Training Course – Basic Course

This booklet is intended for users who either have never used the software and the machine before, and/or for users who desire a refresher course in the basic operation of the software and/or hardware. The intention of this training course is to allow the user to be able to come to a thorough understanding of the software and the machine, and to be able to operate and use the software and machine on their own.

Table of Contents

CircuitCAM Basics – Fundamentals of CircuitCAM

**This section of the training booklet is designed to give the user a thorough understanding of the CircuitCAM software interface and to teach the user how to import and process the various types of cad files. It is the intention that this section will allow the user to operate the software own their own, to be able to understand the proper usage and implementation of the software at the end of this section.**

CircuitCAM Interface And ToolBars

CircuitCAM is a very versatile software program. The software interface is easy to use, and utilizes various toolbars to accomplish it's many functions. In the next few sections, we will run over the tool bars and their functions in the software interface.

Introduction to software interface

File Menu commands

File   Edit   View   Insert   Select   Config   Window   Help
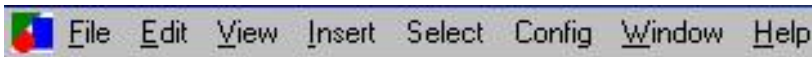
The 'file' menu, consists of the commands for opening and saving your files, as well as importing the basic cad layers. As well as exporting the results of your work out to a format to be recognized by BoardMaster.

Edit, allows you to perform various modification commands on your board drawing, such as cut and paste, and manipulation of your drawing.

View, allows you to change the way your graphics are viewed on the screen, such as, zoom in and out.

Insert, is where you will find the commands that allow you to add in additional elements into your board drawing.

Select, is where you will find the ability to make selectable, various elements in your drawing.

Config, contains the list of Format Configurations, Job configurations, And the General settings box, where you set up the file directories and also enter in your user information.

Keep in mind, that many of the functions found in the file menus, are duplicated with the tool bars. In most instances, you will be doing most all of your processing with the tool bars alone.

Main toolbar

> **The main tool bar**, combine's features from the 'file' menu and the 'edit' menu. With this tool bar, you can open, import and save files. You can also print, copy, paste, undo and redo the last command from this tool bar.

View toolbar

> The **View** toolbar, is used for changing the way your file is viewed on screen. With this tool bar you can, zoom in and out, get an overview of your drawing, and refresh the screen to bring back the graphics.
>
> The last Icon in this group, takes you to the '*layer*' menu, where you can change the way your layers are displayed. In this sub screen, you can change the *color* of an individual layer, as well as the *order* in which the layers are set on top of one another. In addition, you can change the *displayable qualities* of individual layers, like **real view** or **outline view**, and whether a layer is **visible or not**, and even make a layer **selectable** or **non selectable**.

Selection toolbar

> The '**select**' toolbar, contains 3 sets of icons. The first set of icons, allow you to select either all of a certain layer, All of a certain aperture type in all layers of your drawing, Or to select the entire drawing.
>
> The next five icons on this toolbar, allow you to *turn on and off* the *selectable qualities* of the different graphical elements in your board drawing.
> And the last three icons on this tool bar, change the way the selection you highlight on your drawing, will be treated. The obvious correlation is that the '**plus**' adds to your selection, and the '**minus**' takes away from your selection, while the 'equal' sets up for a new set of selections.

Graphics toolbar

> In CircuitCAM, the **graphics toolbar**, is where you will be able to create new elements in the drawing. The first icon, is the '*Selector*' icon, which functions to allow you to 'select' various elements of your board drawing, for further processing and analysis.
>
> The remaining icons are almost self-explanatory, allowing you to insert into your drawing, a graphical element of the shape shown on the icon.

Grid and unit toolbar

> The '**Grid and Unit**' toolbar, has a multifunction in the way it works. The first two icons, deal with setting your drawing's virtual '*zero point*'.
>
> The next single icon in this toolbar, is the '**Match Layers**' icon. You would use this icon to '*match*' up any offset layers.
>
> The next set of four icons, change the way that your cursor co-ordinates are displayed in the *status bar window* at the bottom of the screen. The first set of two icons, switches between *absolute* and *relative* display mode. The second set of icons, switches between '*Cartesian*' and '*polar*' cursor display mode.
>
> The last three drop down windows in this tool bar, are as follows: The first drop down window, changes the **measurement unit** between *metric* and *imperial* measurements. The last two drop down boxes, in order from left to right, are '*display and snap-to-grid*' settings, respectively.

Prototyping toolbar

Now let's take a look at the '**Prototyping**' toolbar. This tool bar is where you will export your drawing out into a file that 'BoardMaster' will be able to import, perform the insulation function, for top, bottom, or both at the same time. Keep in mind, that you would set up the parameters for your insulation jobs, under the '*Edit/Insulation*' menu at the top of the screen. You can also create an *unplated* or a *plated thru-hole* in your board drawing with this function.

The next three icons, allow you to set up what is known as '***rubout***' areas. You utilize these three icons, by selecting the icon to do the rubout on the layer you desire to rubout, and then you move to the section of your drawing you want your rubout to start at. Next, all you have to do is *click and drag* to create a rubout area.

The third from the last icon in this toolbar, is the **Contour Routing icon**. This icon will let you perform either an inside contour routing job, to create *slots* and *cutouts*, or *outside contour routing*, to define a board cutout area.

The next icon, is where you would select and apply any '**breakout**' tabs on your '*cutting outside*' job. This allows your board to be held in place in your base material, by a width of material which is defined in your '*contour Routing*' dialog box.

Front to end toolbar
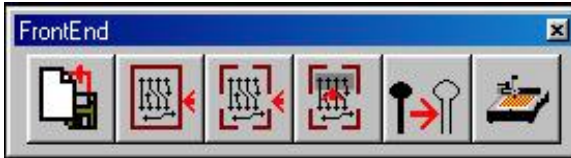
This toolbar contains everything you would need to process your board drawing from start to finish. The first icon is the '**Import**' icon. This is the icon that imports your various cad file layers into your working area.

Next, is the '**Contour Routing**' icon, which you will use to define your '*board cutout*' path.

Next, is the '**Breakout Tab**' icon, where you will set up your defined area's of breakout tabs, used to hold your board in the base material for any further processing after it is cut out.

Following the '*Breakout Tab*' icon is the '**Rubout All Layers**' icon, which lets you define areas of rubout for your board drawing.

Then next icon you will find on the '*Front to End*' toolbar is the '**Insulate All Layers**' icon. This option performs the insulation function on all layers, using the parameters specified in the 'Insulation' settings dialog box.

And last, but not least, is the '**Export LpkfCircuitBoardPlotter**' icon. The '*ExportLpkfCircuitBoardPlotter*' icon, will export your prepared board file, out to an '*LMD*' format which is read by the BoardMaster program.

Importing Files

Now that you have become familiarized with the CircuitCAM interface, let's run through the software in an actual front to end cycle, importing and processing a real circuit board. Let's start out by opening up your CircuitCAM software, from the **Start menu**.

Next, let us begin the process by selecting the '**IMPORT**' icon on the *front to end toolbar*. Navigate to your '**LPKF32**' directory, and choose the '**Data Directory**'. The board drawing we are going to import into CircuitCAM, is named '*Sample*' You will notice that this directory contains four files with the name of '*Sample*', each with a different 'extension'.

Normally, we would recommend that the cad files come with names or extensions that are self-explanatory as to what layer they are. In this case, we can see by the extensions here, 'BOT, DRD, dri and TOP', are 'Bottom side, NCDrill file, Tool List and Topside', respectively.

Importing Top Layer

We will start by importing the Top layer first. Select the '*Sample.TOP*' file and then click on '**OPEN**'. You will notice at the top of the dialog box, the file type is *GerberX*, and the Aperture list reads '**GerberDefault**'. The '*Layer*' box at this time will be empty. In the '*Layer*' drop down box, choose '**TopLayer**'. Click once on the word '**Gerber Default**' in the Aperture list window, and type in the word '**top**'. This will set the name of the aperture list for the TopLayer', to '**top**'. Now, click on '**Preview**', to get a visual view of the board drawing layer you are about to import.
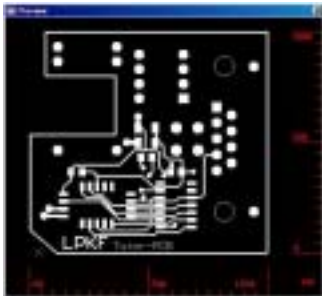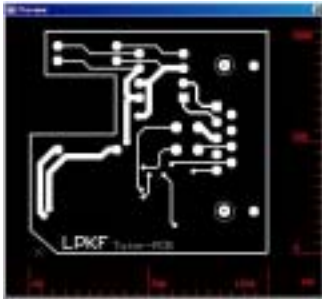

It should look like the one in this picture.

Close the window, and click on the '**Import**' button, to bring the Gerber layer to the CircuitCAM interface.

Importing Bottom Layer

Next, Bring in the *Bottom Layer*, by selecting the **Import** icon, this time, selecting the *Sample.BOT* Gerber file. Change the **LAYER** field, to **BottomLayer**, and change the *Aperture List* to '**bot**'. Now, click on '**Preview**', to get a visual view of the board drawing layer you are about to import

It should look like the one in this picture

Close the window, and click on the '**Import**' button, to bring the Gerber layer to the CircuitCAM interface.

Importing Drill tool list

Next, you will need to import in a *tool list* for the **Excellon** data, which will enable your drill files to actually contain *size information*. Use the **Import** icon once again, to choose the file "**Sample.dri**" to import. What you see displayed now, is the dialog window for importing the tool list.
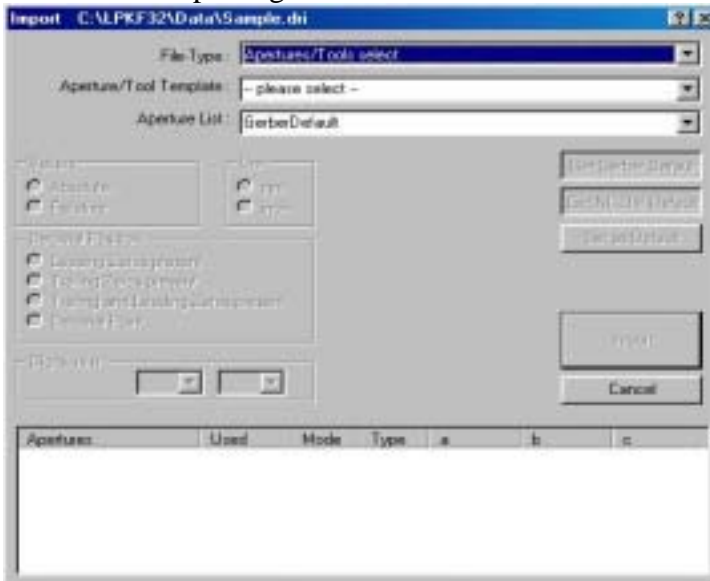
The '**File Type**' box should read, "*Apertures/Tools select*". In order to translate the *Tool list* into a format that CircuitCAM can read, in the '**Aperture/Tool Template**' dialog box, find and choose the selection '*Eagle_Excellon.txt (NCDrill)*. *Click on the words in the dialog box* for the Tool List and type "**drill list**" Click on the '**Import**' icon, and your tool list for your NCDrill data will now be imported into CircuitCAM.

