

**Lawrence Technological University**  
**College of Engineering**  
**ECE Department**

**LPKF® ProtoMat® PCB Plotter Manual**





# Lawrence Technological University

## College of Engineering

### ECE Department

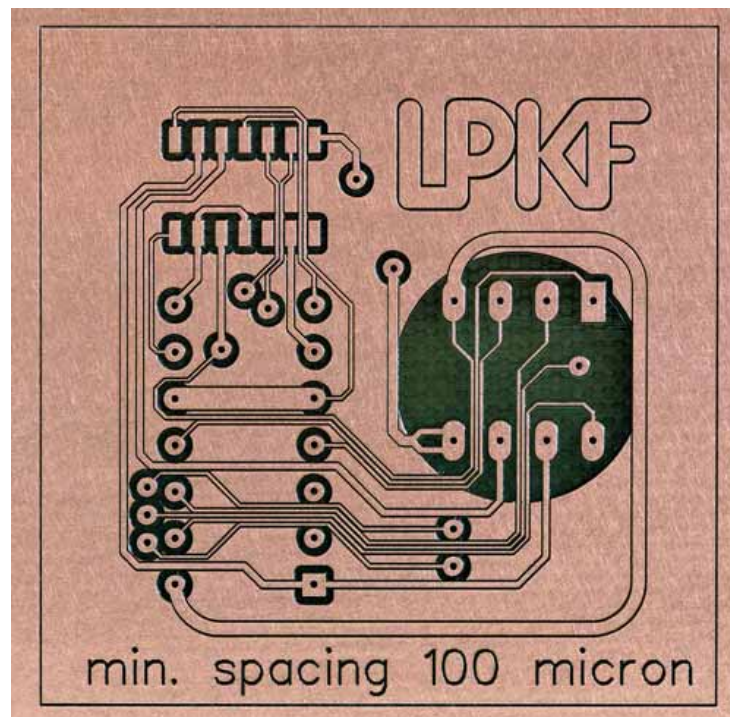
## Table Of Contents

<b>Chapter 1 - Making A Circuit Board:</b> .....	page 1
Introduction .....	page 3
LMD File Creation .....	page 4
Board Master Operation .....	page 8
Changing the Tool Bit .....	page 10
<b>Chapter 2 - Raw PCB Preparation:</b> .....	page 11
Making Plotter Mounting Holes .....	page 12
<b>Chapter 3 - Plotter Calibration:</b> .....	page 13
Introduction .....	page 14
Software Calibration .....	page 15
Hardware Calibration .....	page 16

**Lawrence Technological University**  
**College of Engineering**  
**ECE Department**

# Chapter 1

## Making a Circuit Board



Making your first circuit board can be challenging, frustrating, and gratifying. This manual will help minimize the frustrating aspect of this process. The process may look long but this manual is just written in detail for those who are not able to naturally pick up new software quickly. The entire process consists of Designing a Circuit Board (consult your software), Creating Gerber Files (consult your software), converting the Gerber files into a useable LMD file for the Board Master Software, and finally running the machine to make the board.

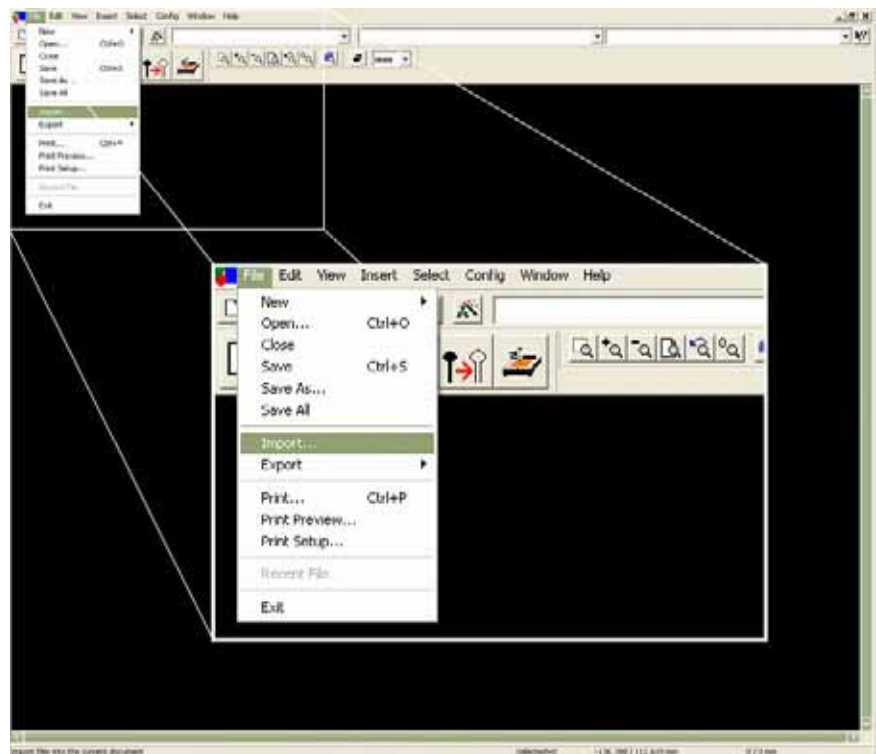
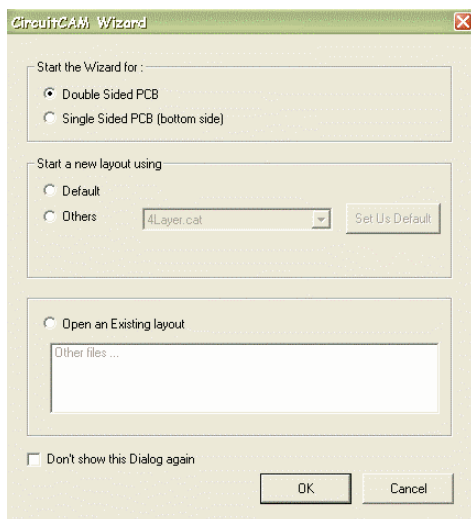
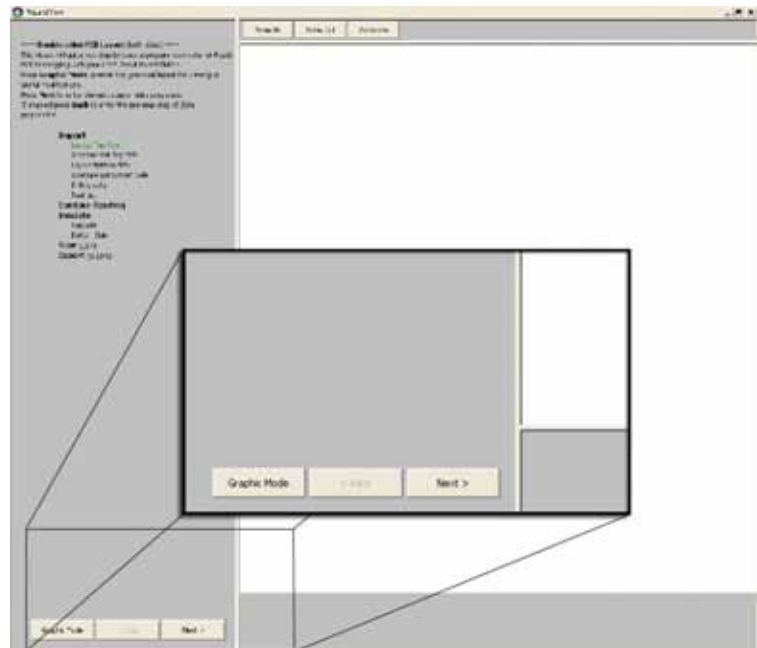
This manual assumes that you have already created your PCB Design and created the required Gerber Files.

The following step by step instructions will show you how to generate the required LMD file from the Gerber files.

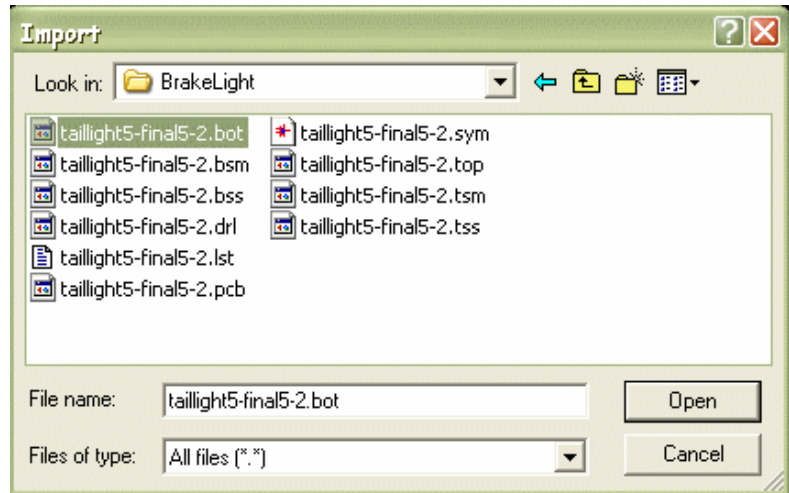
Required Gerber Files (by extension):

- 1 sided board: top or bot & drl
- 2 sided board: top & bot & drl

- 1) Open CircuitCAM 4.0
- 2) Select your board type (1 or 2 sided)
- 3) Click OK
- 4) Click the “Graphic Mode” button
- 5) Go to: File >> Import



6) Select the filename.bot file as shown to the right



7) Now Set the layer to “BottomLayer”

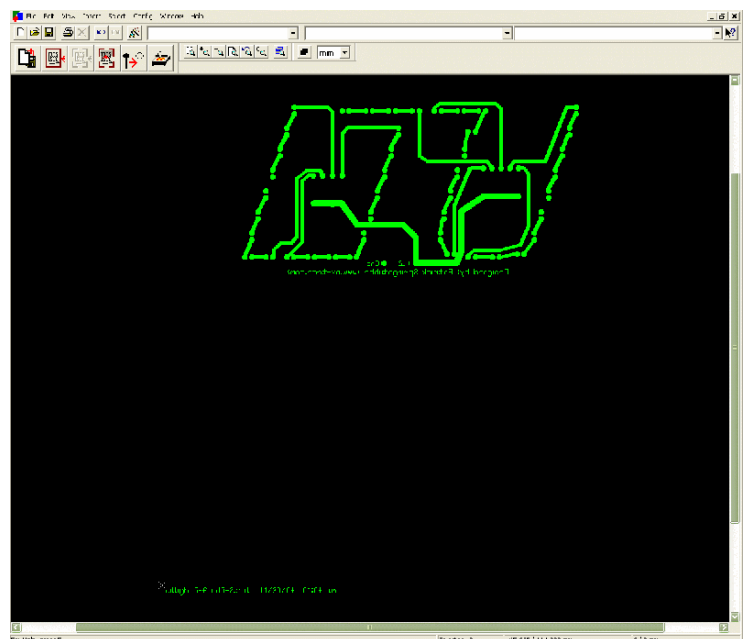
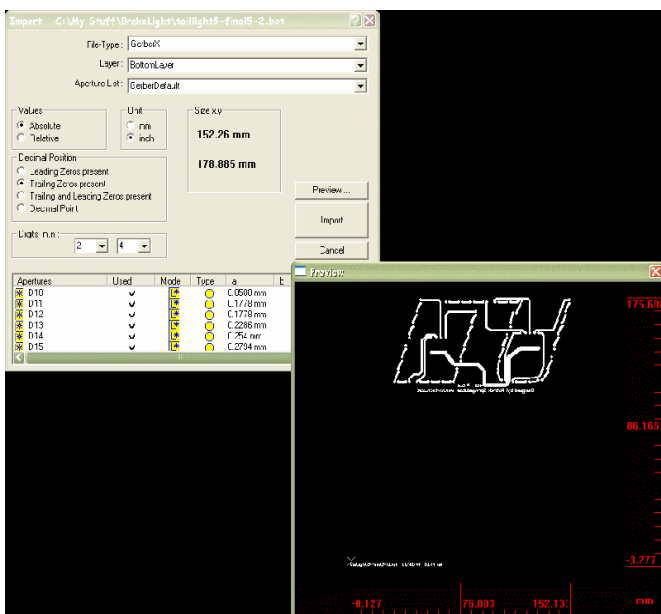
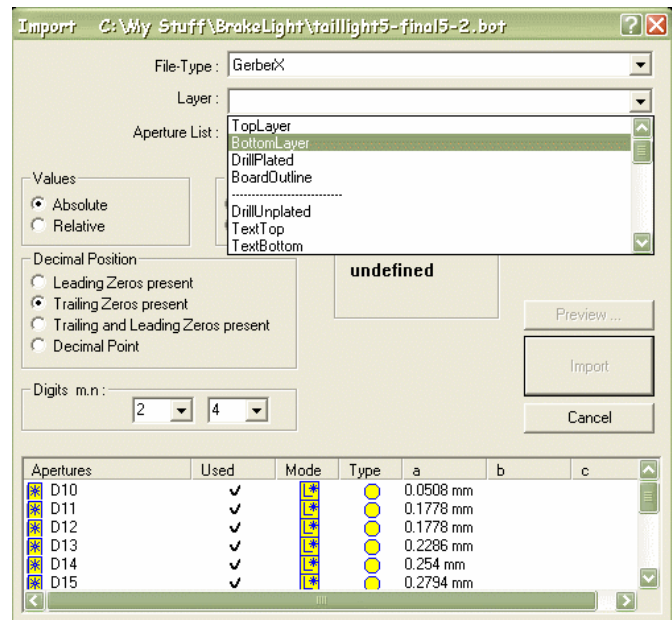
8) Set the “Decimal Position” to “Trailing Zeros Present”

9) Set the “Unit” to “Inch”

10) Now Click the “Preview” Button and make sure the traces look correct. If not then try setting the “Unit” option to “mm”.

11) Now click Import and you should see the layer shown in green (bottom layer) red (top layer) as shown below.

12) Do the same for the top layer for a double sided board.

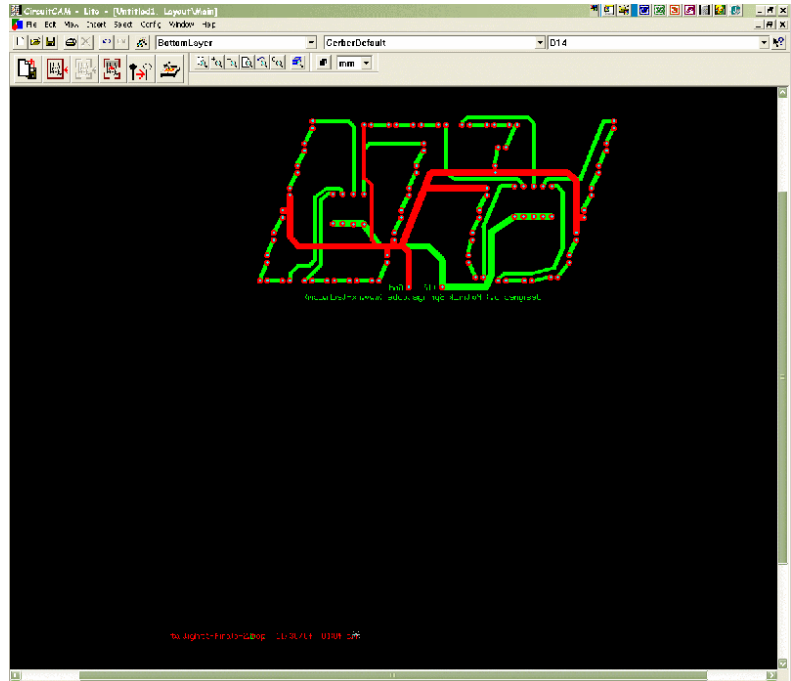


- 13) Now we need to import the Drill Holes from the drl Gerber file. Go To: file >> import select file-name.drl
- 14) Set the layer to “Drill Plated”
- 15) Set the “Decimal Position” to “Leader Zeros Present”
- 16) Set the “Unit” to “Inch”
- 17) Preview and make sure the drill holes look correct
- 18) Click the “Import” Button

Now you should have something that looks like this.

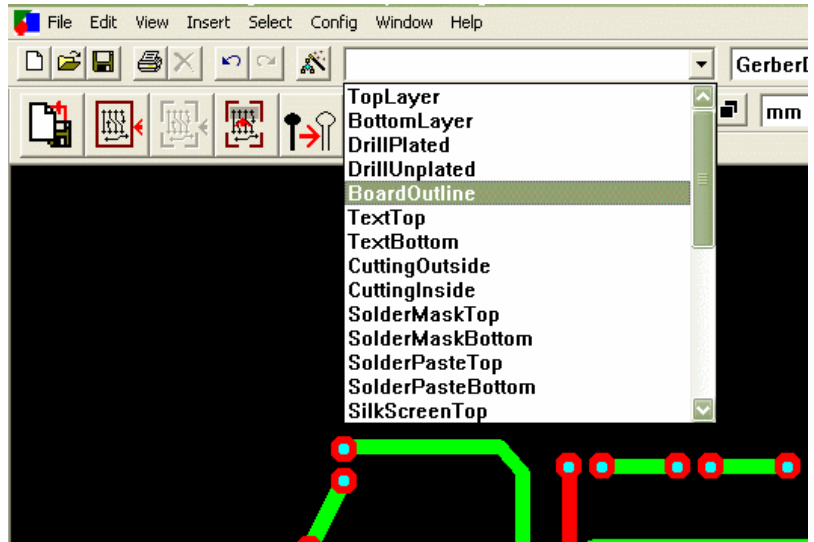
If there is text at the bottom of the black area then you will need to remove it.

- 20) Simply use the cursor to highlight the text then hit the “delete” key on your keyboard



Now that we have all the Required Gerber files imported we now need to create a Board Outline so that the plotter cuts the board out for us.

21) From the first dropdown menu from the top left select “BoardOutline” as shown in the picture to the right.

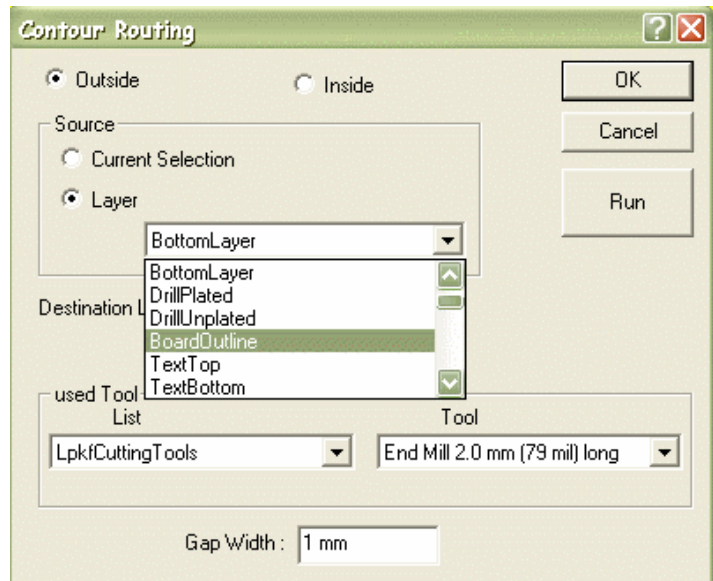
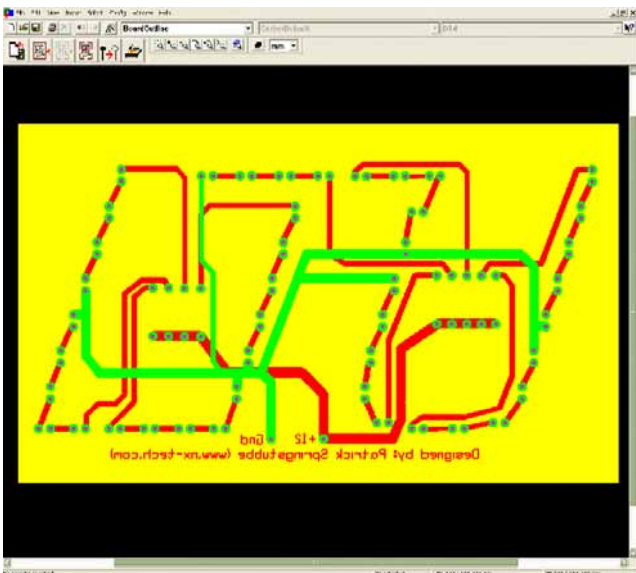
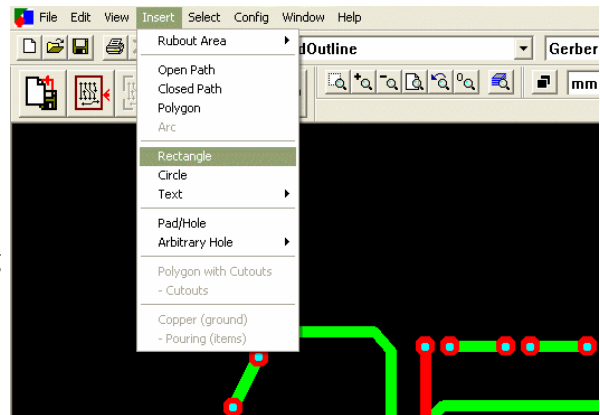


22) Next Select from the menus: Insert >> Rectangle or if you want a circular board select Circle

23) Now define the outline of your physical board as shown in the illustration at bottom left of the page.

24) Now we need to setup the Contour Routing by going to: Edit >> Contour Routing

25) Set the Layer to “BoardOutline” then click Run.





Now you should see a grey outline but the un-needed rectangle is still present. Click one of the corners of the rectangle and hit the “delete” button on the keyboard.

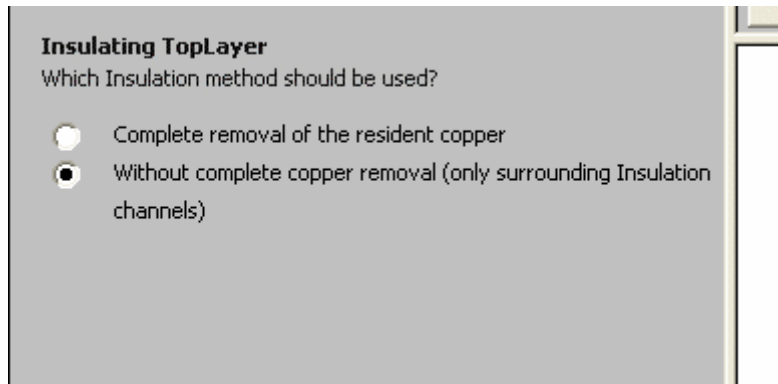
Now that we have done this we need to go back into the Wizard Mode (its easier for this part) to set insulation paths. There are two insulation types, Complete Removal and Partial Removal. For most jobs a partial Removal will work just fine. This will take less time and cause less wear on the drill bits.

26) At the top to the left of the first dropdown menu there is an icon that has 3 little boxes with a magic wand. Click this button. This will bring you back into the wizard mode that we first started in when we started CircuitCAM.

27) Click the Next Button

28) Select the second bullet option as shown to the right (unless you require complete copper removal)

29) Click next for this screen and the next. Using the default tool is in most cases what you will need.

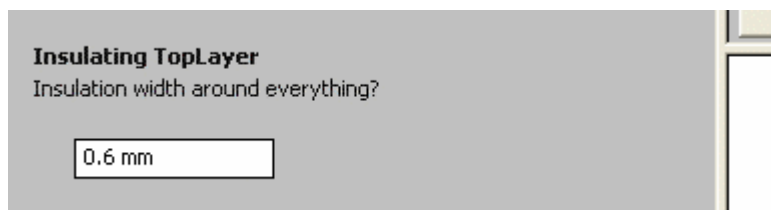


**Insulating TopLayer**  
Which Insulation method should be used?

Complete removal of the resident copper

Without complete copper removal (only surrounding Insulation channels)

30) Set the insulation Gaps to 0.6mm (this will take 3 passes with the Universal Cutter to complete, but will give an easier to solder board), do the same for the Pad Insulation



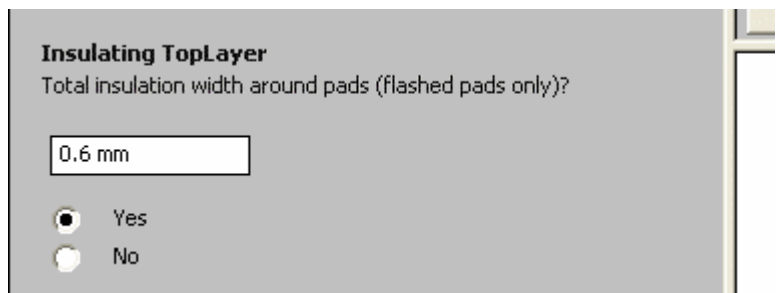
**Insulating TopLayer**  
Insulation width around everything?

0.6 mm

Repeat Steps 28, 29, & 30 for the Bottom Layer

31) now save the CAM file as the wizard directs you to do

Now is the portion where we make the needed LMD file



**Insulating TopLayer**  
Total insulation width around pads (flashed pads only)?

0.6 mm

Yes

No

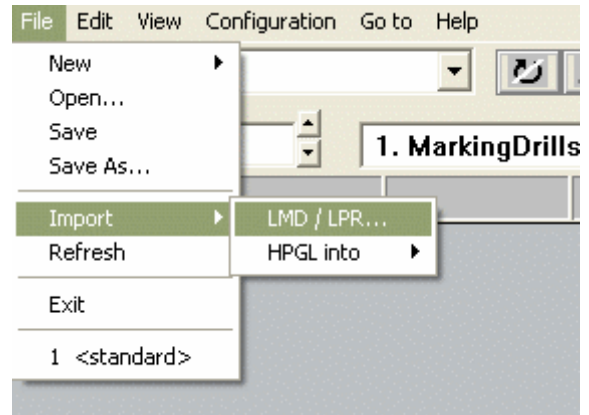
32) in the drop down menu choose the “LpkfCircuitBoardPlotter” Option and click next

An LMD file has been created in the same folder where you saved the CAM file to.

Now you can close Circuit CAM and Proceed to the next section of this chapter to make the board

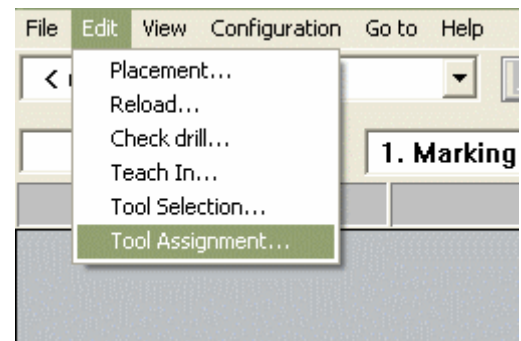
The software that actually controls the PCB Plotter Hardware is “Board Master”. With Board Master you can import the LMD File that was created in the last section of this Chapter

- 1) Open Board Master
- 2) Go To: File >> Import >> LMD / LPR
- 3) Select the LMD file for your PCB



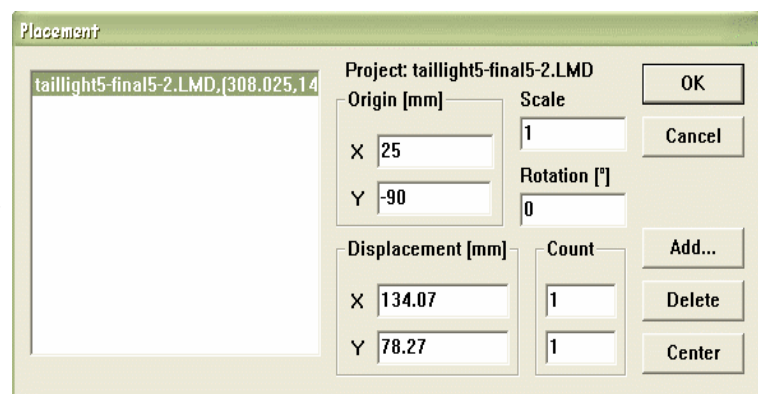
If you receive a Tool Error after you Import the LMD file do the following.

- 4) Go To: Edit >> Tool Assignment
- 5) In the Phase File/Layer section select Cutting Outside and change the tool to the right of that section to one of the larger Contour Router Tools, in this example we will choose the “M Contour Router 3.0mm (118mil)”
- 6) Click OK



Now we need to move the board so that we don't cut right in the middle of the RAW copper board and waste a lot of material. This step may take some trial and error but these settings will get you closer to where you need to be.

- 7) Right click on the PCB Layout
- 8) Change the origin to X = 25; Y = -90
- 9) Click OK and you should now see the PCB layout in the lower left hand corner of the screen. Continue to do this until you get a placement you desire.

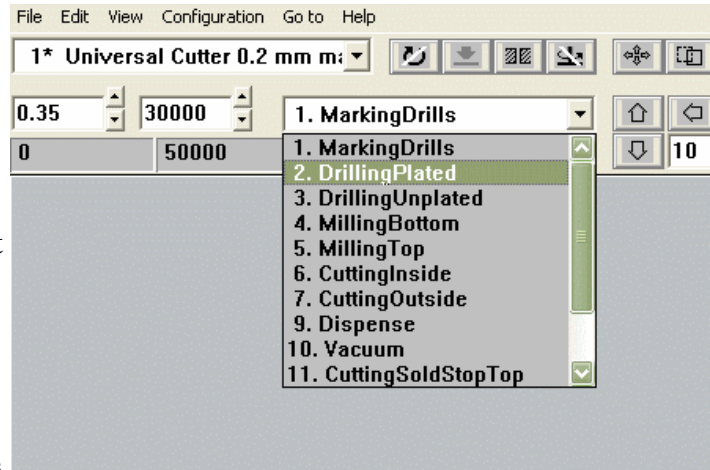


Now for the exciting part, producing the board. This is a 4 Step milling process, Drill Holes, Milling Bottom, Milling Top, and Cutting Outside.

10) Select Option #2 “Drilling Plated” in the dropdown menu as shown to the right.

11) Click on the “All +” button to the far top right of the screen.

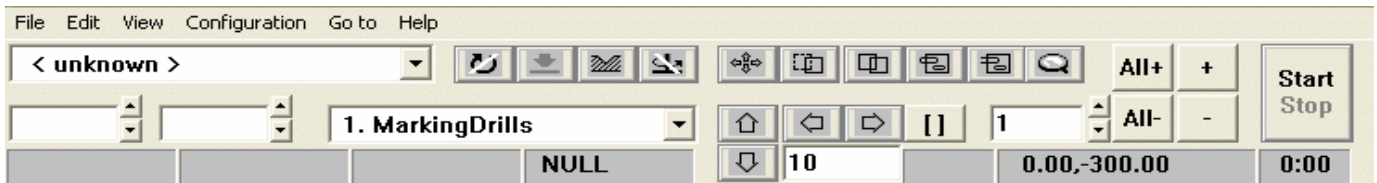
12) Click on the Start / Stop Button. Follow the computer prompts for the tools it requires. For Drilling Holes if you do not have the requested bit it is always better to go with a smaller bit and re-drill it out in the Fab-Lab later.



Now repeat steps 10, 11, & 12 for Option #4

Milling Bottom, Option#5 Milling Top, and Op-

tion #7 Cutting Outside and don't forget to flip the board when changing from the Bottom Milling to the Top Milling as it will show on the screen. (It will not prompt you to flip the board rather it will just show the PCB layout on the other side of the black line.



Now your board should be complete and ready for you to work with. For soldering your board I recommend the following to make it easier and so that you have less problems.

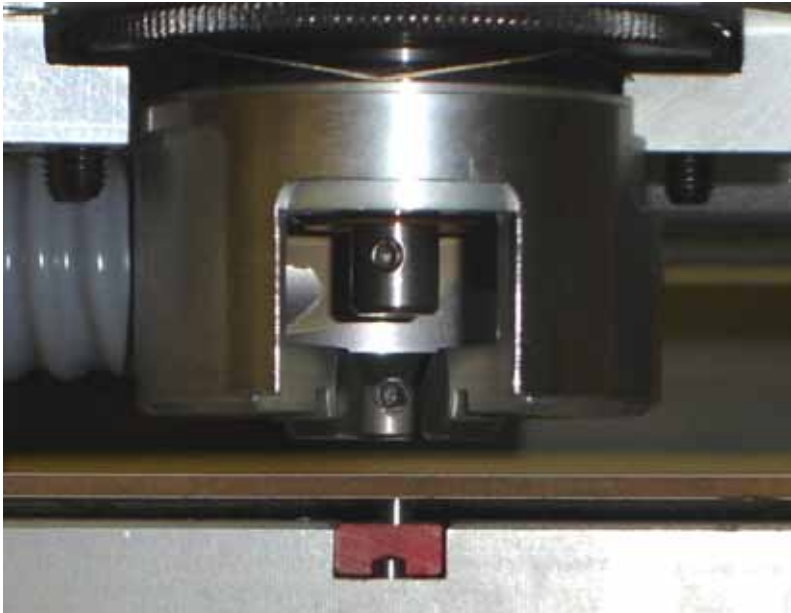
- Tin all areas you wish to solder to (apply rosin and a little solder to component solder points).
- Do a few components at a time and check continuity around the solder point and surrounding copper so that an accidental connection is not made. This will make it easier to find the problem.

Changing the tool bit is a procedure that needs to be done with care. If the bit is not properly placed in the Drill Press then it will cause damage to the copper board and the bit.

- 1) Move the Drill to the Exchange Position: Go To >> Exchange
- 2) Rotate the drill shaft until the black set screw is facing the opening.
- 3) Turn the set screw counter clockwise a half to a full turn to release the bit.
- 4) Remove the bit and place the desired one in making sure it is firmly in place
- 5) While you have the bit firmly in place tighten the set screw by turning it clockwise until it is snug.  
**DO NOT TURN TO TIGHT OR YOU MAY STRIP THE SET SCREW**

Notes:

