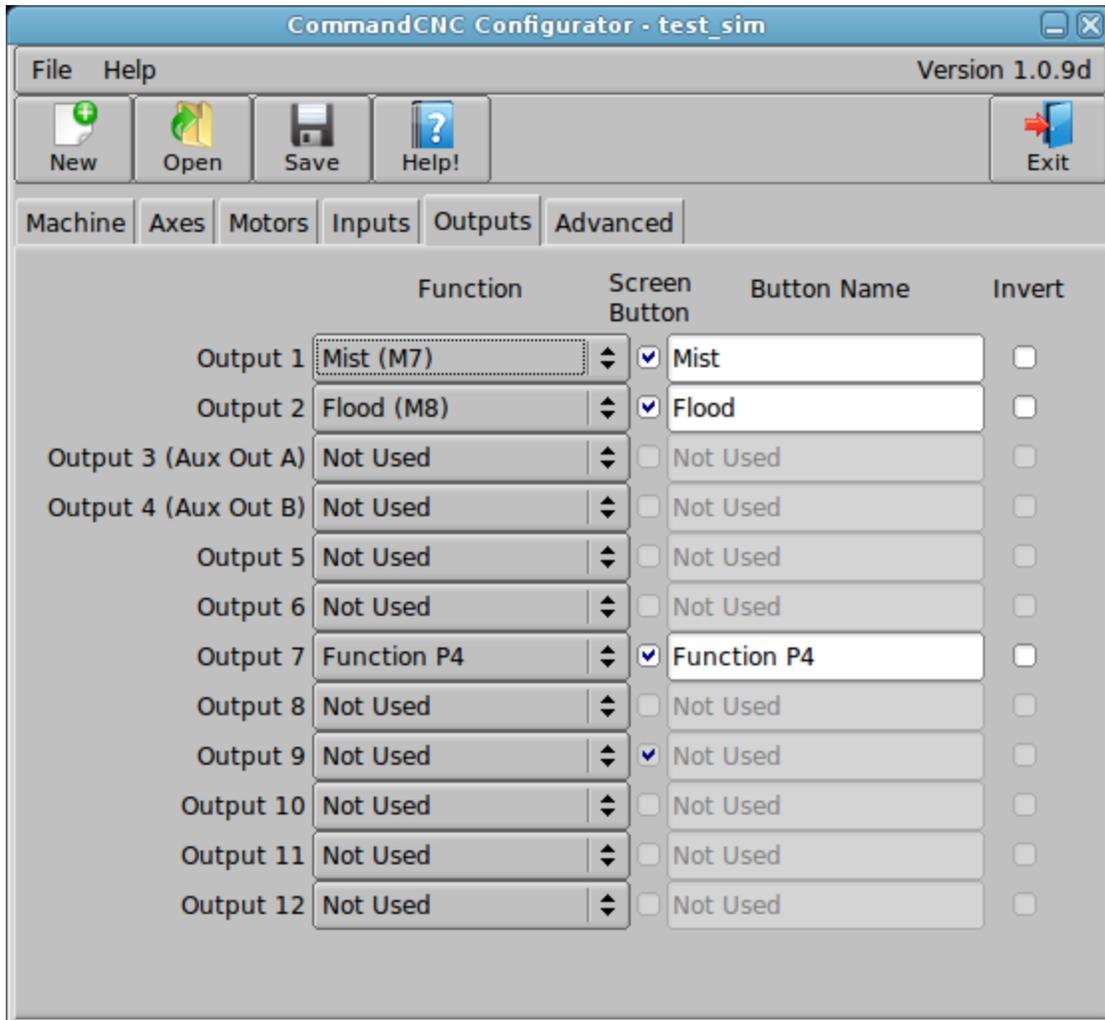


We have already seen some of the user interface changes with the Fixture Offsets page and Auto-Set changes. Several other changes should be noted:

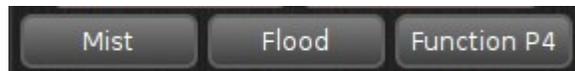
- Feed Override, Max Velocity and Jog Speed controls have been moved from the bottom left corner of the main screen to the bottom of the Manual tab above.
- The Output buttons have moved from under the axis DROs to the left where the Feed, Velocity, and Jog controls used to be.
- Two of the LEDs which were in the THC box have moved to the Diagnostics tab: HyT Online and Hub Watchdog ON

Output buttons can now be customized

Along with the Output buttons having moved, there is now more flexibility in which outputs to show buttons for and the label that should be on the button.