Importing Drill files

Now we are ready to import the **NCDrill files**. Select the **Import** function again, and select the '**Sample.DRD**' file. The information in this dialog box should look like
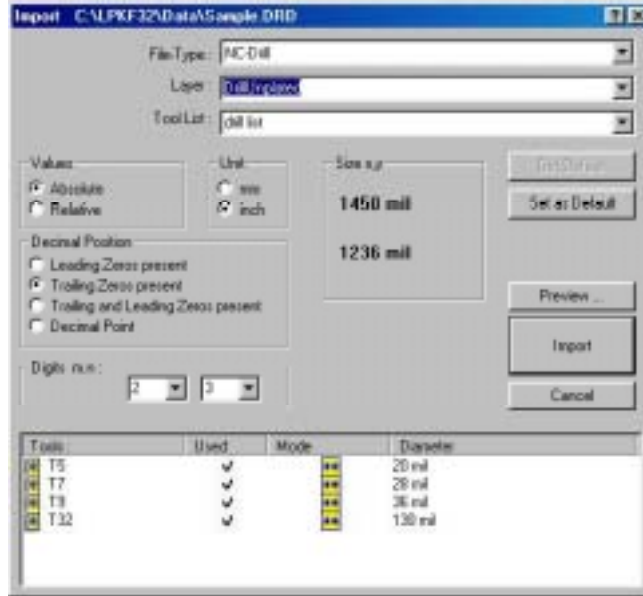
It should look like the one in this picture:

this:

File-Type should read, *NC-Drill*. In the **Layer** dialog box, you will select '*DrillUnplated*' The '**Tool List**' should display '*drill list*', which is the tool list you just imported in the last import session. Now, click on '**Preview**', to get a visual view of the board drawing layer you are about to import

Close the window, and click on the '**Import**' button, to bring the Excellon layer to the CircuitCAM interface.

Contour Routing

Contour routing, is how the software knows to set up the board *cutout path*. This cutting process will take its information from the *layer* which contains your *Board Outline information*, and after calculating the path, will place it on the **Cutting Outside** layer. Select the **Contour Routing** icon,

and a *dialog box* will be displayed. This *dialog box* is also used to perform any *inside routing* for *slots* in your board design

.
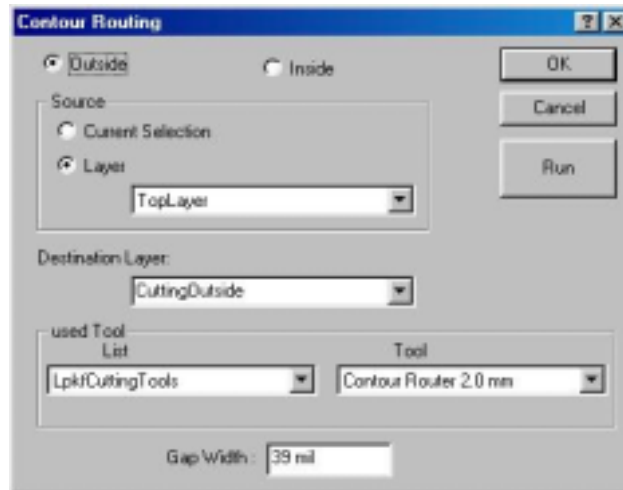
Choosing Inside or Outside

At the top of this dialog box, you will find two radio buttons. They choose which *type* of a routing job you want to perform, in other words, whether the routing will occur on the *inside* of your path, or on the *outside* of your path.. We are going to be performing *outside contour routing*, so we will choose the **Outside** button.

Source and Destination

As our *destination* for the calculations, we will want to choose the '*CuttingOutside*' layer in the *drop down box*. The *Source* for this **contour routing**, will be chosen in the *Source box*. We will choose '**Layer**', and in the *drop down box*, select '**BottomLayer**' as our *source*. A preferred but unnecessary source would be from the '**Board Outline**' layer.

Tool list and Tool Selection

The '**used Tool list**', for most all purposes, will be the '**LpkfCuttingTools**' in the *drop down box*. The '**Tool**' that you will choose to use in the *tool drop down box*, should be decided by the *size of the board* to be cut out. The *larger* the board, the *larger the tool* to use. In the case of this small sample board, we can use a **1mm Contour Router**. The '**Gap Width**', will set up the *size* for any '**Breakout Tabs**' that you may want to attach to the board after you perform the **contour routing**. For our board, a **39mil** or **1mm** size will be good.

Running the Contour Routing Job

Now, we can click on the '**Run**' selection, and the *contour routing* will be calculated and placed upon the chosen *destination layer*.
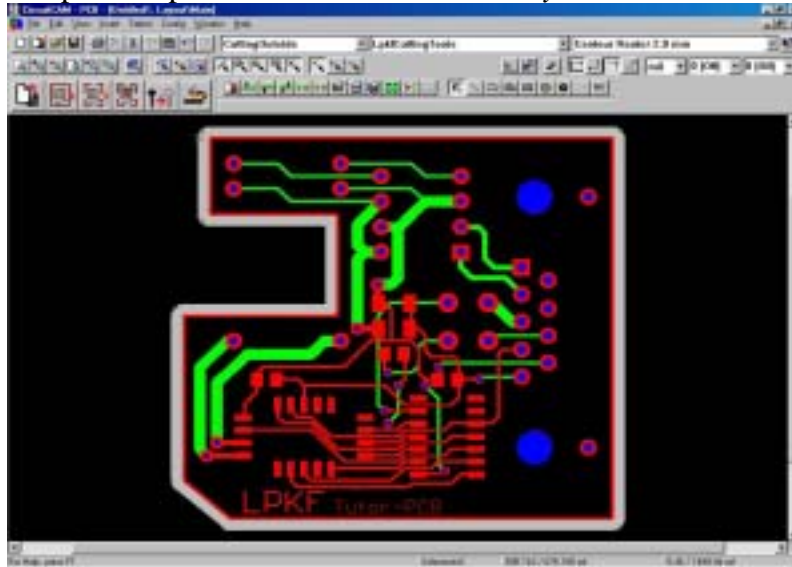
Your Board drawing should now look like this:

Breakout Tabs

After we have our **Contour Routing** completed, we can insert **Breakout Tabs**. These *tabs* will hold the board into the panel, until you are ready to twist it out. To insert **Breakout Tabs**, you must first *select and highlight* the **Contour Routing**.

This is done by clicking your mouse *outside* of the board drawing *next to a corner* on a side you want to place a **Breakout Tab**. Click next to the top left hand corner, and use the **plus** and **minus keys** on the keyboard, to move the selection point *clockwise* or *counter clockwise* until you reach a point where you will want to *insert* a **Breakout Tab**.

Please note that the *numeric keypad minus key* will move it *clockwise*, and the *numeric plus key* will move the selection *counter clockwise*.

Now, click your mouse cursor on the icon in the **front to end tool bar**, labeled

'Breakout Tab'

A gap of the *previously defined dimensions*, will be placed in the contour routing selection. Repeat this procedure until you have inserted all the **breakout tabs** that you will need for your *Contour Routing*. Now, depress the **escape key** on your keyboard to **deselect**, and you are done with the contour routing.

Examine your drawing

Now that you have all of your layers imported in to CircuitCAM, it is time to **examine your drawing**.

Discover Smallest gap

Use the '**Zoom In**' function to view in detail, the various sections of your drawing. Refer back to the section titled '*View Toolbar*' for information on how to do this. You will need to look for the *thinnest gap* between your traces and pads on your drawing. This will help you determine the smallest tool size needed during the '**Insulate**' process.

Use the *context menu*, accessed with a *right mouse click*, to select the '**Measure**' function. Use the *left mouse* button to **anchor** the mouse cursor at the *starting position*. Now *move the cursor to the point* that you want to measure the *distance to*, from the **anchor point**. You will notice, at the *bottom left corner* of your CircuitCAM window, the *measuring distance* will be displayed in the **units** you have set up in your '**Grid and Unit**' setting.

Distance : 0 / 9.036 mil : 9.036 mil / 90.0°   Selected=0

Take notice of the distance, and keep examining your drawing until you are sure that you have located the *smallest gap* on your drawing.

Determine proper tool

You will also have to *examine your drawing*, to determine the **rest of the tools** that you will need to insulate your *pads and traces*. You must know whether you plan on doing any **rubouts** on your board, and on *what layer* you might be placing the **rubouts**. Determine which tool is to be your **standard tool**; this tool will be your *outlining tool*. Also determine if you have areas large enough for a **big & bigger** tool, for functions such as **rubouts**.

Correcting any errors

Take a good look at your project and try to spot *any errors* in the drawing.

Determine Any Rubout Areas

There are a number of different methods for defining a '**Rubout**' area, or in other words, an *area of excess copper removal*. One way is to *manually draw* an **open path shape** around the area you would like to **rubout**.

