Creating a Printed Circuit Board
with the
LPKF ProtoMat C30s
Circuit Board Milling Machine
(Simplified)

An Instruction Manual
Written by
Scott Anderson
April 2008

Updated by
Seth DeCato
November 2009

Introduction

Background

This instruction manual will help you to create one of three types of finished circuit boards with the LPKF Circuit Board Milling Machine (ProtoMat C30s): a simple single-sided board, a double-sided board, or a single-sided board with labels engraved on the component side. The main instructions will refer to the process of making a single-sided circuit board. Additional instructions for making the other types of circuit boards will be noted on each pertinent step with *italic text*. This process will require the use of TraxMaker, CircuitCAM, and BoardMaster software. Slight familiarity with TraxMaker, and general familiarity with Windows-based computers, is required.

> **NOTE:** Other PCB layout software than TraxMaker may be used, if the software generates standard Gerber X and NC-Drill output files, but some adjustments to these instructions may be necessary.

Examples

Throughout this instruction manual, example screenshots and photographs have been provided to give an idea of what to expect. Please take the time to familiarize yourself with the instructions and programs for best results.

Precautions

The LPKF ProtoMat C30s is a well-designed machine, and has very few safety considerations. However, it is necessary to observe the following precautions:

- Never reach into the machine while it is running.
- Be sure the motor has come to a complete stop before changing the tooling bits.
- Do not operate the machine without a tooling bit properly inserted.
- Do not use the control computer for other tasks, while the machine is running.
- Take care when handling the tooling bits, as there is a risk of cutting yourself.
- Keep the cover closed when the machine is operating, whenever possible. If it is necessary to operate the machine with the cover open, wear hearing protection and eye protection.

Definitions

The following definitions may help to familiarize you with the terms used in this manual.

- Trace - A "wire" on a circuit board, used to connect components together.
- Pad - A place to solder components on a circuit board, often has a hole in the center for the leads of through-hole components.
- Via - A spot in a circuit board used to connect a trace on the top side of the board to a trace on the bottom side of the board. Often consists of a pad on each side with a wire soldered between.
- Layer - One set of features in a PCB layout program (for example, bottom side traces, top side traces, board outline, or top markings).
- LPKF ProtoMat C30s - A high-precision CNC milling machine used to produce prototype Printed Circuit Boards.
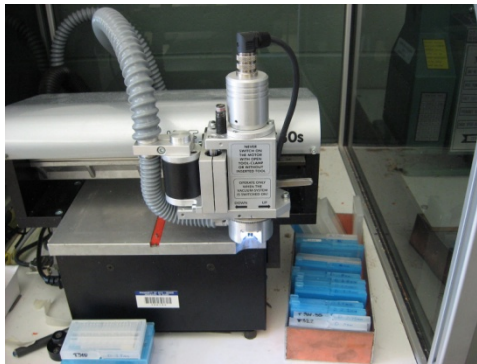

- Working Depth Limiter – The cylindrical part surrounding the tooling bit. It provides a work light, a connection to the dust-collection system, a feeler ring which presses against the circuit board material, and a knurled ring to adjust the depth of the cutting tool.


- Collet - The tool-holding mechanism on the LPKF ProtoMat C30s. The collet is loosened (in order to change tooling bits) by pushing the side handle down, and swinging it towards the rear of the machine.
- Mils - A mil is one one-thousandth of an inch, or 0.001"

- Reference Pin - A steel pin located at the front or rear of the machine table, used to keep the circuit board material from moving around during milling, and also used to line up the two sides of the board, after flipping the PCB material over.
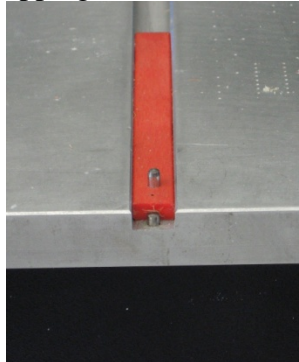
- Spoil Board - A thin white board approx 2mm thick placed between the PCB material and the machine table. It is used to keep drill and router bits from cutting into the machine table.
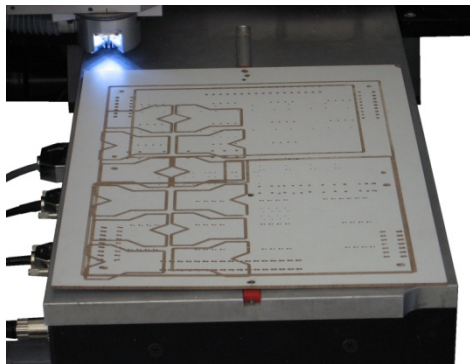
- Home Position - The "zero" position for the machine head, located at the center of the front reference pin. All measurements on the machine and in the BoardMaster software are referenced from this position.
- Tool Exchange Position - The position for the machine head locate at the nearest right-hand corner of the machine table. It has a space cut into it to allow clearance for changing the tooling bits. The machine head in the Tool Exchange Position is shown on the front cover of these instructions.
- Drill Bit - Used for making a round hole through the PCB material.
- Milling Bit - Used for engraving or cutting on the surface of PCB material, to make marks or remove copper.
- Router Bit - Used for cutting a path completely through the PCB material, to make internal break-outs, cut oversized holes, or cut the outside edge of the board.

Left to Right: 1mm Mill, 0.2mm Universal Cutter (UC) Mill, 2mm Router, 0.8mm Spiral Drill, 1.5mm Spiral Drill

I. Section 1 - Design Considerations

NOTE: At this time, it is necessary to use the computer at lab station 202-17, and log on as user: LPKF password: lpkf4work to use the LKPF Milling Machine.