For each output that has the “Screen Button” box checked, there will be a button placed on the screen in CommandCNC to operate that output. The “Button Name” field is a place you can enter the name that you want to appear on the button in CommandCNC. The entries above will produce the following buttons on screen:



Up to eight buttons can be placed on the screen – two rows of 4 buttons each.



NOTE:

- Most systems only have two available outputs, but EBOB equipped units can have expansion relay boxes added.
- The Torch output is not listed because it is automatically connected on a Plasma config, and it uses the relay in the PWM Sensor card. More information on this is available in the release notes for CommandCNC 1.0 on our website.

Support for two more axes

CommandCNC now supports the U and V axes for a total of 8 axes of motion.

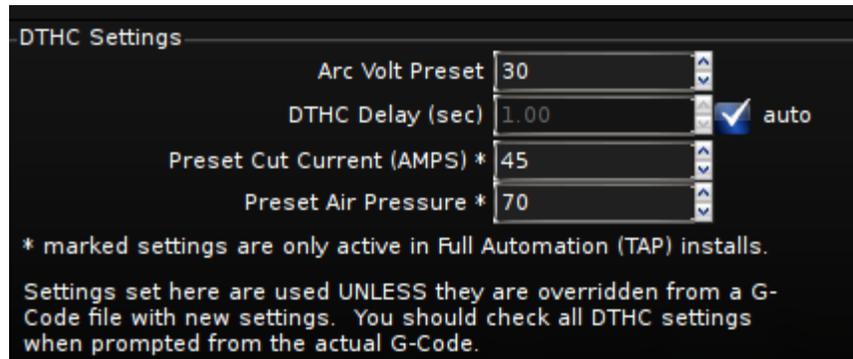
The screenshot shows the 'U Axis' configuration window in the CommandCNC configurator. At the top, there are tabs for X Axis, Y Axis, Z Axis, A Axis, B Axis, C Axis, U Axis, and V Axis. The 'U Axis' tab is selected. Below the tabs, there is a checkbox labeled 'Enable' which is checked. The title 'U Axis' is centered. Under 'Axis type', there are two radio buttons: 'Linear' (selected) and 'Rotary'. Below this, there are three input fields: 'Maximum Velocity' with the value '500' and units 'units / min', 'Maximum Acceleration' with the value '35' and units 'units / s²', and 'Axis Travel Limits: min' with the value '-10000' and 'max' with the value '10000'. A 'Homing' section is enclosed in a box and contains: 'Homing Direction' with radio buttons for 'Negative' (selected) and 'Positive', 'Homing Velocity' with the value '20' and units 'units / min', 'Home Switch Offset' with the value '0', and 'Move to after homing:' with the value '0'.

CommandCNC will add DROs and buttons for each axis which is enabled and set up in the Configurator.



New THC delay “Auto” setting

There is a new feature in the THC Settings → Cut Settings page to automatically calculate an optimal DTHC Delay. This setting will be enabled by default. If you for any reason want to



enter a manual delay as before, simply un-check the “auto” box beside the DTHC Delay setting and enter a value as before.

Miscellaneous other fixes and improvements.

In addition to the more major changes mentioned, there have been a number of bug fixes and improvements:

- PN200 step-jogging with the joystick only moved in one direction – **now fixed**
- CommandCNC now saves and restores window size and position
- Improvements to tolerate higher computer latency and bad packets without causing following errors