

How-To Document

Procedure Description

Generating of extended Gerber- and Excellon data with Protel EDA/Client (version 4.0.16) for CircuitCAM 3.x and 4.x

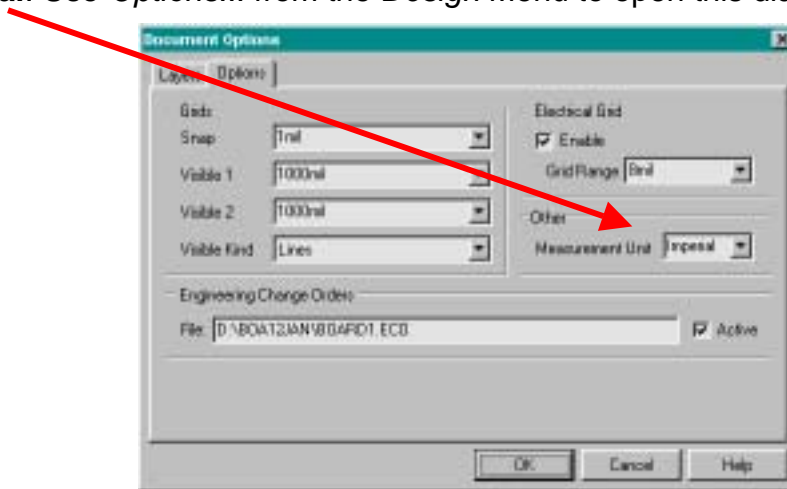
Requirements

Protel EDA software
 CircuitCAM software
 Completed PCB design

Problem / Procedure Solution

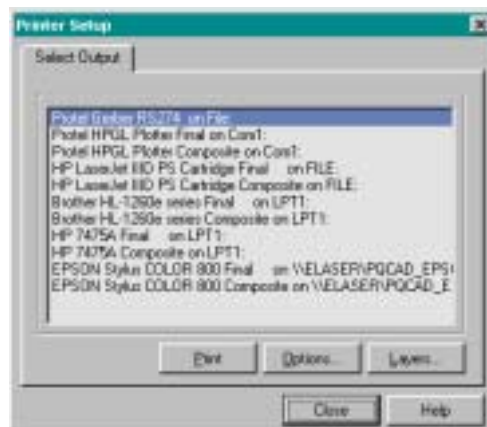
1. Load the finished designed board.

1.a Set the parameter *Measurement Unit* of the property sheet *Options* to **Imperial**. Use *Options...* from the *Design* Menu to open this dialog box.



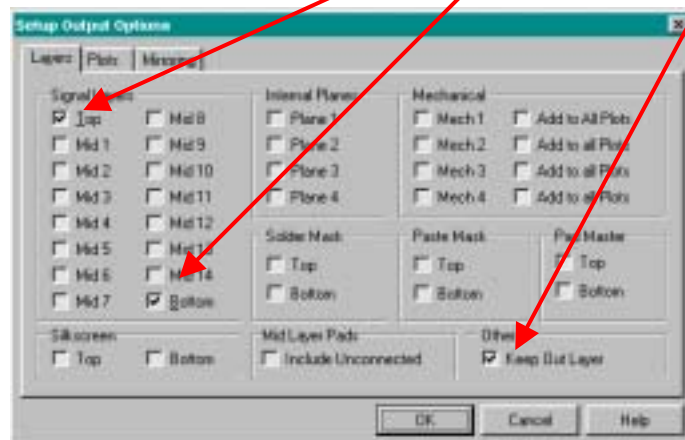
A. Extended Gerber Data

2. Open the *Printer Setup* dialog box by clicking *Setup Printer...* from the *File* Menu. Select the entry *Protel Gerber RS274 on File* from the listing:

