

Volume

1

GUIDE TO FABRICATING PCBS

Using the Bungard CCD

Step-by-Step Tutorial

BUNGARD CCD

Guide to Fabricating PCBs

Blake Robertson
4250 Knox Road • Suite 2123
College Park, MD 20470
Phone 301.226.0482 • blaker@umd.edu

Table of Contents

ABOUT THIS MANUAL	I
IMPORTANT SAFETY PRECAUTIONS	II
FILES USED IN THIS GUIDE	III
SECTION 1: MACHINE SETUP	1
Assembly Requirements	1
Terms Used In Manual	2
Connections Overview	3
Connecting the PC to the Control Unit	4
Connecting the Control Unit to the Machine	5
Automatic Tool Change Connections	6
Mounting the Spindle	8
Assembly Complete!	9
SECTION 2: RECOMMENDATIONS	10
SECTION 3: CONFIGURING THE SOFTWARE	11
Configuration Requirements	11
Configuring Eagle PCB Layout	12
Configuring IsoCam	15
SECTION 4: PREPARING ISOLATION MILLING FILES	16
Section Requirements	16
Exporting Gerber & Drill Files from PCB CAD Software	17
Generating the Milling Data using IsoCam	19
Advanced Features Using IsoCam	27
SECTION 5: MILLING A PRINTED CIRCUIT BOARD	28
Milling Requirements	28
Securing Base Board and Blank Circuit Board to the Milling Table	29

Defining a Milling Offset Position	31
Drilling Fixing Holes	33
Drilling Holes for Circuit Components	36
Routing the Top Side of the Circuit Board	37
Routing the Bottom Side of the Circuit Board for Double Sided Designs.	39
 SECTION 6: ADVANCED MILLING OPTIONS	 40
Milling Requirements	40
Using the Depth Limiter	41
Panelizing Circuit Boards	43
 SECTION 7: DESK REFERENCES	 44
 SECTION 8: PREPARING ISOLATION MILLING FILES	 45
 SECTION 9: TROUBLESHOOTING	 46
 SECTION 10: FUTURE REVISIONS	 48
 SECTION 11: APPENDIX	 50
Appendix A: Glossary of Terms:	50
Appendix B: Technical Defination of Gerber & NC Drill Files	51
 BIBLIOGRAPHY	 53

About this Manual

This manual has been rewritten by Blake Robertson to address a new audience of Universities and companies that lack prior experience using CNC equipment. This new audience will still be engineers and technically minded people who have some experience using PCs and installing software.

Disclaimer: The information in this manual is not endorsed by the company and should be used at your own risk. Before you follow this guide you should read the official manually completely. This manual isn't intended to completely replace the Bungard CCD Manual.

TYPOGRAPHICAL CONVENTIONS

The following list defines the typographical conventions used throughout the manual. The typographical conventions used here were taken from a Dell Computer manual.

- Interface components are windows titles, button and icon names, menu names and selections, and other options that appear on the monitor screen or display. They are presented in bold.

Example: click **OK**.

- Keycaps are labels that appear on the keys to be pressed simultaneously (unless otherwise indicated) to perform a single function.

Example: <Enter>

NOTES, WARNINGS, TIME, AND TOOLS

Throughout this guide, blocks of text may be accompanied by an icon. The icons will be used as follows:



Note: A Note indicates additional information about .



Warning: A Warning indicates a potentially hazardous situation which could cause harm to the machine or personal injury.



Time: the clock is accompanied by a block of text which will explain how long it will take to complete a task..



Tools: A hammer indicates the tools required to complete a task.



Important Safety Precautions!

With respect to the European Community Edict 89/392/EWG (Machines) the following information is given as a supplement to the manual:

0. Application

The BUNGARD CCD machines are manufactured by Bungard Elektronik, Rilke Straße 1, 51570 Windeck, Germany and are brought put into circulation in European countries via local distributors or agents.

