How-To Document

Procedure Description

Generating extended Gerber- and Excellon data from OrCAD 9.00 for LPKF CircuitCAM 3.x and 4.x

Requirements

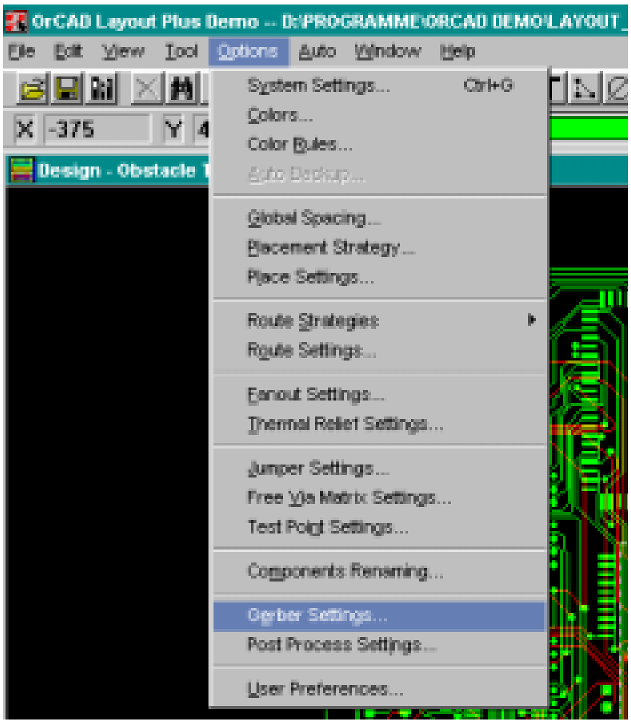
OrCAD Software
CircuitCAM Software
Completed PCB in OrCAD with all necessary default layers.

Problem / Procedure Solution

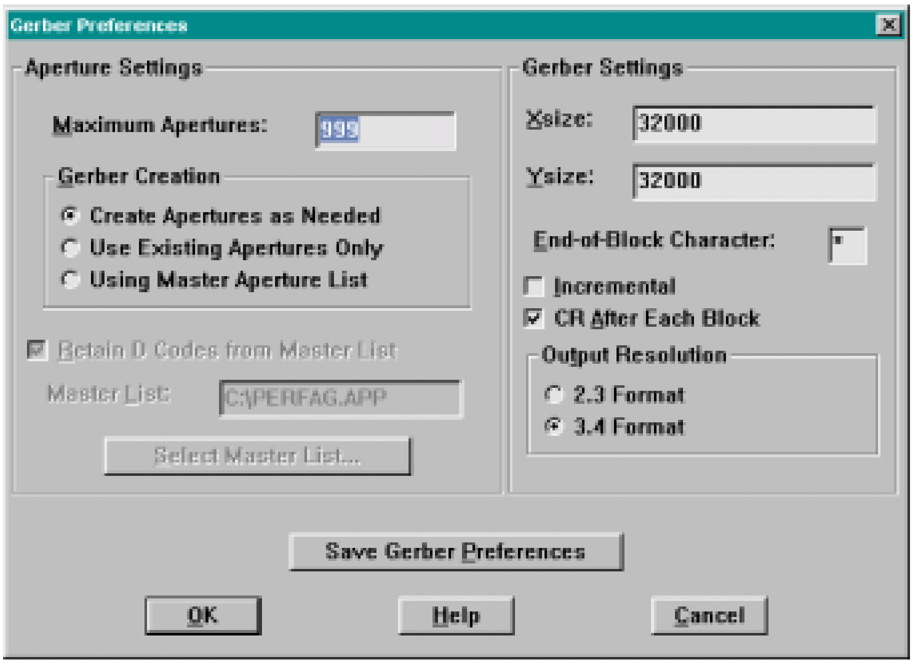
OrCAD Export:

Please follow instructions below:

