CREATING A CAPTURE DESIGN
CREATING A NETLIST FORM CAPTURE
IMPORTING TO LAYOUT
ROUTING IN LAYYOUT
EXPORTING FOR CIRCUIT CAM
SELECT A FOOTPRINT
CREATING A FOOTPRING
SETTING UP A GRID
IMPORTING TO CIRCUIT CAM
BOARD MASTER
TOOL PLACEMENT

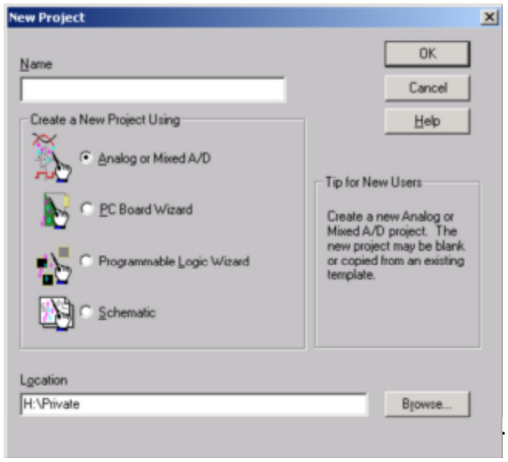
Creating a Printed Circuit Board (PCB) using Capture and the LKPF ProtoMat

By Timothy Frey, Nick Clingler, Marry Nealy

CREATING A CAPTURE DESIGN

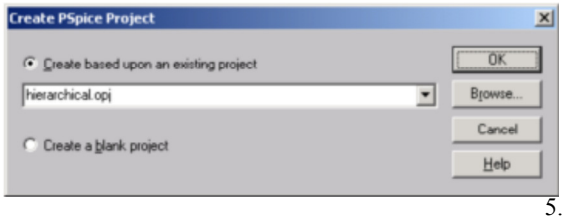
1. Open Orcad Capture; by selecting the Start menu and selecting Cadence PSD, then select Capture CIS.

2. Go to File; New; Project. The following window appears.



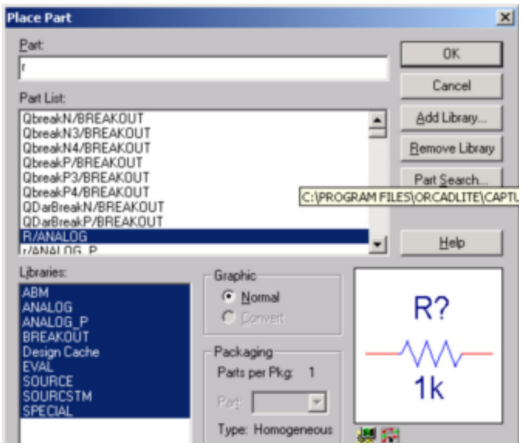
. Type in a name for the project. Then select Analog or mixed A/D. In the location window choose where you want to save the project, and click OK.

4. Now you should see the following window; select Blank project and click OK.



5. You should now have two open windows. One is a blank schematic page, the other is you project manager window. Click on the schematic page and a tool bar appears at the right hand side of the screen. This is the schematic editor bar.

6. To place a part click on the parts button . The following window appears.



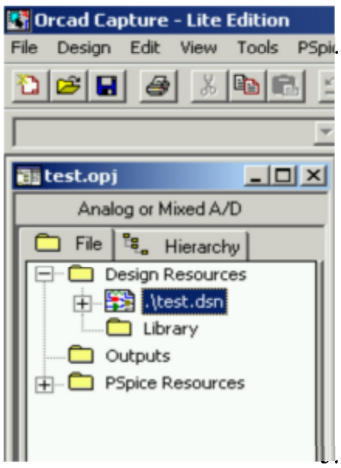
7. Select the part you need and click ok.
8. The part is now attached to your pointer. Click the place on the screen where you want the part to be.

Note: a copy of the part will stay on your pointer until you hit escape, or right click and end mode.

9. To connect the parts click on the wire button . Now click on a lead and drag the wire to the lead it needs to connect to than click again. Hit escape to end the wire, or right click and end mode.
10. Once you have finished click on the project manager window and save the file. This will save the schematic and the project.

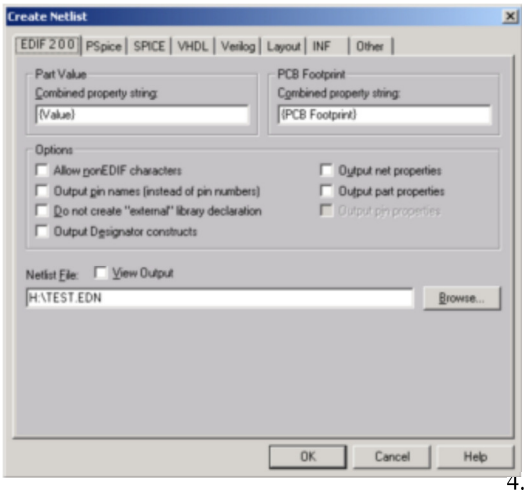
CREATING A NETLIST FROM CAPTURE

1. Save your project by selecting the project manager window and clicking save.

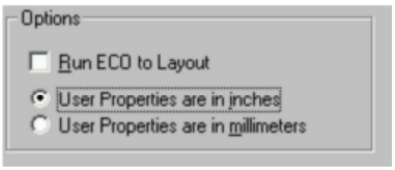


Now highlight the .dsn file in the project manager window as shown.

