

How-To Document

Procedure Description

Generating of extended Gerber- and Excellon data with PCAD for CircuitCAM 4.x and 3.x

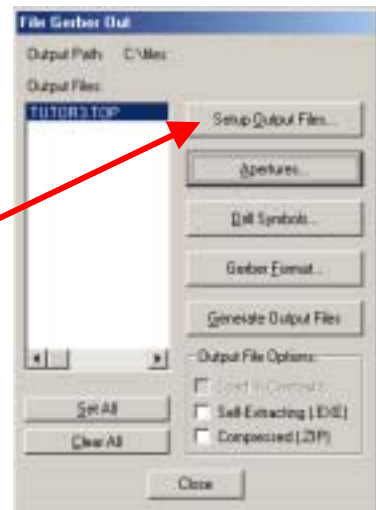
Requirements

- PCAD Accel CAD software
- CircuitCAM software
- Completed PCAD PCB

Problem / Procedure Solution

You will need a completed design in Accel PCB to work with. This example uses the Tutor3 board in the Accel EDA tutorial.

With the completed design loaded in Accel EDA select **File/Export/Gerber**. The Gerber files are made by stepping through the click boxes on the right side of the Gerber Output window. Click first on **Setup Output Files**.



All the layers of the completed PCB design are listed in the center window marked **Layers:**. CircuitCAM only needs the Top, Bottom, and the Board Outline trace layers.

From the **Layers:** list click on **Top**. In the upper right of the **Output File Selections** window is the **File Extension** text box. Enter the letters 'TOP' for the extension with no period before the letters. Leave the **X and Y offsets** at zero.

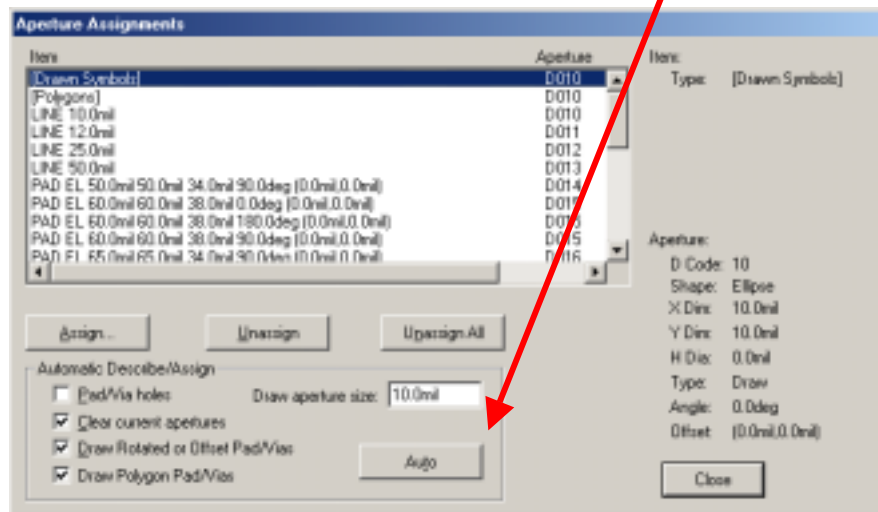


Click on the select boxes for **Pads** and **Vias** only: leave the other boxes unchecked. At the bottom of the window in the **Output Path..** select a folder (or directory) into which the generated Gerber file will get placed. Then click on the Add button located in the left center of the window.

Close this window with the button in the lower right corner.

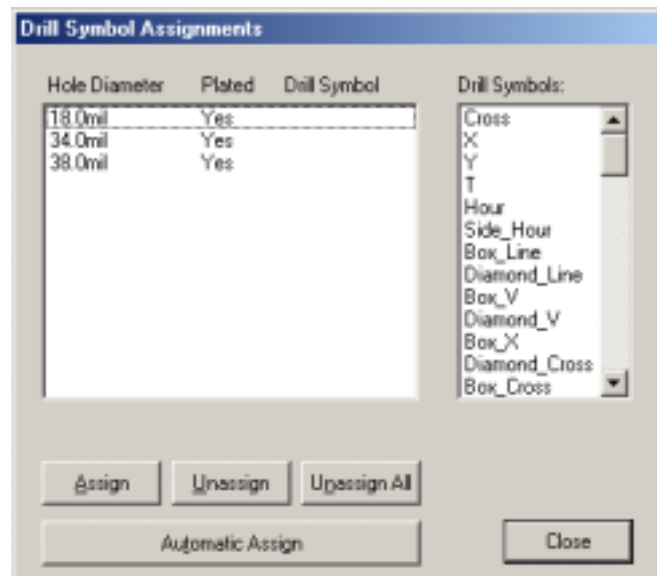
Closing each step of the Gerber File process in Accel EDA returns you to the **File:Gerber Out** window.

