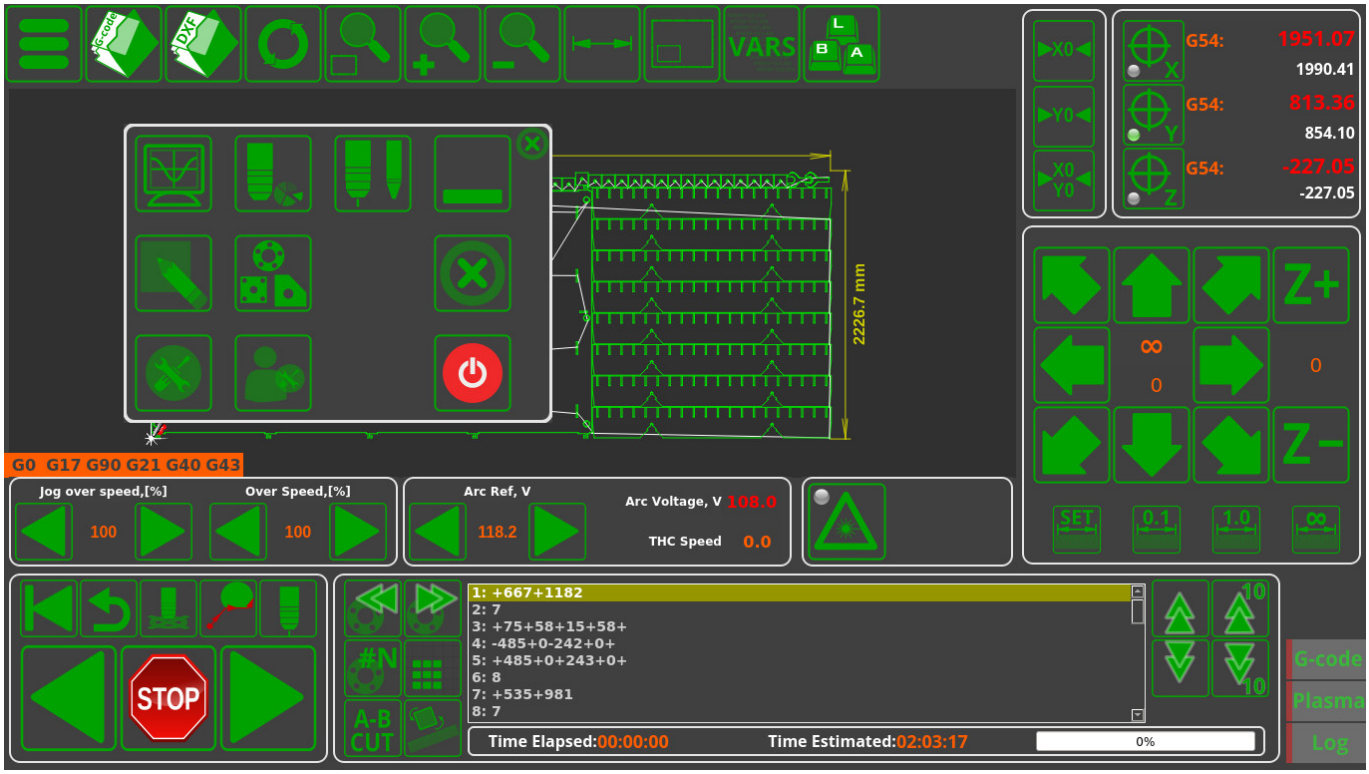
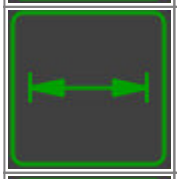
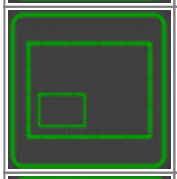
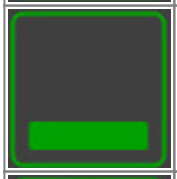
Plasma cutting profile 1366P

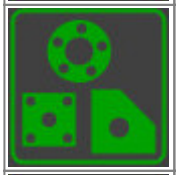
Upon loading myCNC software with the 1366P Plasma Cutting profile, you are presented with the following screen:

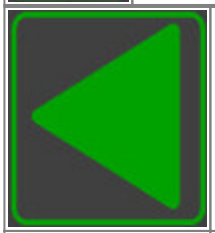
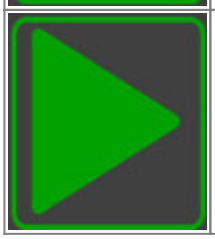