2. Now select Tools, on your toolbar, and select, Create Net List. Then the following window will appear;



4. Click on the Layout tab. In the options block choose ether inches or millimeters.

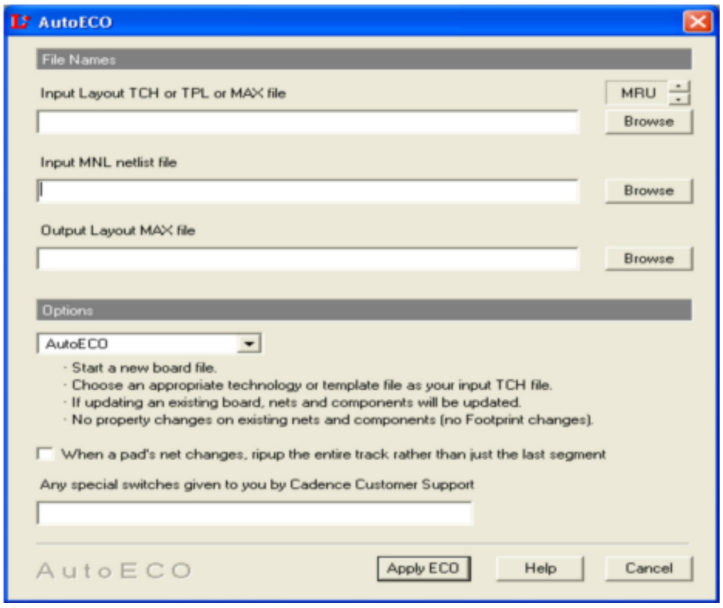


5. Now select where you would like to save the file and click OK.

Note: The program will save the file to the same folder as your project by default.

IMPORTING TO LAYOUT

1. Open Layout, go to File, and select New. The following Auto ECO window will appear.



2. Select the first Browse. You will see the Input Layout window, select default.tch click Open.

Note: Selecting a different .tch file will give you different templates; however you may need to change the form of units in the capture output.

3. Select the second Browse. You will see a second Input Layout window; select the folder your project is in, then select your project .mnl file (example filename.mnl).

Note: The third Browse will be an automatic selection. You do not have to select for this one.

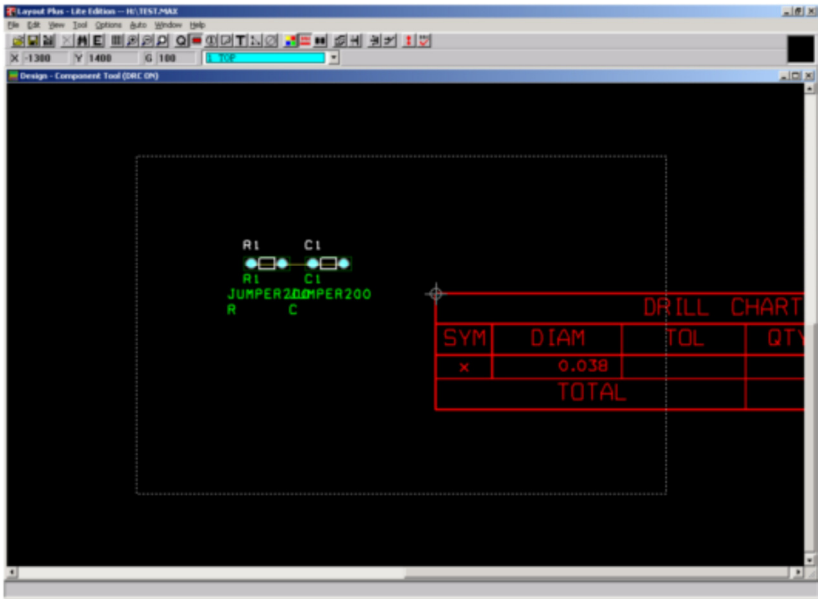
4. Now Click Apply ECO.

5. You will see the ECO Text window if all footprints have been assigned.

Note: If a footprint is not assigned it will ask you SELECT A FOOTPRINT.

6. Click Accept this ECO

7. When small Auto ECO window pops up select OK. You should now see something like the window below, if you don't see this try zooming out until you can.



8. Now click on the component tool button in the tool bar . Now click on a component and drag it to where you want it and click again. Repeat this until you have all the parts in place. You can rotate a footprint by hitting the "R" on you keyboard after the footprint has been selected.

