

Gerber to G-Code, Help needed.

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2006-10/msg00200.html>

- *From:* "chapmjlw" <chapmjlw@xxxxxxxxxx>
 - *Date:* 20 Oct 2006 21:30:09 -0700
-

Hello folks,

I've written software that converts Eagle PCB output to G-Code so you can use any 3 axis mill create a quick circuit board. Every thing works

perfectly from eagle but i'll like to be able to import a gerber file, then convert to G-Code.

I have no problem converting Gerber code and displaying it. The difficulty starts when you have rearrange all intersecting element types

and then generate offset toolpath geometry.

Is there any one out there that has done this?
Is there any one that would like to help with this project?

thanks
jim

.