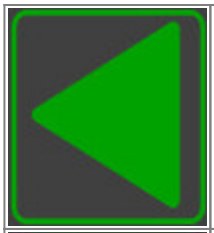
Main Screen buttons

	Go to the main configuration window
	Open a G-code file
	Open a DXF file
	Reload the program from the hard drive
	Zoom - Fit to Window

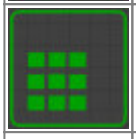
	Zoom In
	Zoom Out
	Show dimensions of the nesting chart
	Show work area with the nesting chart
	Show VARS window
	Show the virtual keyboard
	Minimize the myCNC software
	Close myCNC software
	Shut down the workstation
	Open the System Diagnostics window
	Tool configuration window

	Technology change (plasma to gas cutting)
	Open the editor
	Common parts library
	Open Settings
	Open Custom Machine Settings /User Settings (See below)
	Run the homing for x-axis
	Run the homing for y-axis
	Run the homing for xy-plane
	Reset to zero the working x-coordinate
	Reset to zero the working y-coordinate
	Reset to zero the working z-coordinate
	Machine movement buttons (xy plane)
	Machine movement button (positive z-axis)

	Machine movement button (negative z-axis)
	Set the machine movement step size to a specified value
	Set the machine movement step size to 0.1 mm
	Set the machine movement step size to 1 mm
	Set an infinite machine movement step size
	Reset the operating point to the beginning of control program
	Return to the working point
	Binding of the start of the control program to operating point. NOTE: The machine will bind the coordinates for the axes specified in Settings > Config > Preferences > Start/Stop. If no axes are selected, no binding to the operating point will be performed. Link to video with demo: https://www.youtube.com/watch?v=lgKgMRz-JN8&feature=youtu.be&t=45
	Start cutting from this position
	Enter simulation mode (trial mode) to ignore cutting commands - see description below
	Go back
	Stop the program
	Run the program

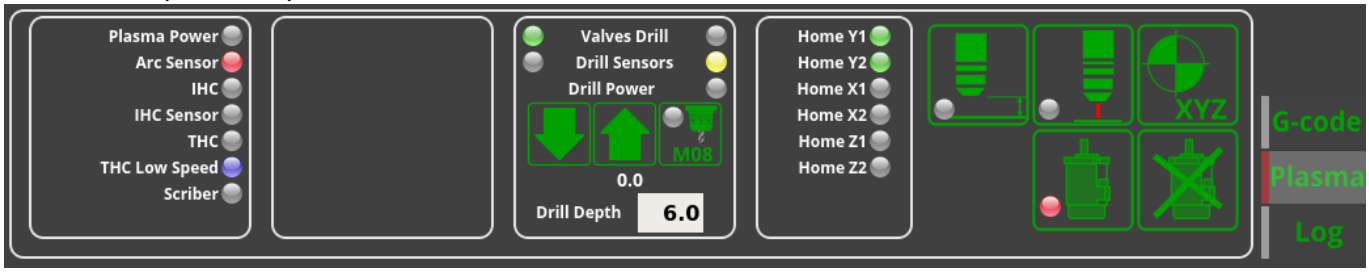
	Go back
	Stop the program
	Switch the instrument to laser pointer. Link to video with demo: https://www.youtube.com/watch?v=pxMQ9XLyOBQ

G-code tab (Part selection and editing)

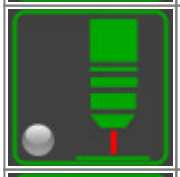
	Go to previous part. The plasma cutting files often consist of a large number of small parts that will be cut out from the working material. This button allows to navigate between the different parts of the G-code file to quickly move back if necessary.
	Go to next part
	Go to a specific part number
	Open part cloning window. This allows to quickly multiply the existing G-code file and to arrange it on the working material in the operator's preferred way.
	Cut from point A to point B. This allows to do a manual straight line cut between two points. Upon clicking the A-B Cut button in the G-code tab, a small window appears which allows the user to set some point, then move the machine away and then click A-B Cut to begin the cutting process.
	Open part rotation window. This window will allow the user to rotate the program file by a set angle or mirror the program file about some axis.

