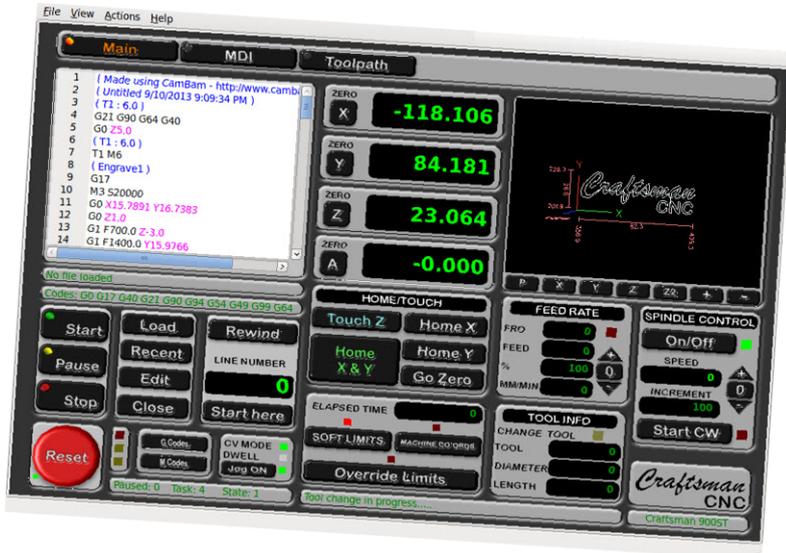


Craftsman CNC

LinuxCNC

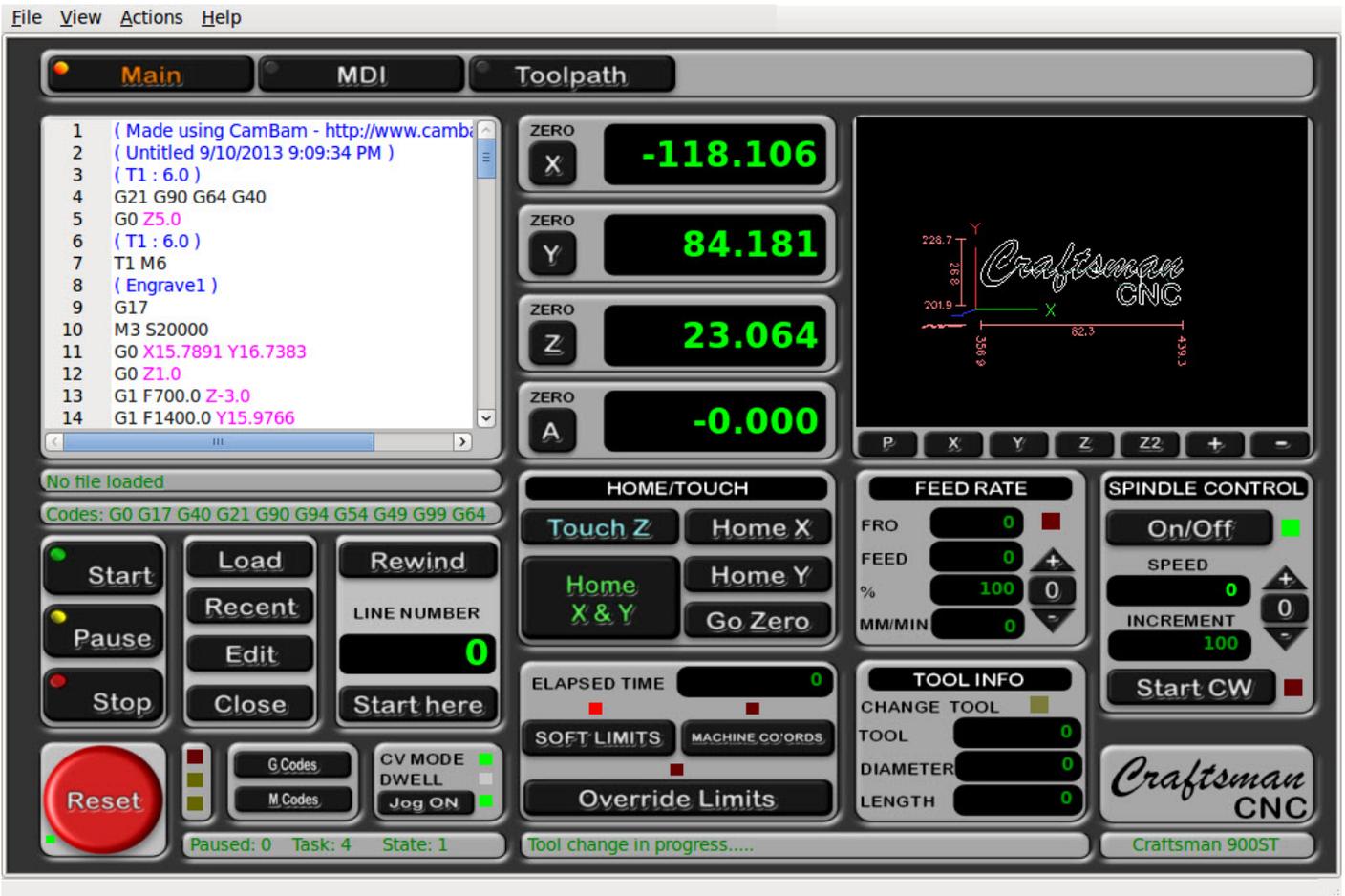
Custom developed for Craftsman CNC Routers