Click on **Apertures..** which is the second switch from the top. It is rarely necessary to alter the aperture list from the defaults that are set up by the PCB design program. You would only do this if you are using a standard aperture list for all of the boards that are being made for a project or throughout the company. For this example, we assume that the aperture listing made by Accel EDA is OK. Therefore, all you need to do on this window, is click on the **Auto Button** in the lower center and then click on the close button in the lower right.



The next switch on the **File:Gerber Out** list is the drill symbols.

This produces what is called a **Drill Guide Layer**. This layer will not be used in the CircuitCAM processing, so we may skip this output and go on to the next step.

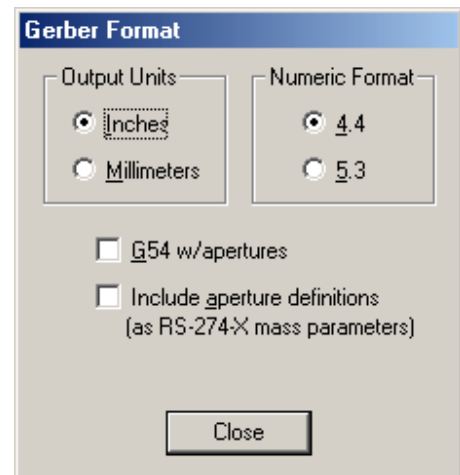


The last step in setting up each Gerber file, is to set the format for the **X-Y coordinate numbers**. To save memory space, the decimal point was removed from all of the numbers in the Gerber format. The decimal must be reinserted into its correct location in order for the board image to be reproduced with the right scale. The numeric format, or **digits m.n**, in CircuitCAM, sets the decimal position from the beginning, or from the end of the number sequence of the X-Y

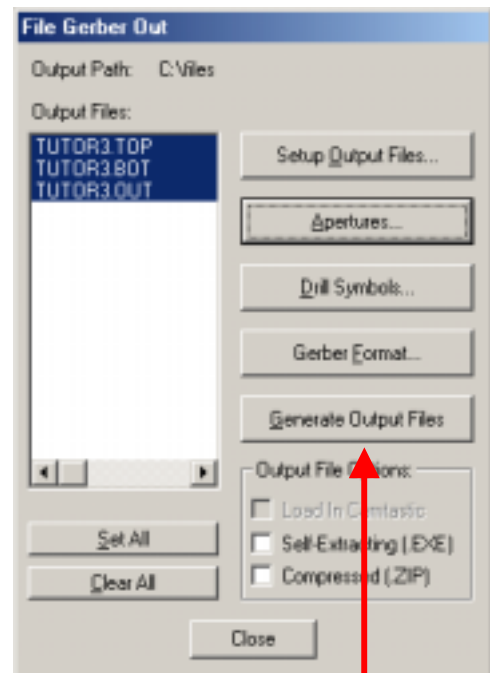
How-To-Document

coordinates for the circuit elements. The lower switches instruct the PCB program to include the circuit element sizes and shapes in a header at the beginning of the Gerber file, in what is known as '**Embedded Apertures**'.

Set the **Output Units** to inches, the **Numeric Format** to 4.4, and the **G54** and **RS-274X** switches on. Then click on **Close** to return to the **File:Gerber Out** window.



The set-up for **Top trace layer** is now finished. The same four steps (**Select layer, Apertures, Drill symbols, and Gerber Format**) need to be done now, for the **bottom trace layer** and the **board outline** of the PCB design. This will place three filenames in the **Output Files** box. Make sure that each filename has a different extension (i.e. TOP, BOT,OUT).