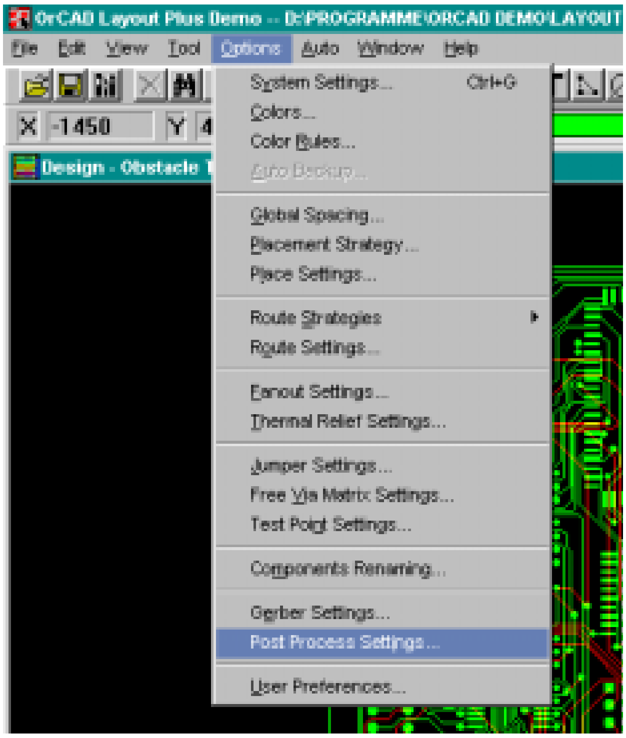
1. Start the OrCAD PCB layout program
2. Open the completed PCB layout
3. Open the **Gerber Settings** Dialog Box from the Options Menu:



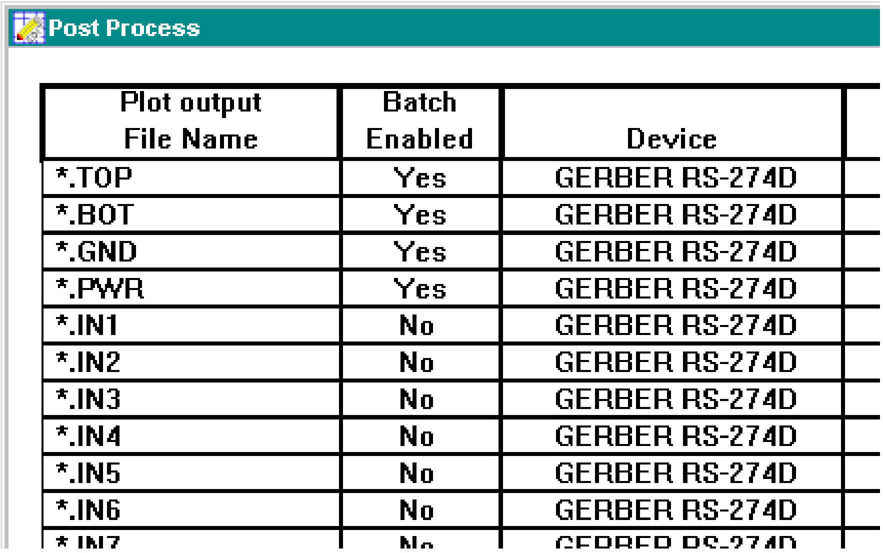
4. Set all parameters as listed in the graphic below and save the settings by clicking the button **Save Gerber Preferences**.



