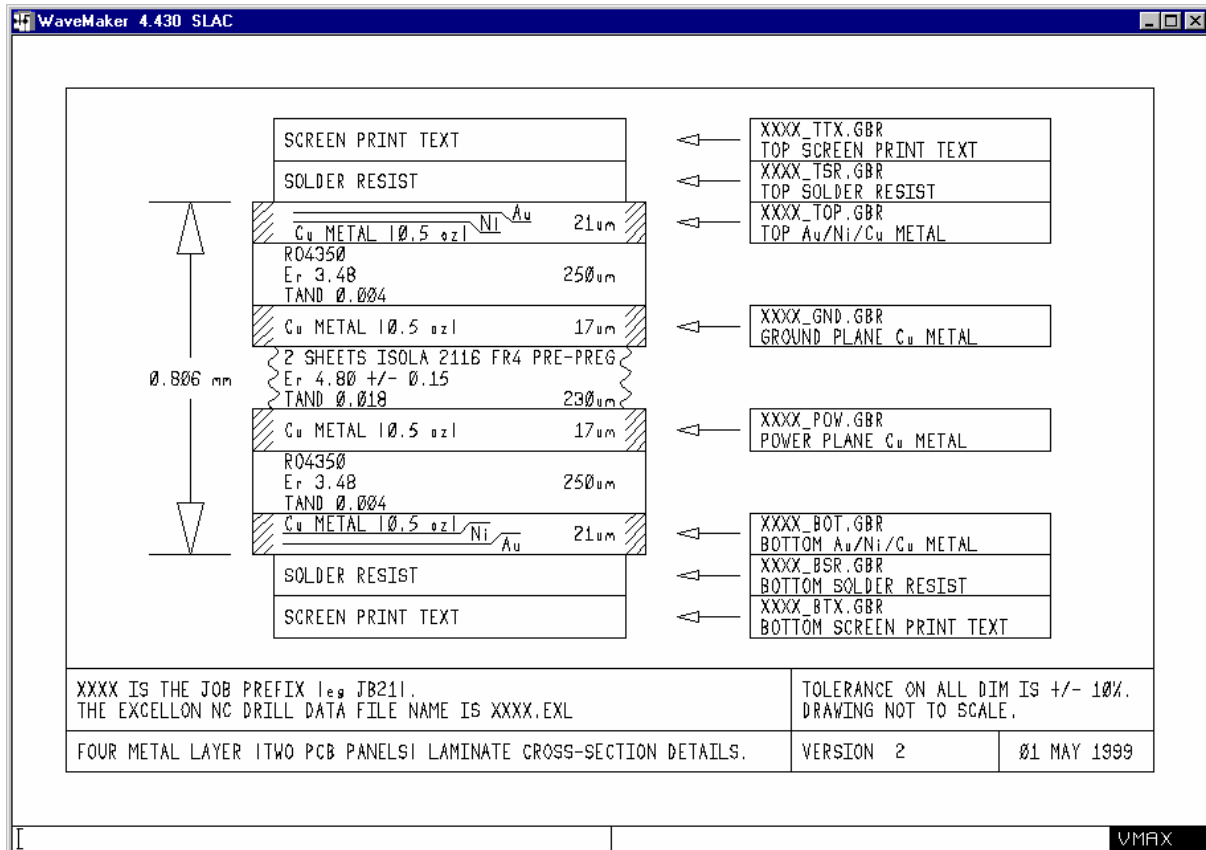


## Application Note 2: Definition of a four layer PCB.

In this Application Note, we discuss the steps we have taken to produce a four layer PCB using Roger RO4350 microwave PCB materials.

This document is saved as appnote2.doc.