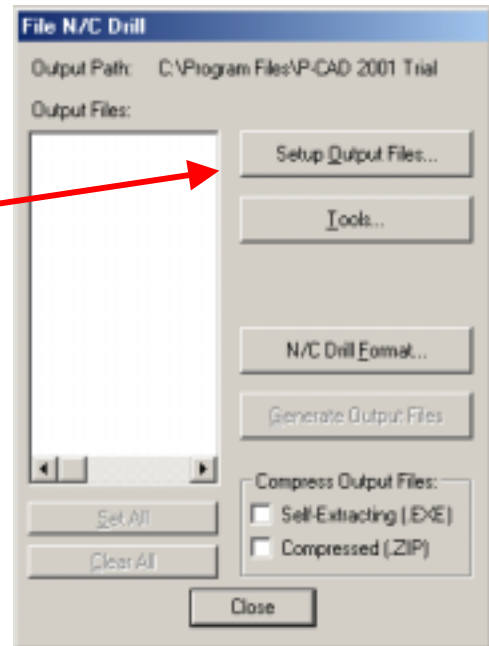
To actually create the files, click on the button marked **Generate Output Files**. Gerber files are all written in ASCII characters and can be viewed with any text editor, such as NotePad or UltraEdit.

Creating the Excellon Drill File from a PCB design in Accel EDA

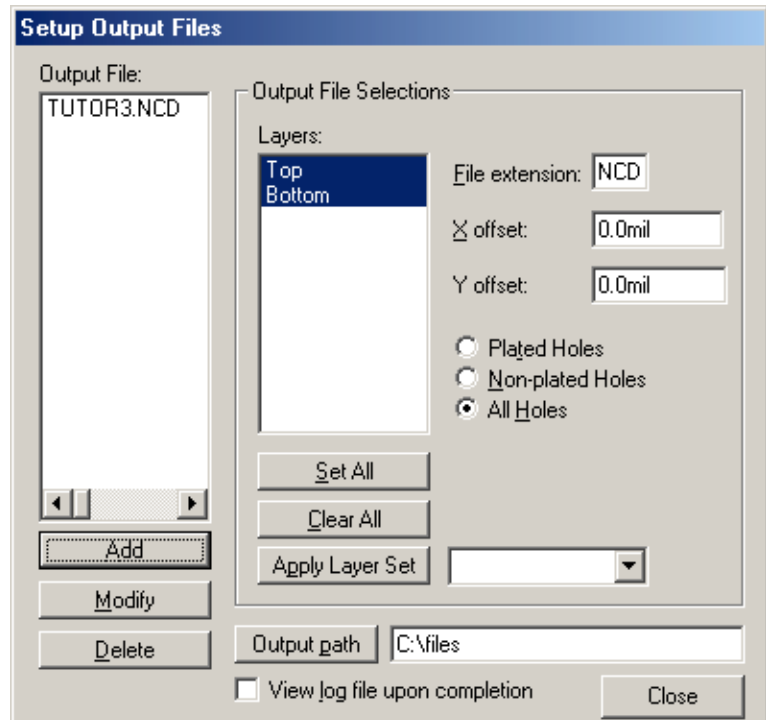
The Gerber file set contains the information for reproducing the image of the board traces and outline but these files do not have the locations and sizes of the drill holes. LPKF's CircuitCAM program needs the drill pattern information in an NCD (Numerically Controlled Device) file. This format (also called Excellon after the company that created it), is much less complex than Gerber. In Accel EDA, making an NCD file is a separate process from making Gerbers.

How-To-Document

Select **File:NC Drill** and then click on **Setup Output Files**.

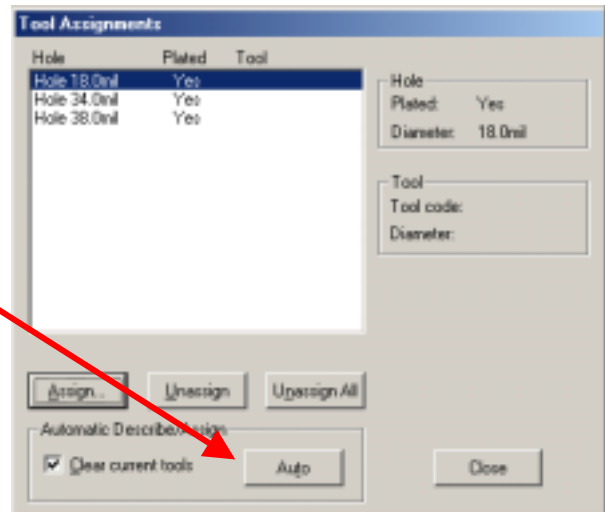
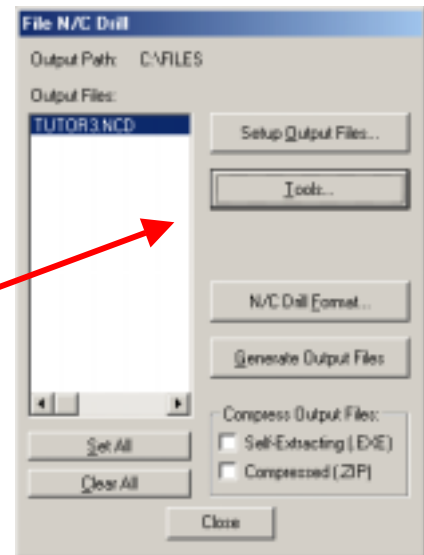


