Fermi National Accelerator Laboratory

FERMILAB-TM-2022

Preparing Printed Circuit Boards for Rapid Turn-Around Time on a Protomat Plotter

James Hawtree

Fermi National Accelerator Laboratory
P.O. Box 500, Batavia, Illinois 60510

January 1998

Operated by Universities Research Association Inc. under Contract No. DE-AC02-76CH03000 with the United States Department of Energy

Reference Number: **TM-2022**

Preparing Printed Circuit Boards for Rapid Turn-Around Time on a Protomat Plotter

Author, Contact: James Hawtree
E Mail: raph@fnal.fnal.gov
MS 121
Ext. 4287

13 January 1998

Fermi National Accelerator Laboratory
P.O. Box 500
Batavia, Illinois 60510

Acknowledgements and Thanks to: Bruce Merkel and David Kindelberger

I. Introduction

This document describes the use of the LPKF ProtoMat mill/drill unit circuit board Plotter, with the associated CAD/CAM software BoardMaster and CircuitCAM. At present its primarily use here at Fermilab's Particle Physics Department is for rapid-turnover of prototype PCBs double-sided and single-sided copper clad printed circuit boards (PCBs). (The plotter is also capable of producing gravure films and engraving aluminum or plastic although we have not used it for this.) It has the capability of making traces 0.004 inch wide with 0.004 inch spacings which is appropriate for high density surface mount circuits as well as other through-mounted discrete and integrated components.

One of the primary benefits of the plotter is the capability to produce double-sided drilled boards from CAD files in a few hours. However to achieve this rapid turn-around time, some care must be taken in preparing the files. This document describes how to optimize the process of PCB fabrication. With proper preparation, researchers can often have a completed circuit board in a day's time instead of a week or two wait with usual procedures. It is assumed that the software and hardware are properly installed and that the machinist is acquainted with the Win95 operating system and the basics of the associated software. This paper does not describe its use with pen plotters, lasers or rubouts.

The process of creating a PCB (printed circuit board) begins with the CAD (computer-aided design) software, usually PCAD or VeriBest. These files are then moved to CAM (computer-aided machining) where they are edited and converted to put them into the proper format for running on the ProtoMat plotter. The plotter then performs the actual machining of the board. This document concentrates on the LPKF programs CircuitCam BASIS and BoardMaster for the CAM software. These programs run on a Windows 95 platform to run an LPKF ProtoMat 93s plotter.

Note: menu and window items will generally be in **boldface** type; the vertical bar "|" will denote a sub-menu item. <u>Underlined</u> names will generally denote recent user keyboard input. *Italicized* names will generally refer to suggested items that can be chosen from a list but that can be edited (although sometimes they may appear later in boldface). Important notes may be both in ***boldface and italics.***

II. Creating the CAD files

Great care must be taken to provide the proper files for plotting. ***Different files are needed for ProtoMat plotter fabrication than for photoplotting***. For photoplotting a graphic file is needed to mask the areas copper is left behind after chemicals are applied to the circuit board. Then another graphic file will provide the graphics for the silkscreen to apply ink to the board at a later step.

By contrast, plotter files must provide information that allows toolpaths to be calculated and must account for several elements that are critical to the CAM process. The CAM software used by the LPKF ProtoMat plotter is called CCAM, or CircuitCAM. The CAM software calculates the tool paths for the plotter. These CAM files must account for tool width, clearance between traces, tool rotation direction (tooling cuts more efficiently when moved in arcs that are opposite in direction to the tool rotation), path length and several other parameters. This means that the CAM software must handle problems that do not occur in photoprocessing due to the non-trivial diameter of the tooling size. The CAD files must not place elements too close to each other because the tool must pass between the graphic elements. If there is not enough space for a tool to separate the pads or traces the CCAD software can force a path between the traces which will result in a smaller pad.

The CAD files must also avoid bitmaps or elements that may appear to be simple geometric figures but are actually composed of multiple lines as fill. Although these can be run successfully, the CAD software will have to compute separate paths for each fill element. This can increase machining time by ten or more times and result in a poorer finished product.