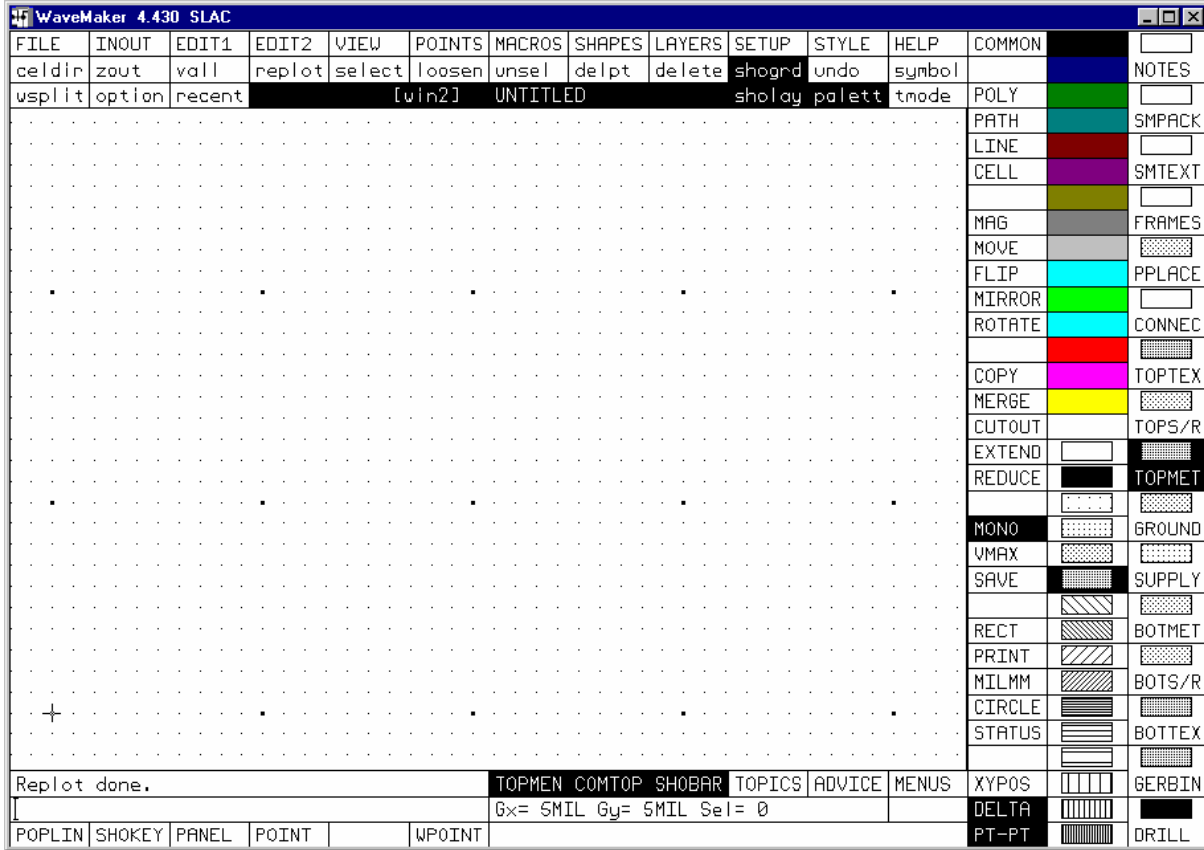


The cross section of the multi-layer PCB assembly is as shown in the above diagram. In order to get a reasonably high definition for the copper metal layers, we have used “0.5 ounce rolled copper” as opposed to 1.0 ounce per square foot of rolled copper. The thickness of the copper layer is worked out from this information as follows:

Density of copper is 8.94 grams / cc.  
Recall that there are 454 grams per pound.

