

# Short notes on generating Gerber artwork in Allegro PCB Editor

Nick Chernyy  
nico@routed.net

v1.0 – September 14, 2008

## Contents

<b>1</b>	<b>Short notes</b>	<b>1</b>
1.1	9/10/2008 – How to set spacing and via size in Allegro . . . . .	1
1.2	9/10/2008 – How to generate PCB artwork in Allegro . . . . .	1

## 1 Short notes

### 1.1 9/10/2008 – How to set spacing and via size in Allegro

Setting the via padstack can be done by selecting *Setup* → *Constraints* and then selecting the *Set value...* button in the *Physical rule set*. The entries in the *Current via list* that have an asterisk are not located and must be found below. The via called *Via* is a 30mil annulus with a 13mil drill.

Although spacing can be set in the same constraint configuration area, it can also be set in the SPECCTRA/Allegro autorouter which is enabled by selecting *Route* → *Route Editor*. Once SPECCTRA is started, individual net clearances can be set by selecting the *Rules* → *Nets* → *Clearance...* pull-down menu. Global clearance setting is done in PCB Editor by selecting *Setup* → *Constraints* and then *Set values...* under *Spacing rule set*. 4pcb requires a minimum of 5mil, however, 10mil is better for larger boards. Again, the wire widths can be set in the *Constraints* setup of the PCB Editor, a fixed width can also be set by selecting *Autoroute* → *Setup* in SPECCTRA.

### 1.2 9/10/2008 – How to generate PCB artwork in Allegro

#### Copper layers

The *Artwork Control Form* can be opened in PCB Editor by either typing *artwork* at the command line or by selecting the *Manufacture* → *Artwork...* pull-down menu. The *General Parameters* is opened by default. Gerber RS274X should be selected from the *Device type* box for best compatibility with 4pcb.

The *Film Control* tab must be selected next and the *Draw missing pad apertures* check box must be clicked. By default, only the copper layers are shown in the *Available films* box. These films will typically contain *ETCH/[layer]*, *PIN/[layer]* and *VIA CLASS/[layer]*. Pressing the *Create Artwork* button will generate a set of .art files containing the Gerber codes for the selected films.

If there is an error about “reading apertures”, be sure that a Gerber RS274X file is created, otherwise the apertures must be generated. This is done by pressing the *Apertures...* button in the *Film Control* tab and then pressing the edit button next to wheel 1. The *Auto* → is then selected in the newly opened *Edit aperture station* dialog window. Rotation does not matter in this case. Pressing *OK* twice to return back to the *Film Control* dialog should resolve this problem.

It appears that there might be a small rendering bug in the *Artwork Control Form* UI (SPB 15.7) where the check boxes in the *Available films* section cannot be selected and the films cannot be modified. Right clicking the entries doesn't work either. Closing and reopening the dialog seems to fix this problem.

### File locations

All files referenced in the subsequent subsections are generated in the same directory as the board (.brd) file.

### Silkscreen

The *Auto Silkscreen* (*Manufacture* → *Silkscreen...*) tool is used to generate a custom silkscreen layer that can then be rendered using the *Artwork Control Form*. The *Both* radio button should be selected in the *Layer:* and *Elements:* boxes. The only items that should be set to *Silk* in the *Classes and subclasses:* box are *Board geometry*, *Package geometry* and *Reference designator*. All others should be set to *None*. This creates classes *MANUFACTURING/AUTOSILK\_TOP* and *MANUFACTURING/AUTOSILK\_BOTTOM*. A new film must then be created to contain these two classes, one per class, and the artwork must be generated to get the silkscreen Gerber files. It is easier to copy one of the films present by right clicking on the name and selecting copy. This creates an additional film which has only the same layers as the film being copied instead of every possible layer as would be created if a new film was added using the add command.

Sometimes the autosilk procedure fails when text overlays a hole or something similar. The *auto\_silk.log* will display the offending text and its location so the user can relocate it. It is also possible to edit the .dra file for the symbol to relocate the default text placement and then update the symbol by selecting the *Place* → *Update symbols...* pull-down menu in PCB Editor.

An additional problem that may occur is that all of the reference descriptors (*Ref Des*) and component outlines may end up having a null line width. The drawn width of all of the text can be adjusted by entering the *Text Setup* by selecting *Setup* → *Text Sizes....* The *Photo Width* parameter determines the

thickness of all of the text to be drawn. This should fix the silkscreen warnings about text size.

### **Soldermask**

The same procedure using *Artwork Control Form* must be used to generate a pair of artworks for the top and bottom soldermasks. A pair of films must be created (or copied) where the end result contains *BOARD GEOMETRY/SOLDERMAS\_[layer]* and *PIN/SOLDERMASK\_[layer]*.

### **NC drill file**

The drill file is easiest to create using the NC Drill toolbox accessible through the *Manufacturing* → *NC* → *NC Drill...* pull-down menu. The appropriate *Root file name:* can be entered, or kept as is, and then pressing the *Drill* button generates the actual drill file.

### **Checking the files**

As of version SPB 15.7, Orcad shipped a licensed copy of GerbTool with their suite, which is similar to the tools that the PCB manufacturing houses use. It can be started by first starting up *Layout Plus* and then from the *Tools* → *GerbTool* → *New..* pull-down menu. Once GerbTool starts, the individual Gerber layers, and the NC drill file, can be imported and overlaid to verify that all of the necessary features are there.