The machines are designed for drilling and milling (routing) printed circuit boards and for milling (routing) and engraving aluminum plates. A special application is dosing of pastes or liquids onto panel-shape work pieces. Any other type of application is permissive only with our consent.

The machines are not determined to be connected or combined with other machines. However, they need to be connected to and be controlled by a compatible computer. The driver software is supplied with the machines and is relevant for security.

The machines may only be operated by qualified operators. Children and animals must be kept off!

1. Emergency Stop Facilities

The ESCape button on the computer stops the spindle and the stepper motors. This is indicated by a red and yellow message on the screen. The main switch on the rear of the control unit cuts the machine from mains supply.

2. Dust

It is necessary that the standard vacuum cleaner or, if special operating conditions occur, a better one, are always being used during operation on printed circuit boards. The vacuum cleaner must be OFF during and after use of alcohol or other inflammable or explosive liquids

(i. e. used for cooling when milling aluminium). Special operating conditions may require that an extraction unit with higher rating be installed.

3. Noise

With the HF-spindle running at full speed and the vacuum cleaner on, the noise pressure level at 1m distance is 86 db(A). Operators have to wear ear protection.

4. Protection against contact with moving parts

There is a transparent, pivoting cover mounted to machines with manual tool change to prevent incidental contact to the moving tool. It is not possible to entirely envelop the tool. This cover cannot be mounted to machines with automatic tool change. The following danger remains:



WARNING: Do not touch the tool when the spindle is operational. Prior to any tool change action, wait until the spindle has come to a complete stop. Close the cover entirely after any intervention to the tool.

Under normal operating conditions and with the given work piece materials, there is no special danger from parts being propelled off the machine. However, special operating conditions or heavy work pieces may require that the machine is encapsulated. A protection cover is available as an option.

5. Documentation

Our distributor / reseller in your country is encouraged and in charge of translating the German and / or English manual coming with the machine into your native language.

Files Used In This Guide

There are a number of files designed to accompany this guide to make it easier to learn how to create your own PCBs. Including:

- CAM files used with Eagle PCB to export Gerber files.
- Examples files which allow you to follow along with certain sections of this guide.
- Example tool rack files.
- IsoCam Quick start Tutorial
- Original Manuals for the Bungard CCD, IsoCam, and Eagle PCB.

If you obtained a printed copy of this manual, you can load the files from the included CDROM.

Otherwise, you can download the files from the following websites.

<http://www.brobertson.com> – primary site

<http://www.wam.umd.edu/~blaker/bungard/> - backup site

Machine Setup

This section will help you setup your new milling machine in about twenty minutes. Shortly after you'll be on your way to producing PCBs in-house!

The setup instructions should be followed in the order they are presented in this manual.

Assembly Requirements

Before you begin assembling the device read the safety precautions on the previous page.



It will take about thirty minutes to complete the assembly.

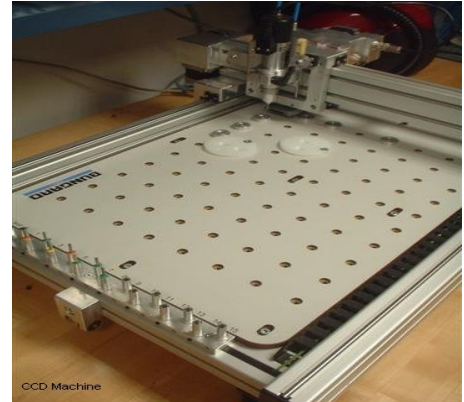


- A table with a flat area of at least 4 feet by 4 feet to set the machine, the control unit and to provide adequate space for the vacuum cleaner.
- The assistance of another person to lift the machine out of the shipping crate.
- A flat head screwdriver to secure the cables
- An adjustable wrench is required if you purchased the automatic tool change model.

Terms Used In Manual

The purpose of this section is to define the terms used throughout the manual. Also see Appendix A which is a glossary of technical terms used in this manual.

Machine – when the manual refers to connections on the machine it referring to the component which is mechanical.



Control Unit – the interface between the computer and the machine.