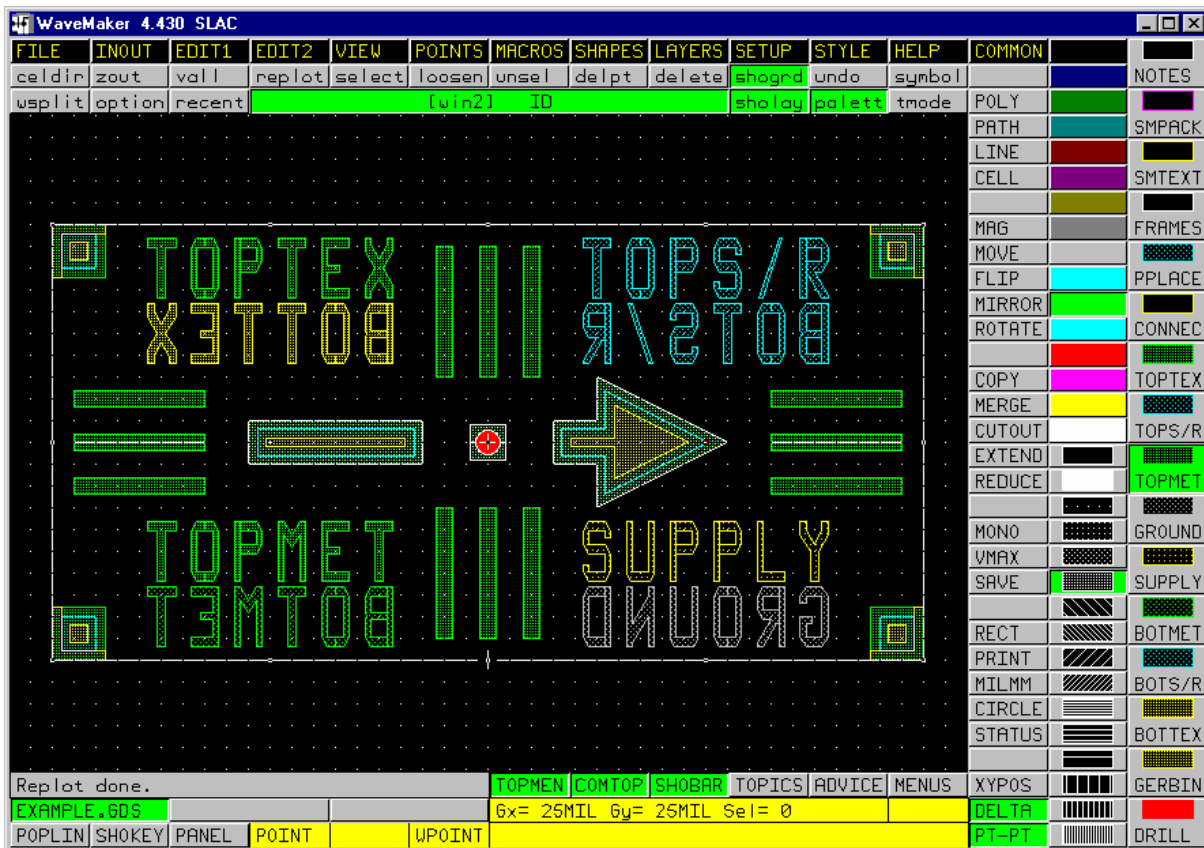
$$\text{Thickness in } \mu\text{m} = \frac{0.5 * 10000}{2.54 * 2.54 * 16 * 144} * \frac{454}{8.94} = 17 \mu\text{m}$$

Two layers of an FR4 type of “Pre-Preg” are used between the two layers of Rogers RO4350. Each layer of “Pre-Preg” is about 115 um thick (pressed thickness), and each layer of Rogers material is 250 um thick. We thus estimate the laminate thickness, not including the thickness of any solder resist and lettering, to be 0.806 mm, as shown in the cross section drawing.



To set the design environment, use the EXEC command to run the PCB.CMD command file (which is supplied in the wamfiles.exe compressed archive), which will set your system to look as portrayed in the screen image shown above.

As a reference, you might like to include the contents of the cell called ID (standing for identification). This cell contains a simple layout plus text for each of the layers you might use in the four layer PCB design. This cell is contained in the example.gds file as supplied in the wamfiles.exe compressed archive.



Since many of the electronic component packages and the dimensions of the package legs are specified in Imperial units, it makes good sense to operate using thousandths of an inch, or “mils” as they are called in the USA.