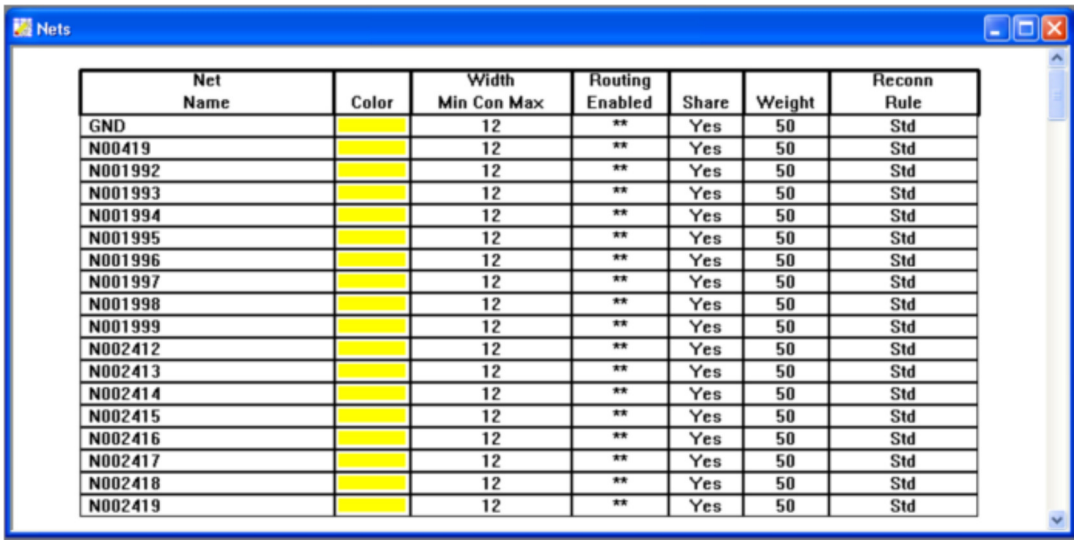
NOTE: If you would like to remove some of the text click on the T button ; then click on the word you would like to remove and hit delete.

If you want a measurement grid go to SETTING UP A GRID.

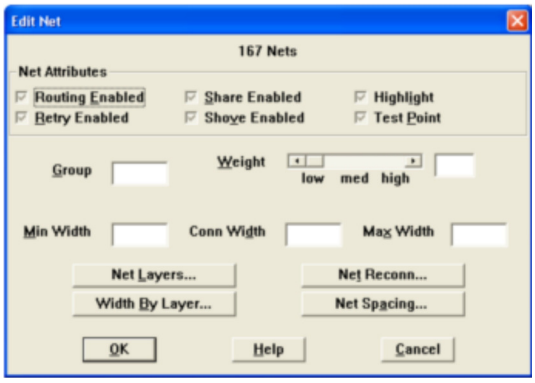
ROUTING IN LAYOUT

Once all footprints have been placed it is now time to rout the board. First we need to set up the net layers that are available to our router.

1. Select the Windows menu and select net. This will open the Net window.

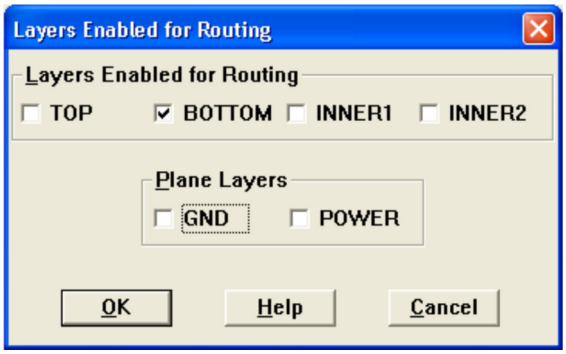


2. Click the Net Name box to select everything. Now right click and go to properties. The edit net window opens.



3. Now click on the check till the box is empty.

4. Next select the button. The following window will open. Click on each block so that it appears just as shown. Click OK.



5. Now reselect Routing enabled so that it appears as so… . Click ok and close the net window.

6. Now reopen the Design window and select the Auto menu; then select Autoroute and board. This will route the board for you. When it is done click ok.

EXPORTTING FOR CIRCUIT CAM

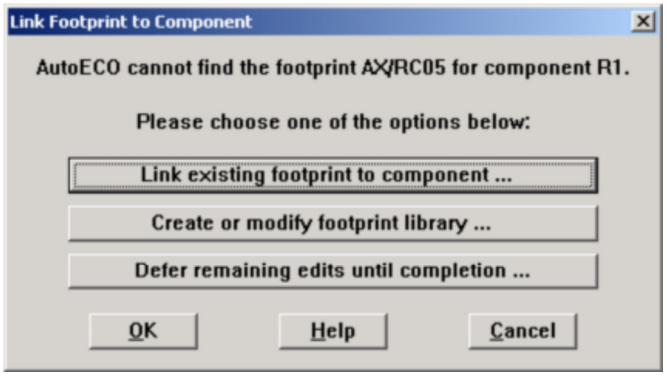
Once your board has been placed and routed we are ready to take it to circuit cam.