Since there is no silkscreen used with the plotter, silkscreen graphics can be milled away on the copper. The CAD files must not put the graphics or other artwork over traces that would cut the traces. And, the text and other artwork must be placed in files separate from the files that describe the actual traces and pads that conduct current. Likewise the files for both the drilling and board outlines must be on separate layers. The files needed are:

1) *Drill file;*

2) *Component-side (or "top") flashes and traces that conduct current;*

3) *Component-side artwork (that does not conduct current) which may include:*

    a) text;

    b) lines, boxes, etc. to separate grounds, planes, etc.;

    c) artwork, logos, etc.;

d) thermal artwork;

4) *Solder-side (or "bottom") flashes and traces (current-conducting);*

5) *Solder-side artwork (that does not conduct current) which may include:*

a) text;

b) lines, boxes, etc. to separate grounds, planes, etc.;

c) artwork, logos, etc.;

d) thermal artwork;

6) *Board outline (which may include slots, voids, large or odd-shaped holes that are removed with a router).*

Please note the following guidelines about these six types of CAD files:

- *Board outline and Drill file are essentially the same as those provided for photoprocessing;*

- *Ground and power planes:* These are, of course, untouched areas that are not included in the above six files. But if you do want to use unused areas as planes, remember that sometimes the insulation process may accidently isolate some areas as paths are milled around traces and pads. Also, connections to pins from these planes must have either no pads around the pins (which would insulate the pins) or must have thermal artwork which would allow an electrical path from pin to plane;

- *Other layers may be added:* For instance rubout areas are an example (areas that have excess copper milled away. Some experimenters prefer to peel away excess copper without specifying rubouts instead);

- *Include a target:* A board outline will do, but an unused drill hole in a corner with a circle around it is best with plots for files **2)** through **6)** above;

- ***Never mix current-carrying and text (or artwork) layers in the CAD files!*** This is critical because the tooling will cut around the traces and flashes to **leave** the copper, but the tooling has to **remove** text, artwork and thermals. ***These are two distinct operations and must be in separate plot files;***

- *Speeding fabrication with careful design:* When large numbers of pins must be insulated on a side, it is far faster to machine and the result is much neater-looking when a grid of straight lines on a graphics layer is used instead of individual pads on a current-conducting layer.

III. Converting the CAD files to CAM files

After the CAD files are created they must be moved to a directory accessible to CCAD. Copy the files from the PCAD directory to a floppy disk (or server subdirectory) to the platform that has CCAM. A typical directory might be called d:\lpkf\data\newpcb. (The file "newpcb" is given as an example here.)

1) Open the *.drl file with Notepad and edit out the comment box at the beginning if it exists. It may have lines that begin with a "*" symbol. Save and close the *.drl file. If the file is too large for Notepad, use WordPad but save it as a TEXT file. Occasionally other CAD files may also have non-command lines that must be edited out. It may be wise to save these lines if they contain aperture data that does not occur in other files. Check your CAD software manual and software defaults for this possibility.

2) Run **CircuitCAM.exe**.

3) Open the **File|Data Input...** window.

4) Under **Job**, select **New**; enter a name such as newpcb (no file extension); hit the **OK** button.

5) Select **-All** in **Files to be Converted** box. This removes files from the previous job.

6) Load new files from PCAD with the **Add...** button. Find the files using the **Directories** window on the right; the files in the directory that you copied will be shown on the left side. Valid file extensions are *.drl and *.gbr for some PCAD documents. Other valid files may have *.lgx, *.pun and *.gdo file extensions. Click on the files you need. Repeat until all are displayed in the **Data Input** window. Select and **Remove** any that are not used in the **Data Input** window. When done use the **Ok** button to exit.

7) Go to **View|DisplayUnit** and select **Inch**. Return to the **Data Input** window and set **EDIF Unit:** to **.001 inch**. The **Flash to Macro** box should remain checked. (These should be defaults from the previous job. Change these values if you use other formats.)

IV. Data formatting: converting data from CAD into the CAM environment

CCAM must now convert the CAD files into a different graphical format. To do this aperture data files must be created and attached before CCAM can run the next step of the file conversion process.

1) When all of the PCAD files are present in the **Data Input...** window then the data format must be specified. Leave the **File|Data Input...** window with **Ok** and go to the **File|Data Format...** window. **The Data Format + Aperture/Tool List** window appears. Apertures and drill lists are entered here;

2) In the **Data Format** box the **Type:** will be **EXCELLON** for drill files (usually in the form *.drl or *.pun) and **GERBER** for the other files;

3) Select **Type: GERBER** from the items listed;

4) For **Name:** either select one of the existing aperture files in the list (and edit it if necessary) or select **New...** and create a new file (or use **Rename...** to keep both the original file and the edited one under a new name);

5) Next manually input the apertures. This has to be done for both the Gerber and Excellon formats. Use the CAD DCODE lists for the **No[D]:** aperture number. "ROUND" refers to **Type: Circle** and "LINE" refers to **Type: Draw**. Ignore the **Min(incl):** and **Max(Excl):** boxes. Use the **Accept** button each time data are entered for each aperture. Only apertures that are used need to be entered in the **Aperture/Tool List** window. Unused apertures can be present. (NOTE: VeriBest CAD generates a different aperture file each time a circuit is run unlike PCAD which may use the same aperture file.) For each aperture enter the **Mode:** as **Flash** or **Draw** (or **Flash+Draw** if you do not need to distinguish between the two modes);