Click on **Set All**. Enter a **File extension** of **NCD**. Select the **All Holes** switch. Leave the **X and Y offsets** at 0.0. Set the folder into which the file will go in **Output Path**. Finally click on **Add**. Click on **Close** to leave this window.

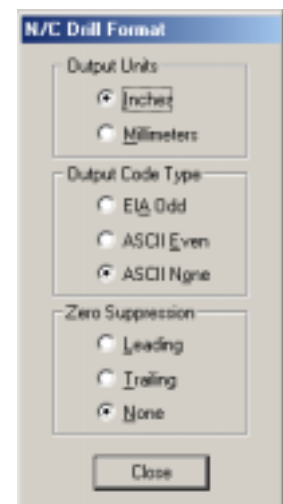


How-To-Document

From the **File:NC Drill** dialog window, click on **Tools**. Click on the **Auto** button and then **Close**. Back in the **File:NC Drill** window, click on NC Drill Format.



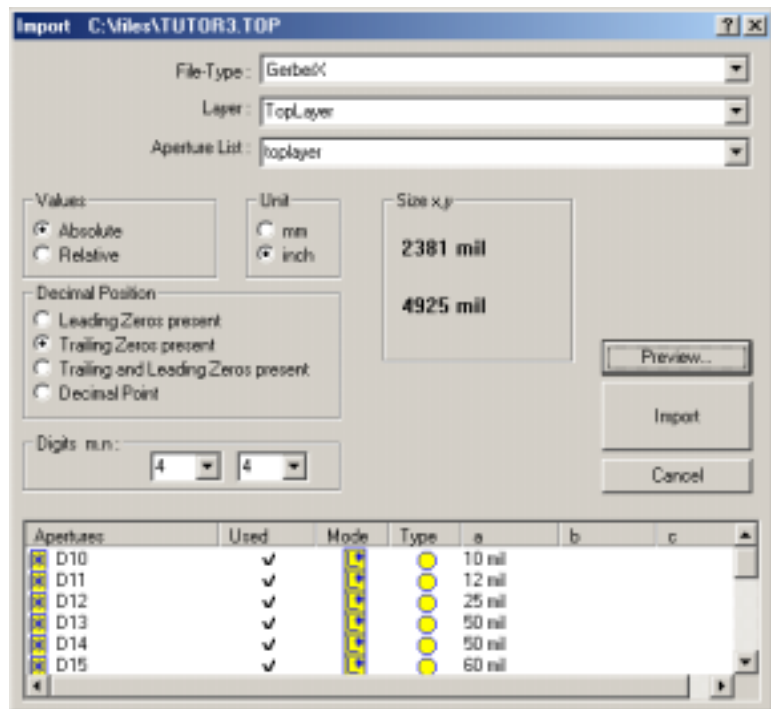
The default options are all good for CircuitCAM. *ASCII None* means that there is no parity bit added to each character in the output file. *Zero Suppression None* means that all of the X-Y coordinates for the drill holes will have the same number of digits. Accel NCD defaults to 2.4 decimal placement. Click on **Close**.



In the **File N/C Drill** dialog window, Click **Generate Output Files** and **Close**.