Plasma tab (LED indicators and drill operation)

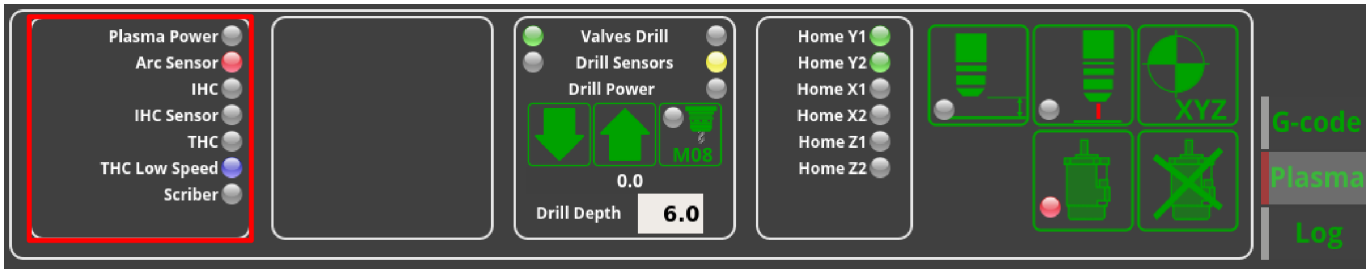
The G-code profile is presented below:



This screen presents the user with the options and indicators for the plasma controls. The five buttons on the right side of the tab are the most commonly used:

	Surface Measure
	Plasma ON
	Homing for the XYZ axes
	Servo ON (done through software PLC)
	Servo OFF

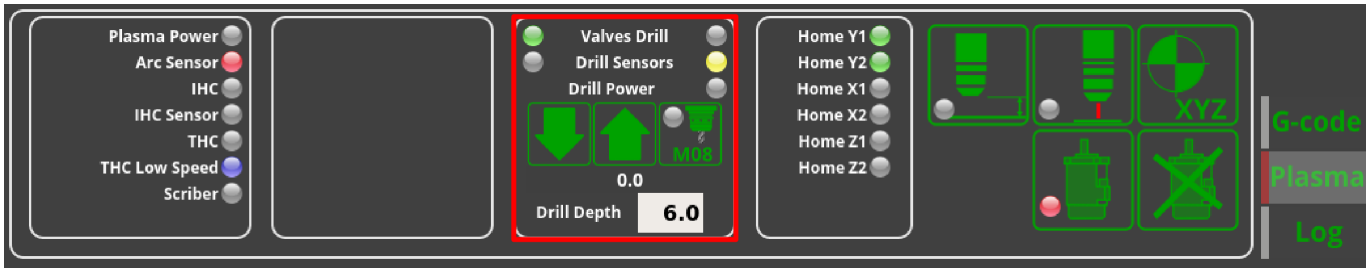
The LED panel on the left of the G-code tab shows the current state of the main plasma components (whether the plasma machine has power, if the arc sensor is on, the IHC indicator for the initial contact with the metal, THC and THC Low Speed LEDs which indicate whether the Torch Height Control is ON or OFF, and the Scriber LED which shows if the marking process is on or off (marking can be done with a smaller plasma torch or with an etching working tool):



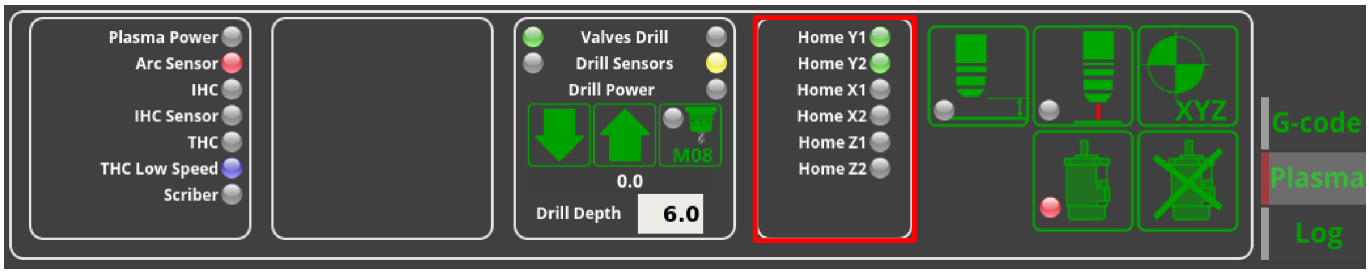
The Drill window allows the user to control the drill attached to some motor on the machine to be used for hole cutting, which is typically operated through the use of some pneumatic or spring mechanism to push the drill module down to the metal. The most common drill modules that myCNC is used on utilize two valves to move the drill up/down, as well as two sensors to register when the drill is in the up/down position. Therefore there are two LEDs for both the valves and the sensors, as well as an indicator LED for whether the drill module is or or not.