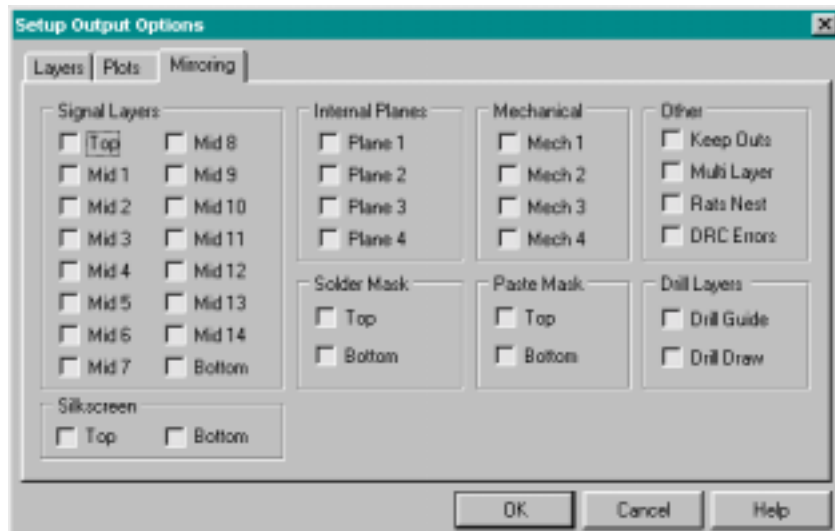
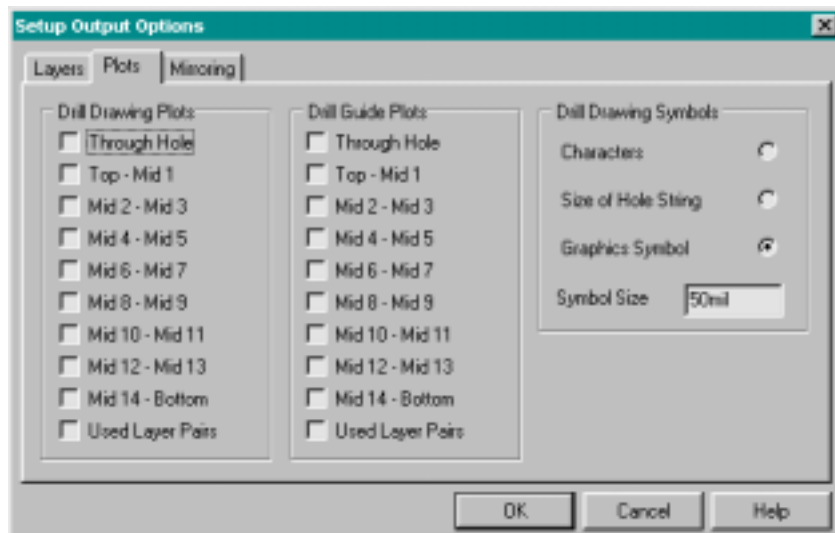


How-To-Document

- Click on the *Layers* tab, to choose the necessary layer. Check the layers shown in the screen shot below and only those: **Top**, **Bottom** and **Keep Out Layer**.

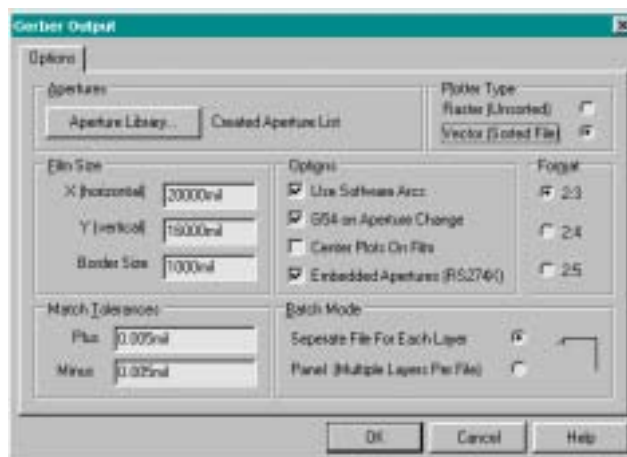


- Disable all check boxes on the property sheet *Plots* and *Mirroring* as shown below.