1. Go to the Auto menu, select Run Post Processor. Two popup windows will appear click OK for both. Then a final window will appear in notepad. This window shows the different tool assignments. Close it and you are ready to go to Circuit Cam.

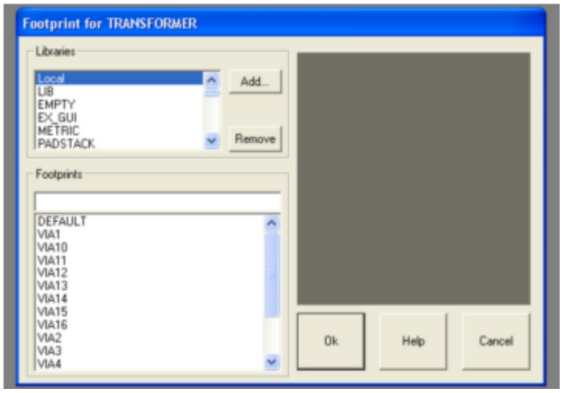
SELECT A FOOTPRINT

If all of your parts have not been assigned this window will appear.

.



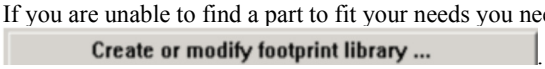
If you think a footprint might be available select link existing footprint to component
The following window will appear.



The top list is a list of libraries; the second list is a list of parts in the highlighted library. The gray window will show you what the part looks like.
Once you have found the part you are looking for click OK and it will connect the footprint to your part.

CREATING A FOOTPRINT

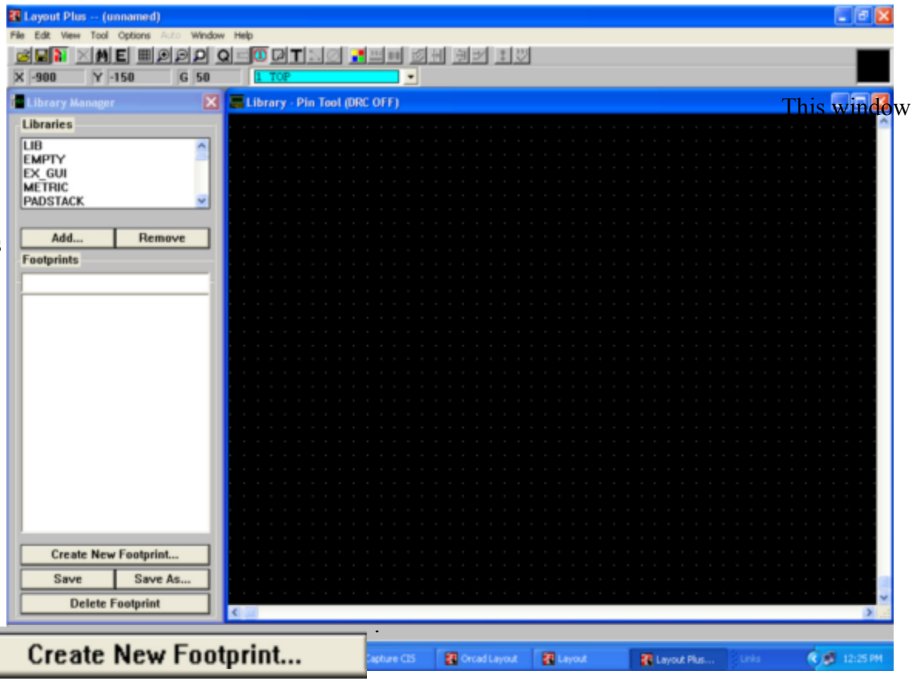
If you are unable to find a part to fit your needs you need to create your own. This is done by clicking



Next you will see the fallowing window.

This window shows the libraries available for the part editing

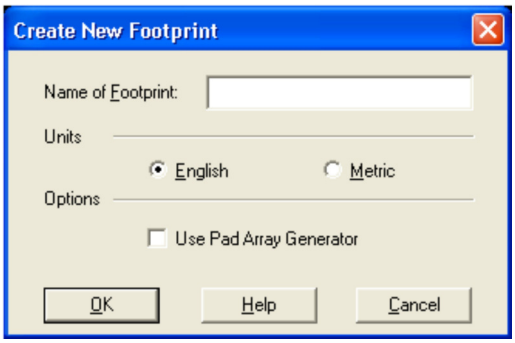
This window shows parts in the libraries selected



This window is

To create a new footprint click on

Then the following window will appear.



Enter the name of your footprint and the unit you are using and click ok.
You should now see something like this…

The blue dot is your first pin for your component. To get the next pin right click and select new. Repeat this until you have as many as you need and have them placed where you want them.

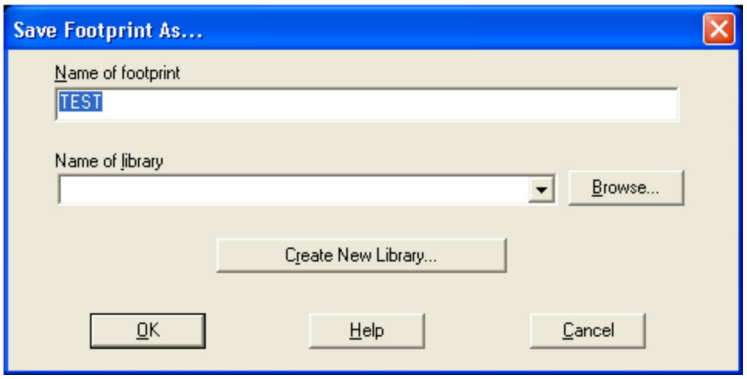
Next you need to place a part outline.
In the tool bar select the optical tool.

(it looks like this)

Now you can out line the part. When you hav

Next click

and the following window will open..



The name of the footprint will already be in place. You can ether save it to an existing library by clicking

or you can

and save it there.

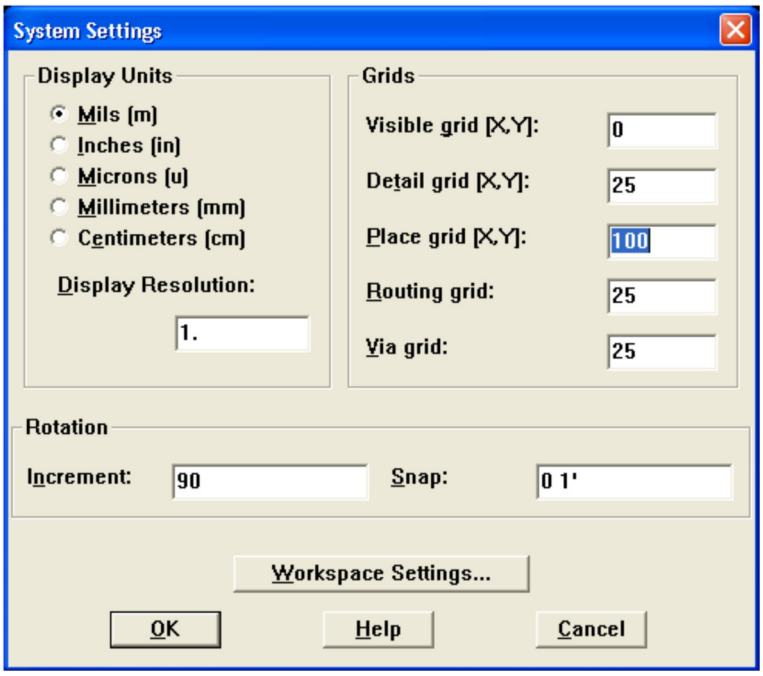
Once you have created and saved the new part you have to go back and click

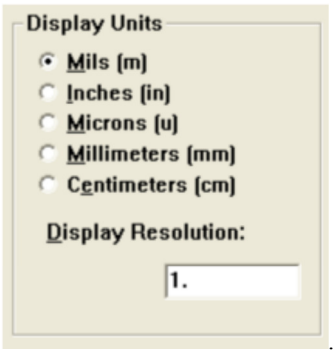
to link the footprint to the part.

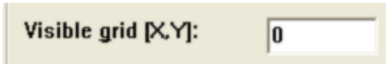
SETTING UP A GRID

1. Open the options menu, and select System Settings.

2. The following appears;



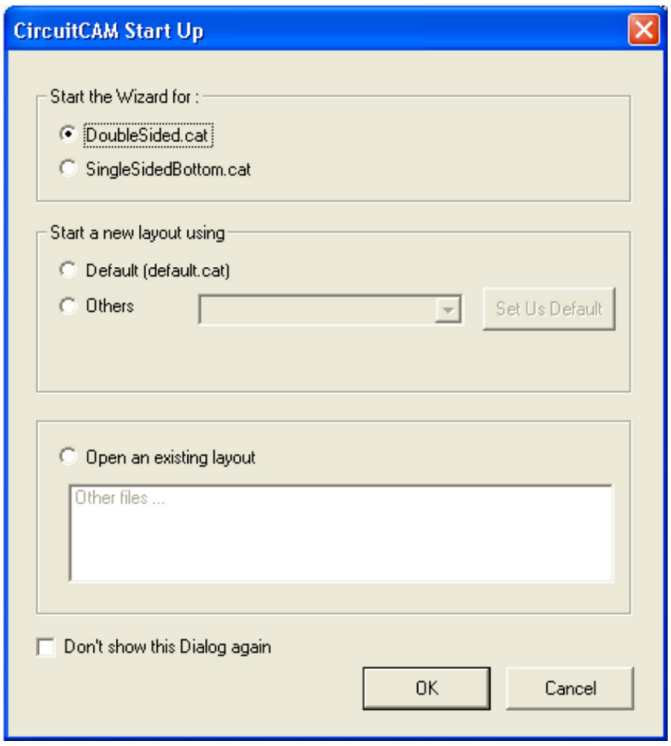
3. Select the units you wish to use 

.
4. Now enter the distance you want each dot in the 

box. The units have to be in decimal form. (.25 in = ¼ in)
5. Now click OK.

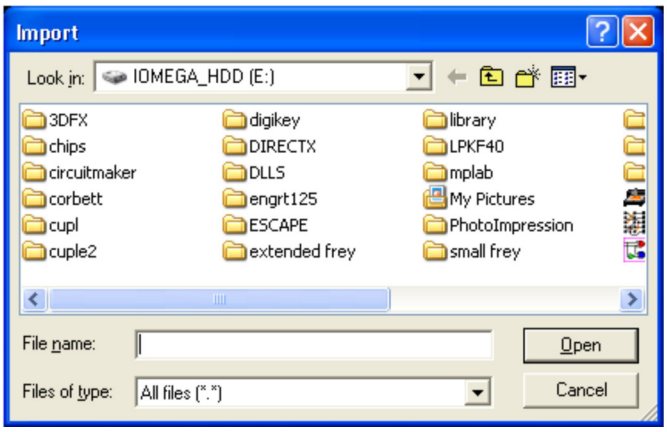
IMPORTING TO CIRCUIT CAM

1. Click on the circuit cam icon and the following window will appear…



2. Select and it will open you in the default editing window.

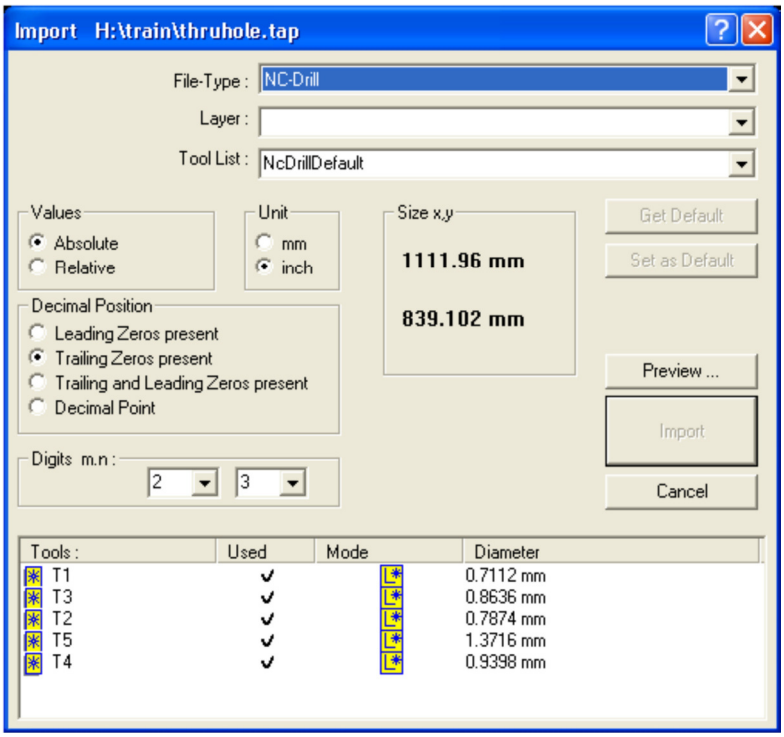
3. Now select the import button and the following window will open.



Go to the location of your project.

4. Now select the following in that folder.
thruhole.tap, .ast, and .bot
The .ast file is your top layer, the .bot is your bottom layer and the thruhole.tap file is the drill assignments.

You should now see the fallowing window



5. Thruhole.tap is your drill plated layer in the layer window select drill plated,
   .ast file is your top layer; in the layer window select top layer.
   .bot file is your bottom layer, in the layer window select bottom layer.

6. In the digits section change the 2 to a 3 and the 3 to a 4 .

7. Now click import.

   Now you should see your board from layout.

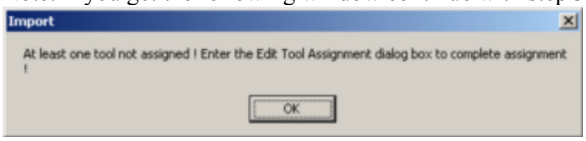
8. Now click on the button to insulate the tracks.

9. Then click on the button to send it to board master. It will prompt you to save the cam file and then tell you that a .lmd file was created. Click ok.