6) After entering all of the apertures for the file you are using select the **More:** button. **Unit** should be **inch**, **Values** should be **Absolute**, **Resolution(m.n)Digits)** are usually either 2,3 or 3,4 or 2,4 depending on the defaults set by the CAD program. Either find out the CAD settings used or read the CCAD manual to learn how to determine what settings were used. Ignore the **Output** and **Size** boxes. Exit this menu with **Ok;**

7) Do the same for **Type: EXCELLON**. Select, modify or create a drill format file as the step above. The drill sizes are listed on the *.mfg list. Enter **Mode: Flash** and **Type: Circle**. Check the **More...** window to be sure the drill list matches the file data. Exit this menu with **Ok.** You do not have to delete unused entries;

8) Do the same for **Type: LPKFMillDrill**. Select **Name: LPKFDrillInch** and modify this file if needed. Exit this menu with **Ok;**

9) Do the same for **Type: LPKFMillDrill** and **Name: LPKFMillInch** and modify this file as needed. Exit this menu with **Ok;**

10) Leave the **Data Format...** window with the **Ok** button and go back to the **File|Data Input...** window.

V. Attaching the format files and creating the *.edi file

The **Job** that was named above will be created as an EDI file (actually an archive of several files). If you called your project "newpcb" and gave the **Job** name as newpcb above, at the top of the CCAD window you will see in the menu bar **CircuitCAM BASIS-[Graphic=C:\CCAD**\NEWPCB.EDI]**. This is the first step in the CAD-CAM conversion process. Other files will be added later to the newpcb.edi file and some will be removed.

1) Select one displayed PCAD file at a time in the **Files to be Converted** box; select **Type:** (Excellon or Gerber) and **Ref:** aperture file from the previous steps. After each **Type:** and **Ref:** filename is selected, hit the **Accept** button each time. Do this for each of the files.

2) Then for each file in the **Files to be Converted** box, choose **Layer Name:** and **Color:** from list for each one. After entering data use the **Accept** button. Unless the imported CAD files were rotated or mirrored, the **Orientation:** should be left **R0**. For **Origin x:** and **y:** leave it **0.0 inch**. (This may be changed later, especially for PCAD Gerber 32 files.)

3) Hit the **Run** button. This creates an EDI file. A log file available under **Window|Log** records the errors if any. Most errors result from failing to edit out the comments at the beginning of the *.drl file.

Here is a list of the suggested **Layer Name:** names to use (these are suggested but they may be edited to add or delete process layers):

Layer Name:	Used for:
Drill	drill file
WiringComp	top (component side) with pads, traces
WiringSold	bottom (solder side) with pads, traces
TextComp	text and artwork that is not insulated
TextSold	text and artwork that is not insulated
BoardOutline	outer edge of board for cutting with router

NOTE: the word "layer" as used immediately above refers to processing of the board with different tools and for different operations only. This usage is different than that in PCAD. For instance the "WiringComp layer" must include only the PCAD layers for the component wiring and pads but not any artwork. Unfortunately the term "layer" is used in very different ways at different times in the CAD-CAM fabrication process.

VI. Viewing the board graphics

The last step ended with running the CCAM file conversion process by entering the drill and gerber files with the CCAM versions of the aperture files, and specifying the output files. This produced a *.EDI file (if there were no fatal errors). If it ran successfully you will see a CCAM window with the CAM version of the project. The (process) layers will have the colors you assigned.

If you get an error message or warning, you may have any of a wide variety of errors. You could have a corrupted file, or have failed to delete appended or prepended data that your CAD program added and the CCAM program could not process. You could have added a file that does not contain drill or gerber files, or did not contain apertures that were needed. CCAM will attempt to create your *.EDI file anyway, unless there are too many errors to create the CAM graphics. Occasionally a warning is innocuous and refers to a "commented out" line that was not processed that you did not edit out. The log file in **Window|Log+Errors** will notify you of problems and will tell you if the EDI file was created successfully.

Often the EDI file will be created but the graphics will not have correct registration or be in the wrong scale or will be severely distorted. In this case, you will have to go back to the **File|Data Input...** window and correct the data and then push the **Run** button again. The first graphic window will contain all of the graphics in the window area. Most common errors are:

- *Registration errors:* Layers do not match up with each other. In **File|Data Input...** the **Origin x:** and **y:** need to have values modified;

- *Scale errors:* One or more of your files has incorrect files attached in the **File|Data Input...** window, or you entered incorrect parameters in the **File|Data Format...** window when creating the aperture files;

- *Missing layers:* One of the aperture files might be so far out of scale that you do not see it. If the **Data Format...** files, for instance, specify a flash or draw diameter equal to or smaller than a pixel, it may be present but invisible; in that case just modify the format file and rerun the data. You could also have a file that is in a format that CCAM cannot recognize and was not converted into the EDI format. Another possibility is that you have one layer completely covering another; in this case go to **View|Top Color** and select the color that you do not see. You might have also chosen white for a layer color and also chosen white as the background; this can be changed in the **View** window. You may also have failed to include flash data in the EXCELLON drill file; this may involve getting a new CAD drill file;

- *Missing features on a layer or distorted graphic elements:* This is usually the result of missing or inaccurate aperture file data. Correct this in the **File|Data Format...** window and **Run** the files again in the **File|Data Input...** window after correcting the errors.