Switchbox – the black box which the vacuum cleaner plugs into. There is a wire that comes out of the box and plugs into the back of the machine that turns the vacuum cleaner on and off when it is necessary.



MTC – Manual Tool Change

ATC – Automatic Tool Change

Note: The terms listed in this section will be expanded significantly when the operations of the manual are described. The reason for providing it now is just to format how the manual will be laid out.

Connections Overview

The images below show what the machine and control unit will look like after the following sections have been completed. Although it may look like a lot of cables to connect, the setup is very easy and the design of the control unit and machine make it impossible to incorrectly hookup the cables.

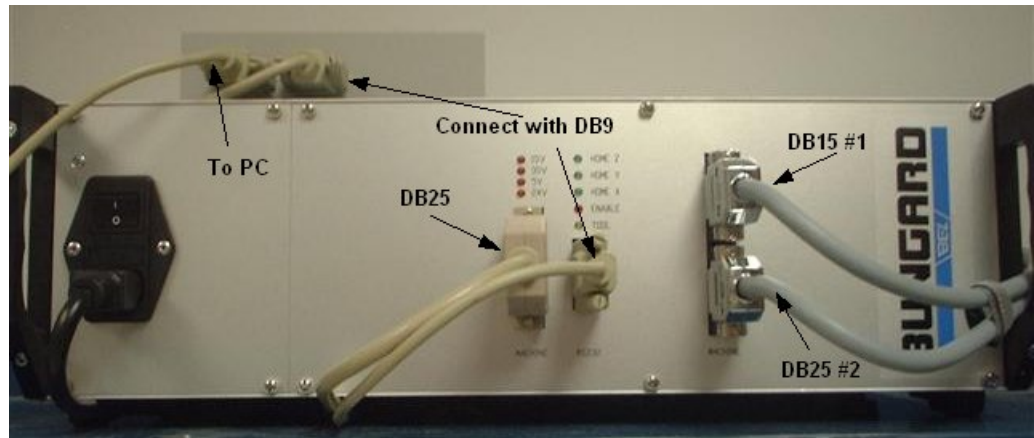


Figure 1: Control Unit Connections

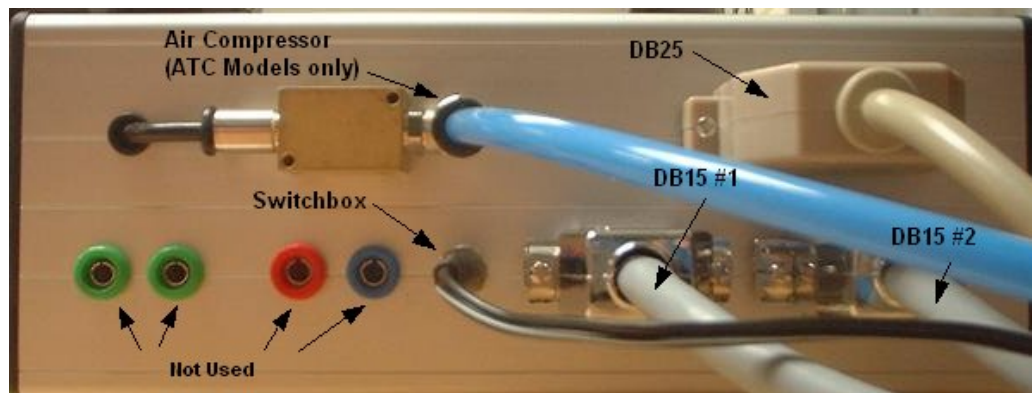


Figure 2: Connections on the back of the CNC machine

Connecting the PC to the Control Unit

If you purchased the RoutePro 2000 upgrade which allows the machine to be operated in a windows environment then skip to the section labeled “For Windows Users”. If necessary refer to the images in the connections overview.

For DOS Users:

The control unit is linked to the computer by use of a standard RS-232 cable. Connect the female end of the supplied RS-232 cable to the computer. The male end connects to the female RS-232 connector on the control unit labeled “RS232”

For Windows Users:

The RoutePro black box is connected between the computer and the control unit. There are two RS232 cables supplied which connect the computer to the black box which connects to the control unit. Connect the first cable between an available 9-pin Serial connector and the female RS232 connector on the left of the black box.. Connect the other cable between the male RS-232 connector (which is on the right) and the control unit’s connector labeled “RS232”.

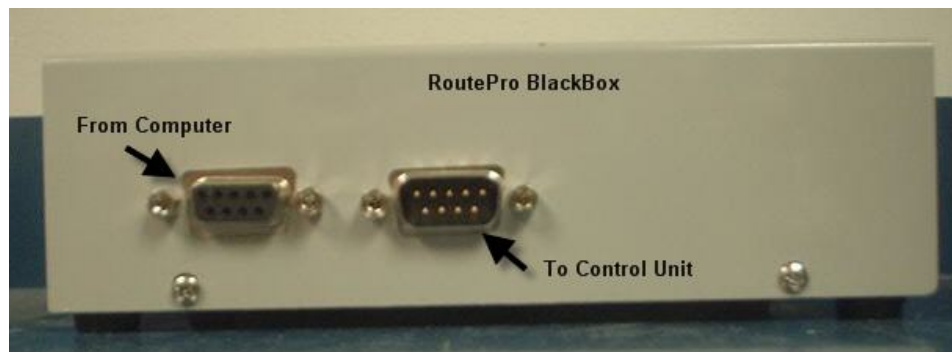


Figure 3: For Windows Users: these are the RS-232 connectors on the RoutePro black box.

Connecting the Control Unit to the Machine

The connection between the control units is done by connecting 3 cables between the control unit and the machine. The cables are a 25 pin Sub-D cable and two 15 pin Sub-D cables. Match the cables with their appropriate connectors. The size and genders of the various connectors and cables have been set so that the cables can't be connected interchangeably. See the diagram below for additional clarification.

NOTE: All the power supplied to the machine is through these 3 cables. Therefore there is not a separate power connection for the machine. The control unit supplies a voltage to the pins to control each of the motors in the machine.

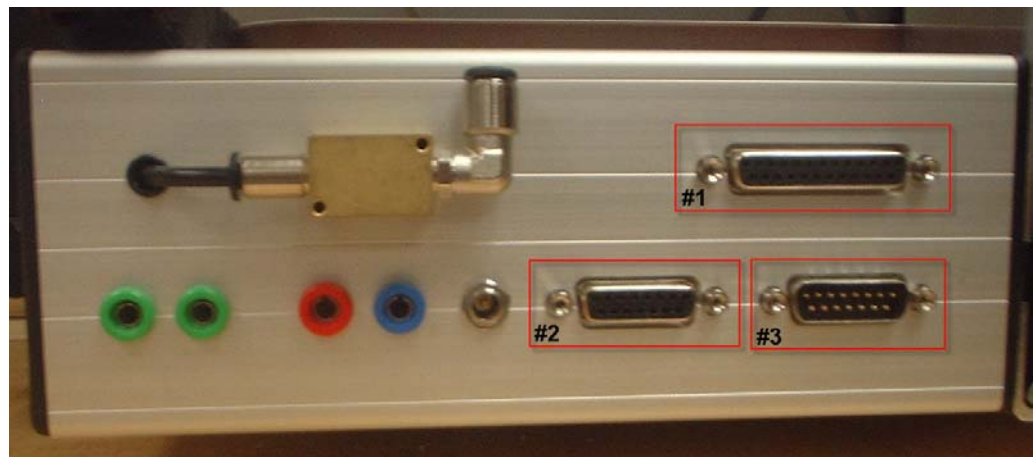


Figure 4: Shows the connections on the back of the machine that need to be connected in this step.

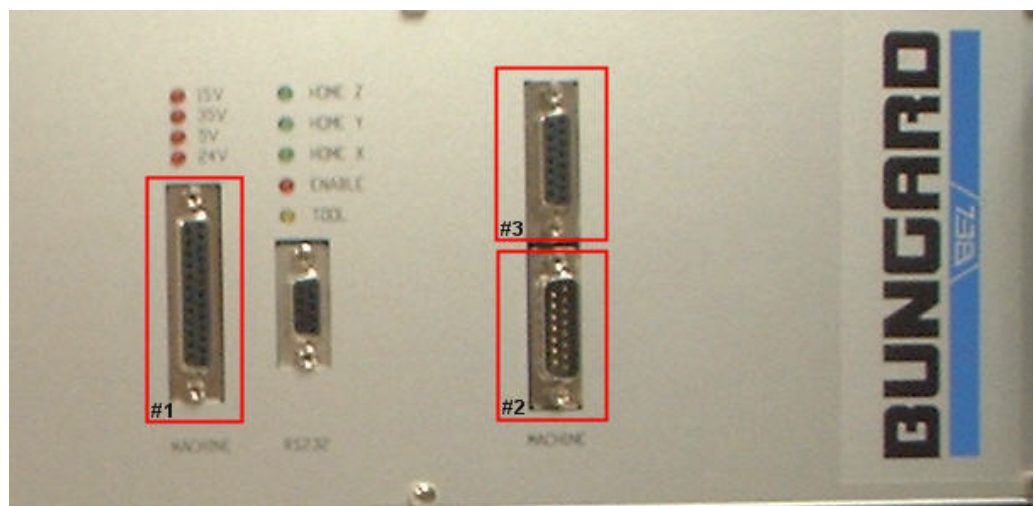


Figure 5: Shows the connections on the control unit that need to be connected in this step.

Automatic Tool Change Connections

This step is only necessary for customers who purchased the Automatic Tool Change model of the machine. If you have the Manual Tool Change (MTC) model then you proceed to the power connections section.

The machine requires compressed air of at least 5 bars to open and close the spindle chuck. A compressor is not included with the machine.

If you purchased an Air compressor with the machine follow the following steps to setup the Air compressor. If you already have setup your air compressor proceed to the section labeled “connecting the air compressor to the machine”

SETTING UP A PURCHASED AIR COMPRESSOR:

1. Set the Air compressor in an upright position
2. Remove the valve cap from the side and replace it with the included air filter.
3. Next, place the adapter valve that is attached to the blue hose, on to the air regulator. This is located near where the power cord enters the compressor and has an air gauge attached to it.
4. Using an adjustable wrench tighten the adapter until its snug.
5. Connect the other end of the blue hose to the machine by pushing it into the connector.
6. To adjust the pressure entering the machine lift the knob on the regulator and adjust until the air pressure gauge on the regulator reads about 5 bar. Turning the regulator clockwise to increase the pressure and counter clockwise to decrease the pressure. Once you’ve finished adjusting the pressure push the knob down until you hear it click.
7. There is an on-off switch located on the grey box where the power cord enters the compressor. To turn the compressor on turn the switch so it’s in the closed position (the straight line).

Note: When you first plug the compressor in, you will hear the motor running as the main tank is pressurized. After the tank pressure reaches a preset level you will hear a quick release of air and the motor will turn off. The motor will turn on once again after air pressure has gone down.

CONNECTING THE AIR COMPRESSOR TO THE MACHINE:

- 1) Connect the long hose's gold, spiral connector to the air compressor. The connector is the standard connector for connecting compressed air outlets to the air Compressor. (See figure on left)
- 2) Connect the free end of the hose to the air valve on the back of the machine. Connection is made by pushing the black plastic piece in slightly and then sliding the hose in. Release the black plastic piece and GENTLY pull on the hose to verify that the hose is held firmly in the valve. (See figure on right)
- 3) Finally, use the short piece of air hose which came with the spindle and connect it to the fitting on top of the Y-Axis slider. Connection to the Y-Axis is the same as in the previous step. On the spindle the hose just slides the hollow piece of metal.



Figure 7: Shows the hose connection on the air compressor

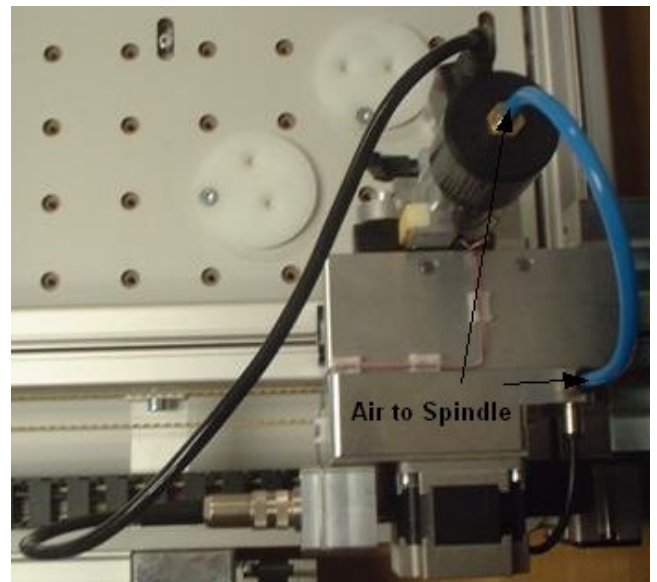


Figure 6: Shows the hose connection between the machine and

Mounting the Spindle

1. Release the spindle holder clamp by a 90 degree left turn of the lever on its right side. Then turn the clamp counterclockwise to loosen.
2. Drop the spindle into the clamp and push it downwards so that the distance flange lies firmly on the holders top.
3. Connect the cable to the connector housing on top of the Z-Axis block. (See figure below-right)
4. Close the spindle clamp by turning it 90 degrees to the opposite direction as before. Then tighten the clamp by turning it in a clockwise direction.

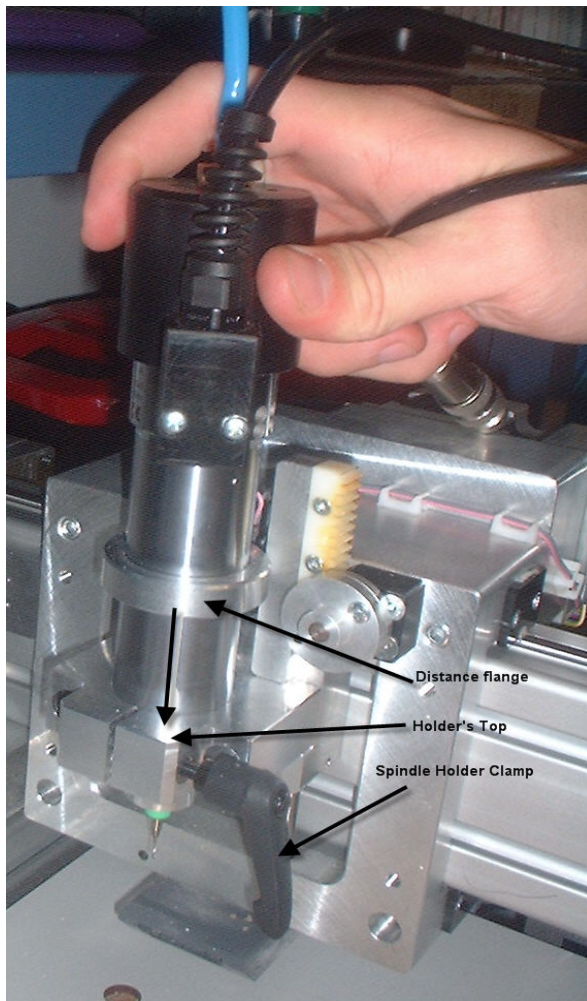


Figure 8: Figure for steps 1 and 2

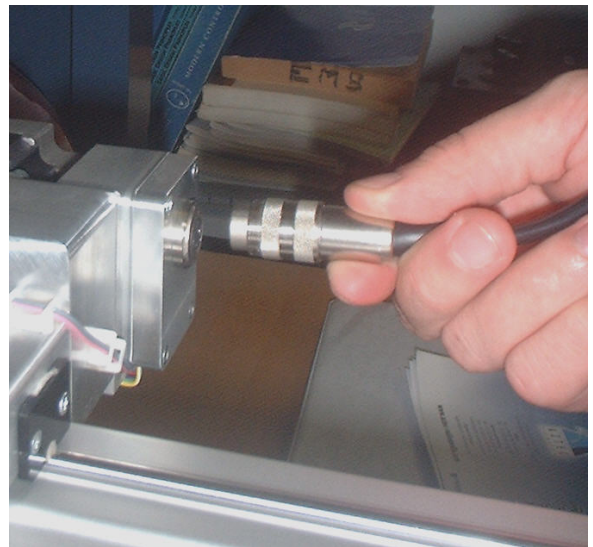


Figure 9: Spindle cable connections in step

Note: the assembly procedure doesn't have sections for setting up the vacuum cleaner and the air compressor setup may need to be more detailed

Assembly Complete!

Congratulations! You've successfully assembled the Bungard CCD. You're one step closer to being able to produce prototype PCBs in-house.

In the next section you will begin learning how to setup the software that you'll use to convert your PCB layout files into HPGL files which can then be processed by the Bungard CCD.

Recommendations

This Chapter provides information on supplies and tools which are helpful for isolation mill prototyping.

DRILL BITS

Even if you purchased a drill bit starter kit from Bungard, you will still need to acquire some additional bits. The starter kit is missing some drill bits which are very commonly used on PCB layouts. If you don't have all the drill bit sizes you need them then you will either have to change your design or substitute another drill bit size. Both are workarounds which will create more work to do and can lead to problems with your PCB. This guide assumes you will have acquired additional drill bits so these workarounds aren't necessary.

When purchasing drill bits make sure that you purchase drill bits with the proper shank length which is 21mm. One of the best places to purchase drill bits is at drillbitcity.com. At the time of writing, on average you can get a 5 pack of the drill bits for \$9 new or \$4.50 resharpened. Another good site for PCB materials is thinktink.com.

COPPER CLAD & BACKING BOARD'S

If you need raw copper clad PCB boards (the materials that you make PCBs out of) and/or backing boards, Ramadan Assani, an associate of Bungard, sells PCB materials at very reasonable prices. Contact him via email at advansys@videotron.ca to get a catalog and pricelist.

MINI-VACUUM

The dust extraction system doesn't take care of all the dust and shavings that are produced in the process of making a PCB. So if you don't like having a messy work area then you should purchase one of these as soon as possible. One such vacuum is the turbo mini-vac from Home Depot. It cost \$20 dollars and comes with a bag of attachments. It's a little loud though.

Configuring the Software

This chapter explains how to setup the software used to prepare the PCB design for isolation milling.

The configuration process will only have to be performed the first time you make a PCB. Setting up the software will reduce the number of actions which must be performed to prepare the data for isolation milling. The configuration process outlined in this section was prepared to minimize the number of mistakes that could be made while converting the files. Consequently, you will be able to prepare the files faster each time you need to make a board.

This shows step by step instructions for how to setup Eagle PCB, IsoCam and Route Pro.

Even if you are not using CadSoft's Eagle PCB Design software you should still read step 1 "creating a drill rack file". This step provides background information on the different types of drill files that you will encounter in the next section "Preparing Isolation Milling Files"..

Configuration Requirements



It will take about twenty minutes to an hour to setup the software.

SOFTWARE REQUIREMENTS:

- Eagle PCB CAD software installed or another PCB CAD software package which documentation on how to export RS-274X compatible gerber files.
- IsoCam software package installed on a compatible PC. See the IsoCam manual for installation instructions.
- RoutePro 2000 software package installed on a compatible PC. See the RoutePro 2000 manual for installation instructions.