Our suggestion is that you create and operate in a new directory for each PCB design. On completion of a design, you should create a copy of the as-supplied nctools.dat file, and then edit it if need to to read:

```
@GERBER
HEAD      NULL
TAIL      NULL
RESIZE    0.0    mil
FORMAT    ANSI
OUTLINE   ONLY
XYUNITS   0.1    mil
! 1.0 mil is 5.3 format
! 0.1 mil is 5.4 format
! Have space between number & units.
!      name      size      overlap
TOOL     G54D10   1    mil    0.0    mil
TOOL     G54D11   6    mil    0.0    mil
TOOL     G54D12   8    mil    0.0    mil
TOOL     G54D13  10   mil    0.0    mil
```

```
TOOL      G54D14      15  mil      0.0  mil
TOOL      G54D15      20  mil      0.0  mil
TOOL      G54D16      30  mil      0.0  mil
TOOL      G54D17      50  mil      0.0  mil
TOOL      G54D18      75  mil      0.0  mil
@END
```

```
@EXCELL
XYUNITS   INCH   2.4  LZ
@END
```

The important thing is that two of the lines should read:

```
OUTLINE   ONLY
XYUNITS   0.1   mil
```

Then you can tell the PCB house that you are using Gerber in outline only, 5.4 format. For the first run, we suggest you ask the PCB manufacturer to fax you a copy of the artwork they plan to use prior to actual use to enable you to check for errors or misunderstandings. Sadly, most people make some sort of mistake, or there is some form of misunderstanding, before the first PCB is correctly made, so the more checking you can perform, the better!

You should also define a standard specification for your PCB manufacturer in which you as precisely as possible define the PCB process parameters. An example of such a definition (whose use does not necessarily guarantee first time success) is as shown in the next few pages.

We make use of the PCB manufacturing capabilities of Cambridge Circuit Company Limited. However, this does not represent an endorsement by us of their suitability to handle your work. Simply for your interest, their details are as follows:

Cambridge Circuit Company Limited

4 Winship Road, Milton, Cambridge CB4 6BQ

contact person: Phil Sanford  
TEL: 01223 425100  
FAX: 01223 425001

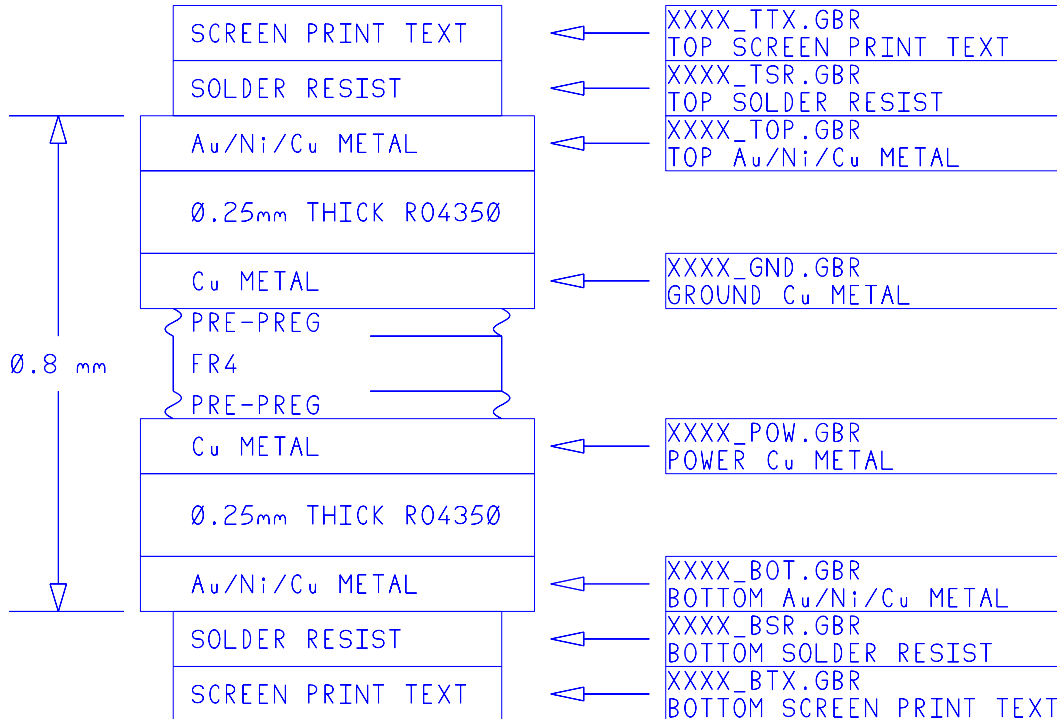
ATTN: Philip Sanford at FAX# 01223 425001  
 FROM: Joseph Barnard  
 DATE: 16 APR 99  
 PAGE: 1 of 10

Reference and job prefix: **JB27**

Job contact person: **Joseph BARNARD**

Delivery address: Barnard Microsystems Limited  
 134 Crouch Hill,  
 London N8 9DX

TEL: 0181 341 0566  
 FAX: 0181 341 0583



XXXX IS THE JOB PREFIX.  
 THE EXCELLON NC DRILL DATA FILE NAME IS XXXX.EXL

**// more on next page 2 →**

PAGE 2

Depending on the nature of the design, not all of the data files may be used. The files to be used are indicated with a .

**XXXX\_TTX.GBR** "TOPTX" white screen print for top  
**XXXX\_TSR.GBR**  "TOPS/R" solder resist pattern on top  
**XXXX\_TOP.GBR**  "TOPMET" copper pattern on TOP  
  
**XXXX\_GND.GBR**  "GROUND" copper on BOTTOM of upper panel  
  
**XXXX\_POW.GBR**  "SUPPLY" copper on TOP of lower panel  
  
**XXXX\_BOT.GBR**  "BOTMET" copper on BOTTOM of lower panel  
**XXXX\_BSR.GBR** "BOTS/R" solder resist on bottom  
**XXXX\_BTX.GBR** "BOTTEX" white screen print for bottom  
  
**XXXX.EXL**  Excellon NC drill data file

The TOP and BOTTOM surfaces of the UPPER and LOWER 0.25 mm thick RO 4350 panels are to be patterned.

The GERBER data in the above file is based on the use of:

- **M5.4** format for position
- **1 MIL diameter ROUND tool** has been used to define the boundaries of all the polygons.

substrate is: 0.25 mm thick Rogers RO 4350, metallised both sides with 0.5 OUNCE copper.

The overall pattern size is: X = 2,550 MIL  
Y = 1,400 MIL

The space between the PCB rows is: 200 mil (approx)  
The space between the PCB columns is: 200 mil (approx)

Please array the pattern out in: 4 columns  
Xtotal = 10.2 inches

by: 10 rows  
Ytotal = 14.0 inches

PLEASE USE HIGH PRECISION LITHOGRAPHY.

Copper line width control is essential. Please ensure that the lithography is as repeatable as possible.

// more on next page 3 →

**PAGE 3**

Please laminate together two 0.25 mm thick ROGERS RO 4350 panels (as supplied) with a combination of PRE-PREG and FR4 board in the middle to ensure a laminate thickness including copper layers as close to 0.800 mm as possible.

The holes defined in the Excellon format NC drill data file must **all be plated through holes.** The Excellon reference hole has a diameter of 28 MIL. **SEVERAL DIFFERENT DIAMETER HOLES ARE USED.**

+++ NOTE +++ Please try to maintain as accurate as possible the alignment of the plated through via holes with the metallisation patterns on the top and bottom of the laminate.

Note on units used: 1 MIL = 1 thousandth of an INCH.

Please first apply a 3 to 6 micron thick electroless nickel layer to the copper, and then deposit a 1 to 2 micron thick layer of electrolytic GOLD on top of the nickel layer. **USE QUALITY PLATING.**

+++ NOTE +++ Please emphasise the need for a **good quality** gold layer, since we have been experiencing problems recently with the soldering of surface mount components to the gold layer.

Where we use solder resist, please ensure that all the copper patterns on the top and bottom of the laminate are Au/Ni plated **before** any solder resist is applied. This procedure is absolutely essential for reliability reasons.

Plots of the expected layouts are as attached.

Turn around time from above date: ASAP, **but** consistent with the application of **VERY GOOD QUALITY GOLD PLATING LAYER**

Method of shipment: Courier

The data has been emailed to: Phil Router at PhotoData.  
Phil Router to forward all data to Phil Sanford at Cambridge Circuits (please).

Work submission co-ordinated by: Dr Joseph Barnard at BML.

**Please send a quotation to Joseph BARNARD at BML as soon as possible.**

Kindest regards,