VII. Editing in CCAM

If the graphics look acceptable then no editing needs to be done. But nearly always the graphics must be edited. Editing is different in CCAM than in most software. To edit a feature first select the feature and then choose the appropriate editing function. Refer to the manual for details. Use the **Edit|Layer...** window to hide layers or allow them to be viewed but not selected (or to change their color or width).

You will notice in the **Edit|Layer...** window that the layers are broken down into sublayers by apertures. This allows, for instance, flashes to have wider isolations to make soldering easier (this is done in the insulation process described later). In subsequent file processing sublayers cannot be processed unless they belong to a layer.

When new layers are added or some are deleted, it may be desirable to declare a sublayer as a layer by checking the **Layer** box. You can modify the layer width here if needed.

These data are all object-oriented and are composed of line segments only. Editing them in CCAM can be challenging. Most of the editing should be done in the original CAD software. This can save hours of time since CAD software is better suited to most editing tasks (for instance there is no "snap to grid" feature).

Occasionally features such as targets or conflicting artwork must be moved or deleted, and this can be done with care at this time. Remember to use **View|Top Color** to examine layers for unwanted coincident objects or to reveal hidden objects. Learn how to make layers invisible or not selectable by use of the **Edit|Layers...** window. There are numerous editing features but they behave in ways that can be very different from those in other programs.

It is highly important to ***save the *.EDI file often during editing.*** Crashes are most common due to stack and RAM limitations. It is also possible to accidentally delete groups of features with a sudden move.

Be sure to view each layer separately as well as in conjunction with the other layers to be sure that the process layers are acceptable. ***A fatal mistake is to place a pad over a pin that is to have a plane connection on that side of the board; or to use a padstack that places a thermal on the same layer as other pads!*** Since the pad will have a cut around it when that layer is processed, the thermal will not connect the pad to the conducting surface! If you want a pin to be isolated then place a pad around it (or draw a circle or a line on the text layer around it). If you want a thermal there, you might have to modify your padstack for the pin to remove the pad so it will not be isolated. This applies to whichever of the two sides of the board you are working with on that layer. Thermals can be either on their own layer or on the graphics layer (with no pad at all on the trace layer). (If thermals are on a special layer, that layer can be combined with a graphic layer in the next file conversion step.)

If a layer appears as a mirror image, go to the **File|Data Input...** window and change the **Orientation:** selection for that layer and re-**Run** the files. Save the file that you are satisfied with and delete the earlier files to save space and avoid confusion.

All the graphics are viewed from the top as if the board were transparent. The decision of which graphic layer will apply to which side can be changed at the final processing of the board on the plotter immediately before machining, but it is preferable to adopt a uniform set of colors and layer names for top and bottom board surfaces. (Once again, the top and bottom board surfaces are also occasionally referred to as "layers.")

VIII. Isolation processing

Now that you have the *.EDI file saved in a form that appears to be usable, it is necessary to generate a couple of additional CAM files. This next step generates the cutting paths for isolating (or "insulating") the current-conducting pads and traces, and generates the tool path for cutting out the finished circuit board from the original copper-clad board on the plotter.

Under **File|Insulate** you will see the **Job** window. (Unfortunately the word "**Job**" is used in various distinct contexts in the same confusing way that "layer" is used.) The names here refer only to macros which are designed and used only in this window for calculating tooling paths. You may select the desired **Job** name or set one up in the window by pushing the **Save** button to store the **Job** macro for later use.

The **Destination layer:** selection in the **Tools** box is the name of the output file that will be run on the plotter in a later stage. This layer name may be edited if desired. It is a default choice when a new **Job** is created. **Diameter:** refers to the width of the tool used to process this Job. Generally only one tool is used but a larger tool could be used as a second entry here for faster machining. **Overlap:** is the amount by which multiple passes overlap the previous pass.

Each **Job** run by the **Insulate...** window has a **Task (Layer)** associated with it. The only use for this box is when more than one process is performed within a Job. Generally this name will default to a (process) layer run by the Job. A **Job** can be set up to run more than one **Task (Layer)** if desired but this is purely optional.