Make sure you draw this open path on the 'rubout' layer for the side you would like to rubout. The simple method is to use the **front-to-end toolbar**, using the icon labeled

'**Rubout All Layers**'. Select this icon, and then *draw a rectangular box* around the area you would like to **rubout**.
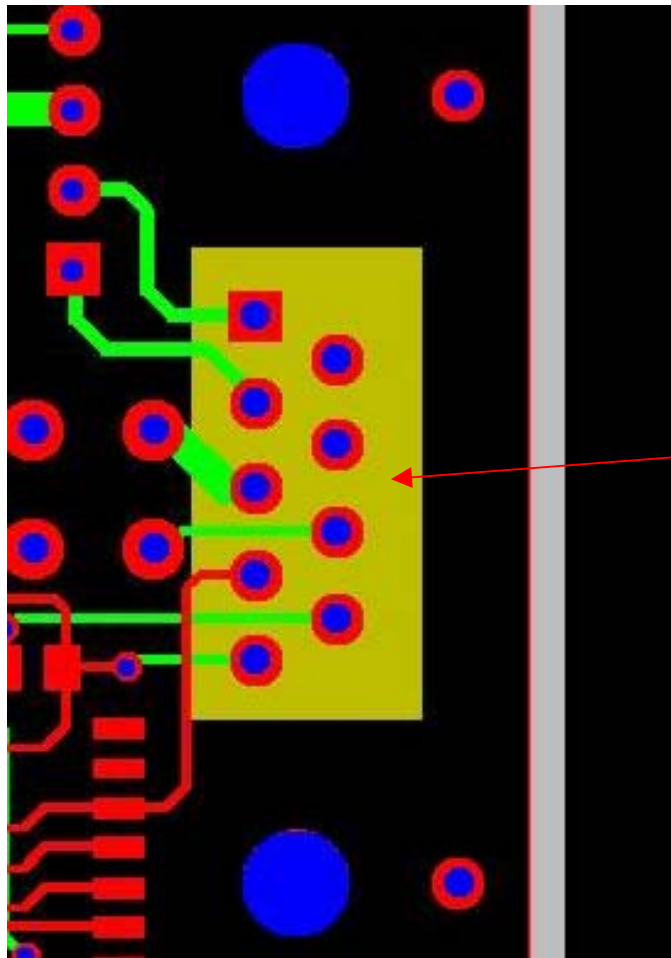
This Rubout area is normally an outline, but has been Changed to a solid for purposes of clarity.

This will place a **rubout** of the specified size in your drawing where you selected it. This will place a **rubout** on *both sides of the board.*

Rubout Top

To place a **rubout** on only the *top layer*, either choose the **Rubout TopLayer** choice from the *drop down* '**Layer Selection**' menu, or choose the **<u>Rubout Top icon</u>** from the **prototyping toolbar**.

Then, choose the *desired shape* of your path for the **rubout** area, by *selecting a shape* from the **graphics toolbar**, or use the *default setting* of an open path rectangle.

Now, simply *draw the shape* of the **rubout** area by *clicking to set your first point*, and then *drag* to **set up the remaining point or points**, and **<u>click to set</u>**.

Hit **escape** to *deselect*.

Rubout Bottom

To place a **rubout** on only the *BottomLayer*, either choose the **Rubout BottomLayer** choice from the *drop down* layer selection menu, or choose the **<u>Rubout Bottom icon</u>** from the **prototyping toolbar**.

Then, *choose the desired shape* of your path for the **rubout** area, by *selecting a shape* from the **graphics toolbar**, or use the *default setting* of an open path rectangle.

Now, simply *draw the shape* of the **rubout** area, by *clicking to set your first point*, and then *drag* to **set up the remaining point or points**, and **<u>click to set</u>**.

Hit **escape** to *deselect*.

Performing Insulation Job

<u>Setup Insulate parameters</u>

Now that you have *examined your drawing* and specified any **rubout** areas, you are ready to **set up the insulate parameters** and **run your insulation job**. One thing to know, is that the *insulation job* only defines the **milling paths**. The **drill data** is defined during the **<u>export function</u>**. You will need to access the *insulate setup parameters* from the '**Edit**' menu by choosing '**Insulate**'
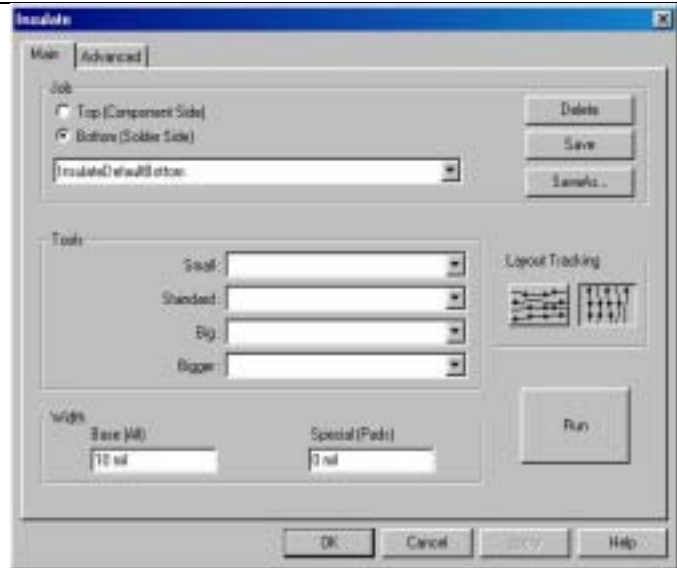
<u>Selecting tools for the jobs</u>

When you first open up the **insulate dialog box**, you will see the '**Insulate Default Bottom**' job displayed. In the *center section*, you will notice either *two* or *four* **drop down windows**, depending upon which *version* of CircuitCAM you have. This is where you will *select the tools* to do your job.
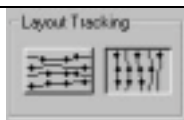
The **Small tool** selection box is the tool that the machine will use to *fit into any areas that are too small* for the **standard tool** to fit into. The *Standard tool* is the **main milling tool**. This is the tool that your machine will use to do all of the pad and trace *outlining*, unless it reaches an area that is *too small* for it to fit into. In this case, the plotter would use the *Small tool*. The *Big tool* will be used to perform any **rubouts** defined, unless **two conditions** are met.

These conditions are as follows: **1)**, if the rubout area is *too small* for the **big tool** to fit into, or **2)**, if the software encounters any large **rubout** area *bigger than the big tool*, and you have defined a *Bigger tool* to use for these areas. The *Bigger tool* is used for rubout areas which are *bigger* than the **Big tool** that is defined. Unless the **rubout** area is too small for the *Bigger tool* to fit into, in which case, it will use either the Big tool or the Standard tool.

Now, make your selections based upon any **rubouts** you may have included in your Board drawing, selecting the *8-mil universal cutter* as your *standard tool*. You will need to set up the tools for both the *top and bottom* layers remembering to select the '**<u>Save</u>**' function after each side is setup. If you do not save your choices, it will change back to the default settings when you leave the dialog box.

Layout Tracking

The '**Layout Tracking**' icons, to the *right* of the **tool selection**

boxes, will add to the insulate decisions that the software will make. It does this by telling the software *which direction* most of your milling paths run on the *layer you are trying to insulate*. Choose whichever one fits the layer you are insulating. In our case, for the *bottom side*, we will choose the **Horizontal Layout Tracking**, because the majority of the tracks run from *left to right*. Do the same for both layers, and remember to save often, just in case you forget or your computer malfunctions for any reason.

Base (All) and Special (Pads)

The **Width dialog box**, allows you to set up a *base isolation width* that will be applied to all of the *pads and traces*. This, in general, will be equal in default, to the size of the *standard tool* that you are using. However, if you so desire, you can increase the width of the insulation for purposes of *solderability* and so forth. The **Special(Pads)** dialog box, allows you to set up a *separate isolation width*, and apply it to whatever selection that you desire. Usually, this will be applied to all pads in your drawing, as a barrier against soldering bridge issues.

Layers

For each insulation layer, there is an **advanced tab**. This is where you will find the further setup options that are available to you. While your '**Bottom (Solder Side)**' radio button is selected, click on the **Advanced Tab**.

You will notice, in the middle of this *dialog box*, there are 5 drop down boxes. These boxes specify from source to destination, how the insulation will occur. The Base (All) selection, tells the software where you want the calculations for this layer to be based upon. In this case, we are working on the bottom layer, so that is where we want the information to come from.

The **Special (Pads)**, tells the software to insulate the layer defined here, with the parameters you set in the **Special (Pads)** box in the *previous tab*. **Rubout-1**, is where the software will take its *first set* of **rubout** information from for the *insulate job*, and the **Rubout-2** is the *secondary layer* that the software will use to perform any **rubout** calculations. **Destination**, is where the calculated insulation paths *will be placed*. This is the *default settings* that you see here, and will be the settings that you will use most often. This is how we are going to set up our insulation job for our board.

Rubout Milling

Now we move on to **Rubout Milling** *dialog box*. This defines the *direction* the milling calculations will occur. The first two radio button selections are the **x-parallel**, and the **y-parallel**. They connect up directly to the *Layout tracking selections* on the **main tab**. The **x+y-parallel** option, is what you will set if you have an even amount of pads and traces going vertical as well as horizontal. And the **Concentric** selection is for any board that has a *circular area* that will need to be milled out in such a *circular fashion*.

Insulation grid, primary overlap and optional strategies

The **insulation grid** represents an *overlay grid*, which the software will use to calculate the milling paths. The *smaller the grid value*, the more *finite the calculations will be*. A *larger insulation grid* will result in a more *coarse calculation*. One thing you must know about this setting, is that the *finer the detail*, the *longer the calculation time* and *milling time* will be. The recommended setting for this is 0.005mm.

Primary overlap is used to determine how much of an overlap will occur on the milling passes, *between different tools in cases of rubout areas*. 80% is a standard default setting.

When the '**Remove Spikes**' box is checked, the software will perform *extra calculations* to *remove any excess spikes of copper* that was not included in the initial insulation paths.

The '**Inner Isolation**' box, when *unchecked*, will **not create** an *inner isolation channel*.

The '**Exact Base Width** ' check box, when *enabled*, keeps the specified **Base Width** for the isolation around all electrical objects

The **Independent Primary** check box, when *enabled*, is only available when there is **no** '**small**' **tool defined**. This will create an *independent primary insulation channel* independent of other areas. This will *guarantee* that **every connected area**, is really insulated from other areas, even if the gaps in the other areas are *smaller* than the **standard tool**. If this box is *not checked*, then, any gap that is *too small* for the **standard tool** to fit into, will ***not be insulated***.

Running the Insulation Job

Now, after setting up the parameters for *both sides* of the project, we are ready to begin the *insulation process*. On the **Front to End toolbar**, you will find the icon to '**Insulate All Layers**'.

Select this [icon] icon, and the software will begin its calculations.

Saving your Drawing

Now, before you proceed any further, you will want to **save the work** that you have performed thus far, to *prevent loss*, and to *specify a working directory* for your **Exporting** to follow. In the **File Menu**, select **Save As**, and *choose the directory* that you will want to save your file into. Give it a name, and click on '**Save**'.