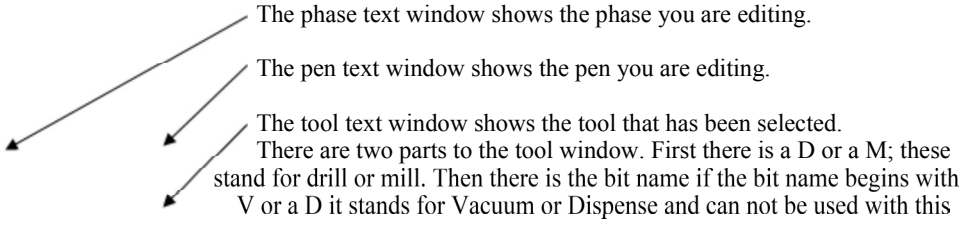
BOARD MASTER

1. Click on the board master icon to open the program.
2. Now click file, import, LMD/ LPR. Then you should see an import project window open; go to the folder with your project and open the .LMD file. This will open you board into board master.

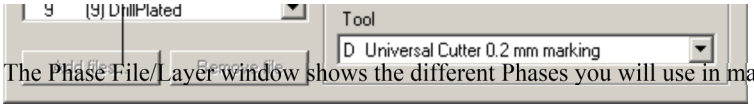
Note: if you get the following window continue with step 3 if not go to step …..



3. Click ok. Then go to the edit menu and select tool assignment and the following window will open.



a



machine.

The Phase File/Layer window shows the different Phases you will use in making the board phases 8-14 do not apply to this machine.

You will need to reselect each tool for each Pen in each Phase.

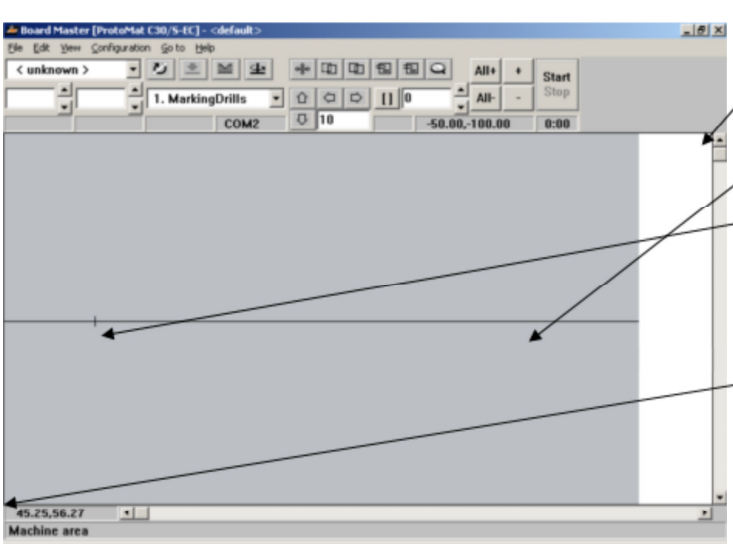
1. Select phase 1; then select the pen scroll button and select the first pen. Now choose the tool you would like to use for this phase.
2. Now select the next pen and tool for that pen. Continue till each pen has been assigned.
3. Repeat step 1 and 2 for each phase.

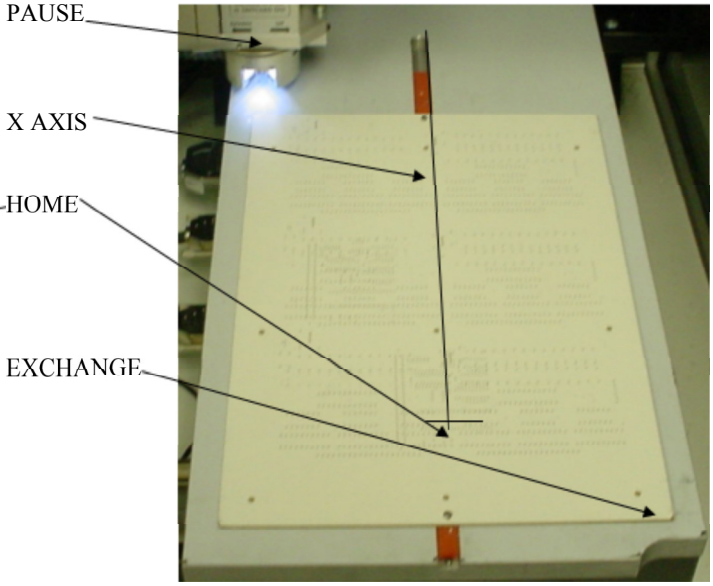
Note: For phases 1,4, and 5 select the D Universal Cutter 0.2 mm marking tool.
For phases 2 and 3 select a bit according to the following table

Bit request	Bits we have
.1-.5 spiral drill	.5 spiral drill
.6-.8 spiral drill	.8 spiral drill
.9-1.1 spiral drill	1.0 spiral drill
1.2-1.3 spiral drill	1.3 spiral drill
1.4-1.5 spiral drill	1.5 spiral drill
1.6-1.9 spiral drill	1.8 spiral drill
2.0-2.9 spiral drill	2.0 spiral drill
3.0 and up spiral drill	3.0 spiral drill