The **Source layer** box has the input gerber file as the entry for **Wiring:**. This box is always used, and is usually a layer name. The **Special:** box is used for putting additional cuts around pad flashes to aid in soldering. Put the sublayer that contains the pad flashes here if this is desired. (**Board:**, **Serpentine** in the **Direction** box and **Rubout:** are not discussed here.)

The **Isolation Width** box gives the minimum width of copper that will be removed by the tools in the **Tools** box. If the width here is equal to or less than the tool width, then one pass will be made. If it is greater, then other passes will be made. The maximum width of two passes is twice the tool **Width:** minus the **Overlap:**. As many passes will be calculated in the **Destination Layer:** file as needed to reach the width unless there is an interference with another conductor.

The **Direction** box gives a choice for tool direction. If the **Wiring:** file is for solder-side (bottom) traces then select **Counter-clockwise**; if the **Wiring:** input file is for the component (top) side, then select **Clockwise**. Tooling makes slightly cleaner cuts when the direction of motion is opposite that of rotation of the tooling.

There are also several other parameter choices. **Independent Oversize** forces a cut between conducting traces and/or pad flashes even if the actual distance between them is less than the **Isolation Width**. Its use is recommended for crowded trace and pad areas. **Inner Insulation**, when checked, causes both the inside and outside of the objects in the **Wiring:** file to be isolated. It is not checked when an output file for a routing tool to cut out the board calculated from the board outline file ("*Cutting*" process layer) and the **Wiring**: width is also left at zero for calculating the board output file. **Max. Overlap (Laser)** is not used. **Spike:** is used for eliminating thin pieces of copper between traces by setting it to a few thousandths of an inch. **Insulation Grid:** is left at **.001 inch** for maximum resolution. **Smooth Radius:** makes little difference in processing.

For a typical project there will be three isolation files produced. One will contain the paths calculated from the component-side traces and flashes (*WiringComp* layer); the other is calculated from the SoldComp layer; and the third is calculated from the *Cutting* layer. Each of these is a separate **Job** and has its own **Wiring:** layer for each of these three process layers. Each **Job** specifies a separate **Destination Layer:** file. (Suggested destination file names are *InsComp* for the *WiringComp* process layer; *InsSold* for the *WiringSold* process layer; and *BoardOutline* for the *Board* process layer. Of course you can edit these names.) Each **Job** must have the proper tooling selected and the proper **Direction** and **Isolation Width** and other appropriate parameters chosen.

For each **Job** confirm that the proper settings are present in the window and push the **Run** button. A pop-up window will display the progress of the file conversion calculation. When the **Destination Layer:** file is complete (this may take many minutes) the pop-up window will disappear and the new layer will appear in the CCAD graphics window. Check the new layer to make sure that it does not conflict with any of the other layers. Go to the **Edit|Layer...** window and give this layer a color to distinguish it from the other layers. You can turn off the *WiringComp* and *WiringSold* layers since these will not be used any further. If it is important not to have any of the isolation paths touch the board outline this is the time to check for this. You also must be certain that the *TextComp* and *TextSold* graphics do not interfere with the new isolation files.

Another possible concern is that ground or power planes (if used) might be cut into parts by the width of the isolation cuts. Occasionally isolated planes must be reconnected with jumpers or traces ending in thermals.

After each isolation **Job** is run successfully you can save the settings by going to the **File|Save Script** menu selection. If you will be using these settings in the future you will have them properly entered each time the **Job** is selected. If you are running a one-time job then choose **Rename** at the top of the window in the **Job** box to preserve the old settings under the previous name. If you choose one **Job** and modify it and then choose **File|Save Script**, that file will retain the modifications.

IX. Task List preparation

At this time all of the necessary files are ready for conversion from EDI format into the LMD format for export to the ProtoMat Plotter for machining. Some of the files described below might not be used, and sometimes others might be produced (*e.g.* for rubout areas). Still other files are no longer used in the process (*e.g.* pad flashes and traces, since they were needed only to produce the files that are needed typically include:

1) *Drill:* This file is essentially unchanged from the CAD program that drafted it;

2) *TextComp:* This contains the line segments that make up any text or lines or artwork such as decorative symbols or thermal artwork that is to be removed by a mill on the top component side of the copper-clad printed circuit board.

3) *TextSold:* This is the same as *TextComp* but describes the line segments on the bottom solder side of the board;

4) *InsComp:* This describes the paths that the mill tool takes across the top of the board to electrically isolate the pads and traces by cutting around them, taking into account the width of the milling tool (or, optionally, tools). This file was produced in the **File|Insulate...** window;

5) *InsSold:* Same as *InsComp* but on the bottom side of the board;

6) *BoardOutline;* This is the path a router tool takes as it cuts entirely through the material to separate the finished board.

X. Output file conversion (*.EDI to *.LMD)

Open **File|Data Output**. There are three boxes in this window: **Job:**, **Task**, and **Task List:**. The six files from the previous step are now to be converted into an output file that is also an archive-like collection of files with the LMD extension. These files are now called "tasks." One task is placed into the **Task List:** at a time. Since the newpcb.edi file has several layers (most of which have several sub-layers) you must choose the proper task layers to be exported to the *.LMD file. Here are the steps to building the LMD file:

Note: LpkfMillDrill format (*.LMD) is a private format in binary form. Its only purpose is to have an optimal connection from CircuitCAM to BoardMaster to drive the mill/drill plotter. This format is used for Data Output at a later file conversion stage. Do not delete or rename or create new files for* Type: *LPKFMillDrill. Only modify these two files for the apertures and drills used. (For metric, use* Name: *LPKFMill and* Name: *LPKFDrill.)

1) *Choose a* **Job** *name:* The **File|Data Output... Job** names are macros, just as the **File|Data Input... Job** names were. Once you create a name and save it with one project, that shell will be available to make the process of file conversion faster if consistent task names were used from one project to the next. **Job** macros do not exist outside the **Data Input...** and **Data Output...** windows in which they were created. Use the **New...** button to create a new macro name (don't type a file extension). For instance you might type in <u>NewPCBOut</u> and then press **Ok**. (If you are running a new project that uses similar kinds of layers to a project and you saved the macro with **file|Save Script** at another time, then scroll down the names in the **Job:** box to select that name and press the **Run** button to create the <u>newpcb</u>.lmd job.) Note that the *.LMD file will not be saved under the macro name but will be saved under the same name as the *.EDI file.

2) *Select the first layer to be moved to the output file:* Usually the drill file is the first file to be run so it will be picked first:

   a) ***For the first task layer file only***, be sure that the **Append** box is ***not*** checked in the **Task** box;

b) In **Layer:** select **Drill(Layer)**;

c) In **Format Type:** select **LPKFMillDrill**;

d) In **Format Ref:** select **LPKFDrill** (this is the format file you edited earlier);

e) In **Phase:** select **DrillingUnPlated** (**Phase** names can be edited or added but it is best to use the same list of suggested names every time to avoid confusion and to save time; these names are simply descriptive headings that describe a complete operation such as drilling or insulating a side of a board);

f) Push the **Accept** button;

g) Push the **Add** button. The selected layer is now a task listed in the **Task List:** to the right.

3) Continue as in the step above until all of the layers that are needed are listed in the **Task List:** box.

Points to remember:

- **File:** is always ***.lmd**.
- **x:** and **y:** must equal **0 inch**, and **mag:** must equal 1.
- **Format Type:** must be **LPKFMillDrill.**
- **Format Ref:** must be **LPKFDrillInch** for the drill file but **LPKFMillInch** for all other files.
- The **Append** box must be checked for every file in the **Task List:** except for the first file.
- To change any item in the **Task List:** listing, select it, modify it in the **Task** box on the right and then use the **Accept** button.
- You may use or edit any name for the **Phase:** for each task but the name must be unique and preferably descriptive. Suggested names are:

Phase Name:	**Used for:**
DrillingUnPlated	Drill layer
MillingCompSide	TextComp and WiringComp layers
MillingSoldSide	TextSold and WiringSold layers
Cutting	BoardOutline layer

(I have combined the **TextComp** and **WiringComp** tasks into the same phase because they use the same tool. They can be separated into different phases if you prefer. **MillingSoldSide** can also be assigned different phases but you will have to edit the **Phase:** list for your new phase names.)

4) Push the **Run** button. A new file (newpcb.lmd) is created.

You are now finished with this stage of file conversion. You can quit CCAM and open BoardMaster.exe to run the new LMD file.

XI. Setting up the ProtoMat plotter hardware

1) ***RTFM! (Read the fine manuals): The warnings, cautions and safety instructions for ProtoMat operations are important! Injury to the operator and machine can result. Careful reading can save many hours of wasted time.***

2) *Start the plotter program* **BoardMaster.exe:** (if you have not already done so). In manual mode turn the spindle on to warm up before use as suggested in the manual.

3) *Set the home position*:(Necessary if there has been a crash, or for adjusting registration on the second side of a board):

   a) Open **Machine|Settings...** menu.

   b) Push the **Unlock** button and then the **Home** button.

   c) After the machine returns push the **Lock** button. Selecting **Machine|Home** sends the head to the factory-setting home position. Check to see that the head is above the centerline of the bed. The home position can be set to another position along the y axis by moving the head *ONLY* along the y axis, opening up the **Machine|Settings...** menu, and using **Unlock**, **Home[inch]**, and **OK** buttons in that order.

