

Gerbers: Food for designers

M. Simon



In my search for good tools that don't cost too much (I like free) I have found an excellent tool for viewing Gerber and drill files. Those are the files produced by most PC board design software and required by most board producers to produce boards. If you want to see what the board will look like without producing one you need a Gerber viewer as a cross check to your board design software.

The [FreePCB](#) [1] (my current layout software) manual suggested [Viewmate](#) [2]. **Viewmate** has a free version (I like that price point) that does not do everything its larger brothers (at \$95 and \$495) will do, but you can view your Gerbers and then edit your board using your layout software as required. **Viewmate** has a very nice set up for viewing lots of layers (**FreePCB** can make 16 layers) and it will also paint drill drawings on your screen. It also allows you to do check plots.

It has already helped me with a design I am working on. Thanks guys!

A note on [FreePCB](#) [1] is in order. In some ways the human interface is kinda klunky compared to some layout software I have used. It took me about a week to get used to its ways. That said - it has the finest footprint maker/manager I have ever encountered. On the other hand the user interface for some functions does not exist. No matter. You can hack the set up files (ASCII text) to change some operating parameters. The excellent users group is ready to help. And if you are really ambitious the source is open.

M. Simon's e-mail can be found on the sidebar at [Space-Time Productions](#) [3].

Source URL (retrieved on 04/01/2015 - 9:22am):

http://www.ecnmag.com/blogs/2012/03/gerbers-food-designers?qt-video_of_the_day=0

Links:

[1] <http://www.freepcb.com/>

[2] <http://www.pentalogix.com/viewmate.php>

Gerbers: Food for designers

Published on Electronic Component News (<http://www.ecnmag.com>)

[3] <http://spacetimepro.blogspot.com/>