Exporting your files

We are now ready to **Export** the file you have just insulated, into a format that will be *imported into the BoardMaster software*, which will run the machine to mill out your artwork. Now that you have *defined the directory* to save the file into, the *exported LMD file* will be saved there as well. Select the '**Export**' icon on the **Front to End toolbar**. You should see the following results:
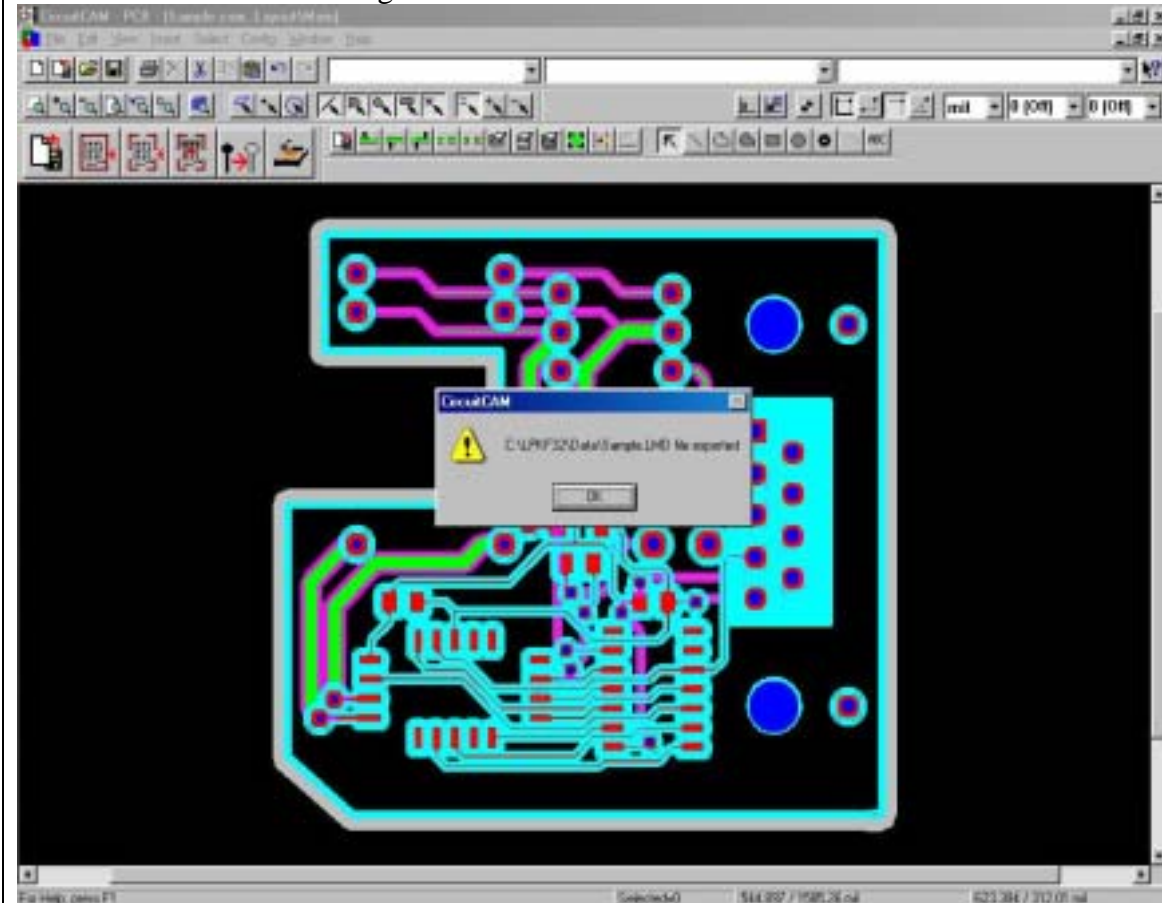
BoardMaster Basics – Fundamentals of the Software and the Hardware

This section of the training document is intended to teach the user the basic fundamentals of the BoardMaster interface and software operations, and the usage of the software to control and operate the LKPK plotter machine. It is intended to teach the user the basic fundamentals of the machine operation, and give the user a thorough understanding of the software interface. The user should be able to operate both the software and the machine after viewing and learning this instructional material.

Setting up Machine for First Use
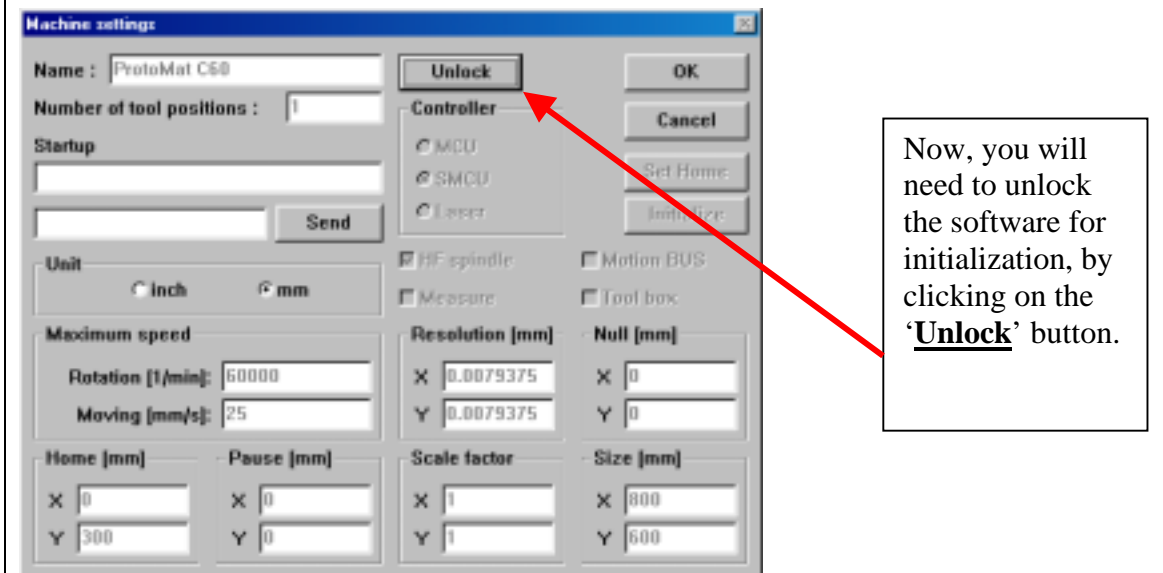
Initializing Machine

Even though you may have received your machine with the initialization disc, along with your red strips and set pins installed, you will need to know how to do this your self eventually.

Before you begin, you will need to turn the power switch on. Reach around by the power cord, and turn on the power switch.

Be aware, that when you turn the power on, the machine will begin to move, and it will come to settle in the forward exchange position.

The first step in setting up the machine, is teaching the machine the *physical limits* of the tabletop. Make sure the machine top is cleared of any and all obstructions before you begin.

Now, you will need to open up the **Configuration menu**, and select '**Settings**'.

Now, you will need to unlock the software for initialization, by clicking on the '**Unlock**' button.

*Before you click on the '**Initialize**' button, be aware that as soon as you do this, the machine will begin to move around the tabletop.*

Click on the '**Initialize**' button, and the machine will begin to seek the outer limits of the table. There are four sets of limit switches mounted in the machine, which will trigger one by one as the machine reaches the outer limits of the table. When a limit switch is encountered, the software will make a note of it, and move on to the next limit, until all four limits are reached.

When the machine has located all four limit switches, it will come to a rest in the exchange position, and the software will calculate three important locations. These three locations are as follows:
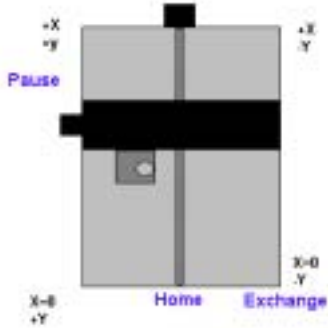

The first, is the **pause** position, where the machine head can be placed out of the way for the user to inspect and work with the tabletop. The next position to be calculated, is the **home** position, which is the central pivot point for the set pins. And the last position to be calculated, is the **exchange** position, which is where all of the tool changes will be performed.

Now, click on the '**OK**' button, to finalize the settings and close the dialog box. The machine is now *initialized*, and ready to have the *set pin* locations drilled out.

Inserting Red Set Pin Strips

In order for the set pins to be inserted, you must have the red set pin strips inserted into the center channel on the tabletop. These red set pin strips, are manufactured to have a *sway* in them, which holds them snugly in place into the channel.
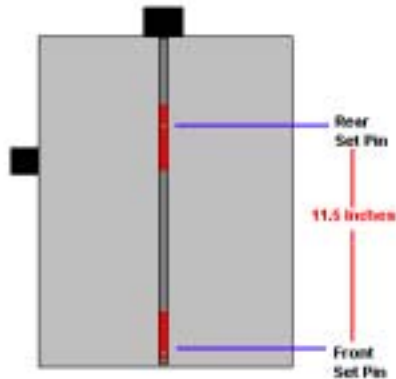

The first strip will be inserted snug up against the front channel limit pin, while the second strip will be inserted at approximately 11 inches back from the first strip. This is for a standard A4 material type size.

The second strip must be situated so that the back set pin can be drilled at 11.5 inches back from the first set pin.

Drilling Set Pin Holes

In order for the set pin holes to be drilled, you must first insert a 2.95 mm drill bit into the tool Colette.
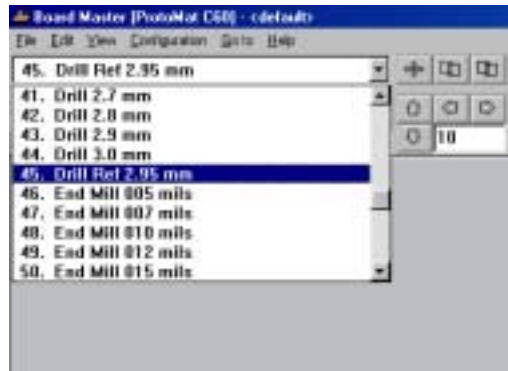
Do this by using the *tool selection drop down box* in the upper left hand corner, and select the '**Spiral Drill Ref 2.95 mm**' drill.

The machine will now move to the exchange position, if it is not already there, and prompt you to insert the proper drill into the *tool Colette*.
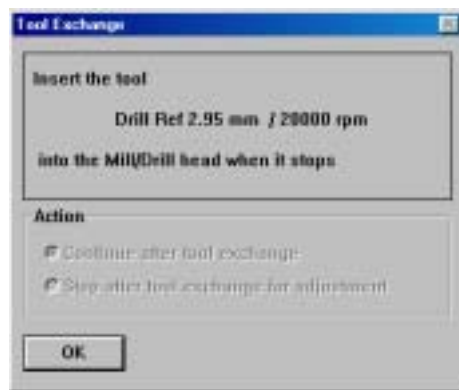
Refer to your manual for the exact method of changing tools, if necessary.