Note that during the machining of the second side of the board, the coordinates of the Home position will have to be adjusted a few thousandths of an inch. This is described later in this document.

3) *Mounting the copper-clad material on the table:*

NOTE: New 6-32 holes have been drilled in the table (for the ProtoMat in HR9E) to accommodate 6/32 x 3/4 flathead screws. They are located along the X axis, 15.5 inches apart to accommodate 16-inch boards. Punch two holes 15.5 inches apart 3/16 inch diameter (preferably with a Rotex punch) and chamfer both sides. This assures better registration and eliminates the chance of a "wreck" during machine alignment and restarts which can occur when tooling hits the LPKF-supplied pins which extend above the board. If these flathead screws are used, ignore steps a) and b).

   a) Check front alignment hole: push the red strip all the way to front reference pin; insert 3mm pin which should be a tight fit, otherwise drill a new hole as in ProtoMat Manual page 19;

insert a 2.95mm drill into collet, position drill over bed (not work piece), and drop the drill to .05 in. from bed; drill hole manually; under **Machine|Settings**, Unlock, set step size to half inch, and then OK; back off 1 inch along **x** and go to **Machine|Settings** and **Unlock** and **Set Home**; back off known number of inches to back of bed and drill manually again a second hole; use this measurement for drilling base material (PC board) and drilling base (masonite or crescent board) holes. IMPORTANT: holes in PC board and drill base must be .118; holes in red alignment strips must be .116 maximum and TIGHT when the two alignment pins are inserted!

b) Insert two 3mm alignment pins; drill .118 holes in drill base and PC board. Be sure that alignment pins are tight.

c) Tape the base material to the table, preferably with Kapton tape (this is cleaner, holds better and is less likely to stretch during machining which would reduce registration accuracy). We have found that "crescent board" available from art and photographic suppliers is superior in cost and performance to masonite. We use two pieces of medium weight board.

d) Attach the material to the base (with pins or screws). Use more tape to keep the corners flat against the base.

e) Enter low and high coordinates under **Materials|Size....** Get these by moving drill over lower right and upper left corners of PCB material as described in the manual. Otherwise for 12" x 16" boards use:

Low corner X[inch] = *0* **Y** = *-5.5*

High corner X[inch] = *14* **Y** = *5.5*

Exit the window with the **Ok** button. Usable area will be grayed on the screen. The plotter is now ready for running the LMD file.

XII. Loading the LMD file into the plotter

1) In the BoardMaster program select **Project...** from the menu bar. The **Project Edit** window opens.

2) Push the **Add Project...** button. The **Add project** directory window opens.

3) Locate the newpcb.lmd (or whatever you named your CCAD file) and select it. Push the **OK** button.

4) The **Add project** window closes and the **Enter new name** window pops up. It will give the name of the LMD file (*e.g.* NEWPCB.LMD) which gives you an opportunity to edit the name. Accept the name as it is and click the OK button.

Note: If you are only running one project (also called occasionally a "job" in the manuals and software) then you are actually adding a

project to the default of "no project." This is why the button is called **Add Project....** ***You may run several LMD files at once if you want by using step 4) repeatedly. Avoid changing names if possible. The option of running several identical copies is available in the*** **Placement...** ***window described below.***

5) All of the files you appended in CCAD in the **Data Output...** window should be shown in the **Phase File/Layer** box. If they are not shown there, then you will have to re-open CCAD and rebuild the newpcb.lmd file (the most common mistake here is having the **Append** box unchecked for the first file only). (If you see the files in the **Phase File/Layer** box properly numbered by phase you can skip the next step.) Choose **OK** to exit.

6) Open the **Phases...** menu. If you are missing the number in front of a file that you will have to process, you will have to add an appropriate name here. The suggested names and normal order of phases are: *DrillingUnPlated*, *MillingCompSide*, (you will reverse the board on the ProtoMat plotter at this point) *MillingSoldSide*, and *BoardOutline*. The last two phases should have the **Reversed side** box checked. If you are doing all operations on the top side of the board, then you will want to leave the **Reversed side** box unchecked. Add phase names by selecting a phase shown in the box on the left above or below where you want the new phase name added, and use the **Ins.below** or **Ins.above** button and typing in the name you want. There may be extra unused phases shown which you may **Delete.** Or, you may ignore the unused phase names for use in subsequent projects. The **save settings as default** checkbox will save your phase name additions for later projects. Select a color to display each phase in the **Color** box. Choose **OK** to exit.