Menu

File	Load gcode	Load a gcode program
	Recent gcode	Shows the 10 most recently opened gcode programs
	Close gcode	Closes the current opened gcode program
	Edit tool table	Opens the tool table to add or edit tools
	Quit	Quit LinuxCNC
View	Show HAL	Show the HAL meter - refer to HAL manual
	Show scope	Show the oscilloscope - refer to HAL manual
	Show shortcuts	Show the shortcut keys help screen
Actions	Reset	Reset LinuxCNC - Use in event that Red button is not responding
	On/Off	Switch machine on or off - Use in event that Red button is not responding
	Clear MDI	Clear the MDI history list
	Clear axis offsets	Clear all offsets set on all axis
	Home Z	Home the Z axis and zero's it with the touch plate.
	Home A	Home the A axis and sets it to zero
	Touch X minus	Touch X axis in the negative direction, make sure that the offset is set in m102.ngc
	Touch X plus	Touch X axis in the positive direction, make sure that the offset is set in m103.ngc
	Touch Y minus	Touch Y axis in the negative direction, make sure that the offset is set in m104.ngc
Touch Y plus	Touch Y axis in the positive direction, make sure that the offset is set in m105.ngc	
Note:	The ngc files can be found at <u>home/craftsman/linuxcnc/configs/craftsmancnc/macros/</u> Offsets are set with gcode, <u>G92</u> and the axis, eg <u>X-5</u>	
Help	About	Display LinuxCNC version information

Main - Screen



From the Main screen the Craftsman CNC Router are controlled and gcode programs are run and controlled.

Reset Switch machine on or off - short cut key - Ctrl + C
 Stop Stops the currently running gcode program - short cut key - Ctrl + S
 Pause Pauses the currently running gcode program - short cut key - Ctrl + P
 Start Runs the loaded gcode program - short cut key - Esc

By pressing F1, the hot key help screen will open to reference other hot keys available

Load Load a gcode program
 Recent Shows the 10 most recently opened gcode programs
 Edit Opens the gcode program in an editor window enabling gcode program editing.
 Close Closes the current opened gcode program
 Rewind Rewinds the current gcode program to it's beginning.
 Line number The currently executed gcode line or when entering a line number the gcode viewer will display the entered line number.
 Start here Start the gcode program from the selected or entered line number.

G Codes Display a list of gcodes and explanation of the gcode's function.
 M Codes Display a list of mcodes and explanation of mcode programming.

CV MODE When the LED is shining green constant velocity mode is active.
 DWELL When the dwell LED is flashing LinuxCNC is in a paused state giving the spindle time to spin up to speed.
 JOG ON Enable and disable manual jogging of the individual axis.

HOME/TOUCH

Touch Z Used to zero the Z axis by placing the touch plate under the tool and pressing the Touch Z button. The Z axis will move down slowly until it touches the plate. The tool tip's position is set to zero. And the Z axis will lift of the touch plate.
 Home X and Y Used to set the X and Y axis's home position simultaneously.
 Home X Used to home the X axis only.
 Home Y Used to home the Y axis only.

Elapsed time	Time elapsed since currently running gcode program started
Soft limits	NOT IMPLEMENTED YET
Machine coordinates	Sets the DRO's to display machine coordinates without offsets or show positional offsets.
Override limits	When limit switches have been activated, the Override Limits button are used to de-activate limits and allow movement away from the activated limit switch.

FEED RATE

FRO	Feed rate override
FEED	Current set feed rate
%	Percentage of feed override
MM/MIN	Current feed rate of the moving axis
+	Override feed rate faster
0	Reset feed override
-	Override feed rate slower