Now, you will need to send the machine to the pause position, by selecting the '**Goto**' menu, and choosing '**Pause**'.

What you will need to do now, is insert two thin strips of masking tape, one on top of the other for depth, into the section of the center channel, near the middle of the tabletop.

Next, send the machine head to the home position, by once again utilizing the '**Goto – Home**' menu command.

Now, use the head movement arrows, to send the machine backwards along the center channel until it is over the strips of tape you have inserted onto the center channel. If you need to change the default movement distance setting, you may double click on the box under the movement arrows, and enter in a new value in the box.

When the drill head is positioned over the tape strips, you will need to loosen up the drill bit in the Colette, and pull downward on the bit until it contacts the tape. Refer back to your manual for information on how to change your tools, if you don't already know.

Now, with the drill bit still loose, press downwards on the spindle motor block, until the block can go down no further. While the block is being pressed downwards, retighten the tool Colette, and then, release the pressure on the spindle block, letting it come back up to the upwards position.

Please observe a warning at this point! Do NOT move the milling head in the y direction with the drill bit lowered. This could result in the table top being scratched, and or the drill bit being broken!

Send the machine head back to the home position, using the 'Goto home' menu command.
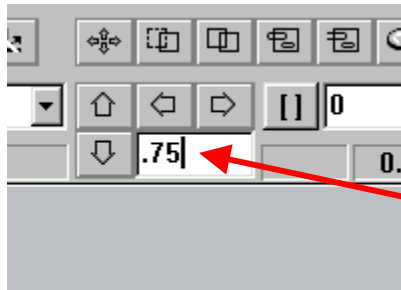
Once the machine settles in the home position, use the '**movement distance settings**' box, to enter in a new value of ¾ of an inch.

Now click the movement arrow once to move the machine head backwards along the x-axis, ¾ of an inch.

Next, place the machine into the **manual mode**, using the **mode control icon** at the top of the screen.

Please be aware, that you may have to wait for the 3 minute warm-up period before you can proceed. This warm-up period is performed once after turning on your machine, when it has been turned off for more than a few hours.

Turn on the spindle motor using the motor control icon, and then, press the head down icon, which is next to the motor control icon. ***You have just drilled your first set pin hole.*** Turn the motor off again, and enter in a new movement value, of 11.5 inches. Press the x-movement button, to send the milling head backwards to this value.

Now repeat the previous steps to drill your last set pin hole. Stop the spindle motor and place the machine back into **auto mode** with the **mode control icon**.

Next, before doing anything else, loosen the tool Colette, and press the drill bit back up as far as it will go, and retighten the Colette.

Now, send the machine to the **pause position**, by using the menu command of '**Goto Pause**'. Now, use the **set pin inserter** to *insert* the set pins into the holes you just drilled. Place the **unbeveled** edge of the set pin up into the inserter, and position the set pin over the red strip, feeling for the end of the pin to lodge into the hole. Now, *press downwards on the plunger*, and insert the set pin into the set pin hole. Repeat this for the second set pin hole.

Preparing Board Material

Drilling Underlay Guide Holes

In order to prepare your underlay material for use, you must first drill two guide holes in the material. Start by selecting the 3.0mm drill bit from the tool selection drop down menu. The machine will move to the exchange position, and you will be able to remove any tools from the Colette, and replace it with the 3.0mm drill bit.

Before you begin, you will have to remove the set pin in the back from the red set pin strip.

Then place two pieces of underlay material on the table top, centering it in the y direction, pressing it up against the front set pin.

Next, send the machine head to the home position, and then move the drill head minus ¼ of an inch in the x direction. This should put you about ¼ of an inch from the edge of the material.

Now, place the machine in manual mode using the mode control icon, turn the spindle motor on, and press the head down icon to drill.

Now, stop the spindle motor, and enter in a value of 11.5 inches into the movement distance box, and send the machine head back in the x-direction the 11.5 inches. Turn the spindle motor on and press the head down button. After drilling is done, turn the spindle motor off, and send the head to the pause position to remove the board material.

Repeat this procedure both for more underlay material and for the copper base material.

After you are finished drilling enough material for your usage, use the set pin inserter to reinsert the back set pin into the red strip.

Interface and tool bars

First, let us begin, by familiarizing you with the BoardMaster interface and tool bar functions. Let's start with the familiar windows style file menu:

File Menu

- **New**, allows you to start a new project.

- **Open**, allows you to open a previously created project.

- **Save**, allows you to save the current project.

- **Save-as**, allows you to save the current project under a different name.

- **Import**, where you will be importing your **LMD** files into your projects.

- **Refresh**, which allows you to re-import your **lmd** file into the project without throwing off your placement of the **lmd**.

- **Exit**, which allows you to exit the BoardMaster software, *saving any needed information* upon exiting the program. You will notice below the Exit option, a listing of any previous jobs, which may be opened if selected.

Edit Menu

- **Placement**, allows you to change the virtual placement properties of your project, such as *Origin*, *Scale*, and *Rotation* of **LMD** file. **Placement** also allows you to *panelize* a project for multiple board etching.

- **Reload**, is used to rework a circuit board which has been previously milled and drilled.

- **CheckDrill**, allows you to check the machine for *broken tool bits*.

- **Teach-In**, allows you to create plotter movements '*on-the-fly*', and either add to the current job, or to create a new job file.

- **Tool Selection**, is used to selectively control which tools assigned to the current phase, will be used, and in what order they will be used in.

- **Tool Assignment**, is used to assign the different tools that will be used in the different layers, as well as allowing you to change how a specific layer is processed in any phase.

View Menu

- **Zoom Area**, allows you to highlight a specific area, and zoom in on that area.

- **Zoom-In 1.5x**, enlarges the whole drawing view by one and one half times.

- **Zoom-Out 1.5x**, shrinks the whole drawing view by one and one half times.

- **Machine**, brings into view the whole programmed table size view.

- **Material**, display's the previously defined *material size*, on the screen, full view.

- **All Projects**, display's in full screen, all of the currently loaded projects.

- **Both Sides**, displays the project as if it were on transparent glass, allowing you to see the artwork on all sides of your drawing at the same time.

- **Real Tools**, display's the graphics in the actual widths of the milling that will occur, instead of in **outline mode**. **Outline mode** is the default selection view mode.

- **Default Font**, displays all milling information in a standardized reading font.

- **Change Font**, will allow you to select any available word font installed on your computer, for the milling information display.

- **Com Report**, enables you to *open, close*, or *copy* to the windows clipboard, the **BoardMaster Event Log information**

Configuration Menu

- **Material**, is where you will define the size of your material, and the physical position of the base material on the machine itself.

- **Settings**, is where you would make any changes to the programmed information of the machine. *It is recommended, that only experienced users make any changes in this screen, as, any incorrect changes made in here, could lead to possible malfunction of the machine.*

- **Tool library**. In this menu, you will *load, edit*, and *save*, the available tool libraries for use in the BoardMaster program.

- The **Heads** menu, allows you to set up an offset for optional features, such as *Auto Contac*, a *video camera*, and the optional *Protolaser*.

- The **Connect** menu, is where you will select which communications port your machine is physically connected to on your computer.

- **Tool Box** is for the machines **Protomat 95s** or a **95sI**I. This option would be where you would make adjustments to the programmed physical locations of your tools, and your tool change mechanisms.

- '**Phases'**, where you can observe and revise the way that the *phases* are configured in the BoardMaster milling cycles

Goto Menu

This menu contains the commands to physically move the milling head to various locations.

- **Home**. This, when selected, will cause the plotter head to physically move to the preprogrammed home position, located at the front of the machine.

- **Pause** does NOT mean that the machine will pause. Instead, it will cause the plotter head to physically move to the *preprogrammed pause location*, allowing you full access to the machine table top.

- **Exchange** will cause the plotter head to move to the *preprogrammed tool exchange position*, which allows you to change out the inserted tools in the *tool Colette*.

- **Camera**-**to**-**Head**. This selection is used in conjunction with the optional *CircuitView* camera installation. Selecting this option, will cause the plotter head to be *physically offset*, to a programmed offset, to allow the optional *camera* to be placed over the position that the milling head is currently at. This allows for *visual inspection* of the milling work being performed.

- **Head**-**to**-**Camera**. This option, will cause the plotter head to move back to the *original milling head location*, to allow the work to continue after being viewed by the optional camera.

Help Menu

The last menu option labeled Help, display's the information about the current version of the BoardMaster software. If you need actual help, the manual will provide these answers. If you don't find the answers to your questions in the manual, please refer to the support section in this training session.

BoardMaster Interface Icons

### Tool Selection control Icons

This first drop down menu.. Is where the tool being currently used, will be displayed..

You can also use this drop down, to manually select any listed tool

When you **manually select** a tool here, the plotter head will actually move to the exchange position, and prompt you to insert the selected tool.

The menu window just below the tool selection drop down..
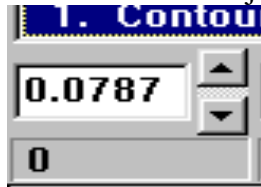
This is set automatically by the tool that is selected to be used.

This indicates and controls the actual x and y milling speeds that the plotter head will move in.

- 31 -

The menu window *to the right* of the **milling speed box**, indicates and controls the actual *spindle rotation speed* on the variable speed spindle machines.
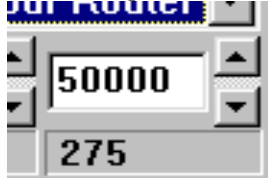
The selected tool also sets this parameter, automatically.

Phase selection drop-down

The third drop down box… Selects which milling phase will be chosen for the machine to perform.

Machine mode Icons

Above the phase selection window, are **four icons**. These four icons control the mode of operation for the machine.

- The **AutoControl** icon, controls whether the machine is in *automatic*, or *manual* operation mode.

- The **Mill**/**Drill** icon, toggles the plotter between *mill*, and *drill* mode. **Drill** mode is *momentary*, **mill** mode keeps the head down until it receives an up command.

- The **Head Control** icon, controls the *vertical position* of the plotter head. Pressing once will lower the head, pressing again will raise the head.

- The **Spindle Control** icon, controls the *spindle motor* itself. This will only become active in the *manual* mode. Pressing once turns the spindle motor **on**, pressing again will turn the spindle motor **off**.

Project Control Icons

The *center group* of icons in the interface, have a combined function of controlling the movement of the plotter head to various locations, and to select various parts of the graphics of the project.