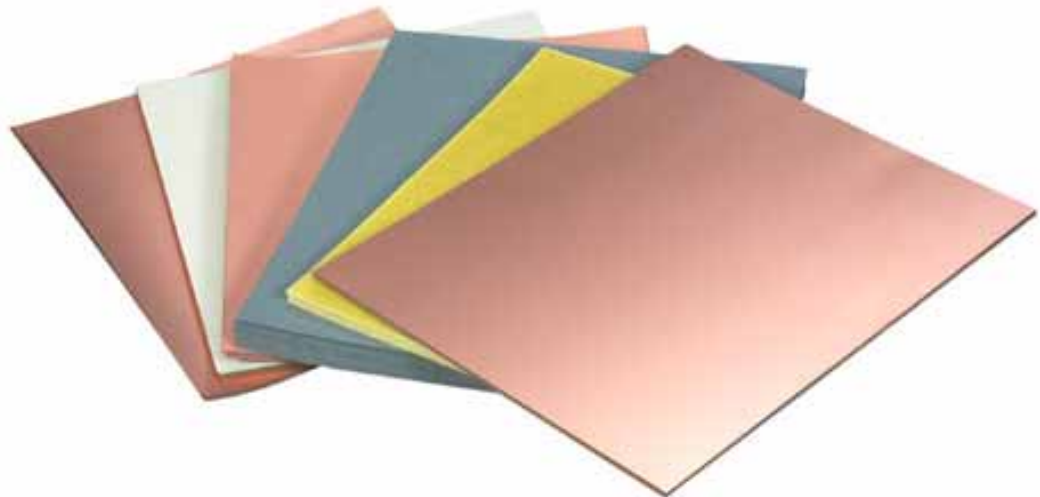
If the set screw seems to be getting worn replace it. If there are no replacement set screws talk to the ECE Lab Coordinator or Department Head to get replacement screws.



**Lawrence Technological University**  
**College of Engineering**  
**ECE Department**

# Chapter 2

## Raw PCB Preparations



When the copper is received it is a sheet of single or double layered copper with no plotter mounting holes. So to make these holes is very simple.

1) Along the short side's of the board we are going to place a hole using the included Drill Bit which is in a Blue Case with a Clear Cover in the middle of the board edge.

It is not necessary to be exact when placing the holes because when you flip the board the axis in which it will rotate on will remain unchanged.

Now when you place the copper board on the plotter you may need to adjust the placemen of the mounting pins to support that board.

**Lawrence Technological University**  
**College of Engineering**  
**ECE Department**

# Chapter 3

## Plotter Calibration



Calibrating the Software and Hardware is key to obtaining a good board. Without proper instruction this process can sometimes be hard to figure out and even confusing as to what you need to do to obtain an aligned board. There are two calibrations that need to be done which give different results. The Software calibration is for adjusting the alignment for double sided PCB's. This aligns the HOME position of the software to the correct location on the hardware. The Hardware calibration is for adjusting the milling depth which is key to making sure that the cuts are not too shallow and not too deep. If at the wrong setting the results can be problematic by cuts making trace widths and pads too small.



The software calibration simply calibrates the X , Y Home Potions (0,0). This is needed so that when you flip the board over to do the 2nd side that the axis of rotation matches.

#### Calibration Steps:

- 1) Import the LMD file c:\LPKF40\Data\CheckHome.LMD
- 2) Place it along one of the edges of the board so that we don't waste a good piece of copper board.
- 3) Run the entire process as laid out in section 3 of chapter 1

Now when you look at the 4 holes that were made with the pads on one side the holes should be mis-aligned. What you need to do is determine (in mm) the distance we need to move the origin so that it is aligned. The use of a micrometer (caliper) will make the process easier.

- 4) Go To: Configuration >> Settings
- 5) Click the “Unlock Button”
- 6) At the bottom left is where you will adjust the home position

Repeat Steps 2 through 6 until the holes are aligned.

The screenshot shows the 'Machine settings' dialog box for a ProtoMat C30/S-EC machine. The settings are as follows:

- Name:** ProtoMat C30/S-EC
- Number of tool positions:** 1
- Startup:** !OC;!SV140;!SM30;!WR0,8,8;!CC;
- Unit:** mm (selected)
- Maximum speed:**
  - Rotation [1/min]: 30000
  - Moving [mm/s]: 25
- Resolution [mm]:**
  - X: 0.0079375
  - Y: 0.0079375
- Null [mm]:**
  - X: 0
  - Y: 0
- Home [mm]:**
  - X: 0
  - Y: 300
- Pause [mm]:**
  - X: 0
  - Y: 0
- Scale factor:**
  - X: 1
  - Y: 1
- Size [mm]:**
  - X: 800
  - Y: 600

Additional controls include a 'Lock' button, a 'Controller' section with radio buttons for MCU, SMCU (selected), and Laser, and checkboxes for 'HF spindle' (checked), 'Motion BUS', 'Measure', and 'Tool box'. Action buttons include 'OK', 'Cancel', 'Set Home', 'Initialize', and 'Send'.

The Hardware Calibration sets the depth of the bit so that it doesn't cut too far into the board and so that it cuts far enough in for a complete insulation cut.



#### Calibration Steps:

- 1) Put the Universal Tool into the mill
- 2) Move the drill to Far Left by clicking on the icon that looks like a 4-way arrow then click on the virtual board to the far left. This will begin to move the drill to that location.
- 3) Now turn the drill on by clicking on the icon that looks like a circular arrow
- 4) Now we need to put the drill down to the board and keep it down. To do this click on the icon two to the right from the one in step 3 till it looks like a valley.
- 5) Click the button to the left of the one in step 4 to put the drill down to the board
- 6) Now click the Up arrow button. This will move the drill 10 units in the X direction
- 7) Adjust the Black disk on the drill housing until the copper is just barely removed from the board.

You will need to move the drill on the Y plane and do steps 2 through 7 until a desired copper removal is obtained