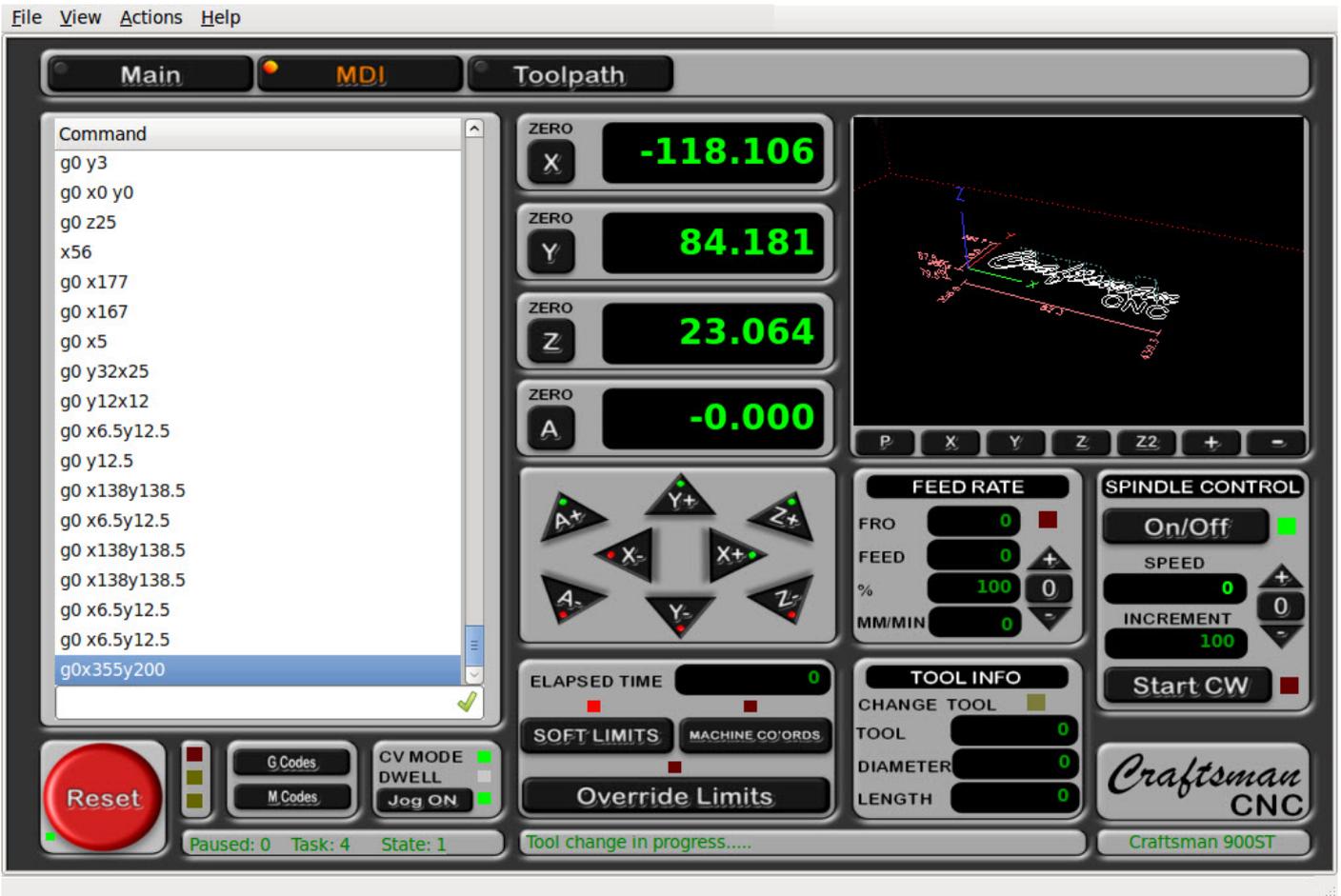
TOOL INFO

CHANGE TOOL	LED flashing when tool change is required.
TOOL	Current tool number
DIAMETER	Current tool diameter (if set on the tool table)
LENGTH	Current tool length (if set on the tool table)

SPINDLE CONTROL

On/Off	Set automatic spindle control
SPEED	Currently set spindle speed in RPM's. The speed can also be set by entering a speed.
INCREMENT	Percentage of spindle speed override
Start CW	Starts the spindle manually clock wise at the set speed.
+	Override spindle speed faster
0	Reset spindle speed override
-	Override spindle speed slower

MDI -Screen



From the MDI screen the Craftsman CNC Router are controlled by manually entering gcode.

Any gcode, mcode or ocode can be entered line by line. Press the enter key to activate the code or click the mark button.

The axis can be jogged manually with the mouse by pressing the axis arrow buttons.

Toolpath - Screen

File View Actions Help

Main MDI **Toolpath**

```
1 ( Made using CamBam - http://www.cambam.com )
2 ( Untitled 9/10/2013 9:09:34 PM )
3 ( T1 : 6.0 )
4 G21 G90 G64 G40
5 G0 Z5.0
6 ( T1 : 6.0 )
7 T1 M6
8 ( Engrave1 )
9 G17
10 M3 S20000
11 G0 X15.7891 Y16.7383
12 G0 Z1.0
13 G1 F700.0 Z-3.0
14 G1 F1400.0 Y15.9766
15 G2 X12.168 Y12.3672 I-30.384 J26.8616
```

AXIS DRO'S

X	-118.106
Y	84.181
Z	23.064
A	-0.000

PROGRAM LIMITS

	356.914	439.250
	201.853	228.668
	79.471	87.471
	0.000	0.000

Simulate program Run from here

ESTIMATED PROGRAM RUN TIME 0

Start Rewind

Pause Stop

Reset

G Codes CV MODE
M Codes DWELL
Jog ON

Paused: 0 Task: 4 State: 1

Tool change in progress.....

Craftsman 900ST

The Toolpath screen display a larger toolpath

Simulate program Not implemented yet
ESTIMATED PROGRAM RUN TIME Not implemented yet

Program limits

The minimum and maximum travel positions for each of the axis.