- **Movement** icon. This icon allows you to move the physical plotter head, to a position that you select on the BoardMaster view screen.

- The **Project Move** icon, to the right of the **movement** icon, allows you to *reposition* your project on the board material. With this icon, you can position your project to take advantage of *free space* on the base material.

- **Project Copy** icon. This icon, when *depressed*, will copy and paste when you click your mouse on the project. Use the **project move** icon to *reposition* the copy on another portion of the base material.

- **Sequence Selection** icon will allow you to select a *rectangular area* of the project, and add this selection to the set of sequences to be performed upon clicking on the **start button**.

- **Partial Sequence Select** icon. This allows you to select a *single* line or *arc* of your project, to be added to the sequence to be run upon clicking on the **start button**.

- The **Magnify** icon you see, allows you to select a *rectangular* area of the project, and **magnify** or **zoom in** on that portion to fill the screen.

Plotter Control Icons

There might be slight confusion, as to the direction each arrow will move the milling head. If you will look at the interface screen, the line running down the middle of your screen is actually the physical center channel on the machines table top. This line runs from the front of the machine, to the back of the machine. The movement arrows, follow the orientation of this center channel line.

- There are **four arrows** on the interface, which allow you to **move the plotter head** in the *x* and *y* axis directions…

- **Step Distance Box** - The numbers entered into the distance box below the **four movement control arrows**, dictates the *amount of movement*.

- The **Sequence Enable** icon, located to the right of the **Step Distance** box, enables the '*Start from Sequence*' mode, while the box *to the immediate right* of it.. Displays exactly **which sequence** the program is currently on.

Sequence control Icons

The **five** final icons, on the ***top right corner*** of the BoardMaster interface, control the *sequence actions* to be performed.

- The '**All plus**' icon, places the *entire sequence* program into the buffer for the phase you are on.

- The '**All minus**' icon, will take the *entire sequence* out of the buffer for the phase you are on.

- The '**plus**' and the '**minus**' icon, *work in conjunction* with the **two sequence select** icons. To utilize these icons, you first select a portion of your drawing with the **sequence select** icons, and click on the '**plus**' button to add them to the **sequence buffer**. To remove a section of the project from the *sequence buffer*, you must first select the portion of your drawing with the **sequence select** icons, then click on the '**minus**' icon. This does **NOT** remove the portion *from the project*, just from the **sequence buffer**.

- The '**Start-Stop**' icon. This icon will *start* or *stop* the actual physical milling of the project. **Please note**, however, that how the machine runs, depends upon the '***mode***' that the machine is placed in **at the time** that you click on the **start** button. If this icon is in **manual** mode, the plotter head will *move from point to point* on the base material, but **no actual milling will occur**. However, to actually begin the milling of your project, this icon must be in the '**auto**' mode.

Main Project Viewing Area

- The **main view area**, display's the current phase view of the project you are working on. This will change the colors of the *project sequence lines* as the base material is milled, to indicate what work has **already** been performed.

- The display box **right below** the **phase selection** drop down box, will display the *communications port* that your machine is connected to on your computer.

If you have no communications set up, it will display the word 'null'.

- You will notice *at the bottom* of the main viewing area, that the wording in this area will change to reveal a description to the function, of the section that your cursor is currently positioned over. This is helpful to remind you what the functions of each icon are.

BoardMaster Machine Operation

Place Material on Machine

Now that you have an understanding of the BoardMaster interface, you have your machine set up, and your material prepared, it is time to begin milling out your board design.

Let's go ahead and place the underlay material and the copper base material onto the installed set pins, underlay first using one sheet.

Next, go ahead and place your double-sided copper base material on top of the underlay, and press it onto the set pins firmly.

Creating a new Project

The first thing we are going to do, is create a new job. Do this, by selecting 'File/New/default' from the file menu.

Defining Material size

Before you import your LMD file, you will want to define the size and the actual physical placement of the base material you are using, on the tabletop. Start by sending the milling head to the exchange position. Next, use the movement arrow icons and the movement distance settings box, to move the spindle head to the corner of the board closest to the exchange position. This is called the 'material low corner'.
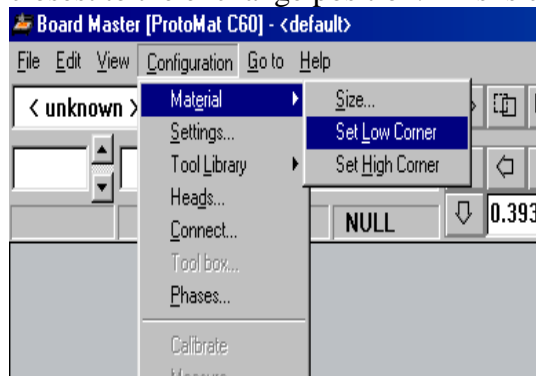

Now, from the configuration menu, select 'Material/Set LOW corner'. This will define the first corner of your material

Now, using the movement arrow keys and the movement distance settings box, move the milling head to the opposite diagonal corner of the material, to set the '**high corner**' of the base material. Now, *return to the configuration menu*, and this time, select the '**high corner**' option.

You have now successfully told the software the size and positioning of your base material on the table top.

Importing an LMD into Project

We will begin by importing the LMD file that was created in the CircuitCAM section of this tutorial. Start by choosing '**File/Import/LMD**' and navigate to the '**LPKF32/Data**' directory, to select the file name of the saved file. In our case, '*sample.lmd*'..
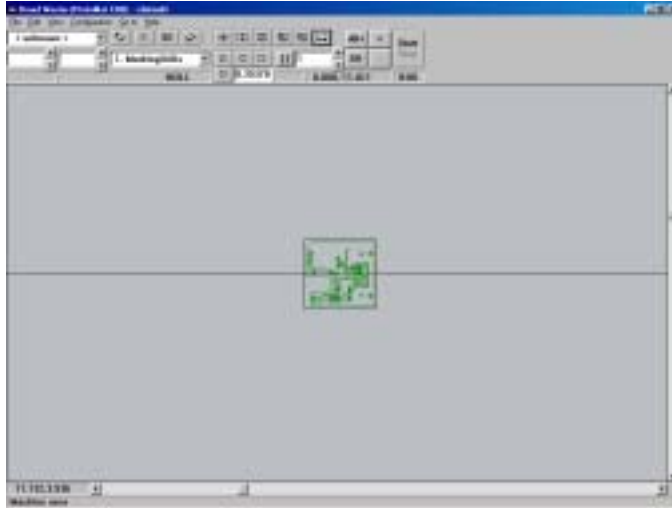
You should see a small board drawing in the center of your BoardMaster interface screen.

This is your LMD file.

Positioning project on Base Material

Position your project in an area of the board surface to maximize potential in order to make the most use of your base material. The way to do this, is, to '*drag*' your project around to an area on the graphic screen, using the '**move project**' icon.

Select the '**move project**' icon, and then click and drag your project to an unused area on your base material. Position it in such a manner, that the maximum area will be left for any other later projects to come. One hint, is to not position the project so close to a corner or another project area. This would allow the depth limiter to fall off the edge of the board or into the area of a previous project. In our case, as we are just beginning to work on a new piece of base material, this will not be a problem. You can reposition your job on the board as many times as you like, up until the time that you actually start to mill your project. Then, you should not move the job any more.

Drilling Phases

Marking Drill Phase

The marking drill phase is included for a very specific reason – to create small guide holes for the drills, to reduce possible drill wander. It is important that you not skip this phase, as it could affect the placement of your drill holes to your pads.

Using the 'phase selection' box, select 'Marking Drills'. Now, click on the 'All +' button, to select everything in this phase to be active.

Important - Before you begin the actual milling of your board, please make sure that your vacuum system is ready to be activated, as the machine should not be run without the vacuum system turned on.

Now, use your mouse to click on the start button, and the machine will move the milling head into the exchange position, and ask for you to 'Insert the tool'.

Check off the radio button labeled 'Stop after tool exchange for adjustment', and insert the tool that it is asking for. After you insert the tool, and click on 'OK', the machine will wait for you to make an adjustment.

You will need to move the milling head to a point on your base material next to your board drawing, place the machine into '**manual mode**', and turn on the spindle motor. Now, click on the 'head down' button, and the head will move down and back up again, and a mark should be made.

You will need to examine this mark for a round width of 8 mils in diameter, with the provided microscope. Turn the spindle motor off, and move the head backwards enough so that you may get a good view of the mark with the scope. Repeat this procedure until you have obtained an 8 mil diameter mark.

With an 8 mil universal cutter, for every four clicks of the knurl nut depth limiter adjuster, the width of the cut will increase by 1 mil.

Now, click on the mode control button to put the machine back into auto mode, and click on the start button again. The machine will now proceed to run the marking drills phase. When finished, the head will come to a rest at the last point that it milled at.

Drilling plated Phase

Our particular board drawing uses only *Unplated* drill holes, so, we will not be using this phase. However, it is performed just the same as '**Drilling Unplated**' phase we are going to perform. The software generates an *extra set of three layers* for any holes that are located on the '**Drilling Plated**' layers. This will not need to be covered in this tutorial. However, if you are going to be utilizing the '*AutoContac*' plating system, then you will find this phase and the other three phases covered in the AutoContac instruction tutorial.

Drilling Unplated Phase

Select the '**Drilling Unplated**' phase from the *phase selection box*, and then click the '**All +**' button. Now we are ready to begin drilling all holes in this phase of your project.

Next, click on the start button, and the milling head will move to the exchange position, prompting you to insert the proper drill bit into the tool Colette. Notice that in this phase, there is no need to stop and adjust the tool depth, as drilling bits drill all the way thru the base material, instead of milling to a specific depth. Keep the *radio button* labeled '**Continue after tool exchange**' highlighted, remove the previous tool, insert the proper tool, and click on the '**OK**' button. The machine will begin the drilling process. Repeat the tool exchange procedure until the drilling unplated phase is finished.

Milling Bottom Phase

Before you begin milling on the bottom side of the board, if you have done any drilling, it is important that you remove any board material left over from the drilling process. Use a non-metalized pad, such as the type supplied by LPKF, to gently deburr the bottom side of your base material.

Now, it is time to begin milling on the bottom side of our project. Select '**Milling Bottom**' from the *phase selection box*, and click on the '**All +**' button. The milling head will move to the exchange position, and ask you to insert the tool '**Universal Cutter 0.2 mm (8 mil)**'. Insert the appropriate tool, and select '**Stop after tool exchange for adjustment**'. Use the same procedure to adjust the milling width as you used in the marking drill phase.