A. **Board Style**
   There are three different styles of board than can be easily produced by the LPKF ProtoMatC30s.
   1. Single-Sided - This board has traces on the bottom side of the board, with nothing on the top side. Surface-mount components can be mounted to the bottom side of the board, and through-hole components and wires mount on the top side. Labels or markings may be placed on the bottom side of the board in empty spaces.
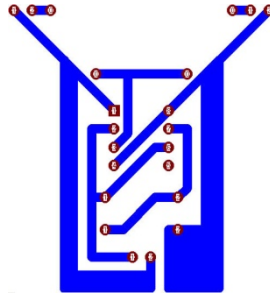
   2. Single-Sided with marking - This board is almost the same as a Single-Sided board, but uses the 0.2mm Universal Cutter to engrave labels and markings on the top side of the board. This can help with assembly when working with more complicated circuits. Labels or markings may be placed anywhere on the top side of the board, or on the bottom side of the board in empty spaces.
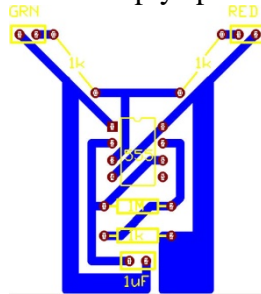

   3. Double-Sided - This board has traces on both sides. Through-hole components should be mounted on the top side, but surface-mount components may be mounted on the top or bottom. Traces may be connected between top and bottom sides by vias or component leads. Labels or markings may be placed on the top or bottom sides of the board in empty spaces.
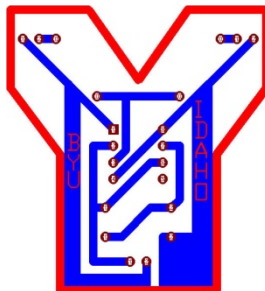
B. **TraxMaker** - The following considerations may lead to quicker, more successful circuit board production.
   1. Track Width – Try to place a majority of tracks and pads with at least 1mm of space between them. Recommended settings for general usage are as follows:
   - o Track Width: 40 or 50 mils (1 mil = 0.001")
      - o Snap Grid: 25 mils
      - o Text Height: 50 or 60 mils
      - o Space between tracks: 50 mils or greater
      - Metric Units
        - o Track Width: 1 or 1.5mm
        - o Snap Grid: 0.5mm
        - o Text Height: 1 or 1.5mm
        - o Space between tracks: 1mm or greater
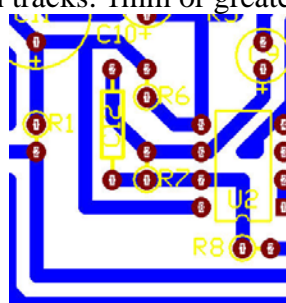
   2. Pads
      - Pads should be created on the bottom side of the board for every component pin or lead, even if the pin or lead is not connected to anything.
      - Additional pads should be created for every external wire connection.
      - Pads can be created to connect components to traces on the top side of the board, but this will not work when the body of the component blocks access to the pin or lead from the top side of the board.
      - If you need a mounting hole for screws, place a pad, double-click on it, and adjust the X-Sixe (and Y-Size) and Hole Size to be the size of the hole you need.

   3. Jumpers/Vias - If it would be necessary for two traces to cross without touching, this can be accomplished by using a jumper (for single sided boards) or creating a via (for double sided boards). A jumper or via must not be placed where it would be covered by a component. A via can be created by using an existing component pad, running a trace from the bottom side of the board to the bottom pad and a trace from the top side of the board to the top pad, and soldering the component pin or lead on the top and bottom sides of the board.
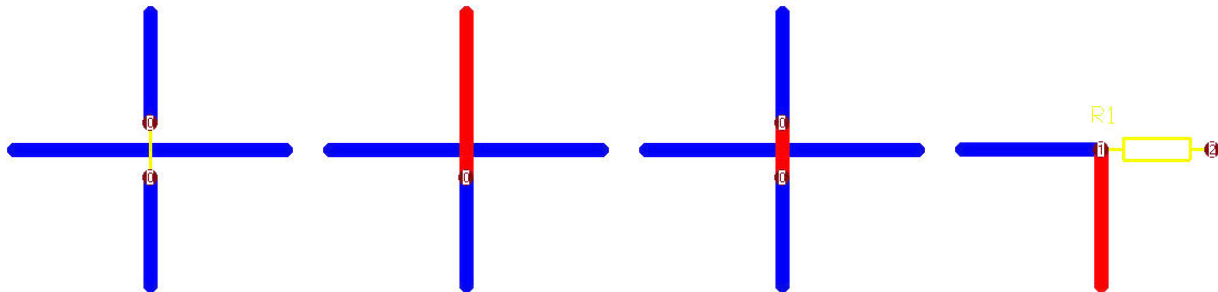
4. Layer Usage - The following features should be placed on the corresponding layers"
   - Bottom Layer (blue) - Traces and text labels on the Bottom Side
   - Top Layer (red) - Traces and text labels on the Top Side *if you are making a double-sided board*
   - Top Overlay (yellow) - Component outlines and text labels *if you are making a single sided board marked on the top side*
   - Board Layer (brown) - Pads for Top and/or Bottom Sides, automatically generated
   - Keep Out Layer (purple) - Traces determining board outline (see below)

5. Board Outline - On the Keep Out Layer, place traces showing the lines where you wish the board to be cut. Double-click each of these traces and change the track width to 1 mil or .01mm. Make sure the traces form a closed loop.

6. If you are going to put text onto the board it is suggested that you use the layer "Top Overlay" and that you also change the size and width of the text to the following sizes as seen in the picture below. When you place the text onto your design it will appear very thick. The text will not be so thick when it is cut on the machine but for a placement reference use this thickness.
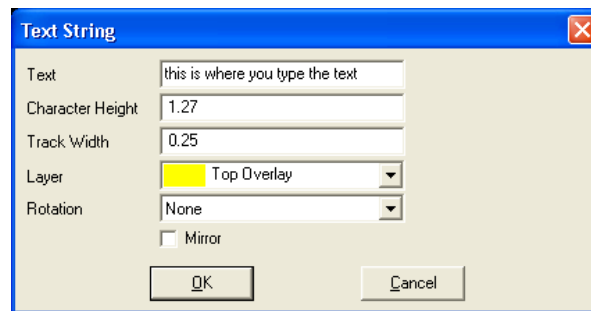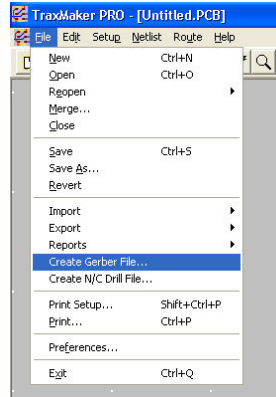
II. Section 2 - Exporting Gerber and NC-Drill files

A. Exporting Gerber Files - Once you have finished your design, you must export the layer data as Gerber Files, to be used in other programs

1. It is recommended that you save your layout in TraxMaker at this point
2. In TraxMaker, click on File > Create Gerber File…

3. In the Gerber Setup window that appears, check that the filename in the "Gerber Filename" box is acceptable. You may wish to change the filename or path.
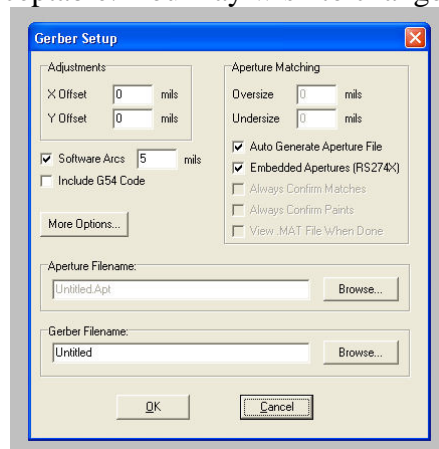
4. You may wish to click the "More Options" button and verify that the following boxes are checked:
   - Top Layer
   - Bottom Layer
   - Top Overlay
   - Keep Out Layer
   - Board Layer
   - Pads
   - Vias
   - Text Strings
5. Click "OK"
6. The files generated will have the following extensions and purposes:
   - *.GBL - Gerber Bottom Layer
   - *.GBO – Gerber Bottom Overlay
   - *.GTL - Gerber Top Layer
   - *.GTO - Gerber Top Overlay
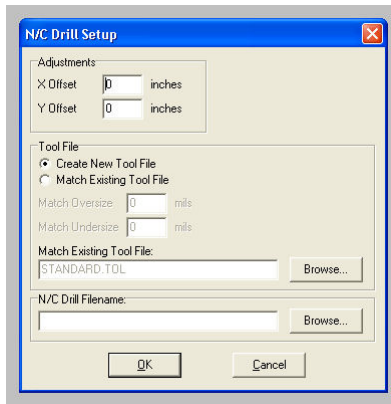
- *.GKO - Gerber Keep Out
B. Exporting NC-Drill Files - If you are making a board with through-hole component mounting, you must export the drilling data as an NC-Drill file.
1. Click on File > Create N/C Drill File…
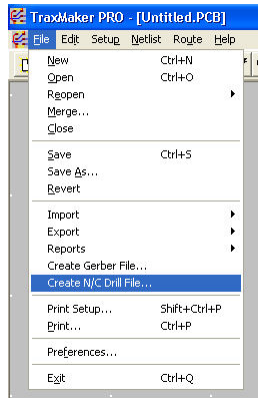
2. In the N/C Drill Setup window that appears, check that the filename in the "N/C Drill Filename" box is correct.

3. Click "OK"
4. The files generated will have the following extensions and purposes:
   - *.TOL - NC-Drill tooling file
   - *.TXT - NC-Drill data file
   - *.DRL - NC-Drill additional data file
C. You may now close TraxMaker

III. **Section 3 – Working in CircuitCam** - Once you have the files exported from
TraxMaker, you must import them to CircuitCam and prepare your design for machining.
   A. Importing Gerber and NC-Drill files - In order to work with your design, you must
   import your Gerber and NC-Drill files to CircuitCam
   1. Bottom Layer
      - Start CircuitCam
      - Click on File > Import…
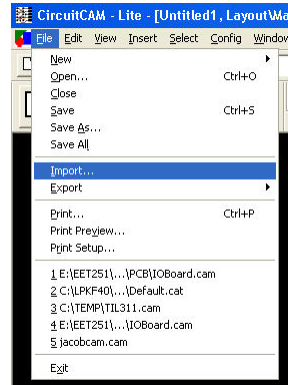
      - In the Import window which appears, navigate to your files, highlight your *.GBL
        file, and click "Open"

      - CircuitCam should automatically import the file to the Bottom Layer (green)
   2. Bottom Overlay – <mark>*You only need to do this step if you have placed text down into the
      bottom overlay layer*</mark>.
      - Click on File > Import…
      - In the Import window which appears, highlight your *.GBO file and click "Open"
      - CircuitCam should automatically import the file to the Text Bottom Layer(purple)
   3. Top Layer - <mark>*You only need to do this step if you are making a double-sided circuit
      board*</mark>
      - Click on File > Import…
      - In the Import window which appears, highlight your *.GTL file and click "Open"
   - CircuitCam should automatically import the file to the Top Layer (red)
   4. Top Overlay - <mark>*You only need to do this step if you are making a single-sided circuit
      board with engraved markings on the top side*</mark>
      - Click on File > Import…
      - In the Import window which appears, highlight your *.GTO file and click "Open"
      - CircuitCam should automatically import the file to the Text Top Layer (pink)
   5. Board Outline
      - Click on File > Import…

- In the Import window which appears, highlight your *.GKO file and click "Open"
- CircuitCam should automatically import the file to the BoardOutline Layer (yellow)
6. Tool Definitions
   - Click on File > Import…
   - In the Import window which appears, highlight your *.TOL file and click "Open"
   - CircuitCam should automatically import the tooling data, and then return you to the main program without any messages.
7. Holes - *You only need to do this step if you need holes drilled into the board*
   - Click on File > Import…
   - In the Import window which appears, highlight your *.TXT file and click "Open"
   - CircuitCam should automatically import the file to the DrillUnplated layer (grey-blue)

B. **Processing -** Having the circuit board information loaded into CircuitCam, you now must specify the machining steps to create your circuit board.
   1. Insulating – Bottom
      - If you wish to remove all the extra copper from the board (**not recommended**):

        - Click on the 4th large toolbar button:
        - Click at the top-left corner of the area you wish to have cleared
        - Click a second time at the bottom-right corner of the area you wish to have cleared
   - Click on Edit > Insulate…
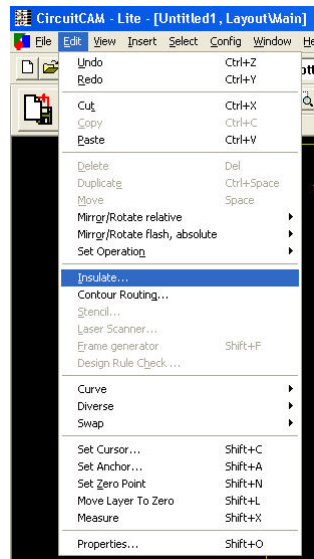
- In the Insulate window that appears, make sure the job is set to "Bottom (Solder Side)"
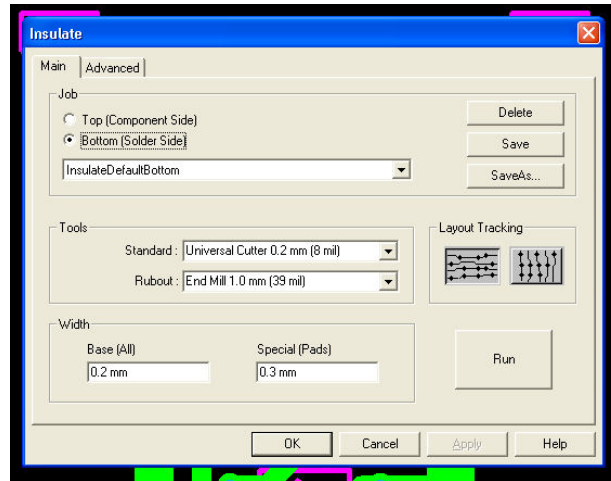

- The default settings are best for most applications, but some exceptions are:
  - If <u>all</u> the tracks and pads are at least 1mm apart, and you desire a large separation between tracks, you may set Tools - Standard pull-down menu to "End Mill 1.0 mm (39 mil)"
  - If you need more or less separation between tracks or pads, you may adjust the values in the Width box as you like, but don't go under 0.2mm
  - If the board has many large spaces between and around tracks, you may set Tools > Rubout pull-down selection to "End Mill 2.0 mm (79 mil)"
  - The Layout Tracking buttons are only used when removing all the excess copper on the board. If you are doing so, click the horizontal or vertical button to match the direction that most of your tracks run. This can result in a slightly faster machining time. However, either setting will work.
- Click on "Run", this will produce dark red lines representing the path of the smaller milling bit, and purple lines representing the path of the larger milling bit.

2. Insulating – Top - *You only need to do this step if you are producing a double-sided circuit board*
- Click on Edit > Insulate…
- In the Insulate window that appears, make sure the job is set to "Top (Component Side)"
- The default settings are best for most applications; see the section on Insulate - Bottom for some exceptions
- Click on "Run", this will produce dark green lines representing the path of the smaller milling bit, and light blue lines representing the path of the larger milling bit.

Inside Cutting - If you need any large holes (other than drilled holes) in the interior of the circuit board, perform the following steps:

- Click and drag a box around the yellow BoardOutline lines representing the area you wish to remove, so they are highlighted
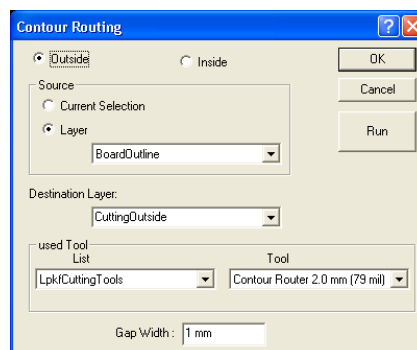- Click on Edit > Contour Routing…
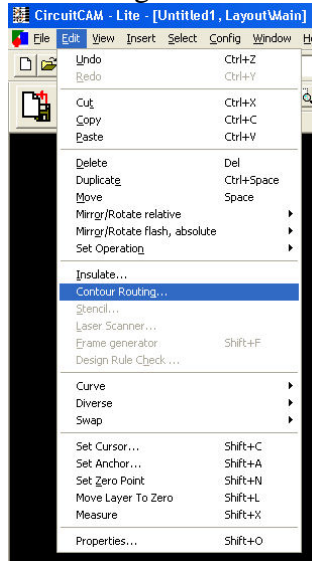
- In the Contour Routing window which appears, select "Inside" and "Current Selection"
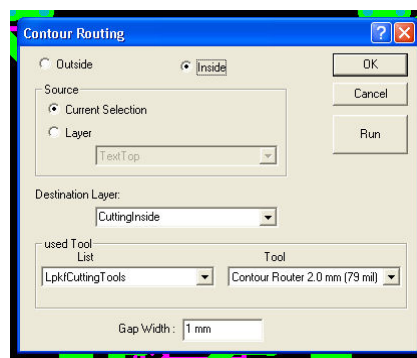

- Click on "Run"

3. Outside Cutting
   - Click on Edit > Contour Routing…
   - In the Contour Routing window which appears, select "Outside" and "Layer > BoardOutline"

- Click on "Run"
4. Text and Marking - *You only need to do this step if you are making a single-sided circuit board with engraved markings on the top side*
    - Click on or near one of the pink traces until one is highlighted
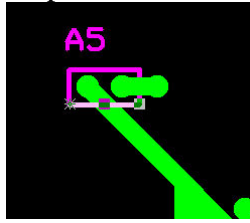
   - Click on Select > Layer (this will highlight all the pink traces)
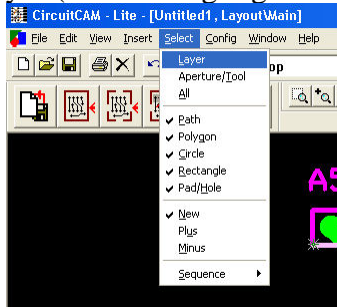
    - In the pull-down menus at the top of the screen, set the three menus to "InsulateTop_Std", "LPKFMillingTools", and "Universal Cutter 0.2 mm (8 mil)". The pink traces will change to dark green, and get very narrow.
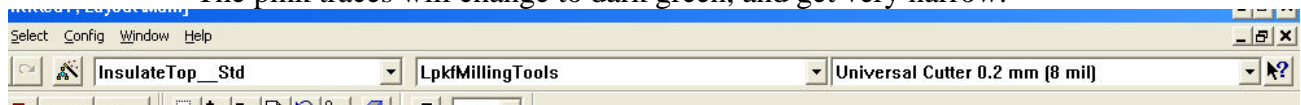
    - Hit the "Esc" key to un-highlight everything
5. Rubout Boarder – *You only need to do this step if want to have a nice copperless boarder on the bottom and/or top of your board*
    - Select your board outline(yellow) layer by clicking on one of the yellow lines and going to Select > Layer
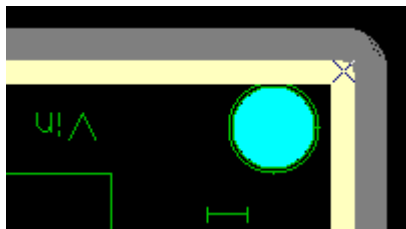
    - In the pull-down menus at the top of the screen, set the three menus to "InsulateTop_Bigger", "LPKFMillingTools", and "End Mill 2.0 mm (79 mil)". The yellow traces will change to a green, and get very narrow and if you have done the contour routing step you may not even see them at all.
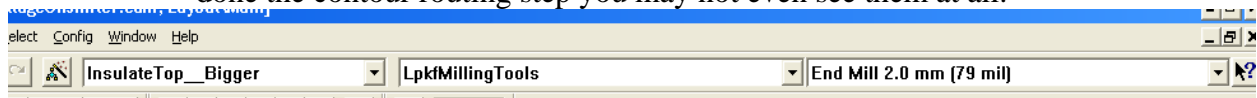

   - Hit the "Esc" key to un-highlight everything

6. Clean Up
   - If any of the dark red or purple lines, or dark green or light blue lines, are in useless places (such as outside the gray contour routing lines) you may click on or near them to highlight them, and press the "Delete" key to remove them.
   - You may remove any of these lines that you would like, if you would prefer the machine not cut the copper from those places.

