OVERVIEW

While not the only option for CNC Plasma CAM, SheetCAM is widely regarded as the easiest and motion efficient solution available for generating G-code files for CNC Plasma Cutting.

SheetCAM features:
- Ability to import DXF and SVG design files
- Rotate, scale, and nesting functions
- Precise control of lead ins, lead out, and start points
- Ability to save cutting parameters based on material thickness and type
- Post Processors available for CrossFire and CrossFire PRO (FireControl) as well as Legacy CrossFire {Mach3).
- Lifetime license when purchased (including updates). No monthly or yearly subscription!
- Windows and Linux compatible
- Optimized G-Code allowing your CrossFire or CrossFire PRO to run fast and smooth

CAM

- There a number of great SheetCAM tutorials available on line including SheetCAM's own User Guide.
- If you run into any issues or have questions, check out the SheetCAM section of the Langmuir Systems Forum for help from other CrossFire users.

Post Processing for FireControl

- The following guide provides instructions for generating cutting programs in SheetCAM software to run in FireControl by using the Langmuir Systems FireControl Post Processor file.

The FireControl Post

- Processor will create g-code instructions specific to your CrossFire machine including IHS sequences, THC activation points, and rapid retract moves.

a. Download Post

-To start, you will need to navigate to the Langmuir Systems Downloads page and download the latest FireControl - SheetCAM Post file. After the file has downloaded, navigate to the Downloads folder on your computer and copy the FireControl-vl.6.scpost file that you just downloaded.

b. Install Post

- Next, you will need to navigate to the folder on your computer where the Post Processor files for SheetCAM are stored so that you can copy the FireControl-vl.6.scpost file into this folder.
- On most Windows computers, the Post Processor folder is located at C:\Program Files (x86)\SheetCam TNC\Posts but you will need to verify this on your own system.
- After locating the folder, paste the copied Post Processor file from Step l into this folder. Your folder should now look like the folder shown below showing the FireControl-Vl.l.scpost file.

c. Setup

Next, open SheetCAM software and click Options> Machine from the top Menu bar.

- Under the Machine Type tab make sure that Jet Cutting is selected and Rotary Cutting is de-selected.

- Next, under the Post Processor tab make sure that your desired Output file units, desired Output folder, and Z Zero (should be Top of Work) are configured correctly. Next, click the drop down bar under Post Processor and select FireControl-Vl.l like shown below. Finally, click [OK] to exit the menu.

d. Settings for CrossFire With Z-Axis

- After selecting the FireControl Post Processor, we are now ready to create programs to cut in FireControl using SheetCAM as normal.
- If you are using the CrossFire PRO or a CrossFire machine with a powered Z-axis, you will need to input settings for Pierce Height, Plunge Rate, and Cut Height when creating a plasma tool in order to activate the IHS sequence.
- For pierce height and cut height, please consult your plasma cutter manufacturer for the appropriate values.

- Note: If you are cutting on thin gauge material, we reccommend adding additional height to your Cut Height in order to compensate for the material springback during IHS. For the plunge rate used for the IHS sequence, we recommend using a rate of 700 1PM.
- Please also note that the FireControl post for SheetCAM has a hard-coded l inch rapid retract move after each cut loop is completed by default (after MS torch OFF).

e. Settings for CrossFire Without Z-Axis

- If you are using a CrossFire machine without a powered 2-axis, be sure to set both Pierce Height and Cut Height to 'O' in the Jet Tool settings menu shown above in order to omit the IHS sequence from the generated g-code file.