Refer back to that section if you have any need to review the process.

When you have adjusted the depth limiter for the proper milling width, press the start button to begin the milling process in auto mode. The machine will continue milling until either the life of the tool is ended, or the end of the milling process is reached.

When the machine has reached the end of the milling cycle for the bottom side, it will stop where it finished, and bring up the message that it is finished. Click on '**Ok**', and then manually send the milling head to the pause position, using the '**goto –pause**' command.

Use the top end of the supplied tweezers, to slide under the end of the *base material*, and remove it from the *set pins*, and keep the front end in mind, so as to not turn the board backwards when you re-install it onto the *set pins*.

Now, flip the board over, keeping the front in the front and the back in the back. In other words, *flip it along the y-axis*, and place the *base material* back onto the *set pins* the way you did the first time.

Now, we are ready for the next section known as '**Milling Topside**'.

Milling Top Phase

To begin milling on the *topside*, you will need to choose the '**Milling Top**' option from the phase selection *drop down box*. You will notice, depending upon where you have placed your job, the image will shift to the section of the *base material* on the opposite side of the milling bottom. This is because it will be milled directly over the position of the bottom side milling, as is needed in order to have the board line up with the bottom side.

Now, click the '**All +**' icon to select all of the work on the topside for milling. Click on the '**Start**' icon. The machine will move to the exchange position and prompt you to insert the appropriate tool, or, if it knows that it already has to correct tool it, will begin milling the topside. Since you just finished milling the bottom side without changing a tool, there is no need to reset the milling width, so, you can continue on with the milling.

If you had a need to reset the tool width, just follow the procedure that you learned in the '**Marking Drills section**'.

Now, the machine will continue to mill the board design into the *base material*, until either the tool life has ended, or the end of the phase is reached. When the machine is finished with the **milling top phase**, click on '**Ok**', and manually *send the machine* to the **pause position** using the '**goto pause**' selection.

Next, inspect your project on the base material, and if necessary, give it a small amount of sanding with a non-metallic scrubbing pad, like the pads supplied by LPKF. This will remove any extraneous material from the base material.

Now, you are ready to go on to the cutting phase of the job.

Cutting Outside Phase

Now comes the final phase in our project, the '**Cutting Outside**' phase. This phase will use a *contour router* to actually cut all the way thru the *base material*, leaving the board attached only with the '**breakout tabs**' that we designed into our project.

Using the '**phase selection**' *drop down box*, choose the '**Cutting Outside**' phase of the project. Now, click on the '**All +**' icon, and depress the '**start**' button. The milling head will once again move up into the exchange position, and prompt for you to insert the appropriate tool into the Colette.

Insert the correct tool into the Colette, and choose to '**Stop after Tool exchange for Adjustment**' option before hitting the '**OK**' button. This will give you a chance to adjust the tool for the correct height before you begin the cutting process.

Adjust the depth limiter so that the length of the tool below the depth limiter foot, is the depth of the base material plus half of the depth, for a total of one and a half the base material depth. This will assure that the base material is cut thru all the way, but that the underlay material is not.

Once you have adjusted the depth limiter for the correct cutting depth, click on the **start button**. The milling head will move to the starting point of the '**Board Outline**', and begin cutting thru the *base material*. You will notice that the milling head will lift up and reposition once it reaches the '*Breakout Tabs*', and begin cutting again on the other side of the tab. This will continue until all of the '**Contour Routing**' is completed.

When the end of this process is reached, click on the '**OK**' button, and manually send the machine to the *pause position* so that you may remove the base *material* from the tabletop for further inspection. This is the end of the machining process.

Removing Project from Base Material

Now that you have removed your base material from the machine, and have inspected it for the last time, you may desire to coat it with a solderlac spray to keep it from oxidizing.

To remove the project from the base material, simply twist out the project in the case of two breakout tabs, or simply clip it out with a pair of cutters, in the case of more than two breakout tabs.

This concludes the BoardMaster software basics section of this training session.

Template Editor Basics – Fundamentals of the Template Editor

This section of the training document is intended to teach the user how to utilize the template editor software to create and update the CircuitCAM aperture translation files. It is the intention of this section to teach the user the usage of the template editor software, and allow the user a thorough understanding of the software after viewing and learning this section of the training document.

Template Editor

Why use the Template Editor?

The 'Template Editor', is used in cases where your aperture files cannot be accurately read into the CircuitCAM software by any existing translation files. The Template Editor can be found from the Windows Start Menu, in the 'LPKF32' directory. This option is available to you if you cannot find a suitable translation file for any aperture file you are trying to read in. The Aperture files are only needed for 'NON-GerberX' or 'NON-NCDrill' files.

Where the Template Editor is used

Let's start by opening up the CircuitCAM program from the start menu. In a non-GerberX set of files, or a NON-NCDrill file, we will need to import an aperture list. We are going to pretend that there are no existing translation files available to properly import the aperture list we have.

Using the import function, navigate to the 'LPKF32' directory, and go to the subdirectory called 'data'.
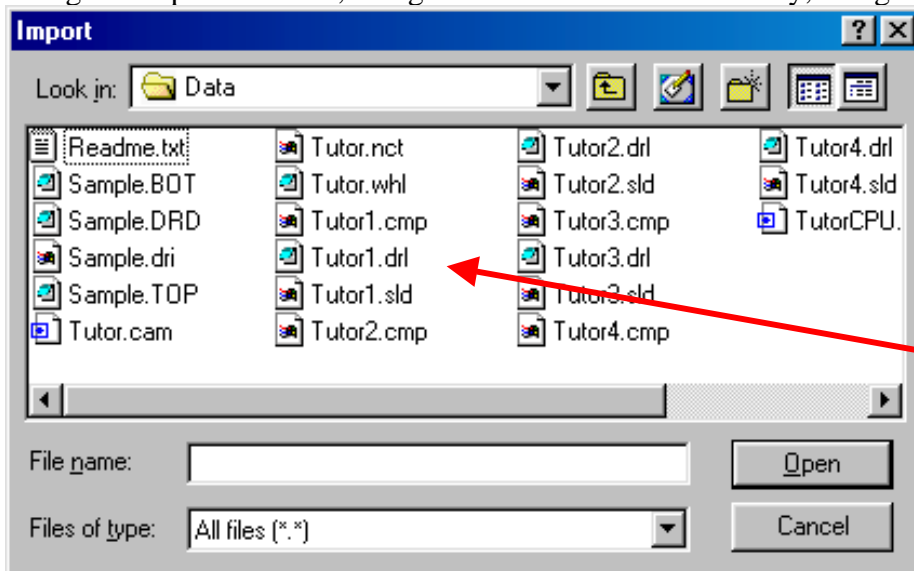

In this directory you will find a set of three files that begin with the name 'Tutor1'.

You will see that from the extensions, one is the solder side data, and another is the component side data. If you try to import either of these two files, you will notice that the aperture list information at the bottom, has question marks in it. Before you can read this type of Gerber data, you will need to import in a list of aperture information, which this file you see here, will use for it's aperture list.

Cancel out of this import dialog box, to exit out of this importation. Open the import dialog box and select the file named 'Tutor.whl'. This is the Aperture list for all of the tutor Gerber files that you see in this directory.

Reading in an Aperture file
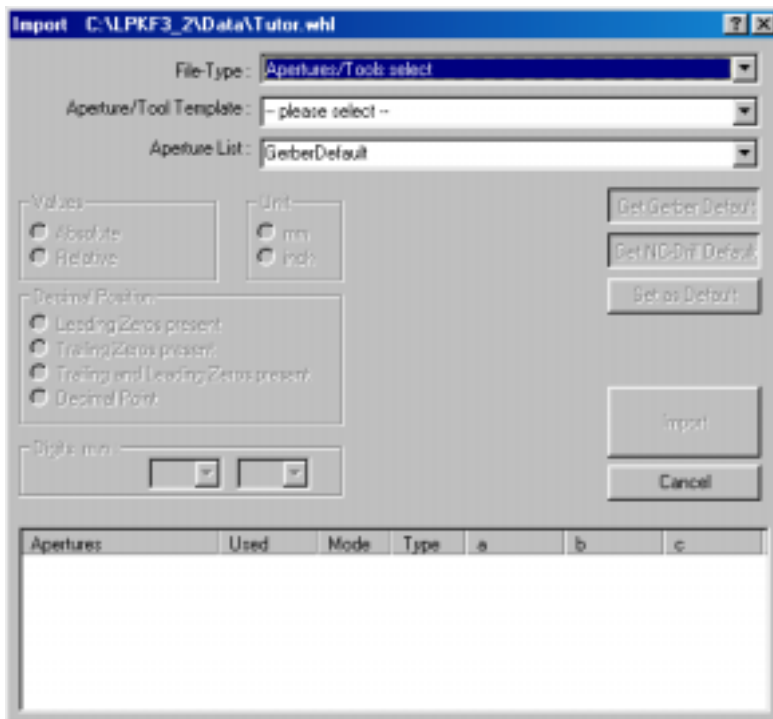
Select this file, and click on the 'open' button, to import the file. You will notice that there is no aperture information showing, and the drop down box labeled Aperture/Tool Template displays the words 'Please select'.

 If we knew which translation template to use, and we knew that it was the correct one to use, we would simply use the drop down menu to select the appropriate one, and give it an 'Aperture List' name.

However, the purpose of this tutorial is to teach you how to use the Template editor program, to create your own template translation files should no translator exist. Lets assume that no translation file currently exists.

Utilizing the Template Editor

Let's start the Template Editor, which will begin processing this file that we are trying to import.

Click on the 'Template Editor' selection at the bottom of your import dialog screen. What you see, is the Aperture file you are working on, with all of the information displayed also including the coding information that you will need to instruct the Template Editor program to include in the translation file.

The information we are most concerned about are the apertures with their 'D-Codes' situated near the bottom of this file.
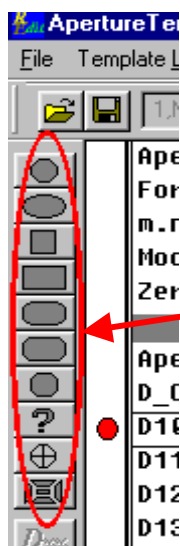
Defining an Aperture definition Line

The first thing we need to do, is to select, in the vertical column to the left, the particular aperture that we want to work on at this very moment.
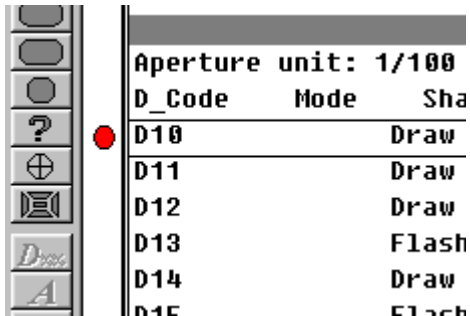
Select the line with 'D10' in it, and click in the column to the left of it.