C. **Saving and Exporting from CircuitCam** - Finally, you must export the machine instruction file as *.LMD file so that the BoardMaster software can make use of it.
   1. Click on File > Export > LPKF > LpkfCircuitBoardPlotter
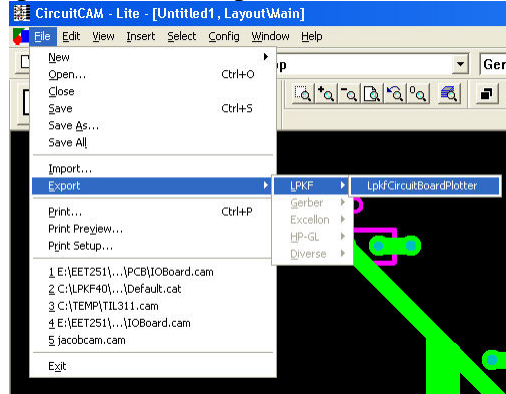
   2. At this point, if you have not saved your work yet, it will prompt you to do so
   3. CircuitCam will produce a message indicating if the export was successful or not
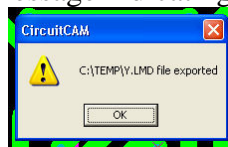
D. **You may now close CircuitCam**

IV. Working in BoardMaster - BoardMaster is the software that controls the LPKF ProtoMat C30s. Using this software you can have the milling machine produce your circuit board.

A. Starting BoardMaster
1. Turn on the LPKF ProtoMat C30s
2. The machining head of the milling machine will move slowly to the Tool Exchange Position, then "kerchunk" for a moment, before stopping
3. After the LPKF ProtoMat C30s has been turned on, you may start the BoardMaster Program
4. If the machine head has not yet reached the Tool Exchange Position, the "Waiting to transmit…" window will appear. Simply wait until the machine head has finished moving, and the program will continue to load.

B. Importing and Placement - You must bring your project into the software and tell it where to produce it on the circuit board material
1. Importing
   - Click on File > Import > LMD / LPR…
   - Navigate to your saved *.LMD file, highlight it and click "Open"
   - Your project will appear in the main window
2. Placement
   - Assure that the blank circuit board material has an adequate space for your project, is foil-side up, firmly affixed to the machining table. If not:
     - ○ Click on Go to > Pause
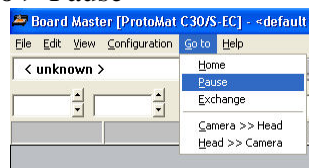
- The machine head will move to the rearmost left corner of the machine table.
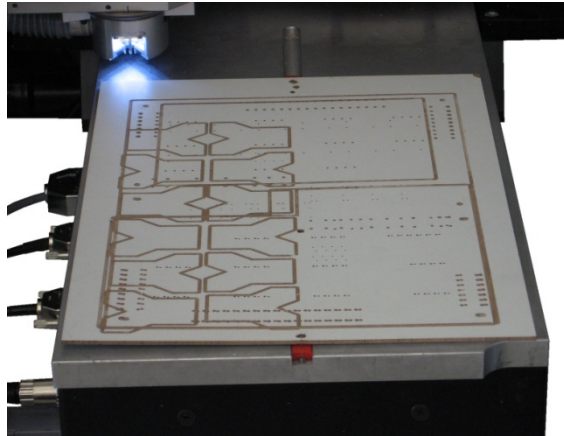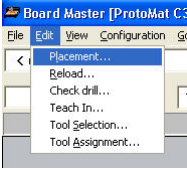

- Remove any tape holding the circuit board material down, and lift it from the reference pins
- Replace the circuit board material with the foil side up, placing the hole over the front reference pin, and the slot over the rear reference pin

NOTE: The circuit board material must lie flush to the machine table. If the circuit board material or spoil board lifts up at the center, away from the machining table, you may be able to fix this by removing the circuit board material, flipping the spoil board over, and replacing the circuit board material.

- Secure the edges of the circuit board material tightly down to the table at any point they are raised by running a short piece of masking tape from the edge of the board, over the side of the table. **Make sure the masking tape is at least a half-inch away** from the area where your project will be.
- Locate a suitable area on the circuit board material for your project. Your project must not cover the front reference pin.
- On the circuit board, measure the location of this area from the home position (in mm) in the X direction (the long dimension of the machine table, with +X being away from the home position) and the Y direction (the short direction of the machine table, with +Y being away from the tool exchange position)
- There are two ways to adjust the placement of the project

NOTE: As displayed in the program, the X direction is left-right on the screen (+X being right) and the Y direction is up-down (+Y is up when viewing the bottom side of the project, and +Y is down when viewing the top side of the project)

- Method 1
  - Click Edit > Placement…

Board Master [ProtoMat C3

File Edit View Configuration G

<

| Placement... |
| Reload... |
| Check drill... |
| Teach In... |
| Tool Selection... |
| Tool Assignment... |

- o In the Placement window which appears, in the Origin fields, enter the values you have measured for X and Y, enter an angle (multiples of 90°) then hit "OK"
  - If after changing the rotation angle you may not be able to see your board if this happens then go back to placement and click on the center button and press okay to have it re-centered so you can see the circuit board placement again
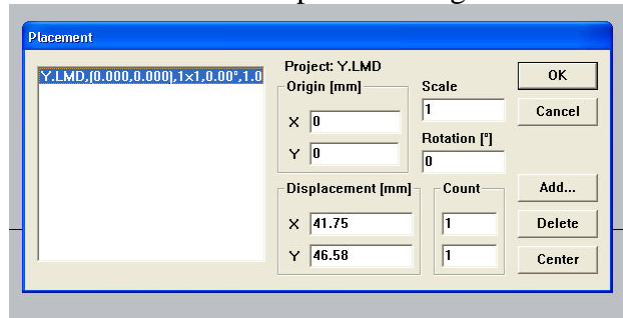
- o Move your mouse cursor near the corners of your project, and read the position from the bottom left corner of the screen (in mm)

- o If the project is in the wrong place, click Edit > Placement… again and adjust the X and Y Origin values until the project is in the right place.
- Method 2:
  - o Click the Move button on the toolbar
  - o Click on project and drag it to the desired position
  - o Move your mouse cursor near the corners of your project, and read the position from the bottom left corner of the screen (in mm)
  - o If the project is in the wrong place, click the move button again and drag the project until it is in the right place.
3. Multiple Projects - The milling machine can load multiple projects and work on them at the same time.
   - If you wish to make a duplicate of the existing project, click on the Copy button on the toolbar click on the existing project, and drag the new instance of the project to the desired location
   - If you wish to add a different project, import it as a LMD file as described above
   - Adjust the placement of the new project instance using either of the above methods.

C. Machining Processes - This section details the steps of the actual machining process

---

NOTE: In each small case of bits, the bit on the far right is the one you should always use because the machine tracks the number of uses each bit has had and so we need to consistently use the same bit for the count to be accurate. When the machine tells you to replace the bit, throw away the old one and the next bit you use will go into the far right slot when you put it back into the case.

---

1. Marking Drills - The first step is to mark pilot plots for the tips of the drill bits. This will improve the accuracy of hole placement, and reduce the chances of drill bit breakage.
   - In the toolbar at the top of the screen, click on the pull-down menu and select "1. MarkingDrills".
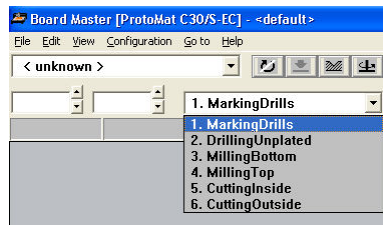
   - In the toolbar, click on the "All +" button . All of the orange dots in the center of the holes will change to yellow, indicating that they are enabled.

   - In the toolbar, click on the "Start/Stop" button
   - The Tool Exchange window will pop up; load the 0.2mm UC tool into the machine and press "OK"
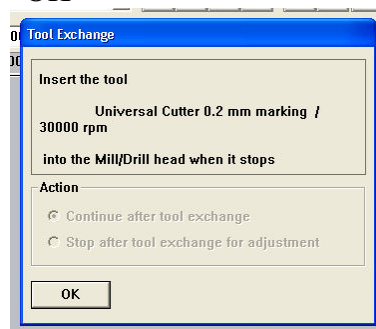

   - If the machine has not run for several hours, the Warm Up window will pop up. Wait approx 3 minutes and the window will disappear and the machine will continue.

   - The machine will move the head and make pilot plots for the drills. As each mark is made, the corresponding yellow dot will turn orange.

- Once the machine has stopped, it will display an End of Phase window and play a repetitive low tone through the speaker. Hit Enter or click OK.
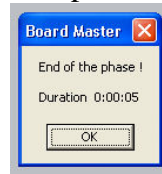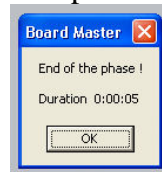

- Examine the circuit board and make sure that the marks have been cut deep enough across the board. If not, turn the depth adjustment ring about 5 "clicks" to the left (in the direction of the "Down" arrow) and repeat this step.

2. Drilling Unplated
   - In the toolbar at the top of the screen, click on the pull-down menu and select "2. DrillingUnplated".
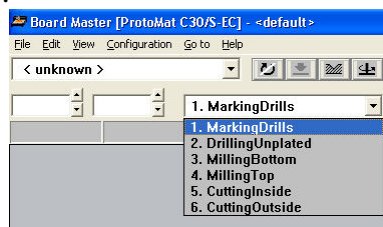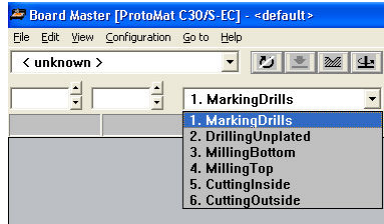

- In the toolbar, click on the "All +" button . All of the dark blue dots representing the holes will change to bright blue, indicating that they are enabled.

- In the toolbar, click on the "Start/Stop" button

- The Tool Exchange window will pop up; load the indicated tool into the machine and press "OK"

- The machine will move the head and drill the first set of holes. As it drills each hole, the corresponding bright blue dot will change to dark blue.

- If there are holes of larger sizes, the machine will stop, move to the tool exchange position, and the Tool Exchange window will pop up again. Insert the indicated tool and click OK, until the machine finishes the drilling phase.

- Once the machine has stopped, it will display an End of Phase window and play a repetitive low tone through the speaker. Hit Enter or click OK.

3. Milling Bottom
   - Make sure that there are no burrs or residue on the circuit board material, within a half-inch of your project.
   - In the toolbar at the top of the screen, click on the pull-down menu and select "3. MillingBottom".

- In the toolbar, click on the "All +" button **All+**. All the dark green lines will change to bright green, indicating that they are enabled.

- In the toolbar, click on the "Start/Stop" button **Start Stop**

- The Tool Exchange window will pop up; load the indicated tool into the machine and press "OK"

- The machine will move the head and cut insulation paths around the circuit traces. As each cut is completed, the corresponding line will change from bright green to dark green.

NOTE: Adjust the depth of the machine head until the milling bit just barely cuts completely through the copper, across the circuit board. If you are using a brand new 1mm milling bit, set the depth so that the two blue marks on the depth adjustment ring line up. As milling bits wear, you will need to adjust the milling depth down.

- If, at any point, the machine produces an inadequate cut, perform the following steps:
   - Click the Start/Stop button **Start Stop**. (It may be necessary to click it more than once; if the machine is performing very short or very long instructions, it will not recognize the click.)
   - If necessary, move the depth adjustment ring to adjust the depth of cut.
   - If necessary, click the Go to > Exchange menu to bring the machining head to the tool exchange position, and replace the milling bit.
   - To re-enable tooling paths (to re-cut them):
      - Click the Select Elements button **□**.
      - Click and drag a box across the paths you want to re-enable.
      - The paths you have selected will turn bright white.
      - Click the + button **+**, and the paths will turn bright green, indicating that they are enabled.
      - If there are paths enabled that you do not wish cut, click the Select Elements button, highlight the paths, and click the - button **-**.
   - Once all the desired paths are enabled, click the Start/Stop Button again.
   - These steps may also be performed while not in a machining phase, in which case do not click the Start/Stop button initially.
- If the program requires a milling bit of a different size, the machine will stop, move to the tool exchange position, and the Tool Exchange window will pop up

again. Insert the indicated tool and click OK, until the machine finishes the milling phase.


NOTE: When using the 1mm milling bit, as the bit becomes slightly dull, the machine sometimes lacks sufficient downward force to plunge the bit through the copper. It may be necessary to operate the machine with the cover open, and press down on the top of the motor each time the head lowers onto the circuit board. **While operating the machine with the cover open, wear hearing and eye protection.** Discard and replace the 1mm milling bit when it starts leaving ragged edges on the copper as it cuts.

- Once the machine has stopped, it will display an End of Phase window and play a repetitive low tone through the speaker. Hit Enter or click OK.

- Examine the circuit board and make sure that the marks have been cut deep enough across the board. If not, refer to the above instructions to re-enable cutting paths.
- Click on Go to > Pause to send the machining head to the back of the machine table.
- Remove the circuit board from the reference pins, and remove any masking tape on the board. Flip the board over and replace it upside down, placing the hole over the front reference pin and the slot over the rear reference pin. Re-apply adequate masking tape to hold it down tight to the table.

4. Milling Top - *You will need to do this step if you are making a double-sided circuit board, or a single-sided circuit board with engraved markings.*
   - Make sure that there are no burrs or residue on the circuit board material, within a half-inch of your project.
   - In the toolbar at the top of the screen, click on the pull-down menu and select "4. MillingTop".
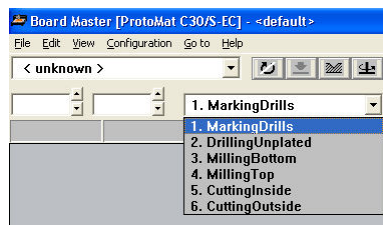
- In the toolbar, click on the "All +" button . All the dark red lines will change to bright red, indicating that they are enabled.

- In the toolbar, click on the "Start/Stop" button

- The Tool Exchange window will pop up; load the indicated tool into the machine and press "OK"
- The machine will move the head and cut the programmed paths. As each cut is completed, the corresponding line will change from bright red to dark red.

NOTE: All the considerations and adjustments that apply in the previous step, 3. Milling Bottom also apply in this step.

- If the program requires a milling bit of a different size, the machine will stop, move to the tool exchange position, and the Tool Exchange window will pop up again. Insert the indicated tool and click OK, until the machine finishes the milling phase.
- Once the machine has stopped, it will display an End of Phase window and play a repetitive low tone through the speaker. Hit Enter or click OK.


- Examine the circuit board and make sure that the marks have been cut deep enough across the board. If not, refer to the above instructions to re-enable cutting paths.

5. Cutting Inside
- In the toolbar at the top of the screen, click on the pull-down menu and select "5. CuttingInside".
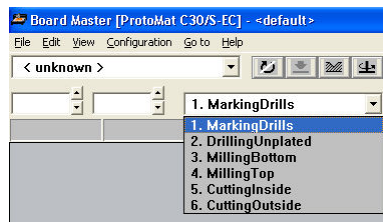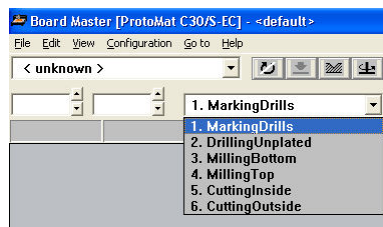

- In the toolbar, click on the "All +" button . All the dark yellow lines for this phase will change to bright yellow, indicating that they are enabled.
- In the toolbar, click on the "Start/Stop" button
- The Tool Exchange window will pop up; load the indicated tool into the machine and press "OK"
- The machine will move the head and cut the board around the inside path. As each cut is completed, the corresponding line will change from bright yellow to dark yellow.
- Once the machine has stopped, it will display an End of Phase window and play a repetitive low tone through the speaker. Hit Enter or click OK.
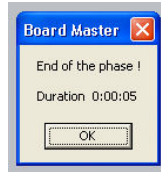
NOTE: At this point, it may be desirable to remove the board from the machine and inspect the traces and cuts. Click on Go to > Pause, remove the circuit board from the reference pins, and use the Motic 60x Lighted Inspection Microscope to examine anything that may be imperfect. If any re-work is needed, you can replace the board with the proper side up, secure it with tape, and follow the instructions for that phase again.

6. Cutting Outside
   - In the toolbar at the top of the screen, click on the pull-down menu and select "6. CuttingOutside".


   - In the toolbar, click on the "All +" button . All the dark yellow lines for this phase will change to bright yellow, indicating that they are enabled.
   - In the toolbar, click on the "Start/Stop" button
   - If the proper tool is not already in the machine, the Tool Exchange window will pop up; load the indicated tool into the machine and press "OK".
   - The machine will move the head and cut the board around the outside path. As each cut is completed, the corresponding line will change from bright yellow to dark yellow.
   - Once the machine has stopped, it will display an End of Phase window and play a repetitive low tone through the speaker. Hit Enter or click OK.
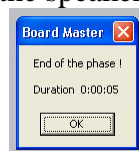
   - Click Go to > Pause to move the machining head to the rear of the machine.
   - Remove your circuit board from the machining table.
7. Turning Off the Machine
   - Click File > Exit.
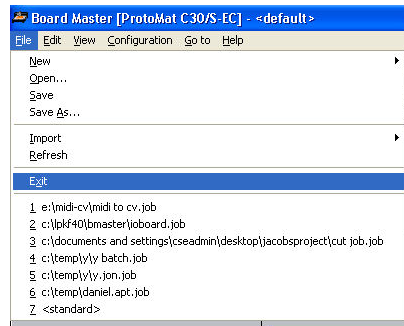
- The BoardMaster program will prompt you to save the job you have created. If you wish to make the same board(s) again later (in exactly the same position(s)) then you may save it, otherwise hit Cancel.
- Switch off the LPKF ProtoMat C30s.

8. Final Preparation - You have finished machining your board. There are a few final steps that you may wish to perform for professional looking results.
   - The corner where the CuttingOutside phase finished may have a small "tab" protruding on the edge. You may remove this with a fine file.
   - The inside and outside cut edges may have a copper burrs on them. You can smooth these quickly with a fine file applied at a 45° to the face of the board.
   - You can polish the board with steel wool.
   - If you have polished the board, you may apply a tinning solution to the board.

V. Trouble Shooting
   A. Circuit Cam: When you are importing the files if they aren't automatically loaded then you will need to go into Config > General Settings and add or change the following file extensions to match the following

| File – Spec	Layer	Aperture/Tool List	Type
*.GBO	Text Bottom	ApeTutor	Gerber X
*.GBL	Bottom Layer	ApeTutor	Gerber X
*.GTO	Text Top	ApeTutor	Gerber X
*.GTL	Top Layer	ApeTutor	Gerber X
*.GKO	Board Outline	ApeTutor	Gerber X
*.TOL	TraxTools.TXT (NcDrill)	NcDrill Default	Apeture/Tool Select
*.TXT	Drill Unplated	NcDrill Default	Excellon
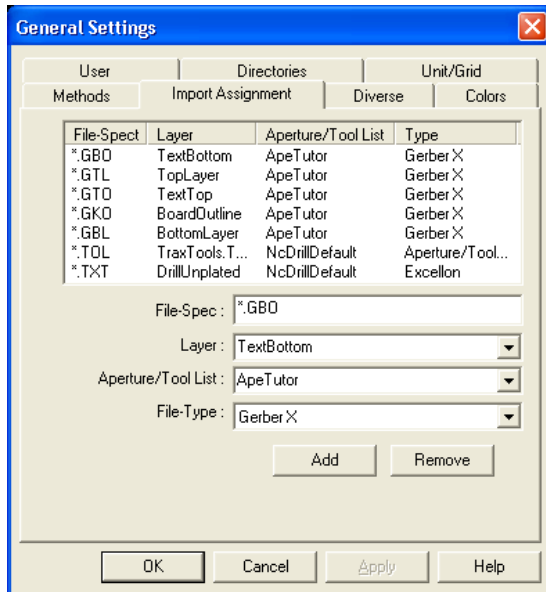
B. Board Master:
   1. If there are any problems with alignment make sure that the board is properly inserted on the reference pins and that it is taped down
   2. Check the machine dimension settings by going to Configuration > settings and verify that all the settings are correct
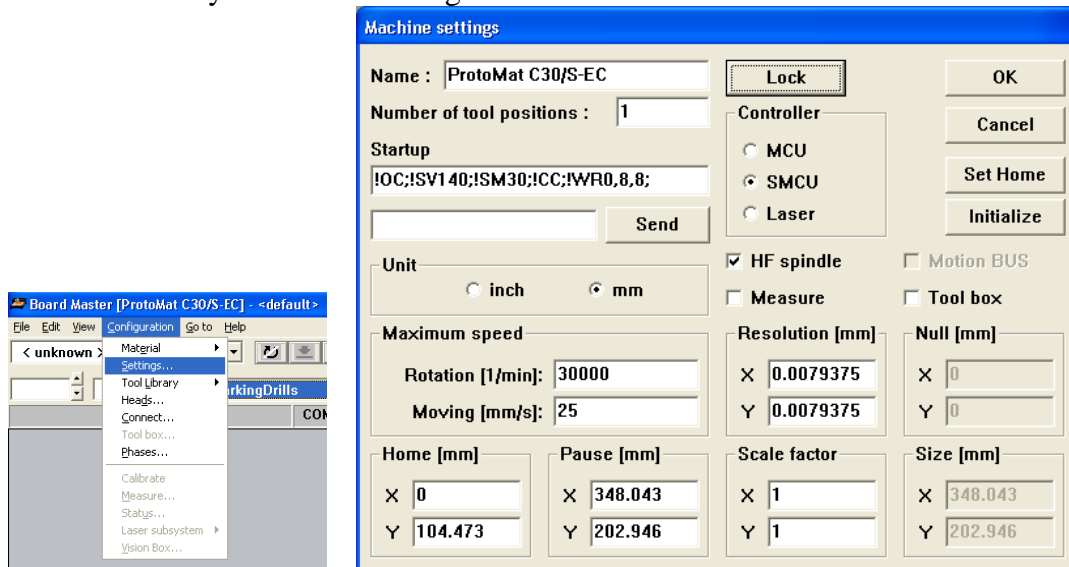
   3. Check that the project you are working on is within the boundaries for milling by going to configuration > material > size and input the following dimensions
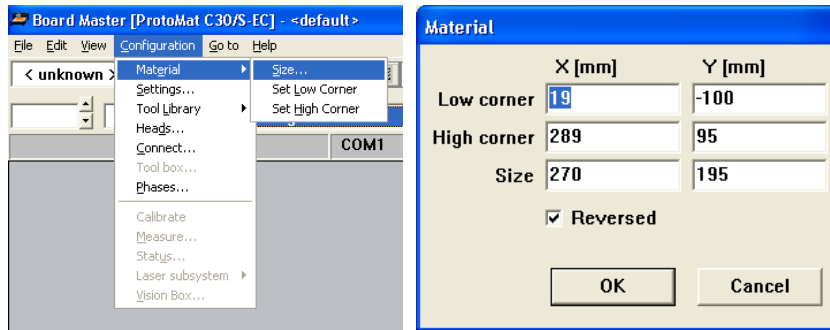
When you hit "OK" you will see a dark square appear on the machine mat. Verify that all projects are within this dark area.
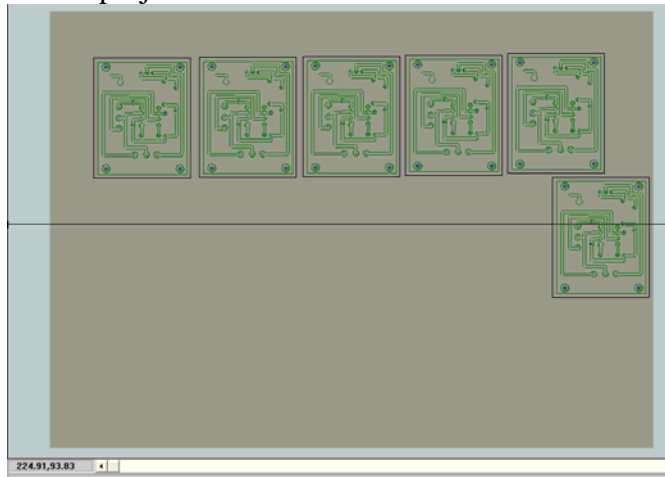
C. Reference manual / tutorial – if you want to you can go to the complete trainers manual for CircuitCAM and BoardMaster are here http://www.ee.cuhk.edu.hk/pcblab/manual/CCAM40tut-e.pdf

D. Additionally the school has a website with several useful links for the LPKF milling machine and CircuitCAM at http://www.byui.edu/csee/cadence/relatedLinks.htm