4. Once all of the tools have been assigned click ok.

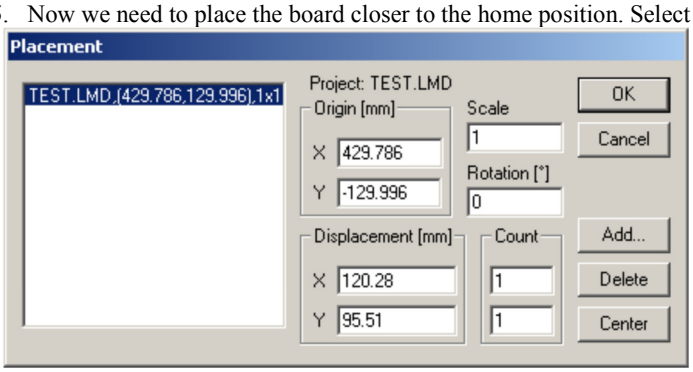
Now we need to place the board.





As seen on the screen
machine

5. Now we need to place the board closer to the home position. Select the edit menu, and placement. Then the following window should open.



6. In the origin [mm] section decrease the X value until your board is only a short distance from the home position on the screen.
Note: If you have enough space and need more than one of the same boards increasing the count value will add boards.

7. Once your board is in place make sure there is a copper board on the machine. When making a capper board for the machine You need to cut it to fit the white base board and put a hole at each end for the metal studs. Then place the board on the machine and tape down each corner. Make sure the tape is as close to the edges as you can and still hold the board down.

Once the board is in place you are ready to start. The following buttons will help you in the process.

Start motor and stop motor

Lowe bit manually

Mill and drill

Move bit head manually to a location

Move bit in x directions

Move bit in y directions

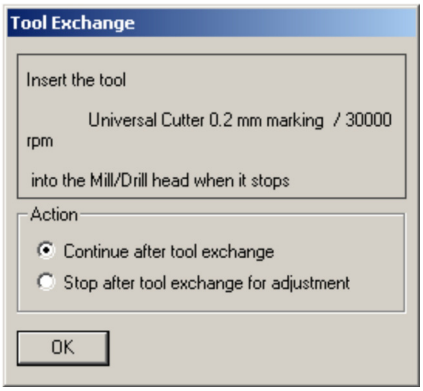
Zoom in

Selecting a phase

Select all

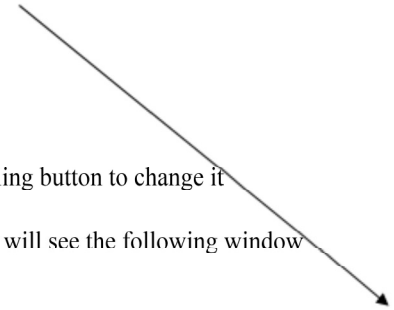
Start and stop

8. Now turn the motor on. And select the first phase. Select all and start. The following window should open.

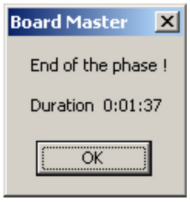


9. Insert the tool it is asking for and select stop after tool exchange for adjustment. Then click ok.
Not to insert a tool go to Tool Placement.

10. Now we will set the bit depth for the remainder of the board. Select the mill or drill button so that it looks like . Now select the head placement button and click on the screen in a place where your board is not and turn the motor on. Now lower the bit head and move the head in the x direction and raise the head and move it out of the way. Look at the track it just made if it is too deep raise the bit if it is to shallow lower the bit.
Note to raise and lower the bit depth move the dial as shown in the picture.



11. Once the bit depth is set to the depth you want select the mill/drilling button to change it back to drill. and click start. When the Phase is done you will see the following window



Click OK

12. Now select phase 2, select all and start. It will ask for another bit change the bit and select start. If there is nothing in the phase then nothing will happen; move to the next phase.
13. Repeat step 12 for phase 3.
14. When you select phase 4 it will ask for a tool. Place the tool and repeat step ten. Once the bit depth is set make sure that the milling/drilling button still looks like this . Now click start.

1. Once phase 4 is done you will need to flip the copper board, make sure when you flip the copper board your circuit stays close to the home position and the metal studs stay in the same holes as before.
2. Select phase 5.Now we need to realign you circuit board with the computer. To do this select view and both sided, then zoom in on a drill hole in one of the corners of the board on the screen. Now select the placement button and click on the hole. This should move the bit head on the machine to where that hole should be. It will be off select edit placement and in the origin section adjust the origin x and y till the bit lines up with the hole. You can change a little as the third decimal point.
3. Once the bit is lined up to the hole select phase 5 again, then select all and start. When this phase is done you board is done.

TOOL PLACEMENT

To insert a tool lower arm at the side if the bit head and move it to the back. Remove the bit that was there and place the next one all the way in then move the arm back to place.