<u>Defining the shape of the Aperture</u>

Now, you will need to tell the program what kind of shape that this aperture is, by examining the line for the shape keywords, such as 'Circle' in it's third column.

You will see a set of 10 shape buttons vertically located at the upper left corner of your screen.

The Topmost icon is the representation of a circle.

You will also notice that at the bottom left corner of your screen, a definition dialog will appear as your cursor runs across the icons. There will also be a pop-up definition that will appear if you stop the cursor long enough over one of the icons.

Click on the circle button to define this as a circle shaped aperture.

Defining Aperture D-Code Names

Now, you will need to tell the program, what wording on the selected line, defines the aperture number, or what 'D-code' it is. Do this, by double clicking the left mouse button in the middle of the D-code, 'D10'.

You will notice several things have happened.
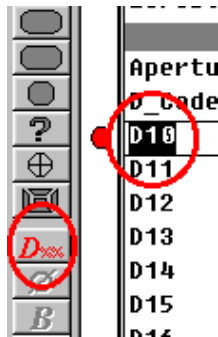
- The first, is that the D-code, is now black, to signify that the characters are now ready to be defined.

- The second thing you will notice, is that the set of icons below the shape definition icons, have changed colors.

We want to tell the program that this is a D-code, so, click once on the icon that is Red. Notice that the D10 in the line you are working on, is now changed in color. The D is a blue color, indicating the software recognizes it as a D code line, and the number will be red, meaning the program recognizes it as an integer of the D-code
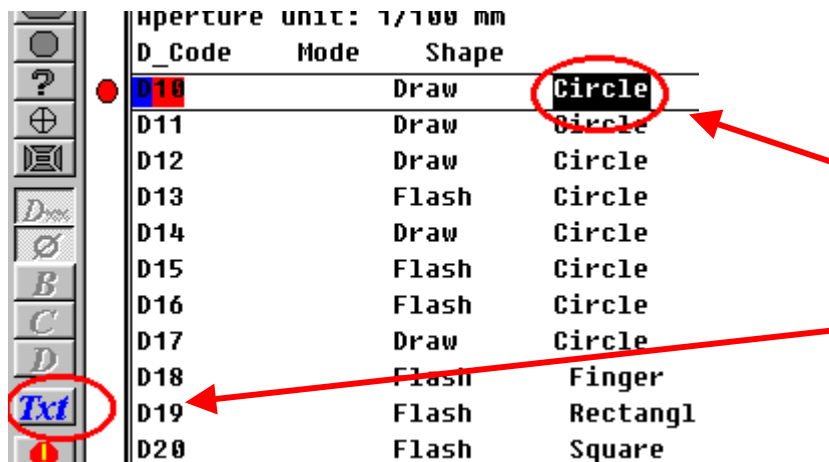
Defining Aperture size

Next, we will want to define the size information for the aperture. In the case of either a circle or a square aperture, there will only be one argument needed. In our D10 line, you will find the numerical argument of '20'.

Aperture unit: 1/100 mm

D_Code	Mode	Shape	X-size	Ysize
D10	Draw	Circle		20
D11	Draw	Circle		100
D12	Draw	Circle		30
D13	Flash	Circle		110
D14	Draw	Circle		150
D15	Flash	Circle		130
D16	Flash	Circle		160
D17	Draw	Circle		5
D18	Flash	Finger		250

Double click on this and it will turn black, then click once on the argument parameter, located under the red D code icon.

Defining Text Identifier for D-Code

Now, the last significant definition we need to make in this line, is a text identifier. This will tell the program to select all lines with this text definition, as being the same aperture type as this line.
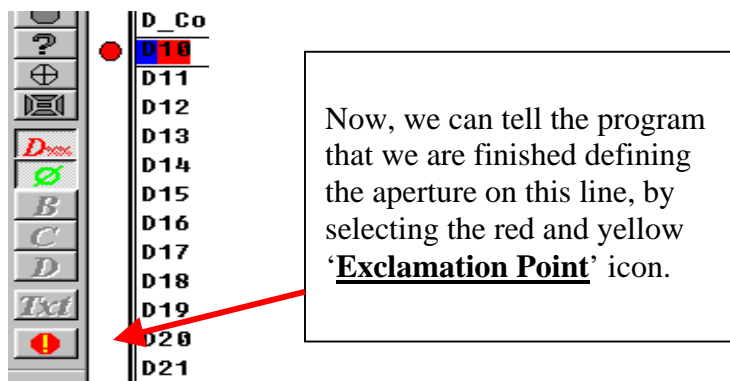

Double click on the word 'Circle' in this D10 line, and it will turn black. You will need to tell the program that this is the defining text of the aperture, by clicking on the icon with the 'Txt' label.

We could go even further, and define the word 'Draw' as identifying text also, however, in this case, it is not needed, as the Gerber files already have aperture definitions of which ones are flash and which ones are draw.

Accepting the defined Aperture Definition

We have the four necessary definitions for this aperture now; **shape**, **D-code**, **size**, and **identifying text**. This will now code all of the similar lines as being the same aperture type.


Now, we can tell the program that we are finished defining the aperture on this line, by selecting the red and yellow '**Exclamation Point**' icon.

Notice that the lines of similar type, D10 through D17, are now greyed out. What this means, is that the program has recognized these apertures as being of the same type that you defined the D10 line as, and that you will not need to re-define these lines.

Defining the remaining D-Codes

Now, we define the rest of the lines, as to their shape, D-code identifier, size, and Text identifiers, each in turn, until all of the D-Codes are identified and none are left un greyed.
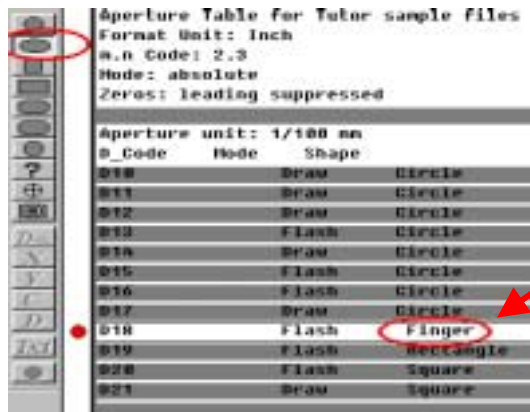

The D-code of 'D18' has a unique identifier, called 'Finger'. This aperture type, we will define as an 'Oval'. Note that on oval or rectangular shapes, you will need to define two size arguments, instead of just one.

Examining the Remaining information

Before you go to the next section of the program, you will want to make a note of all pertinent information that might be displayed in this aperture file, such as the information at the top of the file. When you are finished with all of your definitions, you will then be clicking on the 'Save' button located near the top left hand corner of the screen.

Defining Template Parameters
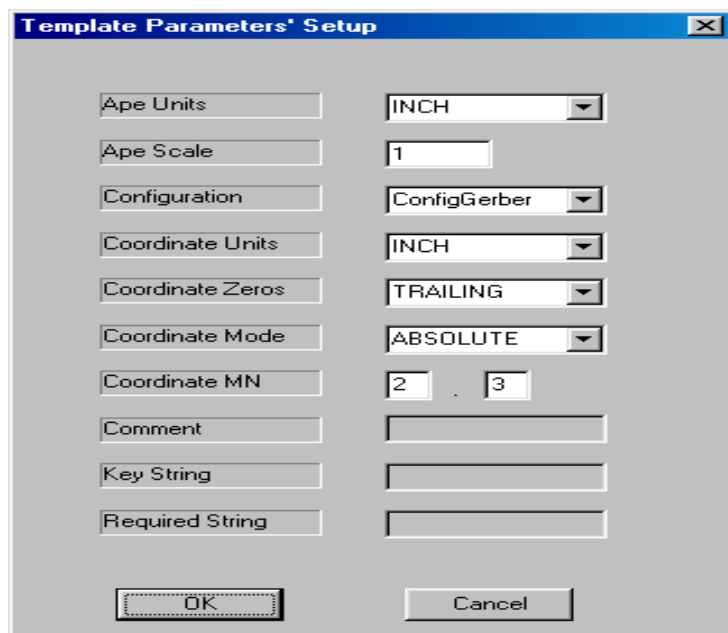
The dialog box that should now be displayed, is where you will be entering in the information about the format of the aperture translation file.

You will find some of the requested information here, listed at the top of the aperture file we are working on.

- The first line, Ape Units, refers to the measurement type used for the apertures. Notice in our file, that there is a line that says 'Aperture unit: 1/100 mm. That means, the units are in millimeters. Use the drop down box to find and select the 'mm'.

- Ape scale should be '.01', since it is 1/100th mm, and since this is a Gerber file, Configuration should be ConfigGerber.

- Co-ordinate units refer to the unit of measurement that the drawing is created in. In this case, it is in Inches, so, make the Configuration read 'INCH'.

- Coordinate Zero's, refer to the format that your aperture units are created in, whether there are decimal points, whether there are leading zeros, or trailing zero's in the numbers. In our case, it is stated that there are 'Leading zero's suppressed', in other words, trailing zeros would be appropriate here.

- Coordinate mode refers to how the points in the drawing are based, whether from a steady absolute reference point, or 'relative' to the last known point. In our case, it is in 'Absolute' mode.

- Coordinate MN, refers to the number of digits before the decimal point, and after the decimal point. In our case, it states at the top of our aperture file, that it is in 2 point 3 format, so , that is what we will enter in here.

- The comment, key string, and required string, are used to specify any keys to allow CircuitCAM to automatically recognize and select the translation file for an aperture list that has those key figures embedded into the aperture file.

Accepting and saving the new Template

After all information is entered correctly, click on 'OK'. You will then be asked for a name and a location to save this newly created translation file into.

Choose a name that refers to the cad package that you are using, including in the information about the translation file, such as coordinate.

Now, save your template in the directory where the other Aperture Templates are stored. This happens to be 'LPKF32/Ape_Templates'. The software will always look in this directory for the Aperture translation files.

Using the new Template in CircuitCAM

You will now 'Import' this list into CircuitCAM, also importing the Top and the Bottom layers, utilizing your aperture list for the aperture information of the layers.

The same method is used to create a translation file for reading in tool list for Excellon files. In an Excellon tool list, all tools are defined as circles!