LANGMUIR SYSTEMS

FireControl is the Machine Control software for the Langmuir Systems CrossFire CNC Plasma Tables. Langmuir Systems supports Post Processors for Autodesk Fusion 360 and SheetCAM but you may want to write a post processor for your particular CAD/CAM system.

Note: This guide is intended for someone needing the specific Gcode commands used in a FireControl post processor, NOT someone starting from scratch in post development. Please use this document to help flesh out the requirements for generating FireControl compatible G-code.

We recommend generating programs in these existing post processors and viewing these other post processors code directly to aid in developing new Posts.

Please reach out to support@langmuirsystems.com for any concerns relating to this documentation.

CrossFire GCode

The CrossFire line of CNC machines uses most basic GCode commands as well as some custom commands specifically developed for FireControl and the CrossFire. The table below outlines the accepted generic commands (some of which do not apply to plasma). The default state is highlighted in bold.

Modal Group	Member Words
Motion Mode	**G0**, G1, G2, G3, G38.2, G38.3, G38.4, G38.5, G80
Coordinate System Select	**G54**, G55, G56, G57, G58, G59
Plane Select	**G17**, G18, G19
Distance Mode	**G90**, G91
Arc IJK Distance Mode	**G91.1**
Feed Rate Mode	G93, **G94**
Units Mode	G20, **G21**
Cutter Radius Compensation	**G40**
Tool Length Offset	G43.1, **G49**
Program Mode	**M0**, M1, M2, M30
Spindle State	M3, M4, **M5**
Coolant State	M7, M8, **M9**

CrossFire Specific Gcode commands are outlined in the sections below.

IHS

The IHS or Initial Height Sensing sequence is a set of probe commands designed to determine the height of the material using the torch as a probe switch. It uses G38 commands to seek the material and determine its height so that the torch starts to cut at a precise relative cut height.

> G38.2 Z-5.000 F100.0 (Probe Seek Down)
> G38.4 Z.5 F20.0 (Probe Seek Up)
> G92 Z0.000 (Zero Z)
> G0 Z**0.15** (Move to Pierce Height)
> M3 (Fire Torch)
> G4 P**0.4** (Pierce Delay)
> G1 Z**0.06** F**100.0** (Move to Cut Height)
> H1 (Enable THC Control)
> G1 X0.6941 Y5.530 F129.0 (First line of Cut)
> …

The bolded portions of the above are likely going to be configurable via your new post. Generally users set **Pierce Height, Cut Height, and Pierce Delay** per program. Optionally the speed at which the torch moves from pierce to cut height is often configurable.

Supported FireControl posts also include an **offset** value that can be optionally included to otherwise account for springback caused by the torch weight bending the material during the probe sequence. Please see generated programs for this optional setting.

THC

FireControl accepts H0 and H1 modal commands to control the activation of the LS-THC system. These commands determine which of the two controllers controls Z-Axis motion.

H0 - Machine Controller in Control of Z-Axis.
H1 - Torch Height Controller in Control of Z-Axis.

H0 is written into the preamble of every program to ensure that the machine controller has control of the Z-Axis to perform the IHS sequence.

H1 is written once the torch has fired, delayed, and moved to Cut Height - before starting a cut loop. Once the cut loop has finished and the torch must retract for the rapid to the next cut, the Machine controller needs to be in control of the Z-Axis so an **H0** is again written, generally before the Torch Off (M5) command.

Ex.

```
…
M3
G4 P0.4
G1 Z0.06 F100.0
H1
G1 X0.6941 Y5.5301 F100.0
G1 X3.4441 Y5.5301
…
G1 X0.6656 Y5.5016
G1 X0.6656 Y5.5954
H0
M5
…
```

Pre-amble

Version Comment

The first line in a FireControl Posted Program contains a comment that identifies the version of the post processor that generated the program. This comment is intended for use with the two Langmuir Systems supported Posts: Autodesk Fusion and SheetCAM.

If you are developing your own post, be sure to use either **(v1.6-af)** or **(v1.6-sc)** in the first line of your program to ensure that FireControl will accept it. *If you publish your post online, please make Langmuir Systems aware by contacting our support team and we can assign a character code for your post so that future versions of FireControl will accept it without needing to mask as Fusion or SheetCAM.*

Ex.

```
(v1.6-af)
G90 G94
G17
G20
H0
…
```

Machine Settings

The rest of the program preamble are machine settings and initial states (THC) apart from the units setting G20/G21.

```
G90 G94
G17
G20
H0
...
```

We recommend using the above (with your desired units) verbatim in your post processor.

Program Speed Comment

For use with the LS-THC system, all programs loaded into FireControl must have what is called the PS Value comment as the last line of the program. This is a comment that looks like:

```
...
M5
G0 Z1.0
M30
(PS100)
```

At the end of a program. This value must exist in a comment and must be on the last line of the program. The comment is structured with the letters PS followed by the linear feed rate of the program in the units of the program. For example the PS value above would apply to a program at which the fastest speed (generally the linear feed rate) within the program is 100 IPM or mm/min.

This PS value is used by the THC system in conjunction with the Program Speed Cutoff value in order to determine the torch speed at which LS-THC should and should not make Z motion corrections. Please see LS-THC documentation for more information.

If you are not using LS-THC we still recommend coding your Post Processor to produce this value. A Program Speed comment is required for all programs that are loaded into FireControl but this value will not be used unless you are using the LS-THC system.

Special Situations

Circles

Circles can be coded as a single line of GCode, however, due to nuances in the controller firmware, programmed circles should be broken into two or more half circles (multiples lines of gcode) to have different start and end points.

If your posted programs are **missing** circles when loaded into FireControl, this is likely the reason.

Rounding

Keep your values rounded to a **maximum** 4 digits after the decimal. More than this can cause unnecessary bloating to the Machine Controller buffer.