The M08 button begins the M08 PLC procedure, which involves turning the drill on and operating the valves to press the module into place. The up/down buttons allow the user to move the entire module in the z-axis via the motor that is connected to operate the drill module (note that this is separate from the valves which operate only when smaller distances are involved). The drill depth from the top

of the working material can also be set in mm:

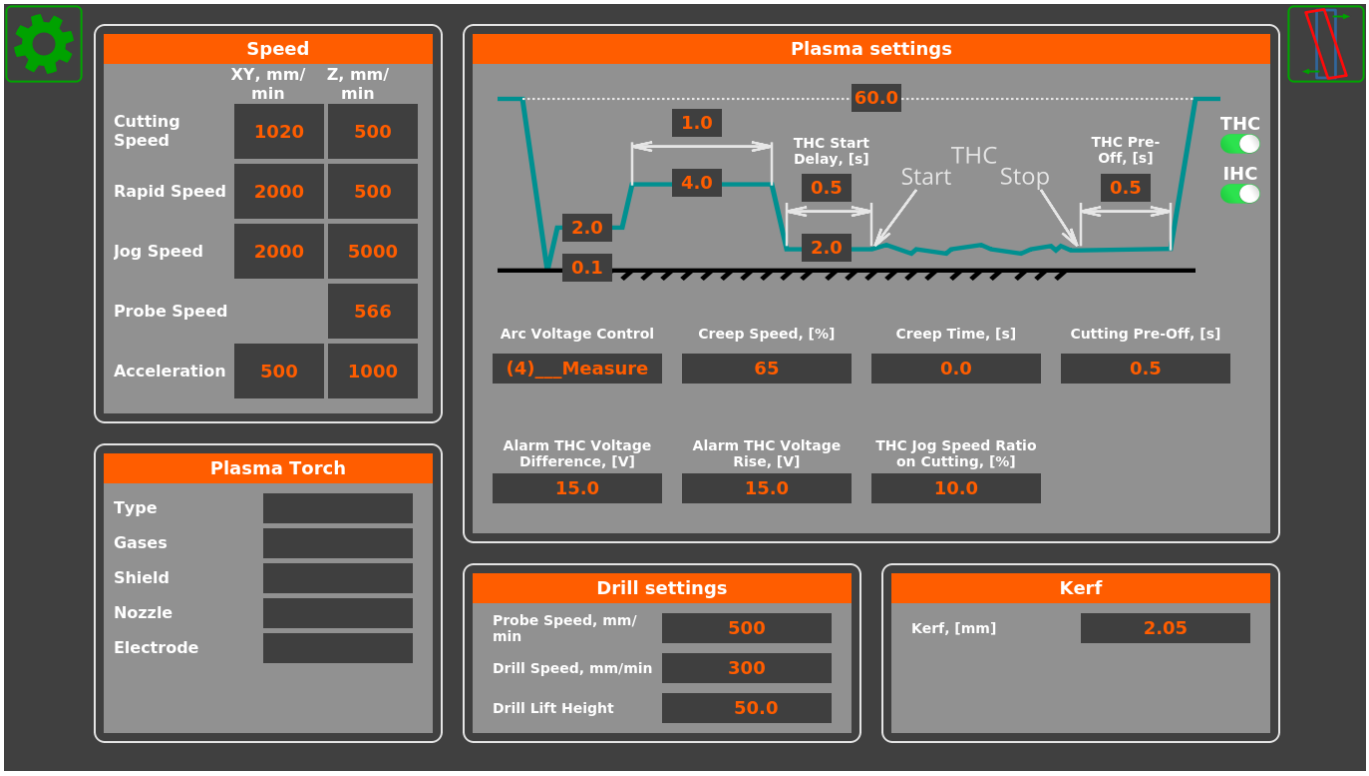


The homing sensors LED show two sensors per the X, Y and Z axes. Not all of these sensors are commonly used during the homing procedure on different machine setups, so the user is free to edit the `x-plasma.xml` file in the X1366P profile folder to not display the unnecessary LEDs:



Custom Machine Settings (User Settings)