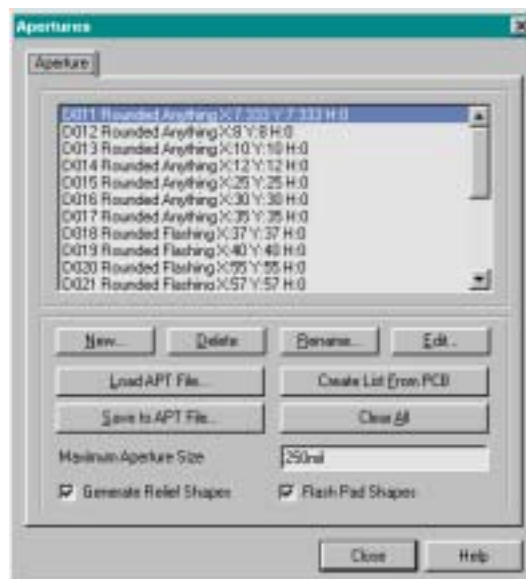


How-To-Document

- Click on the *OK* button, and then click on the *Options* button in the *Printer Setup* dialog box.
- Check the parameters of the *Gerber Output* dialog box, as shown in the screen shot below.
Very important are:
 under Options: Use Software Arcs enabled
 G54 on Aperture Change enabled
 Center Plot On Film disabled
 Embedded Apertures (RS274X) enabled
 under Format: 2.3 enabled
 Under Batch mode: Separate File For Each Layer enabled



- To define the used apertures click to the push button *Aperture Library*. Set all parameters of the *Apertures* dialog box as shown in the screen shot below, and click to *Create List From PCB*. All apertures used in the current layout will be detected and listed. They are available for exporting the data.

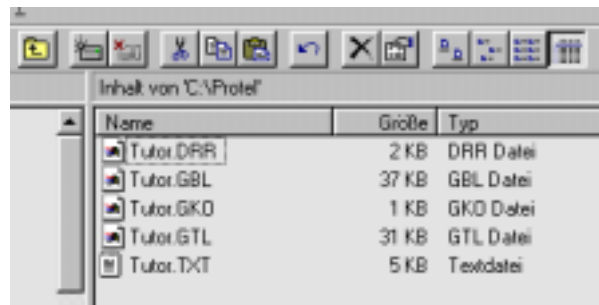


8. Close the *Apertures* dialog box by clicking *Close*. Close the *Gerber Output* dialog box by clicking *OK*.
9. Click the push button *Print* in the *Printer Setup* dialog box. The *Save Gerber* dialog box appears. Click on *Save*, to start the export of the Gerber files. The program creates 3 extended Gerber files automatically and writes them to the directory of the opened layout:

name.GTL	Gerber data of the component side (Top Layer)
name.GBL	Gerber data of the solder side (Bottom Layer)
name.GKO	Gerber data of the board frame (Keep Out Layer)

B. Excellon drill data:

10. Start the Excellon output by clicking of *NC Drill* in the *Reports* menu. The created files will be saved in the directory of the opened design. After successful export of the data, a report window appears which shows some information about the created data. This information is also stored as an ASCII file **name.DRR**. This file includes information about the used tools and if so, information about processed holes. In addition, a file with the name **name.DRL** is created. This is a binary file, which is not necessary for the further work with CircuitCAM. The file **name.txt**, includes the original Excellon drill data with the drilling coordinates. This file and the file **name.DRR**, will be imported into CircuitCAM.
11. The export of the production data from Protel EDA/Client is completed now.



All 5 generated files (**name.GTL**, **.GBL**, **.GKO**, **.DRR** and **.TXT**) will be imported into CircuitCAM now. After importing the files the Gerber and Excellon data will not match on the screen. This happens because of Protel EDA/Client uses different data origins for Gerber and Excellon data. The data can easily be matched as explained in exercise 2 of the CircuitCAM Tutorial.

References

CircuitCAM manual
Protel reference manual

Author

Craig Kniskern