Easel G-code Spec

[Follow](#)

Changelog
- 12/18/2015 : Add note to always include unit system (G20/G21), Published

This document describes the format of g-code files that can be sent by Easel. The range of allowed options is currently small because we want to be sure that any g-code that is sent is correctly parsed, visualized and safely executed by the machine.

Any g-code commands not listed here will cause Easel to reject the entire file.

The following commands are allowed exactly as shown

G0, G1, G4, G17, G20, G21, G40, G54, G61, G90, G94

M0, M1, M2, M3, M4, M5, M6, M7, M8, M9, M30

The following commands are allowed, in this order, followed by a decimal (or whole) number

X, Y, Z, F, S

Implementation Notes:

- **ALWAYS** include a G20 or G21 depending on your coordinate system to avoid confusion.
- Only linear movements are allowed (arcs must be linearized).

- Tool changes are not allowed, so no more than one tool can be written to a file at a time.

- All settings commands with only one option (G54, G17, G90, etc.) are the default. There is no need to include those commands.
- Comments in parenthesis or lines ending in (;) will be ignored.

Special note for Carvey:

- G-code must be absolutely positioned inside of the machine, and not exceed the work area and Smart Clamp Area. Any movement outside of these bounds, or inside the clamp area, will be rejected.
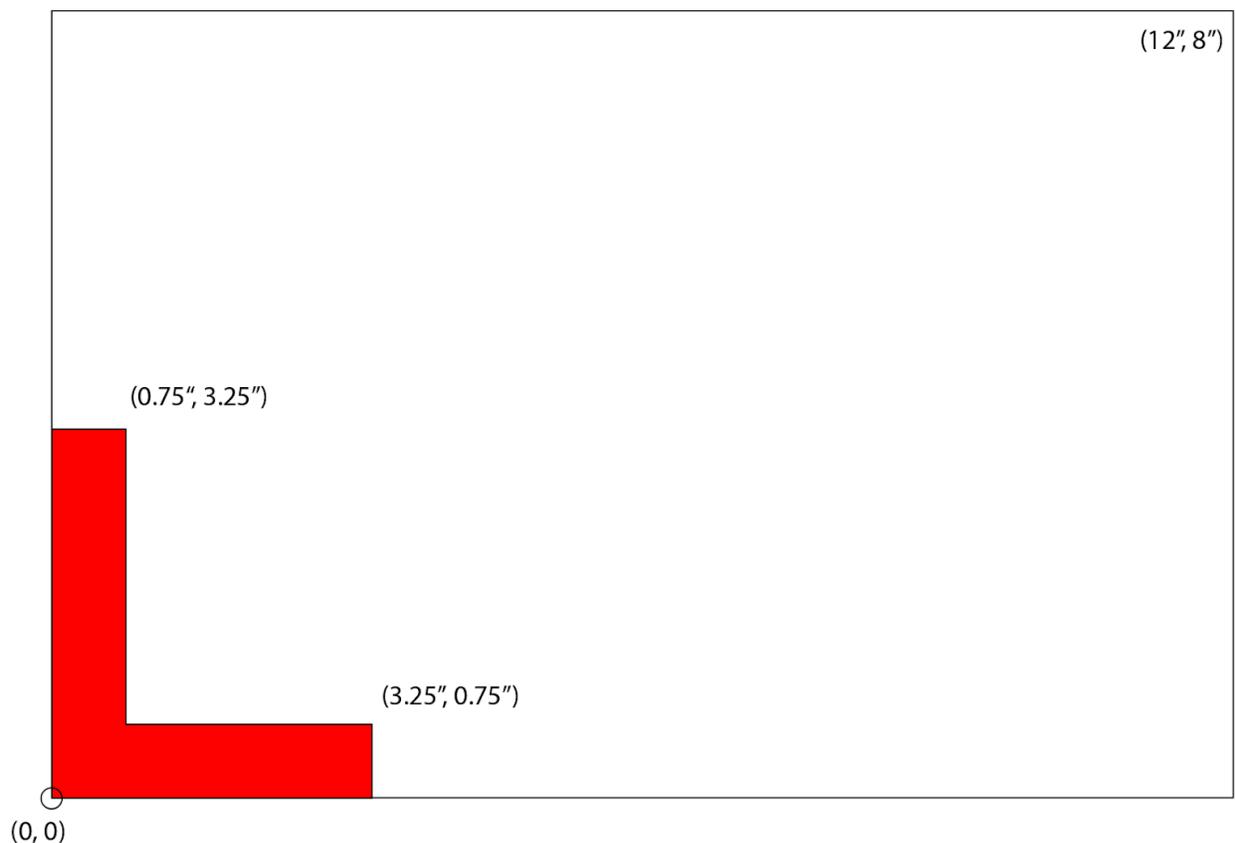
- The G-code will be sent between a header and footer that ensure that the machine is homed and will touch off the Smart Clamp, setting the Z=0 at the material height.

Was this article helpful?

454 out of 670 found this helpful

Have more questions? [Submit a request](#)

Return to top

Recently viewed articles

[Installing Fusion 360 Post Processor](#)

Related articles

[File Import Guide](#)
[Start Carving in Easel](#)
[Add Easel's post-processor to your Vectric software](#)
[Can I make my X-Carve larger than 1000mm x 1000mm?](#)
[Easel Keyboard Shortcuts](#)