Upon opening the Custom Machine Settings for the 1366P profile, you are presented with the following screen:



This settings window allows to specify the distances within the plasma cutting cycle, as well as functions such as THC and IHC. It also serves as the quick settings window which is always accessible to the operator, as opposed to the general settings, which might be open or closed to the operator

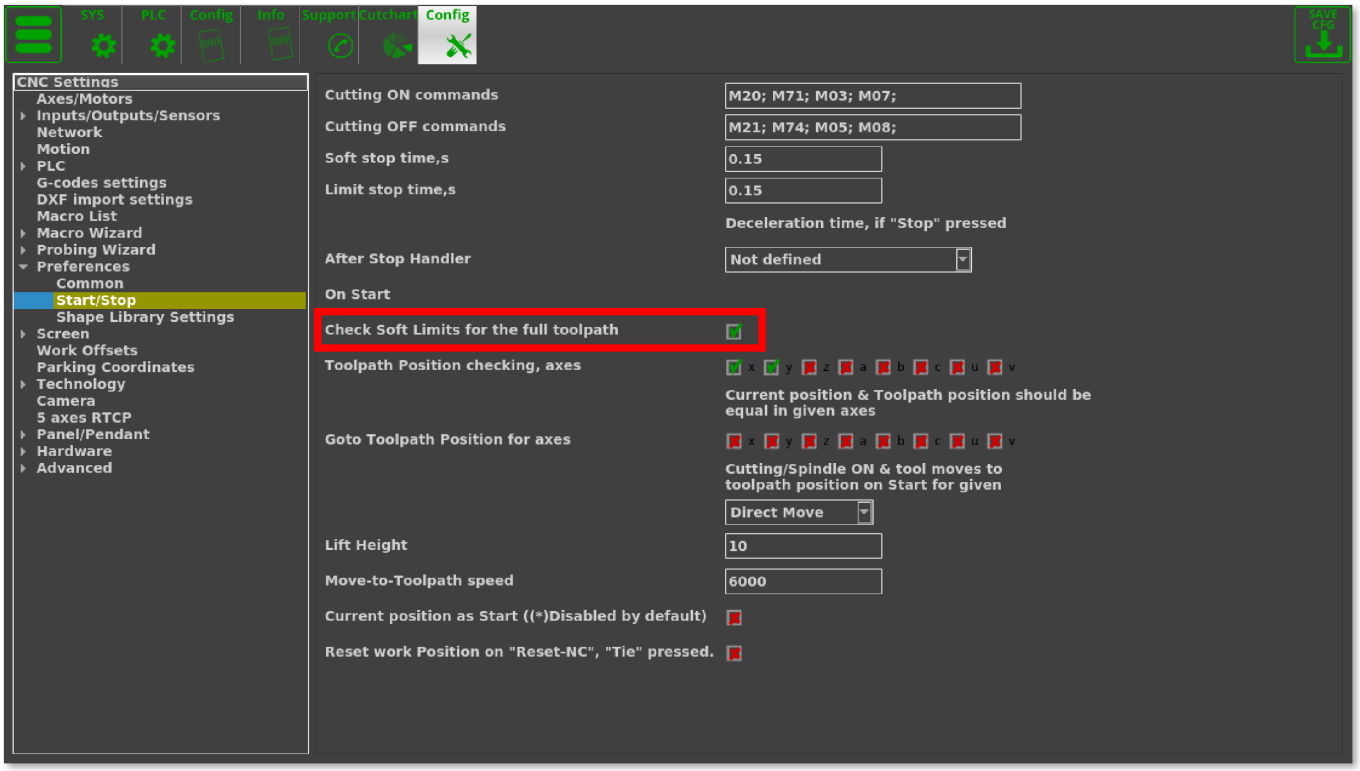
depending on the software configuration that the user decides to employ.

Progress bar

The progress bar for the control program is present in all the recent versions (X1366 series) of myCNC plasma cutting software.

The bar shows time elapsed, estimated time remaining for the entire program, and the percent progress bar for quick visual reference.

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



Simulation mode (Trial mode)

With the simulation mode turned ON, the software will ignore the cutting commands, simply moving the working bit (plasma torch, laser pointer, etc) around to give the user an option of running the program before the actual cutting begins. The commands which the myCNC software will ignore can be specified in the **Settings > Config > Preferences > Start/Stop**, as seen in the image below: