Guide to Designing and Fabricating Printed Circuit Boards

Rev 1.0

© University of Toronto
January 2006

Contact for ECE496 students:
Olivier Trescases

trescas@vrg.utoronto.ca

Outline

Glossary

Term	Meaning
Milling	Process used to mechanically remove copper lines in order to form isolated wires.
PCB	Printed Circuit Board.
Ground Plane	Sheet of metal used to form a low-resistance ground connection.
Pitch	Centre-to-centre separation between two pins of a package.
Package	Mechanical housing for an integrated circuit.
Trace Insulation	Layers (top and bottom) generated around the PCB traces by *CircuitCAM*. The copper region designated by the insulation layer will be milled to achieve electrical isolation.
Contour Routing	Process used to cut the completed PCB out of the bare copper board.
Surface mount package (example: TSOP)	The pins are soldered on the same side as the package body.
Through-hole package (example: DIP)	The pins travel through the board and are soldered on the opposite side of the PCB board.
Footprint	Overlap pattern between the package and the board.
Via	Electrical connection between a top and bottom trace.
Mil	American unit for distance (1 mil = 0.001 inch = 0.0256 mm).

I. Introduction

This guide was originally developed to help 4th year undergraduate students complete their design project. The guidelines should also be helpful to all undergraduate and graduate students who wish to design and fabricate a single-side or double-side printed circuit board (PCB). With traditional DIP packages being replaced by fine-pitch surface mount alternatives, making a circuit "by hand" may not be practical in many cases. Making a two-sided PCB in the ECE Design Centre facility provides students with a rapid, low-cost alternative to using third-party PCB fabrication services.

1.1 What is a "Milled" Circuit Board?

The term "Printed Circuit Board" (PCB) has been loosely used to describe boards fabricated using a variety of methods. The most common method of commercially manufacturing high-quality PCBs relies on a photolithographic process, whereby the undesired copper is chemically etched away from the copper substrate. The regions to be etched are defined optically using a photolithographic mask, which explains why traditional PCBs are considered to be "printed". Alternatively, simple PCBs can also be made by removing the unwanted copper using a purely mechanical process known as milling. In a milled PCB, a milling machine is used to remove the unwanted copper from a bare copper substrate to form the desired traces. Note that the resulting boards are still referred to as PCBs despite the fact that they are not "printed". A sample milled PCB that has been removed from the original copper board is shown in Figure 1.
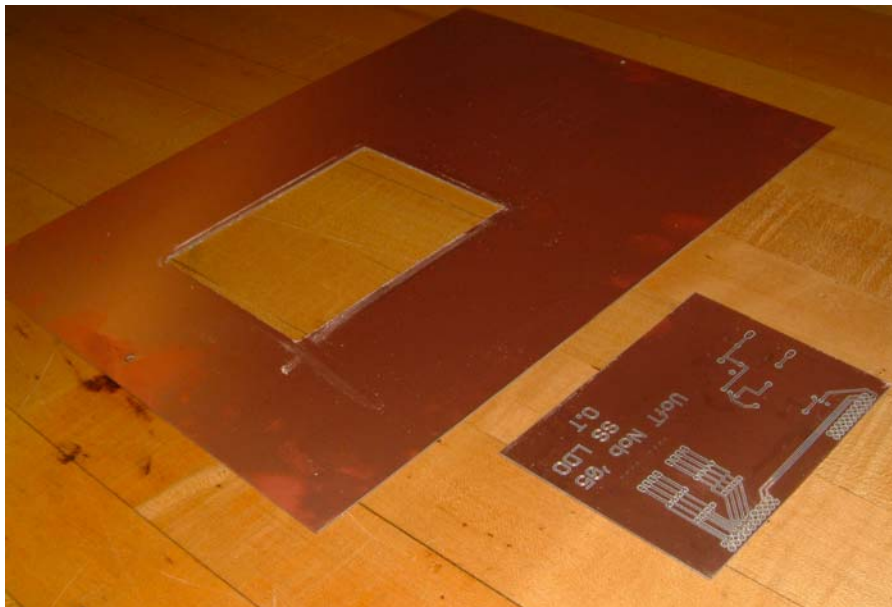

Figure 1: A completed milled PCB.

1.2 PCB Fabrication Flow

The high-level design-flow for the milled PCB fabrication covered in this document is shown in Figure 2. Numerous CAD design packages are available. The process involves three separate software packages. The CAD flow will be described in detail in the following sections. After careful consideration *Eagle PCB* was selected for this guide. *Eagle* offers the following benefits:

- Available for the three most popular OS platforms (Mac OSX, Linux and Windows)
- Script-based functionality (command line)
- Numerous features (autorouter tool, DRC etc.)
- Excellent selection of pre-defined parts and package footprints
- Connectivity is maintained between schematic and layout

1. PCB Design (Section II):
- *Eagle*
- *CircuitMaker*
- *OrCAD*

2. CAD File Preparation for Fabrication (Section III):
- *CircuitCAM*

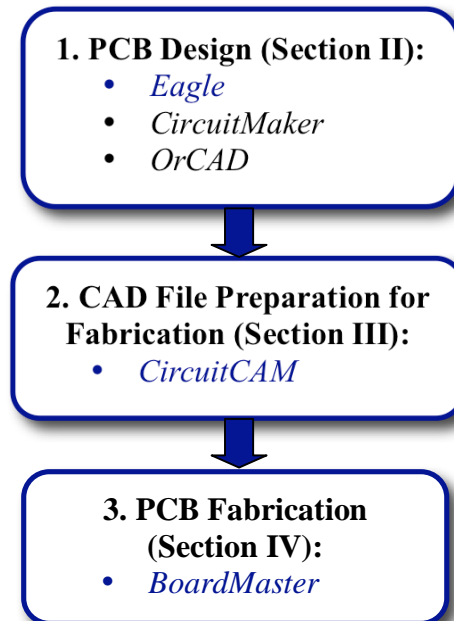**3. PCB Fabrication (Section IV):**
- *BoardMaster*

Figure 2: High-level CAD flow for the PCB fabrication process. The CAD packages supported in this document are highlighted.

The standard version of *Eagle* is installed in the ECE Design Centre and the free student version can be downloaded from cadSoft:

www.cadsoft.de

II. PCB Design in *Eagle*

This section is intended as a supplement to the *Eagle* user manual (available at *www.cadsoft.de*), where specific functions are explained in more details. The reader

should be familiar with *Eagle's* basic functionality before proceeding in this section. The CAD flow for PCB design is shown in Figure 3. The circuit should be fully designed with all the parts sourced before proceeding to the PCB design. The steps will be described in the following section.
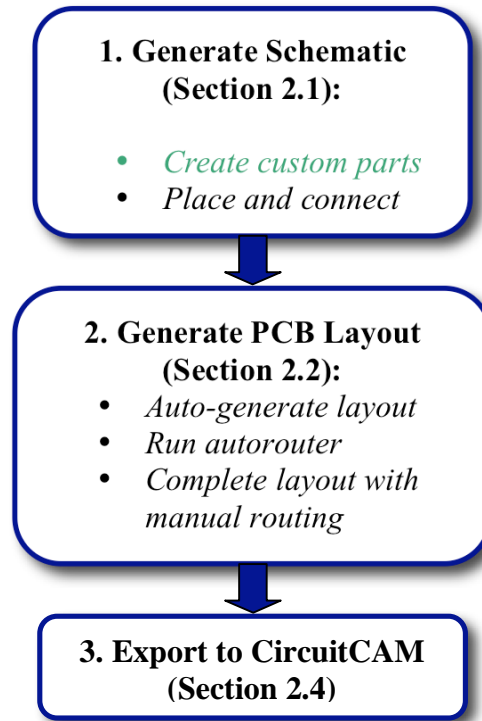

Figure 3: CAD flow for PCB design.

2.1 Generating a Schematic

Refer to the *Eagle* manual (available at www.cadsoft.de) for specific instructions on creating a schematic for your project. When deciding on which parts to use in the design, special attention must be paid to the package dimensions, as described in the following section.

2.1.1 Package Selection Guidelines

The smallest recommended package dimensions are shown in Figure 4. The most important package dimension is the pin "pitch", defined as the separation between the centre of the pins. Though a pitch of 0.5 mm has been successfully milled with the LPKF machine, it is not recommended due to the extreme difficulty of manually soldering the IC. If at all possible, SOIC or DIP packages should be selected as they typically have a large pitch and are much easier to solder than SOP packages.
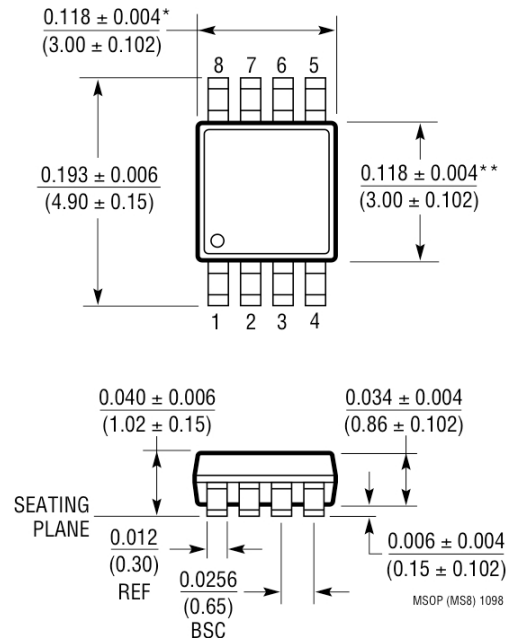
Figure 4: Smallest SM8 8-pin MSOP. Dimensions are in inches and (mm). The smallest recommended pitch is ~26 mils (0.65mm).

2.1.2 Creating Customs Parts

Eagle contains a vast collection of parts in the default libraries however it is likely that certain parts may not exist. If this is the case, a custom device must be created. The most time consuming part of creating a custom device is defining the package footprint. In the *vast* majority of cases, an existing footprint can be re-used from a device in the *Eagle* library. Before wasting effort creating a custom footprint, ensure that it does not already exist in the *Eagle* library. An example of a custom device is shown in Figure 5. The following high-level instructions explain how to define a custom device in *Eagle*:

1. Create your own *Eagle* device library.
2. Open the new library, click the *package* menu, enter the name of your package (ie: DIP8, SOIC8 etc.) and click *new*.
3. Paste the desired package from an existing part in the *Eagle* library (almost any imaginable package is already pre-defined).
4. Save the package and create a new symbol by clicking on the *symbol* button in the Library window.
5. Draw and save the schematic representation of your device, including all the pins.
6. A custom device can be created by linking the new symbol to the package definition, as shown in Figure 5. Create a new device by clicking the *device* button in the Library window.
7. Instantiate your custom symbol by clicking *add* in the main window.
8. Link the desired package by clicking *new* at the bottom of the window.
9. Connect the symbol pins to the package pins by clicking *connect*. This connectivity information is critical and must be entered very carefully.
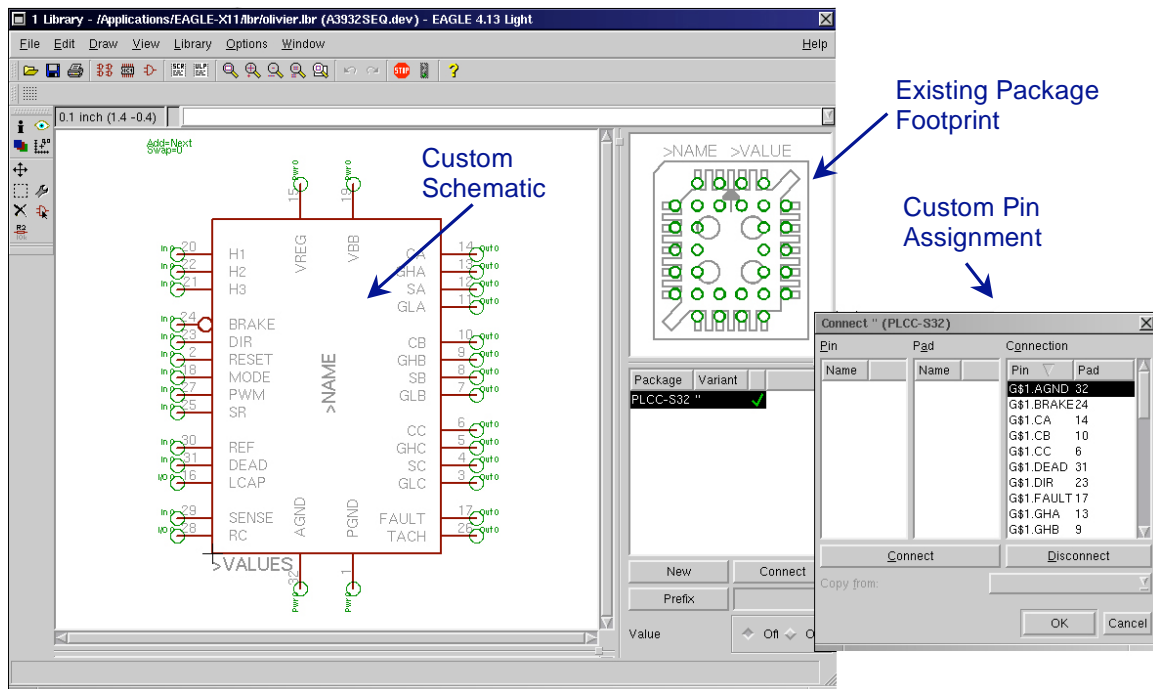
7

Table 1: Recommended Grid and Routing Settings

Setting	Value	Comments
Grid Size	50 mil	Convenient for standard footprints
Grid Alt	25 mil	
VIA drill diameter	24 mil	
VIA shape	circular	
Minimum Routing thickness	16 mil	Minimum reliable size for the milling equipment. Should be used for all traces that must be routed *between* standard 50mil pins.
Minimum Recommended Pin Pitch	25.6 mil (0.65 mm)	
Mode for Text Labels	Vector	(Do not use proportional fonts)

2.2.2 Important: Routing Restrictions Due to Unplated VIAs

The *Eagle* PCB software is a generic PCB tool and it therefore assumed that plated VIAs are available in the fabrication process. Plated VIAs allow traces to connect to a through-hole pin from either the top or bottom layer. The fact that plated VIAs are *NOT* available for the LPKF milling machine must be considered in the design process. The routing for certain through-hole components can only be done on the bottom layer. This is due to the fact that the part itself will block soldering access from the top layer. This concept is illustrated in Figure 6. *Eagle* does not recognize the fact that certain components must be routed from the bottom layer; it is the designer's responsibility to ensure that the layout is compatible with non-plated VIAs.
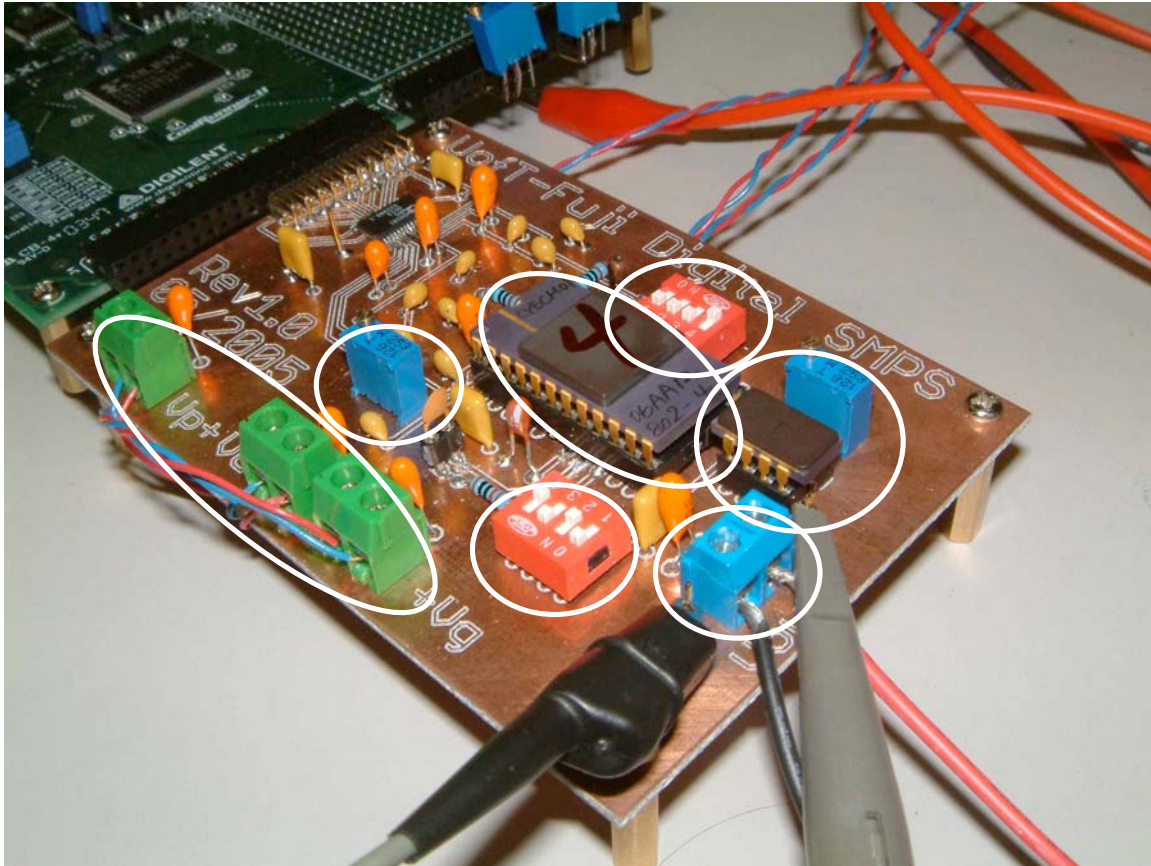
Figure 6: Sample PCB showing which parts (circled) must be routed from the bottom layer since the solder access from the top layer is blocked.

2.2.3 Creating a Ground Plane

In a typical two-layer PCB, the top and bottom copper layers are sandwiched between a fiberglass substrate, as shown in Figure 7. In most designs, both copper layers are connected to the ground node. The proper use of a ground plane can vastly improve the performance of an analog design. The low-resistance ground plane offered by the un-milled copper sheet of the PCB is used to avoid unwanted ground noise, as shown in Figure 8.
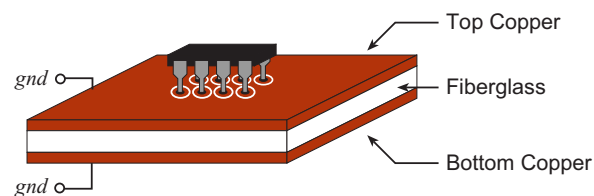

Figure 7: The top and bottom copper layers are sandwiched between a fiberglass substrate.
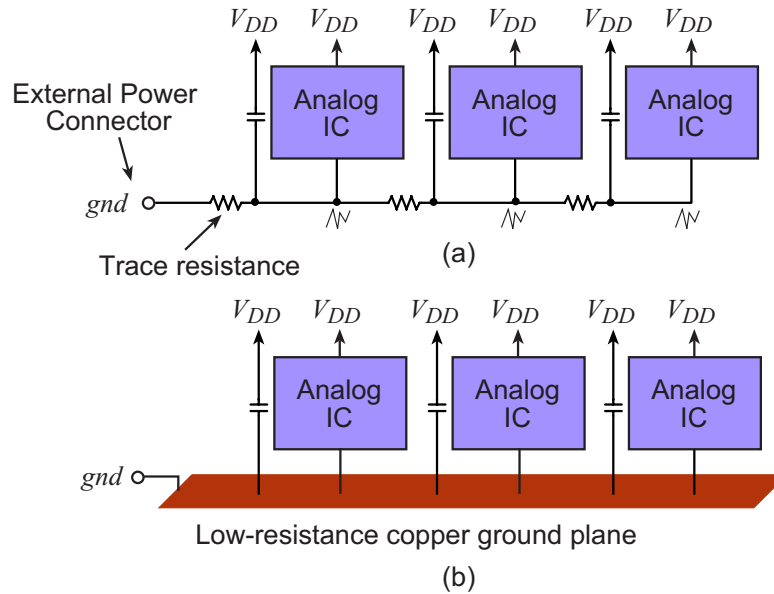
Figure 8 (a) A traditional ground connection, which results in a noisy ground potential and (b) a low-resistance ground plane.

You can connect a pin to the ground plane simply by leaving it un-routed in *Eagle*. An additional step will be required in *CircuitCAM* to ensure that the pin is properly connected, as explained in Section 3.1. If the auto-routing routine is used, you can selectively "*ripup*" (un-route) the ground node by entering the following command in the *Eagle* at command line:

Ripup gnd

The pins that remain connected to the *gnd* net are now connected using yellow "air wires" as shown in Figure 9.
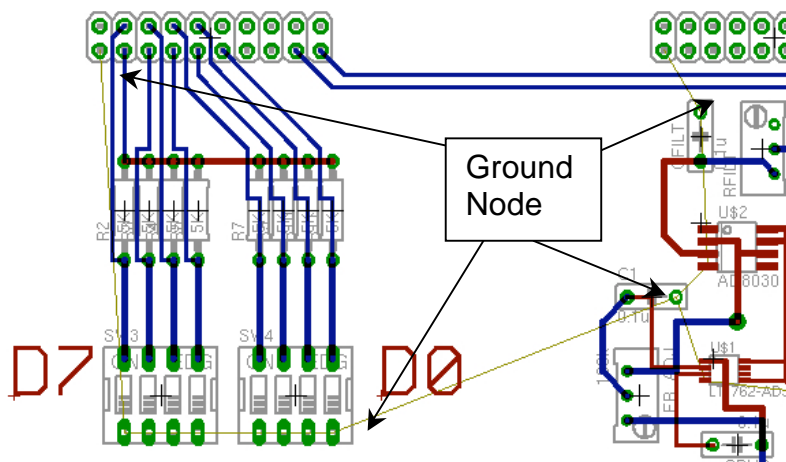

Figure 9: The ground node is ripped up after the auto-routing has been completed.

Typically, both the top and bottom layers of the PCB are used as a ground plane, which minimizes the number of unconnected "islands". The two planes should be shorted together at several locations by soldering the ground pin to both the top *and* bottom ground planes, as shown in Figure 10.

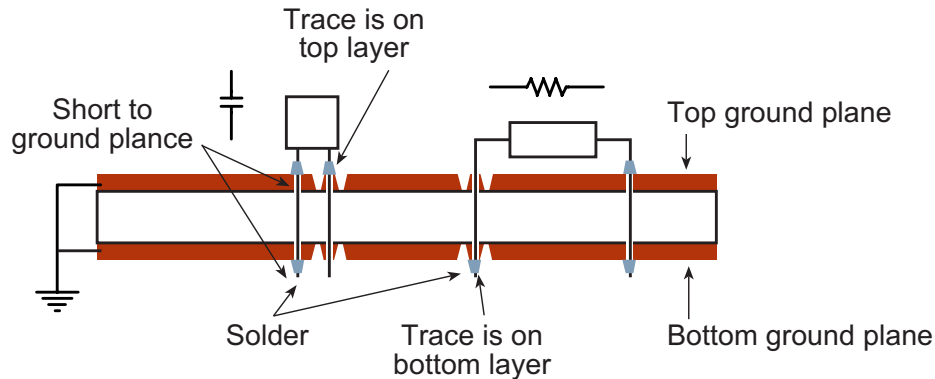Figure 10: Connecting the top and bottom ground planes.

2.3 PCB Example

A sample PCB schematic is shown in Figure 11. The simple circuit contains two surface-mount ICs, passive components, connectors and two DIP switches. The completed *Eagle* layout is shown in Figure 12. Notice that the ground pins are left un-routed since they will shorted to the ground plane.
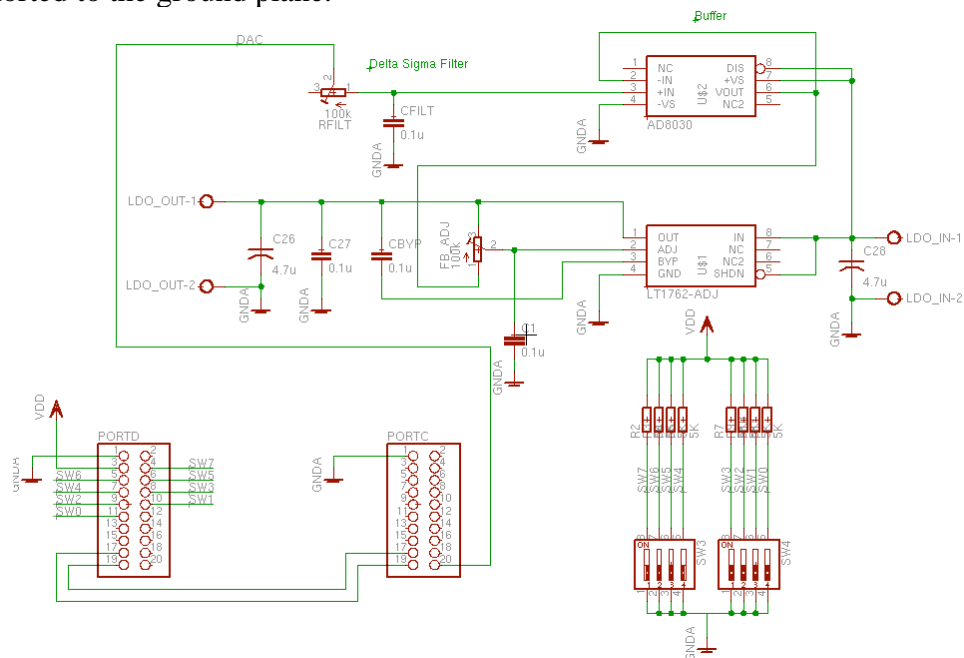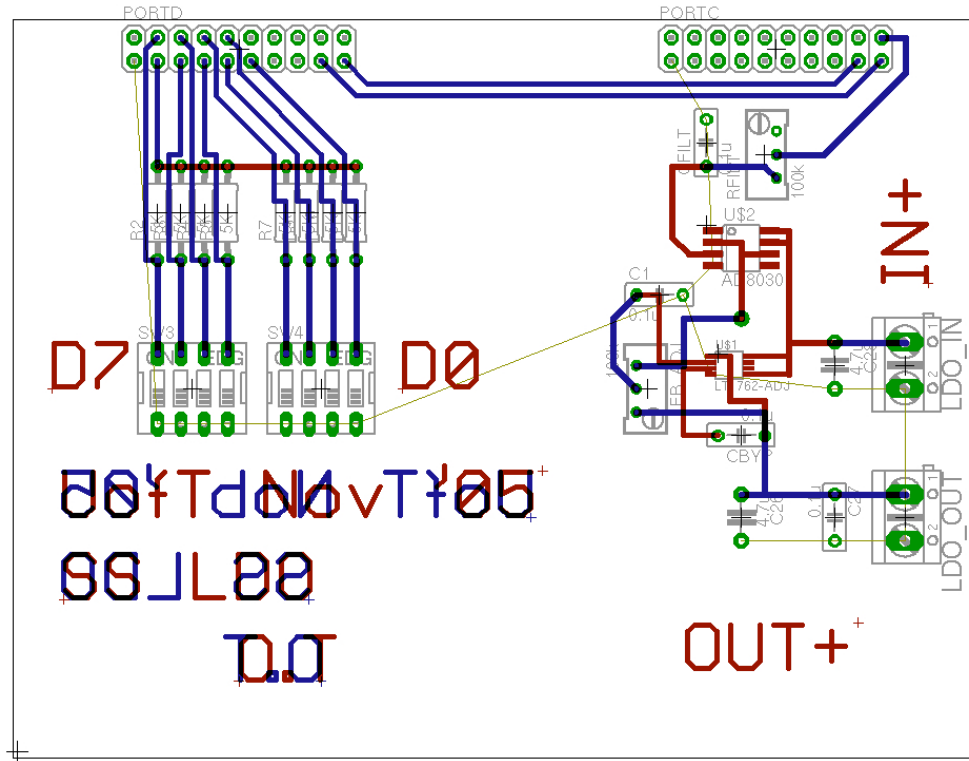

Figure 11: Schematic for sample PCB.

Figure 12: Layout for sample PCB.

2.4. Exporting to CircuitCAM

The five files listed in Table 2 are required to completely transfer the design from *Eagle* to *CircuitCAM*. Any additional files exported by *Eagle* (including silkscreen layers) should be ignored since they are not required by the milling machine.

Table 2: Required Files for Export to *CircuitCAM*

File	Data Contained in File
board.cmp	Component side, *TOP* layer
board.sol	Solder side, *BOTTOM* layer
board.otl	Board outline
board.drl	Drill configuration/aperture file
board.drd	Excellon drill file

The following steps are necessary to generate the files listed in Table 2 in *Eagle*. Several configuration files with the proper settings for the LPKF machine are available online.

Generate Files: *board.cmp, board.sol, board.otl:*

1. Create a folder in your local directory where the *GERBER* files will be stored

2. Open the CAM processor: *file > CAM Processor*
3. Open the sample export script gerber274X_import available at:
   www.vrg.utoronto.ca/~trescas/PCB
4. For each of the tabs, enter the path of your *GERBER* directory in the *file* box
5. Click *process job*

Generate Files: *board.drd:*

6. Open the sample Excellon export file available at:
   www.vrg.utoronto.ca/~trescas/PCB
7. Enter the path of your *GERBER* directory in the *file* box
8. Click process job

Generate File: *board.drl:*

9. Open the board layout file
10. In the command prompt at the top of the window, type: *run drillcfg.ulp*
11. Select *mm* for the units
12. Click *OK* when the aperture list appears and select your the target directory

The board design is now ready to be imported by *CircuitCAM*. Note that the connectivity information is not exported by *Eagle*. The exported files only contain the coordinates of the desired traces for both layers, as well as the required drilling diameters.

III. CAD File Preparation in *CircuitCAM*

The PCB files exported from Eagle are compatible with various fabrication methods. *CircuitCAM* is used to prepare the design data specifically for the LPKF milling machine available in the Design Centre. More specifically, *CircuitCAM* is used to merge the exported *Eagle* files and add an insulating layer to the traces. The procedure is facilitated using the *CircuitCAM* Wizard described below:

1. Open *CircuitCAM*
2. If the Wizard does not automatically appear, select *file > new*
3. Click on the Wizard icon which opens the wizard window
4. The "Layout Top Side" should be highlighted in green, click *next*
5. You will be prompted to select a file; choose the *board.cmp* file from your GERBER directory
6. The top side of your PCB should appear as shown in Figure 13.
7. Click *next;* you will be prompted to select a file for the layout bottom layer; choose your *board.sol* .
8. If the displayed layer look correct, select *yes* and click *next*
9. Both layers should now be correctly superimposed, as shown in Figure 14. Click *next*.

10. You will be prompted to select the "NC-drill file"; select your *board.drd.* The drill located are not shown in the wizard window. Click *next*.
11. You will be prompted to select the "Tool List for the PCB drills"; select your *name.drl* file. Click *next.*
12. Select *choose an existing translation file* and click *next*.
13. The translation file directory should popup; select the *eagle_excellon* template. The correct tools list should appear will your drill diameters, as shown in Figure 15.
14. If the layers and drill coordinates are properly aligned, proceed until *contour routing* is highlighted.
15. At this point, exit the wizard by clicking *graphic mode* to finish the remaining steps manually. Follow the instruction in the following section.
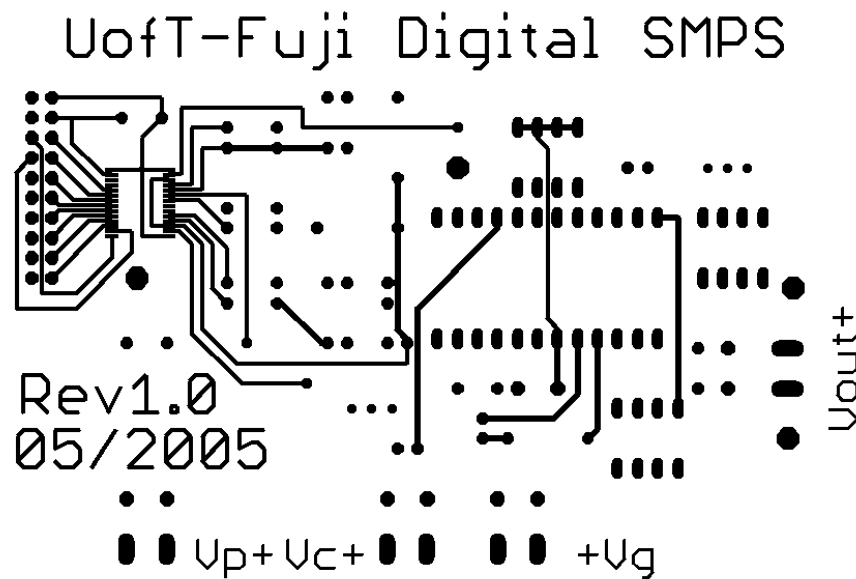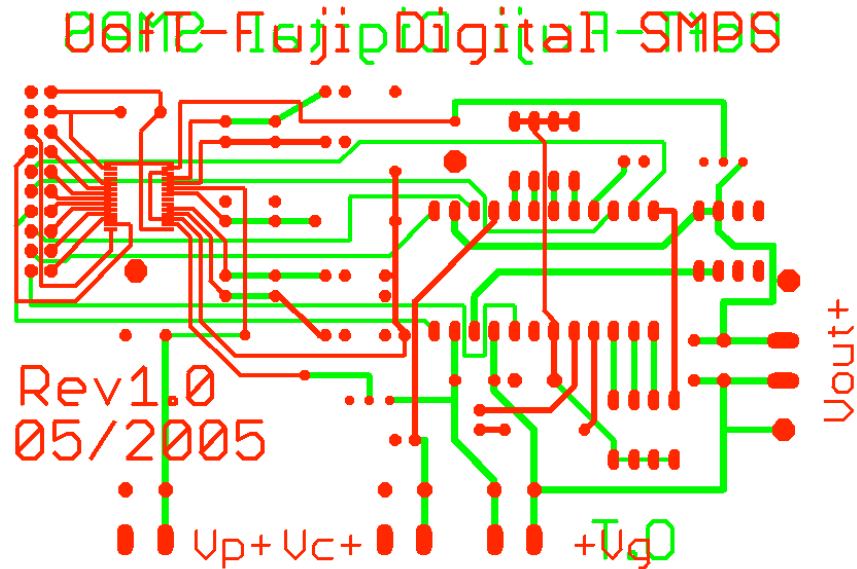

Figure 13: *CircuitCAM* imported top layer.

Figure 14: *CircuitCAM* imported top layers are superimposed.

Figure 15: *CircuitCAM* imported tool set.

3.1 Insulating the Traces, Defining Ground Pins and Contour Routing

The PCB traces must be insulated to avoid being shorted to the ground plane. *CircuitCAM* uses a built-in algorithm to automate the insulation. The insulation lines are used by the LPKF machine to mill around the PCB traces. In addition, a layer will be added to cut the PCB out of the copper board using contour routing. Follow the instructions below:

1. Click *file > import* and select the *board.otl* file from your GERBER directory.
2. Use the default settings (*CircuitCAM* should recognize that this is a "BoardOutline" layer) and click *import*
3. Select *edit > contour routing*
4. The default options should be chosen; a contour routing layer will be created around the outside of the board outline layer.
5. Select *edit > insulate*

6. The bottom layer should be selected. Keep the default settings and click *run* (not OK)
7. Once the process has completed for the bottom layer, redo step 5-6 but select the top layer in the insulation window. Once completed, your board traces should appear as shown in Figure 16.
8. The insulation must now be removed around the pins that should be shorted to the top and bottom ground planes. This is achieved by clicking the **FOUR** insulating layers and clicking *del* one-by-one. All four layers must be deleted (two of them are invisible until they are selected) as shown in Figure 17. Surface-mount pads only have two insulating layers.
9. Repeat step 8 for each pin that should be connected to the ground plane.
10. Click *file > export > LPKF > LpkfCircuitBoardPlotter.* Choose a file name and click *save*. This will save both the *CircuitCAM file* and the .LMD file that is used to mill the PCB.
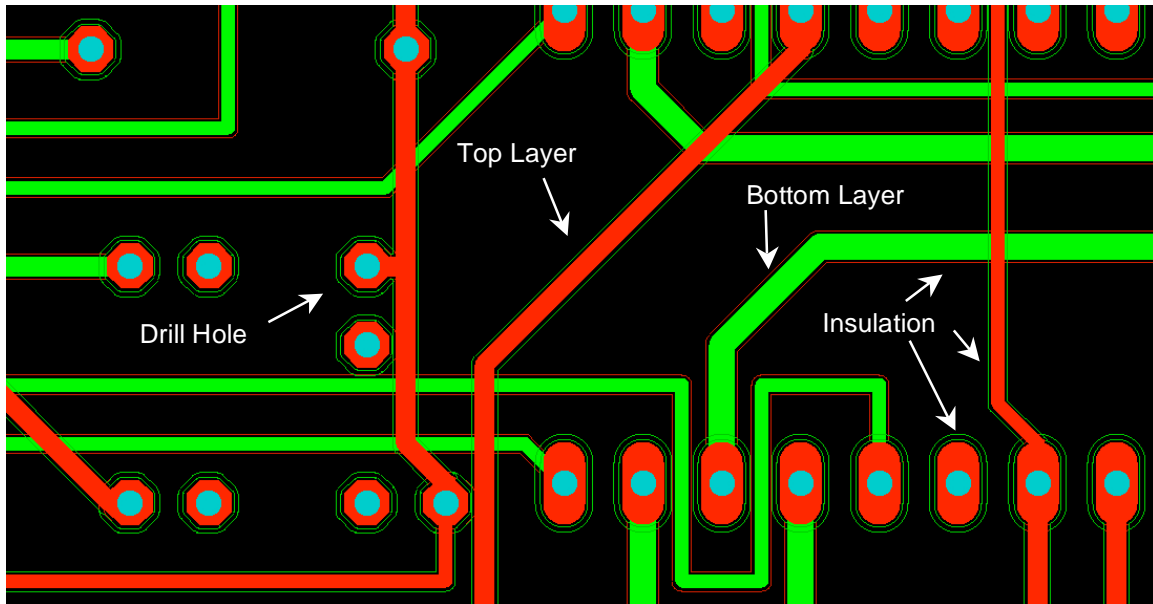

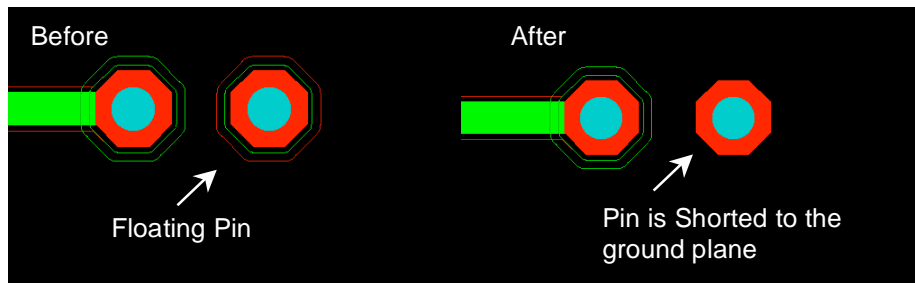Figure 16: PCB traces after running the insulation algorithm.

Figure 17: Grounded pin, before and after the insulation is removed.

3.2 Rub-out Regions

A rub-out region can be defined around a small-pitch surface-mount package to make soldering easier. The rub-out region is used to define the area where the ground plane should be completely milled away, as shown in Figure 18. Without a rub-out region, it is common to accidentally short a pin to the ground plane during the soldering phase. Large rub-out regions should be avoided as they cause unnecessary wear in the milling bit. It is left to the user to learn how to define rub-out regions within *CircuitCAM*.
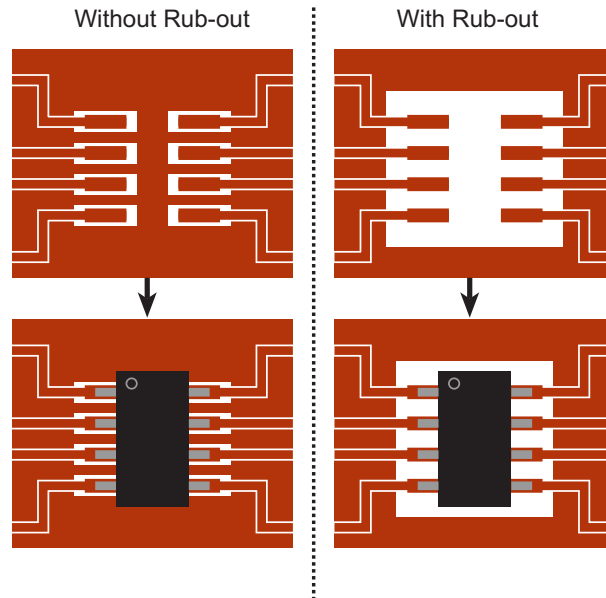
Figure 18: Milled package footprint shown with and without a rub-out region.

IV. PCB Fabrication using *BoardMaster*

The PCB fabrication flow is shown in Figure 19. The steps will be described in more detail in the following sections. In an attempt to accelerate the PCB fabrication process, the design centre will make copper boards available to students. These boards are already clean and the marking holes have already been drilled. In this case, students should proceed directly to step 3.
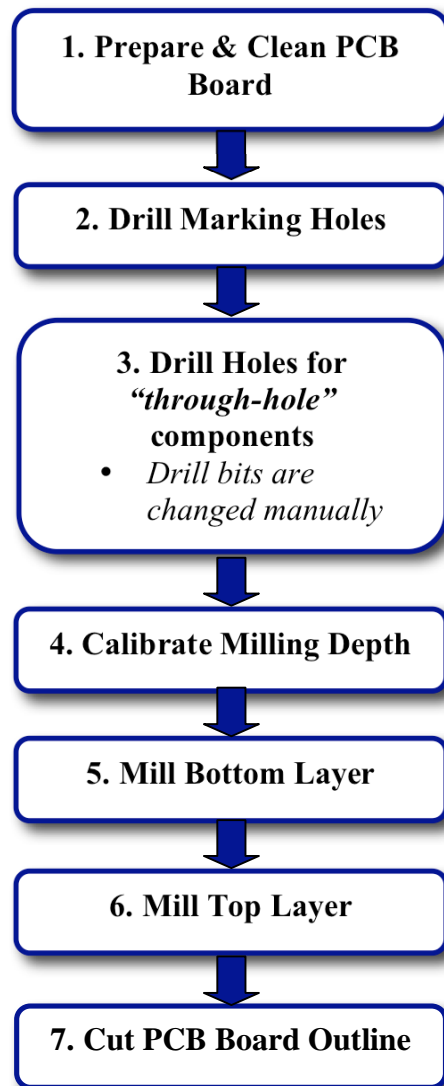
```
┌─────────────────────────────┐
│  1. Prepare & Clean PCB     │
│          Board              │
└─────────────────────────────┘
              │
              ▼
┌─────────────────────────────┐
│   2. Drill Marking Holes    │
└─────────────────────────────┘
              │
              ▼
┌─────────────────────────────┐
│     3. Drill Holes for      │
│      "through-hole"         │
│        components           │
│   •  Drill bits are         │
│      changed manually       │
└─────────────────────────────┘
              │
              ▼
┌─────────────────────────────┐
│  4. Calibrate Milling Depth │
└─────────────────────────────┘
              │
              ▼
┌─────────────────────────────┐
│     5. Mill Bottom Layer    │
└─────────────────────────────┘
              │
              ▼
┌─────────────────────────────┐
│      6. Mill Top Layer      │
└─────────────────────────────┘
              │
              ▼
┌─────────────────────────────┐
│   7. Cut PCB Board Outline  │
└─────────────────────────────┘
```

Figure 19: CAD flow for PCB fabrication.

4.1. Brief Description of LPKF Milling Machine

The computer controlled milling machine is shown in Figure 20. The milling machine is a sensitive piece of equipment and must be treated with great care to avoid damage and loss of milling accuracy. The vacuum is automatically controlled by the computer to keep the copper surface free of dust during the fabrication process. A piece of cardboard is placed on the PCB platform to avoid drilling into the aluminum below. The milling head can be automatically positioned anywhere on the cardboard PCB platform. The tool set for the milling machine is shown in Figure 21.
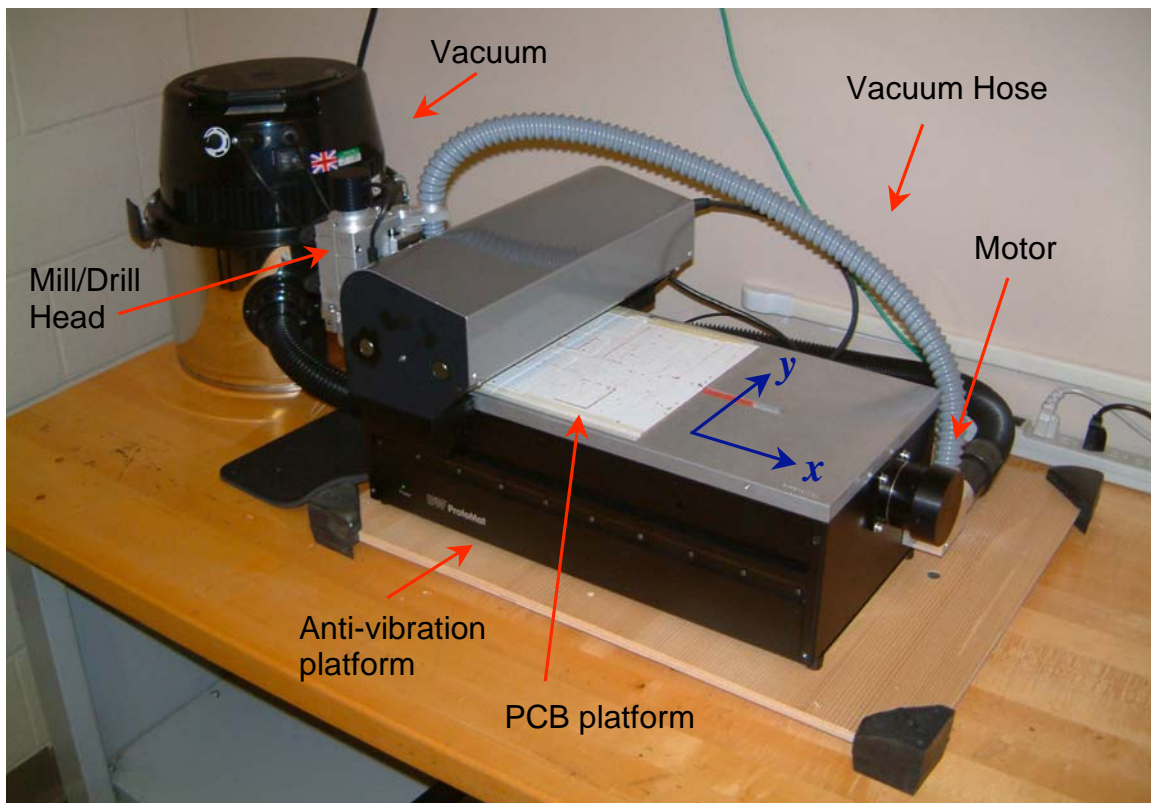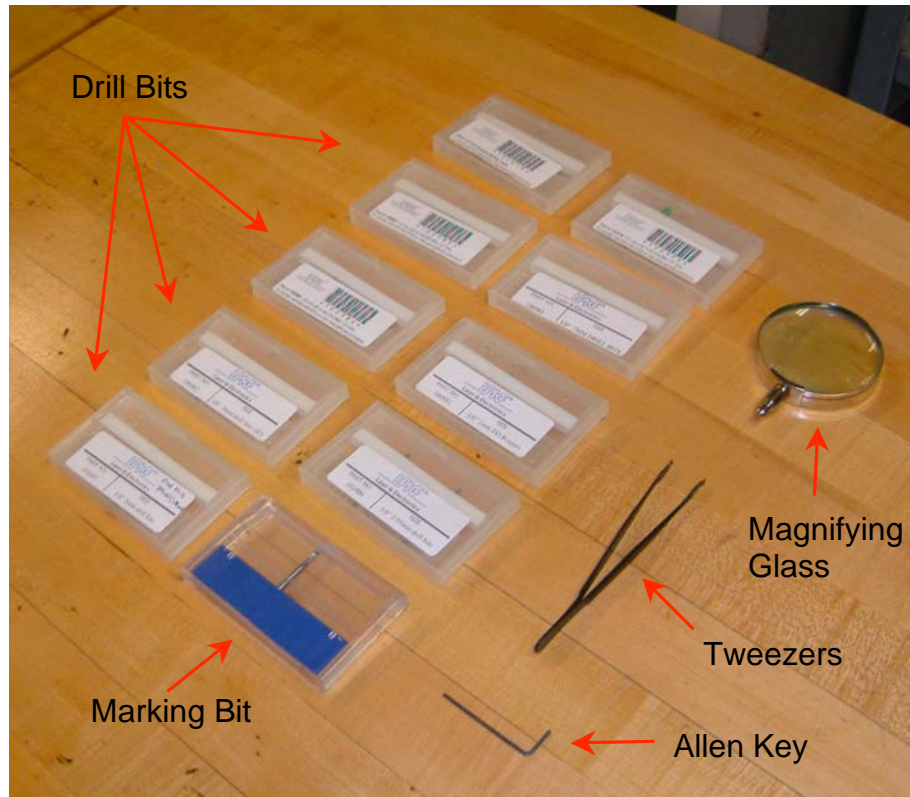

Figure 20: LPKF milling machine.

Figure 21: Tool set for the LPKF milling machine.

4.2. Performing Basic Tasks

This section describes how to perform certain repetitive tasks that are required throughout the PCB fabrication process using *BoardMaster*.

4.2.1 Moving to Predefined Locations on the PCB Platform

The drill/mill head can be positioned to one of the three pre-defined locations as shown in Figure 22. The *home* position corresponds to the x,y origin of the PCB platform. The head must be in the *exchange* position to load/unload bits into the shaft. The drill head can be moved to the *pause* position to view the entire PCB platform. To position the drill head, simply click: *Go to >exchange/pause/home* in the *BoardMaster* window.
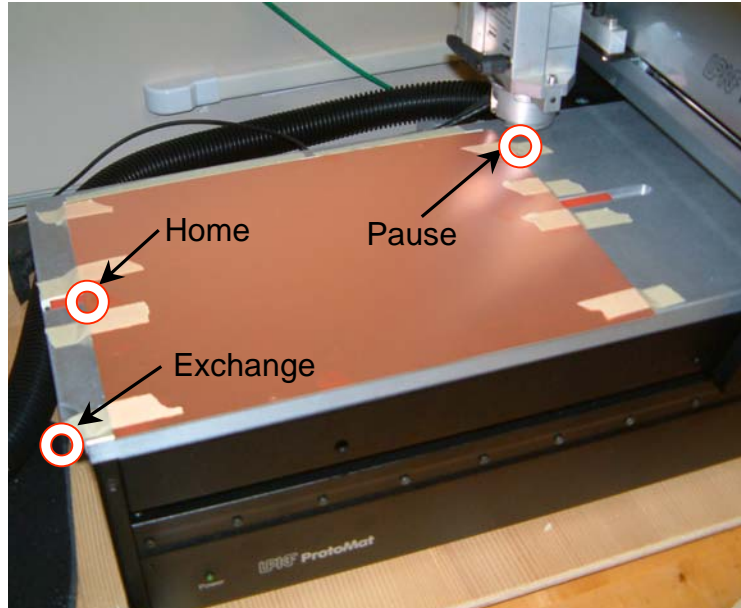
Figure 22: Pre-defined positions on the PCB platform.

4.2.2 BoardMaster Buttons

The most important buttons in the *BoardMaster* window are numbered in Figure 23. The button functions are outlined in Table 3. The head can be positioned very precisely using the positioning buttons as shown in Figure 24. The current fabrication phase can be selected using the drop-down box shown in Figure 24.
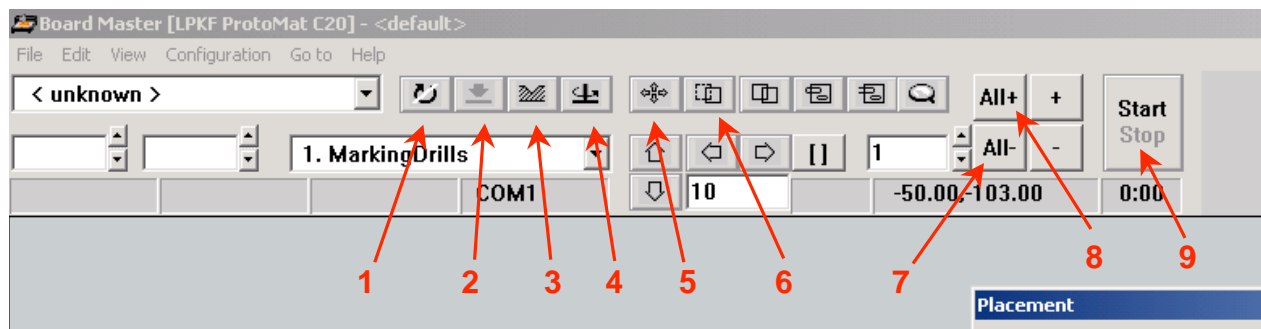
Figure 23: *BoardMaster* window.

Table 3: *BoardMaster* Buttons

#	Button Name	Function / Description
1	Motor Enable	Turn motor *on*/*off*
2	Drop Motor Head	Lowers the drill-head to drill a hole or start milling
3	Drill Mode Select	Selects between milling or drilling mode
4	Motor Control	Set motor control to manual or automatic (CAD controlled)

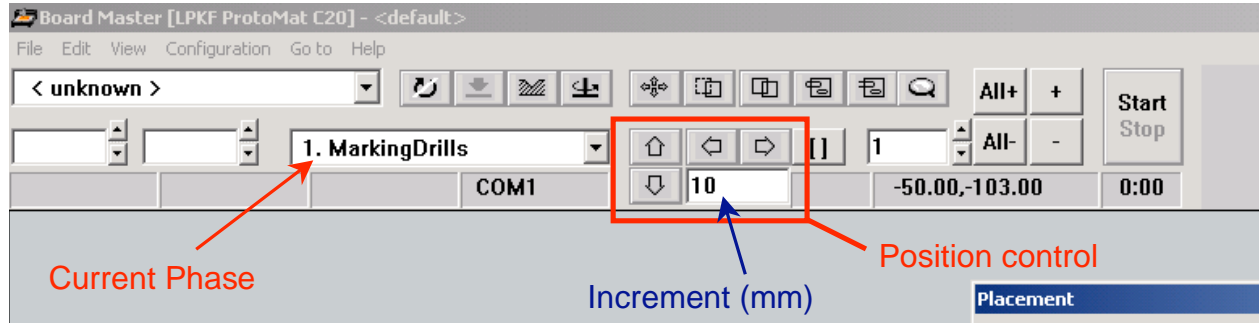| 5	Manual Position Control	When clicked, the mouse cursor is used to manually position the drill head on the PCB platform
6		
7	All -	De-select all designs present in the window (w.r.t. performing a specific task)
8	All +	Select all designs present in the window (w.r.t. performing a specific task)
9	Start/stop	Start or stop one of the fabrication phases

Figure 24: BoardMaster window showing the manual positioning buttons.

4.2.3 Loading the Drill/Mill Bits and Adjusting the Depth

The LPKF milling machine does not support automatic tool loading and therefore each drill/mill bit must be loaded and unloaded manually. This process is illustrated in Figure 25. The Allen key is used to loosen the set screw in the drill shaft while the drill/mill bit is gently inserted into the shaft using the tweezers. **Be careful** not to over-tighten and damage the set-screw. The drilling depth must be adjusted to avoid damaging the PCB platform. The height adjustment screw should be used a shown in Figure 26. To estimate the necessary drilling depth, place a test copper board flush with the drill head. Turn the adjustment screw until the drill bit barely protrudes below the test board. This ensures that the drilling depth is sufficient without damaging the PCB platform.
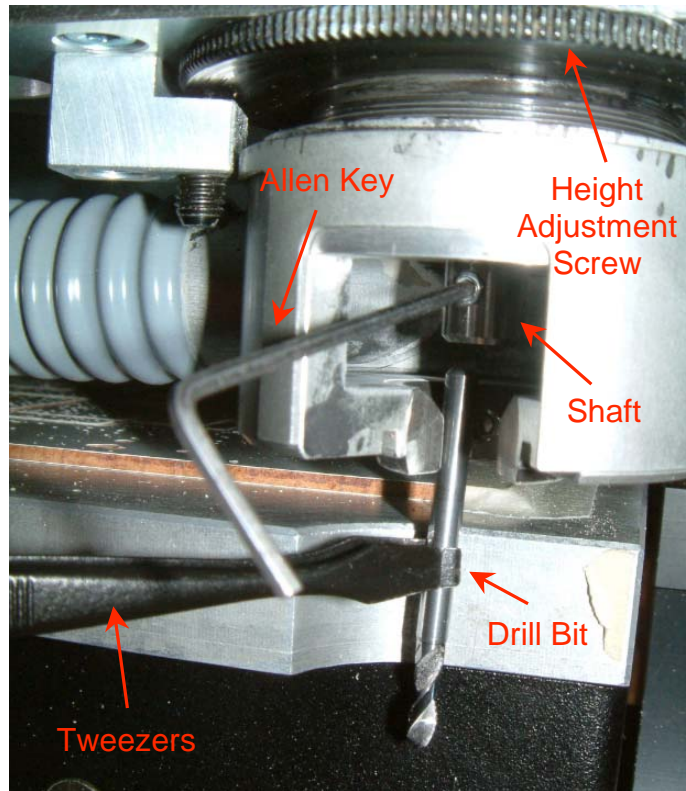
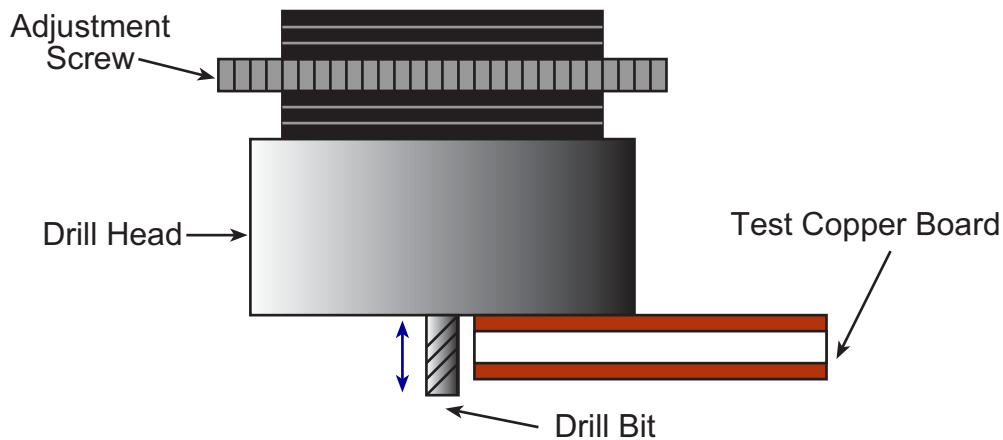Figure 25: Loading a bit into the drill head.

Figure 26: Adjusting the drill depth using a test copper board.

4.2.4 Calibrate Milling Depth

Move to the *exchange* position and load the universal milling bit. Adjust the height so that the tip of the bit barely protrudes from the drill head, otherwise the milling depth will be much too deep and **the milling bit will be destroyed**. Properly adjusting the milling depth is the most chAllenging step in the fabrication process. The milling depth must be adjusted manually. If the isolation groove is too shallow, traces will not be properly

isolated, as shown in Figure 27. If the groove is too deep, copper lines that are closely spaced may be completely milled away. It is especially critical to set the proper depth when fine-pitch packages are used in the design.
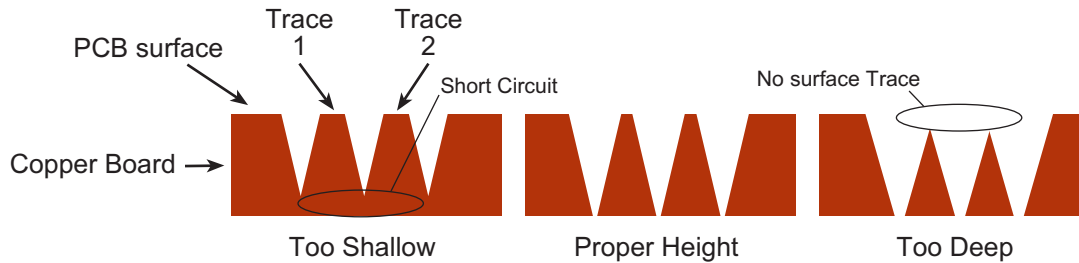

Figure 27: The milling depth must be carefully adjusted.

The milling depth should be adjusted using the procedure illustrated in Figure 28. The following steps can be used to systematically determine the proper depth:

1. Move the milling head to the edge of the copper board, away from the area allocated to the PCB board.
2. Mill a 5mm trace using the buttons/commands: *motor on > drop head > move 5 mm in x direction > motor off > raise head*.
3. Move the drill out of the way for a visual inspection using the buttons/commands: *move 40 mm in y direction.*
4. Inspect the trench. If the trench is too shallow, bring the head down 40 mm, lower the milling bit by 15 clicks and repeat steps 2-3 until the depth is calibrated. A magnifying glass is available for the visual inspection, as shown in Figure 29.
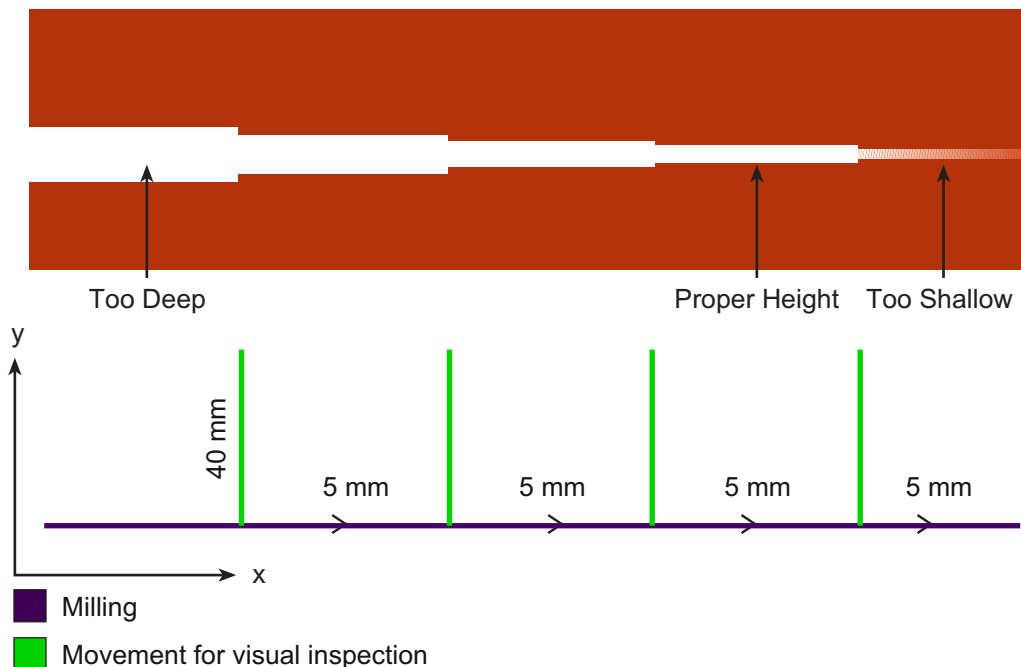

25

Figure 28: Graphical representation of procedure used to calibrate the milling depth.
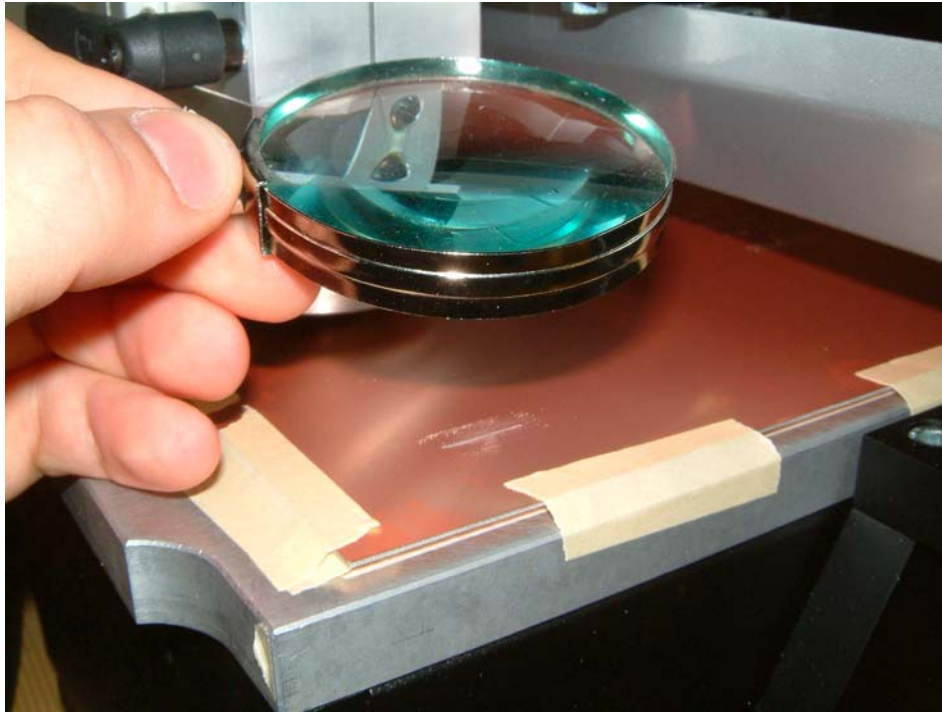

Figure 29: The magnifying glass can be used for the visual inspection of the milling depth.

4.3 Fabrication Steps

This section includes the PCB fabrication steps. Refer to Section 4.2 for detailed instructions on the specific tasks.

4.3.1 Import the PCB Board

1. Obtain a clean two-sided copper board from the design centre. Note: copper boards from *Supremetronics*/ *Active Surplus* are **NOT** acceptable.
2. Open BoardMaster and click *file > import > LMD/LPR*.
3. Choose your .LMD file and click OK.
4. Your design should appear as shown in Figure 30; only the bottom layer should be visible. Right click on the board to change the location if desired. The coordinates can be entered directly.
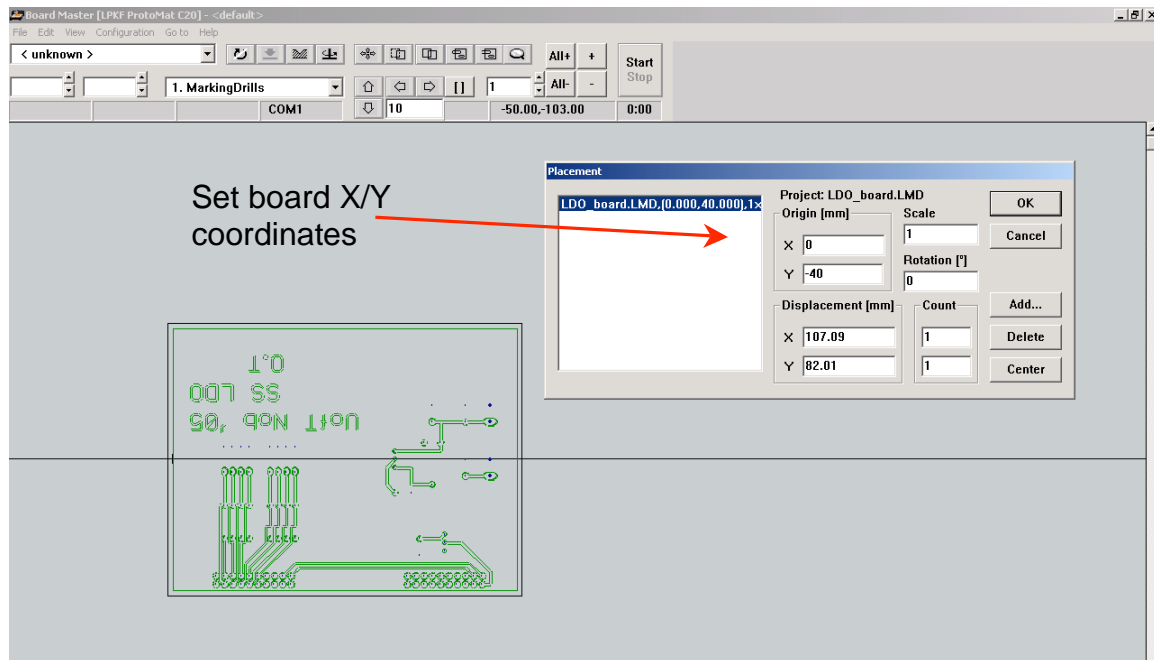
Figure 30: The PCB design is imported into BoardMaster. Only the bottom layer is visible. The drill holes are shown in blue.

4.3.2 Drill Marking Holes

1. Click *Go to > exchange*.
2. Load the marking hole drill bit.
3. Click *Go to > pause*.
4. Using a pencil, draw a dot on the left edge of the copper board (see Figure 31) and tape the board such that the dot is offset from the alignment stubs by ~ 10mm as shown in Figure 32.
5. Position the drill head above the drawn dot on the left of the board using the manual positioning function, as shown in Figure 33.
6. Drill the first marking hole using the button sequence: *motor head on > drop head > raise motor head > motor head off*.
7. Move the drill head **287 mm** to the right using the manual displacement button. The head will now be positioned exactly over the second alignment post, as shown in Figure 34.
8. Drill hole the second hole using sequence of buttons: *motor head on > drop head > raise motor head > motor head off*. Remove the tape and place the board on top of the alignment posts and Re-tape the board securely in place as shown in Figure 35.
9. Click *Go to > exchange*.
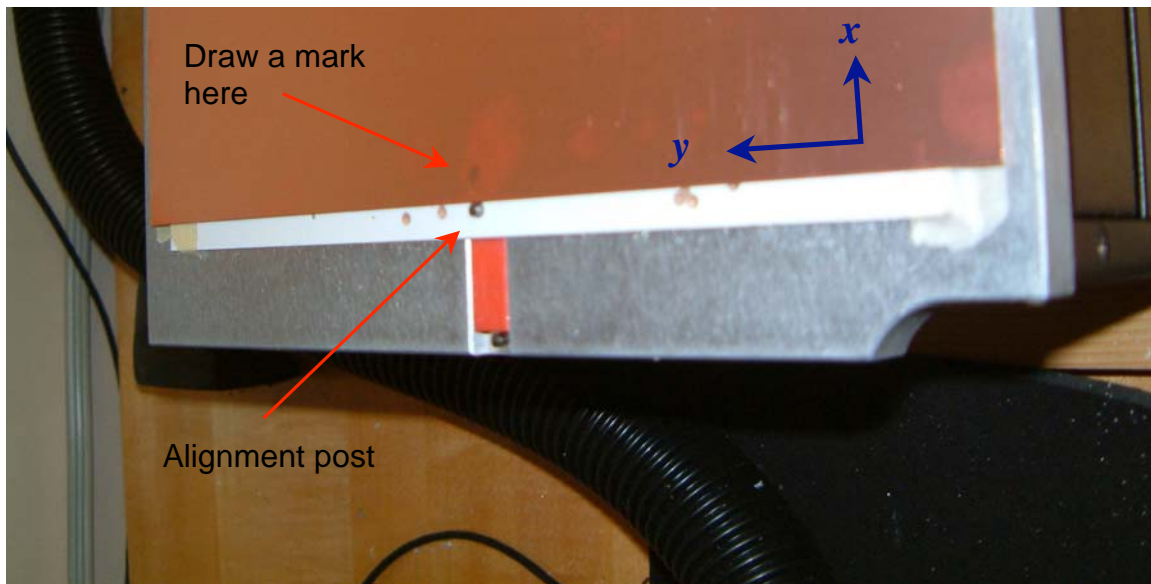10. Remove the marking drill bit.

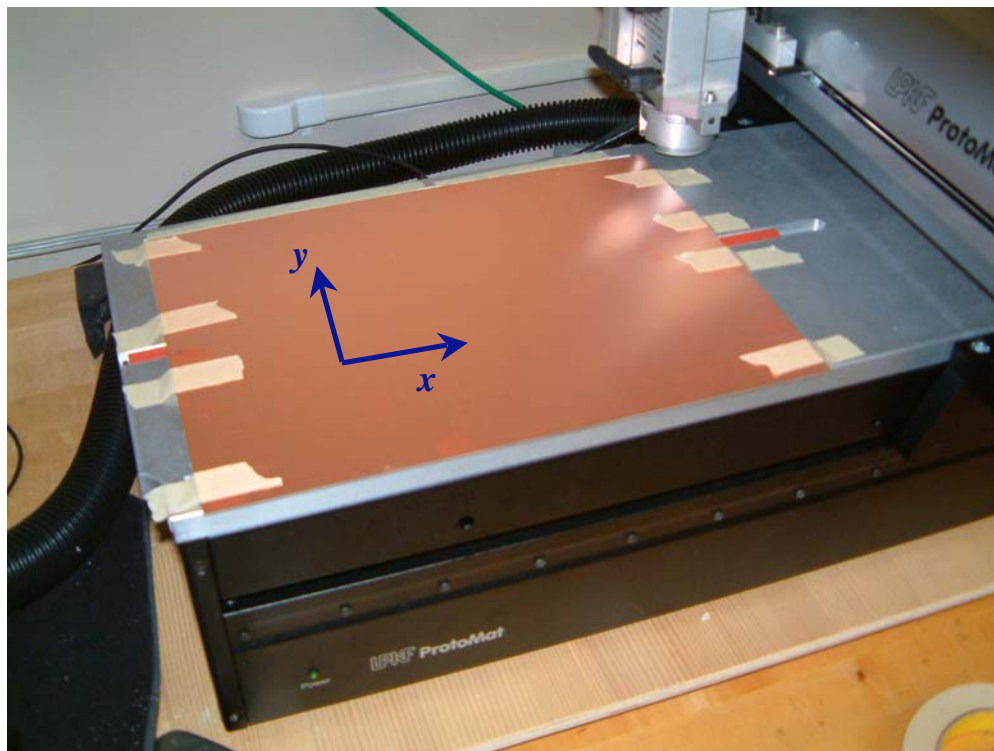Figure 31: Position copper board beside alignment post and draw mark.

Figure 32: Position copper board to drill marking hole. The marked dot must be offset from the alignment post.
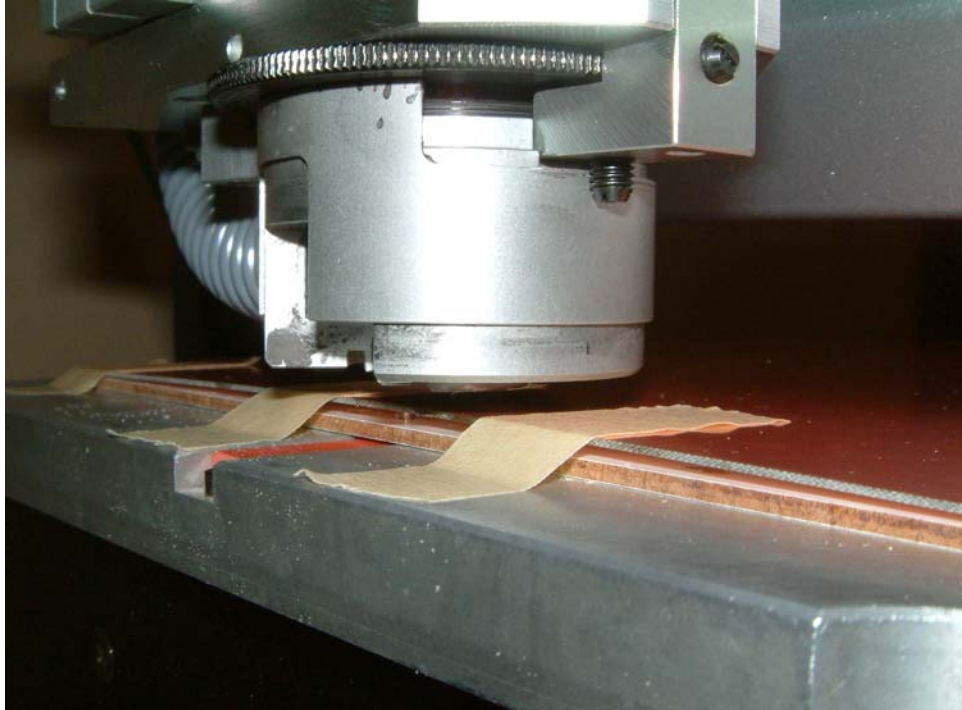
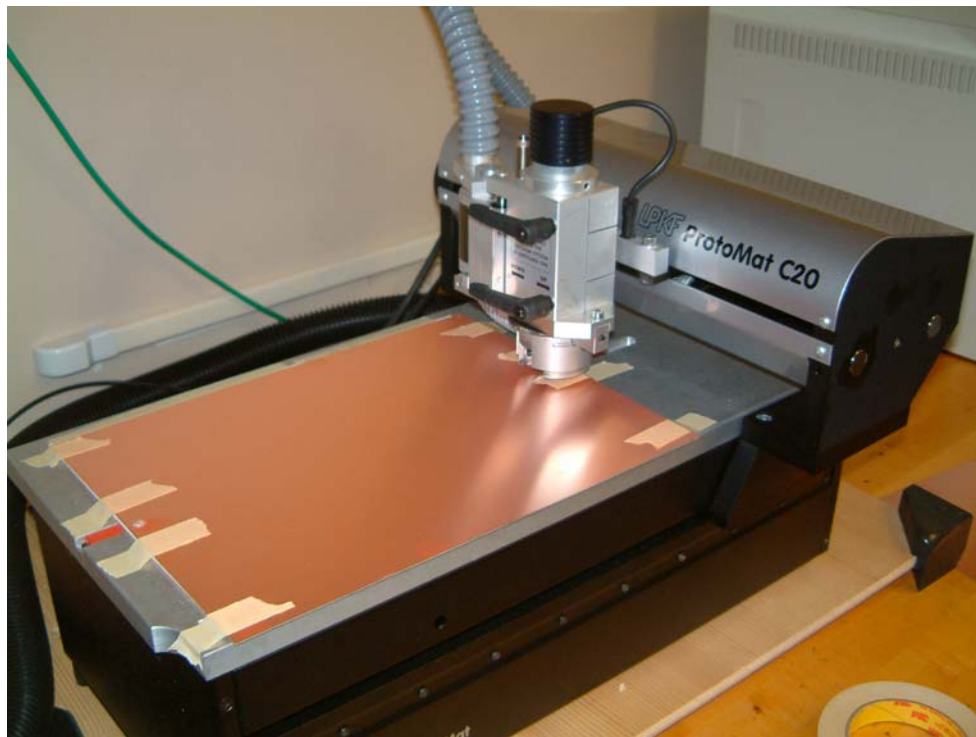Figure 33: Position copper board to drill marking hole. The marked dot must be offset from the alignment post.

Figure 34: Drill the second marking whole.

4.3.3 Drilling Phase

1. Before proceeding, the drill head should already be in the *exchange* position and the board securely taped on he alignment posts, as shown in Figure 35.
2. Check that the PCB board coordinates are as desired by manually moving the drill head to the coordinates of the edge of the PCB board as shown in Figure 36. This will tell you exactly where on the board the PCB will be milled. This is especially important for milling several PCBs in one session.
3. Change the mode to *drilling unplated*.
4. Click the *auto drill head control* button.
5. Click *all +* and click *start*.
6. You will be prompted to load the desired drill bit. Adjust the drilling depth as described in Section 4.2.3.
7. Follow the instructions until all the holes have been drilled. If the exact bit diameter is not available, use the next smallest bit.
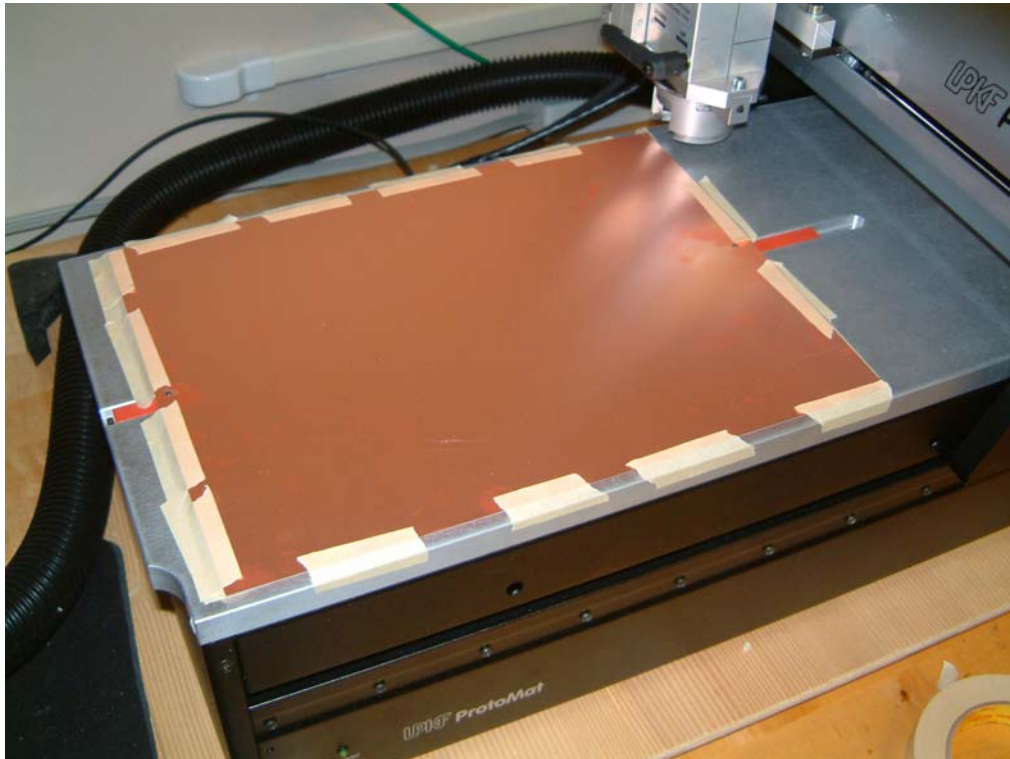

Figure 35: Once the marking holes have been drilled, the board should be placed on the alignment posts and securely taped.

4.3.4 Milling Layers and Contour Routing

Once the milling height has been adjusted, follow these steps:

1. Change the mode to "*4.Milling bottom phase*".

2. Click *all+* and then *start*. You will be prompted to load the milling bit. Select *pause to calibrate*.
3. Calibrate the milling depth as described in Section 4.2.
4. The drill head will be automatically controlled to mill the board. During this lengthy process, the progress will be visible on the BoardMaster window as shown in Figure 36.
5. Once the process has completed, click *goto > pause* and inspect the board (see Figure 38. If the milling was not performed correctly, halt fabrication and seek assistance.
6. Remove the tape, flip the board *vertically* (in the y-axis) and re-apply the tape.
7. Change the mode to "*5. Milling top phase*". The top layer of your board should appear in the BoardMaster window.
8. Click *all+* and then *start*. Click *continue* (calibration has already been completed in step 3).
9. Once the process has completed (see Figure 39), click *goto > pause* and inspect the board. If the milling was not performed correctly, halt fabrication and seek assistance.
10. Change the mode to "*7. Cutting Outside*". Click *all+* and then click *start*.
11. When prompted, remove the milling bit and load the 2mm DD router bit. Adjust the height to ensure that the router bit will not drill beyond the cardboard layer.
12. Once completed (see Figure 40) click *goto > exchange* and remove the router bit.
13. Click *goto > pause*. Remove the tape and the copper board. Clean the workstation and shutdown the design center PC.
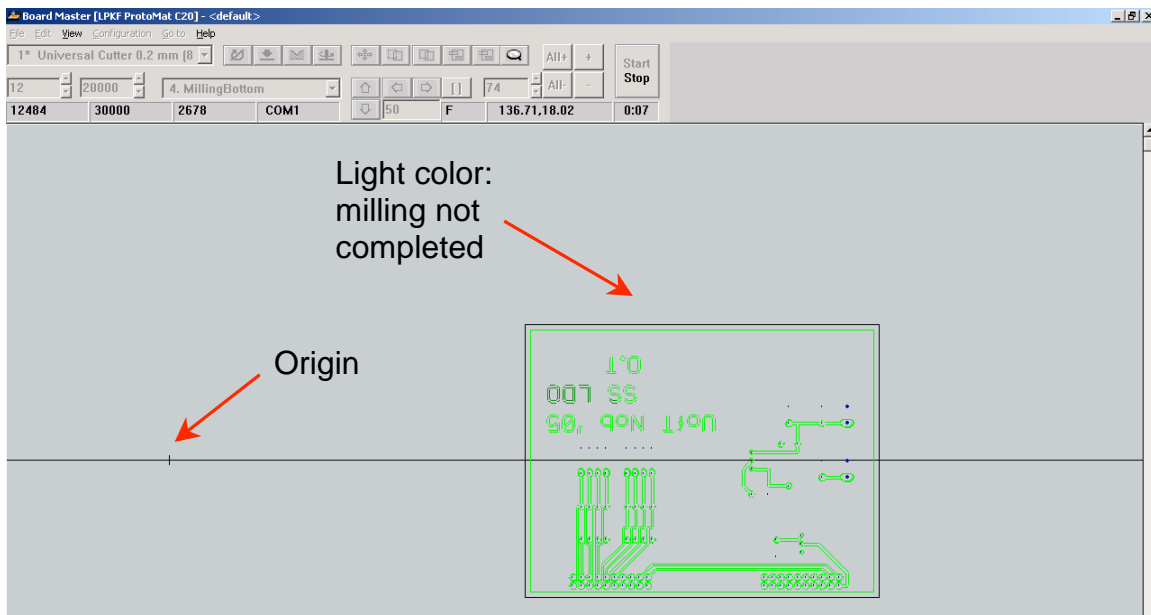

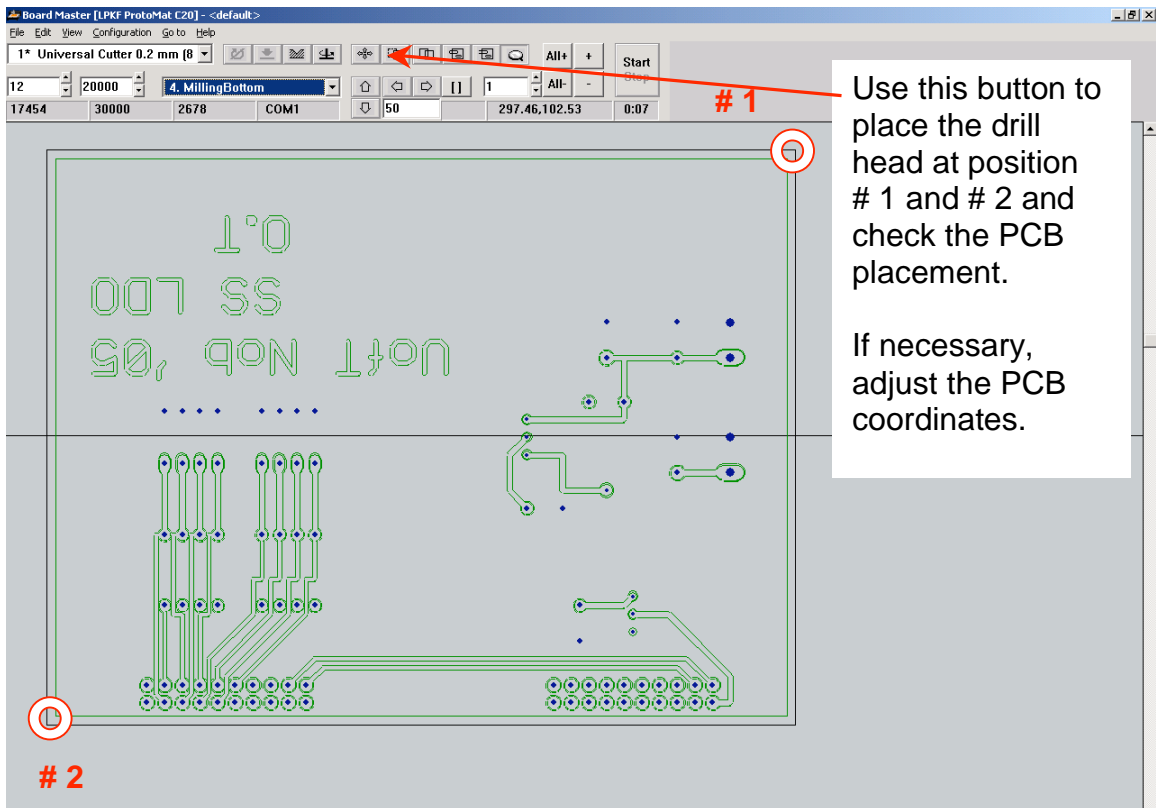Figure 36: The milling progress is displayed in the *BoardMaster* window.

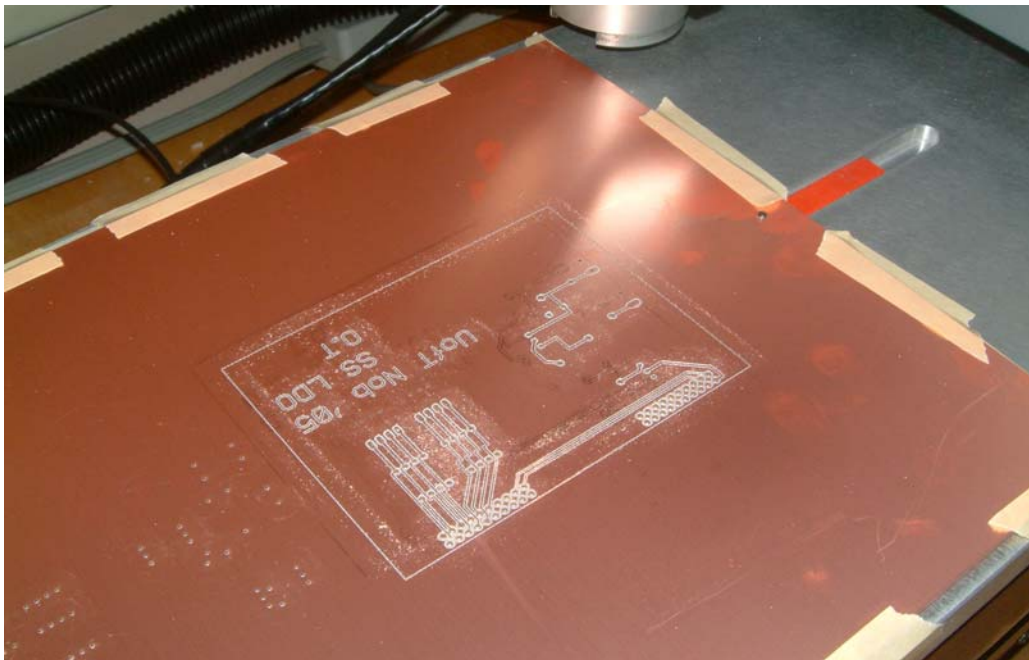Figure 37: Check the PCB placement on the copper board.

Figure 38: The bottom layer has been milled. Inspect the board before proceeding to milling the bottom layer.
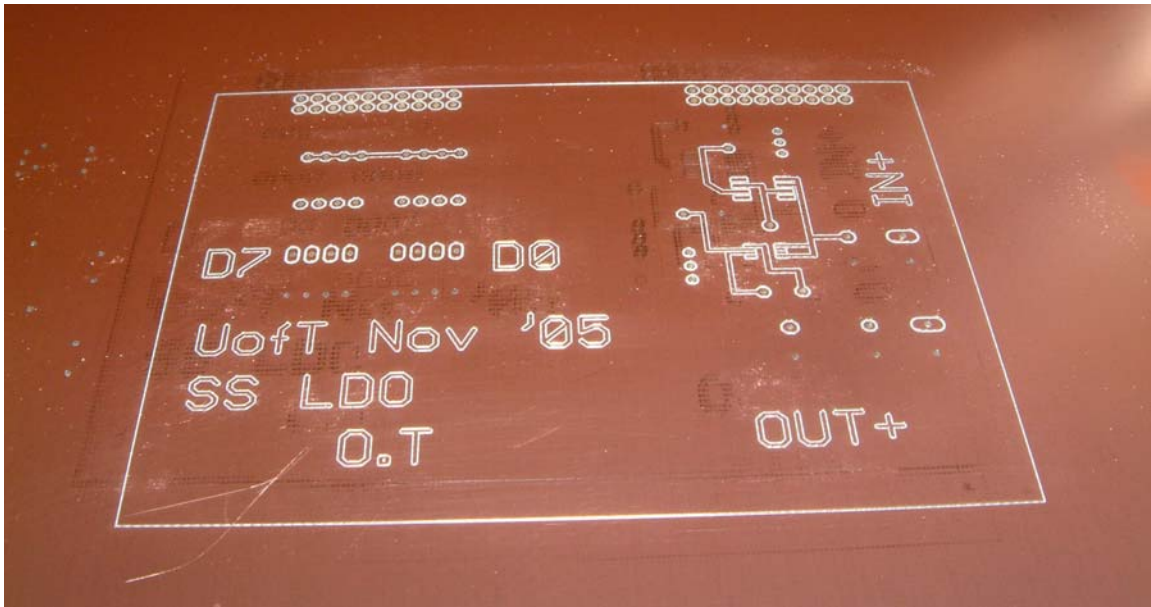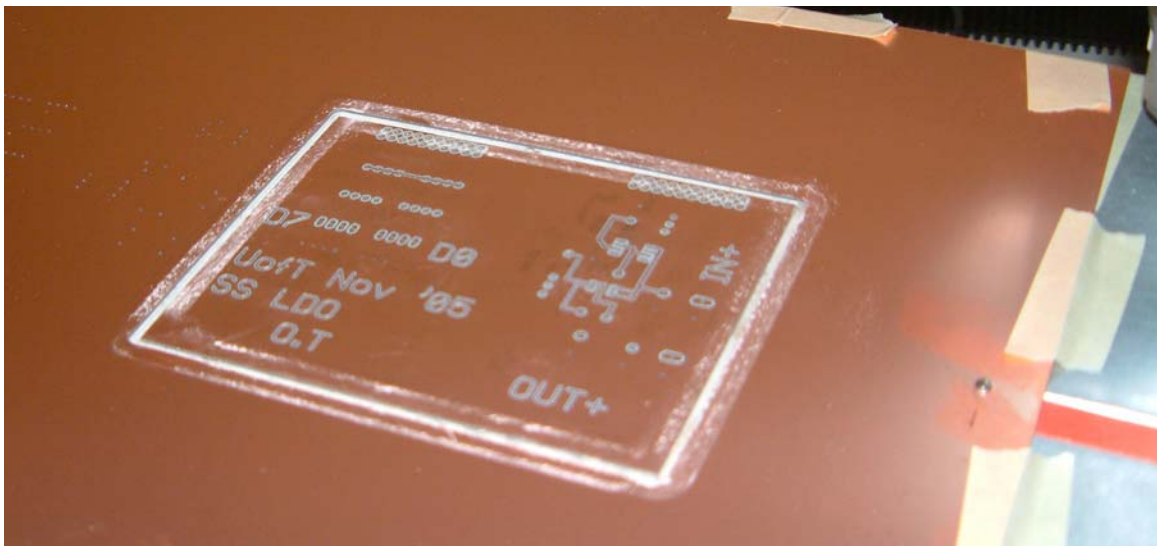
Figure 39: The top layer has been milled.

Figure 40: Completed PCB board. The PCB has been cut out of the copper board.

V. PCB Quick Reference Sheet

This section is intended as a quick reference guide for experienced users only.

Eagle → CircuitCAM

1. Open the CAM processor: *file > CAM Processor*
2. Open the sample export script gerber274X_import available at: www.vrg.utoronto.ca/~trescas/PCB
3. For each of the tabs, correct the file path and click *process job*
4. Open the sample Excellon export file available at: www.vrg.utoronto.ca/~trescas/PCB
5. Correct the file path and click *process job*
6. Open the board layout file
7. In the command, type: *run drillcfg.ulp* (Select *mm* for the units)

CircuitCAM → BoardMaster

1. Launch import wizard
2. Import "Layout Top Side" (*board.cmp*)
3. Import "Layout Bottom Side" (*board.sol*)
4. Import "NC-drill file" (*board.drd*)
5. Import "Tool List" (*name.drl*) with *eagle_excellon* template
6. Select *choose an existing translation file* and click *next*
7. Exit wizard (*graphic mode*) to finish the remaining steps manually
8. *file > import* : *board.otl*
9. *edit > contour routing* (default options)
10. *edit > insulate* , select bottom layer and click *run*
11. Repeat for top layer insulation
12. Remove insulation around ground plane nodes
13. *file > export > LPKF > LpkfCircuitBoardPlotter* to generate .LMD for *BoardMaster*

PCB Fabrication

1. Fill out PCB fabrication request form (signed by supervisor)
2. Import .LMD file into *BoardMaster* and set the desired XY coordinates
3. Tape board beside alignment post and drill first marking hole
4. Move 287mm in x direction and drill second marking hole
5. Tape board securely onto marking holes and proceed to drilling unplated phase
6. Load the requested drill bits as necessary
7. When completed, proceed to milling bottom phase and calibrate milling depth
8. Rotate board in y direction, re-tape the board and proceed to milling top phase
9. Proceed to contour routing phase; remove board from copper substrate
10. Clean workstation and shut-off PC