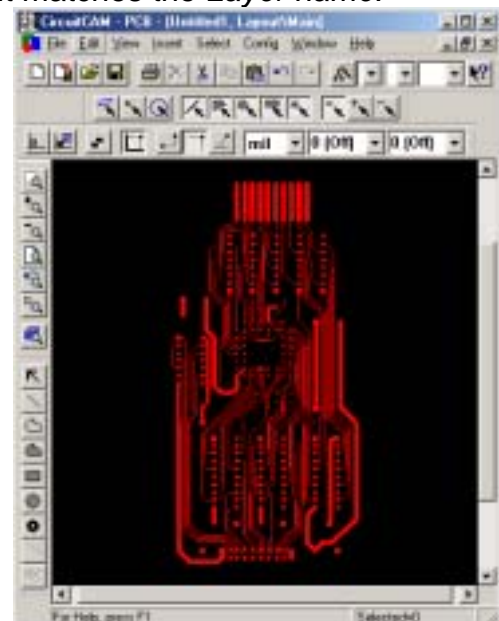
Loading Accel Gerber and Excellon files into CircuitCAM

Load CircuitCAM and select **File: Import**. Select the Gerber for the Top layer from the Open list. The Import window as shown above appears. The file has not been imported into CCAM yet: only the parameters for recreating the PCB image from the Gerber file are loaded for your verification.



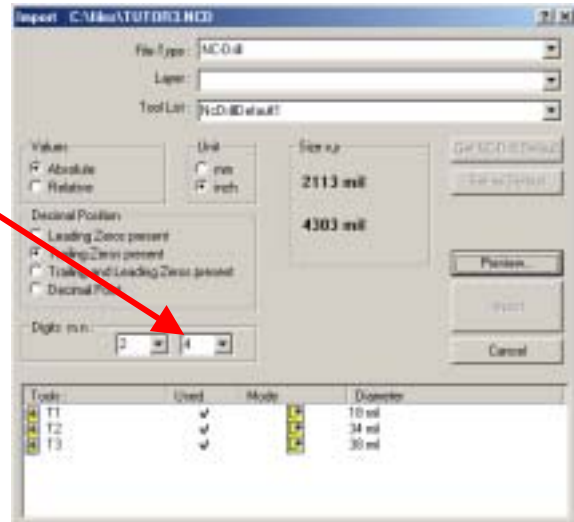
CircuitCAM selects GerberX as the File-Type by opening the file and examining its structure. The apertures are read into a 'format configuration', which is an internal structure to CircuitCAM, of the data needed, to make the image of the PCB. This *format configuration* will have a name that you enter in the **Aperture List** text box (third from the top). The other information, such as *measure units* and *decimal position*, is in the header of the Gerber file. Verify that the size of the board is correct. Also ensure that the *Layer name* is applicable to the file that you are importing. You can check the *title bar* at the very top of the window, for the *extension* of the Gerber file, to make sure that it matches the *Layer name*.

When all looks OK, click on **Import** to read the Gerber file, and create the board image on the CircuitCAM main window.





The other two Gerber files for Bottom and Board layers are loaded in the same manner. Be sure to select the correct layer in CircuitCAM for each file. The drill layer gets imported the same way as the Gerber layers with one exception. Accel EDA defaults to 2.4 as the decimal position for NCD files and Circuit CAM defaults to 2.3.

On the *File Import window*, change the **Digits m.n** value of the right text box from **3** to **4**. Now click on **Import**. This will bring in the *NCD (Excellon)* file of the drill holes into CircuitCAM correctly scaled.



Load the NCD file into the **DrillUnplated** layer. Do not use the **DrillPlated** layer unless you will be making the PCB with LPKF's *AutoContact* system for plating the drill holes. Having data in the *DrillPlated* layer, will force the creation of BoardMaster phases that are *specifically designed for AutoContact use only*. If you are using the *LPKF MiniContac Electroplating* system for thru-hole connection, then also *place all NCD drill data* into the **DrillUnplated** layer.

The Accel demo will overlay the drill layer directly into correct position onto the Gerber's drill pads. However, it sometimes happens that there will be an offset between the drill pattern and the Gerber pattern. This results from the user's having set a different zero reference point in the PCB layout program from the PCB program's internal zero reference point. The Gerber's get written from the user's reference point and the NCD file gets referenced from the other internal default point.

This is easy to fix in CCAM. First click on a pad for which you know the correct dot in the drill pattern. Then click on the Make Zero Point icon.  Now click the drill hole dot and click the Move Layer icon.  This will shift the entire drill pattern into to the correct position over the pads.

References

CircuitCAM manual
 Accel PCAD manual
 LPKF Technical Support

Author

Craig Kniskern