5. Open the **Post Process Settings** Dialog Box from the Options Menu:



Below listing appears on the screen:



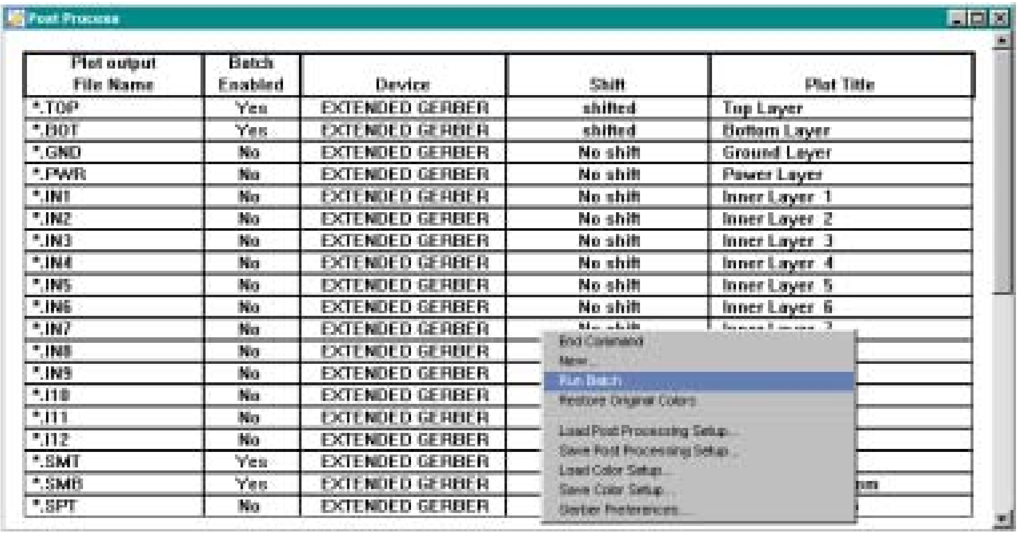
Post Process

Plot output File Name	Batch Enabled	Device
*.TOP	Yes	GERBER RS-274D
*.BOT	Yes	GERBER RS-274D
*.GND	Yes	GERBER RS-274D
*.PWR	Yes	GERBER RS-274D
*.IN1	No	GERBER RS-274D
*.IN2	No	GERBER RS-274D
*.IN3	No	GERBER RS-274D
*.IN4	No	GERBER RS-274D
*.IN5	No	GERBER RS-274D
*.IN6	No	GERBER RS-274D

This list shows each single layer, which is enabled for export by the Post Processing Batch Job. In addition the data format and the file name to be created are shown. By clicking on the first column of a line, the parameters (or entries) can be modified.

5. Double-Click on the first line, and choose ‘Extended Gerber’ in the dialog box that shows up. This will set all the outputs to Gerber274X output, with embedded apertures.

7. Click on OK, and you should now have a setup looking like the image below. Click with the right mouse button into the still open listing, and select **Run Batch** from the Pop Up menu. Confirm all messages about generation of several files.



By running the batch job, all files necessary to produce the PCB prototype with the LPKF circuit board plotter, will be created.

These are the files (**name** stands for the *name* of the file, which may vary from PCB to PCB):

name.TOP	extended Gerber file of the component side
name.BOT	extended Gerber file of the solder side
THRUHOLE.TAP	Excellon drill file

All other files generated are not needed.

The export from OrCAD has been done. The files will now be used to continue with LPKF CircuitCAM.

CircuitCAM Import:

1. Start CircuitCAM.
2. Choose the template **Default.CAT** from the **File New** menu
3. Click the **Import** icon (**Front-To-End** tool bar) or choose **Import** from the **File** menu
4. Select the above-mentioned files from the working directory and click the OK button. Import the files and place them into the appropriate layers. If you would like to *Automatically Import* your files, refer to the Import Assignment How-to document.

All selected files will be imported and appear graphically on the screen.

For all the following steps use the various exercises of the CircuitCAM manual.

Tip and Tricks:

You should define the PCB outline in your design as a continuous line around the whole board (Obstacle Type **Free Track**). This should be done on the topside and the bottom side of you PCB but at least on one of them. This board frame may be used as limitation for removing the residual copper of your PCB (Rubout).

References

OrCAD Manual
CircuitCAM Manual

Author

Craig Kniskern