7) Open **Project....** The **Unit[inch]:** should remain **0.001**. In the **File/Layer** box ignore the **Pen** box and leave the **y=>-y** checkbox unchecked. The proper sequence of files should be shown with a **Phase** number for each in the **Phase File/Layer** box on left. Select each of the phases in this box, and then select the proper tool for each phase. (Tool types can be edited later if needed in the **Tools...** window.) For the *Drill* file this is done automatically. For the *MillingCompSide*, *MillingSoldSide*, *TextComp*, and *TextSold* files, the usual tool is the unimill. (These tool widths as well as the router chosen to cut out the finished board or add special cutouts or details should correspond to the width specified in CCAD in **Isolate....**)

8) Open **Materials|Size....** Enter low and high coordinates of the board material under Get these by moving drill over lower right and upper left corners of PCB stock as described in the manual. Otherwise for 12" x 16" boards use:

Low corner X[inch] = *0* **Y[inch]** = *-5.5*

High corner X[inch] = *14* **Y[inch]** = *5.5*

The usable area will be shown in light gray on the screen. Exit with **Ok**.

9) Using the **Placement...** window:

a) Open the **Placement...** menu window.

b) Leave the **Scale:** at **1**. (Let the software calculate the entries for the **Origin[inch]** and **Displacement** boxes. Only enter data in these if unavoidable.)

c) Open the current project with the **Add...** button, select your project and exit with **Ok**.

d) Select the number of "panels" to be run on **X** and **Y** axes (a panel is an image of the project on the board. Other projects if selected earlier also can be run at the same time.) The software automatically puts the panels in a rectangle adjacent to each other with minimum clearance.

e) Push the **Center** button to arrange them in the middle of the work area.

f) Exit with **OK** and check the graphics to see if the panels fit on the work area. If it is not satisfactory open **Placement...** again and use the **Rotation** box, **Count** box and **Center** buttons again to adjust the placement. The buttons on the top of the main program screen (overlapping squares) can be used to move or duplicate panels after the **Placement...** window is closed to graphically adjust the position. The **Displacement:** window gives values of the width of the board plus an offset to allow the cutting tool to clear the adjacent boards.

XIII. Machining the boards

When the placement of the panels is acceptable, the project can be run. Here are a few suggestions and hints:

- Select the first phase and insert the tool specified with an asterisk (*). You may want to do a trial cut to make sure that the tool is at the right depth. Proper feeds and RPM will be shown. Select the segments to be machined and they will be highlighted as described in the manual. When the head's motor is under auto control the machining will be automatic.

- The default tool settings (**Tools...** menu) are excellent for drills and milling with the "Unimill" universal cutters at full feed and RPM (this does not apply for ProtoMat plotters that are not capable of 60000 RPM). Routers may need to be run at a slower feed; and be careful that they do not loosen and work down the collet. Our plotter will do this occasionally. It digs into the table as it slowly drops before wrecking. To avoid this I remove the smooth finish on the tool shank with a honing stone and keep my hand near the OFF button to stop it if the pitch begins to suddenly lower as the tooling begins to jam.

- To check for insufficient depth due to tooling wear for an insulation cut, try setting **Machine|Settings...** to **Step[inch]** to *.2* and cut a square, and test that square with a continuity checker. After about 1200 inches of use a unimill tool may need to be lowered to continue cutting at the proper depth.

- Running a tool at too slow a feedrate can decrease tool life because all the wear is concentrated on the extreme front of the cutting edge. (Of course running a tool too fast so that it snaps will also suddenly decrease its life!)

- Drills that leave a long "stringer" of metal are too sharp and need to be dulled by the use of a honing stone.

- Drills that leave a hole that is too large for their size are too dull. Unlike mills, as they become more dull they take off more metal, not less metal.

- If you want to check a tool before running it (very advisable) be sure that at the end of the previous phase you have turned off the automatic control of the head. If you have selected the segments to be run and you change the tool, it will immediately begin to run the next series of segments and will not stop even if you press the **Stop** button!

- Be extremely wary of using tooling that is not recommended for the ProtoMat plotter. Lost time and broken tooling can rapidly overwhelm any savings in tooling cost.

- Data are sent from the BoardMaster program to the plotter in serial chunks. So if there is problem, the plotter may still machine several segments of the program after the **Stop** button is pressed. For a true emergency stop, the power switch of the plotter must be turned off.

- The setup can be saved in the **Job** menu if desired. Jobs that run similar layers can call on one **Job** that can be modified to save time.

- Before machining the bottom a test cut should be done to ensure that the board is centered. A good way of doing this is the following:

Mill around a drill hole and check it visually. (Select one pad and run it; then back off the head to examine the milling for centering.)

Is the hole too far to the right?

Move the mill to the right by decreasing the* Y Home *value in the* Machine|Settings... *menu selection. Use the* Unlock *button first, change the settings, then use the* OK *button to exit. The registration error will probably be under 0.010 inches. Repeat as necessary.

Is the hole too far down?

Move the mill down by decreasing the* X Home *value.