Configuring Eagle PCB Layout

STEP 1: CREATING A DRILL RACK FILE

When exporting the drill files to the standard Excellon format you are limited to the number of drill sizes you can make by the number of drill bits you have. You can tell the CAD software the drill bits that we have available by making a drill rack file. A drill rack file is a text file which has two columns. A column for the tool number and a column for the diameter of the drill bit. When the software is exporting the drill data, it checks to see that an appropriate drill bit is available for each hole in the design. A certain tolerance can be set so that if you have a drill bit that is slightly smaller or larger than the actual drill size necessary, the drill bit may be substituted without effecting the design. For example, if your design calls for a .61mm drill bit and has a minus tolerance of 2.5% and a plus tolerance of 10%, then drill bits in the range .59mm - .67mm will be acceptable substitutions.

Math from example:

$$0.61 - (0.61 * .025) = 0.59 = \text{min drill size}$$

$$0.61 + (0.61 * 0.10) = 0.671 = \text{max drill size}$$

Note that the 0.61mm bit is a result of converting a 0.024" drill size to metric units. The 0.024" drill size is a very common drill size.

If using MTC, the drill rack file could list every drill bit that you have. Users who have ATC might want to limit the number of drills to 15 so that they never have to change the tools in their ATC sockets. With fifteen tools you can cover nearly every possible drill size you will encounter when making PCB's. An example ATC drill rack is shown below which works for 95% of standard PCB designs.

The exported drill file (.drl file) is composed of a line specifying the tool number followed by a list of x & y coordinates which tell the machine where to drill the hole. The drill file only contains drill numbers not the drill diameter. The purpose of the drill rack is to map the drill tool numbers to a drill diameter.

Procedure:

1. Open Notepad by left clicking start->programs->accessories->notepad.
2. For each drill bit you want to include in your drill rack, type a tool number followed by two spaces and then the diameter of the drill bit.
3. White space (additional lines after your last entry) should not cause any problems but just to be safe leave the cursor at the end of the line on your last entry.
4. Save the file as 'myDrillRack.drl' in a convenient location. During configuration you will use it frequently.

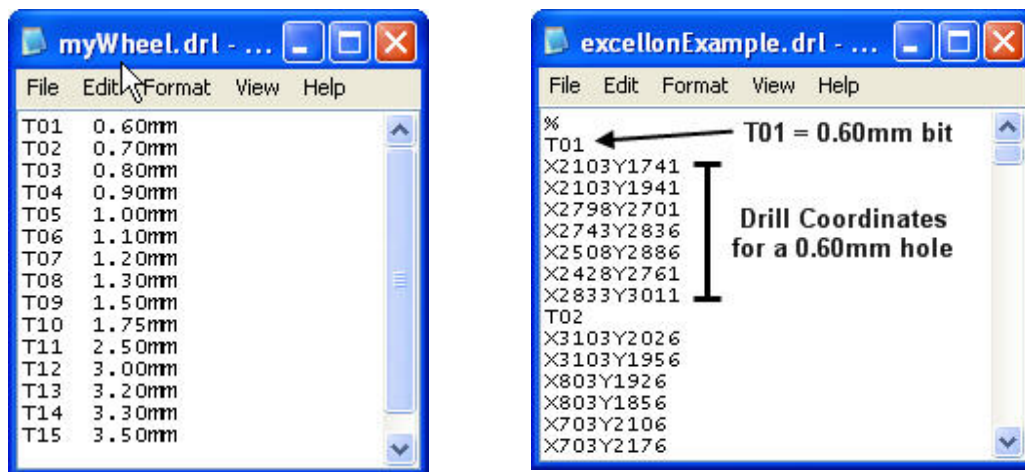


Figure: On the left is an example of a drill rack file which could be used for ATC. On the right is the Excellon drilling file that was generated as an example to explain the correlation between the rack and Excellon drill files. Notice: the diameter of each tool isn't included in the Excellon file. It's implied that T01 will be a .6mm drill bit.

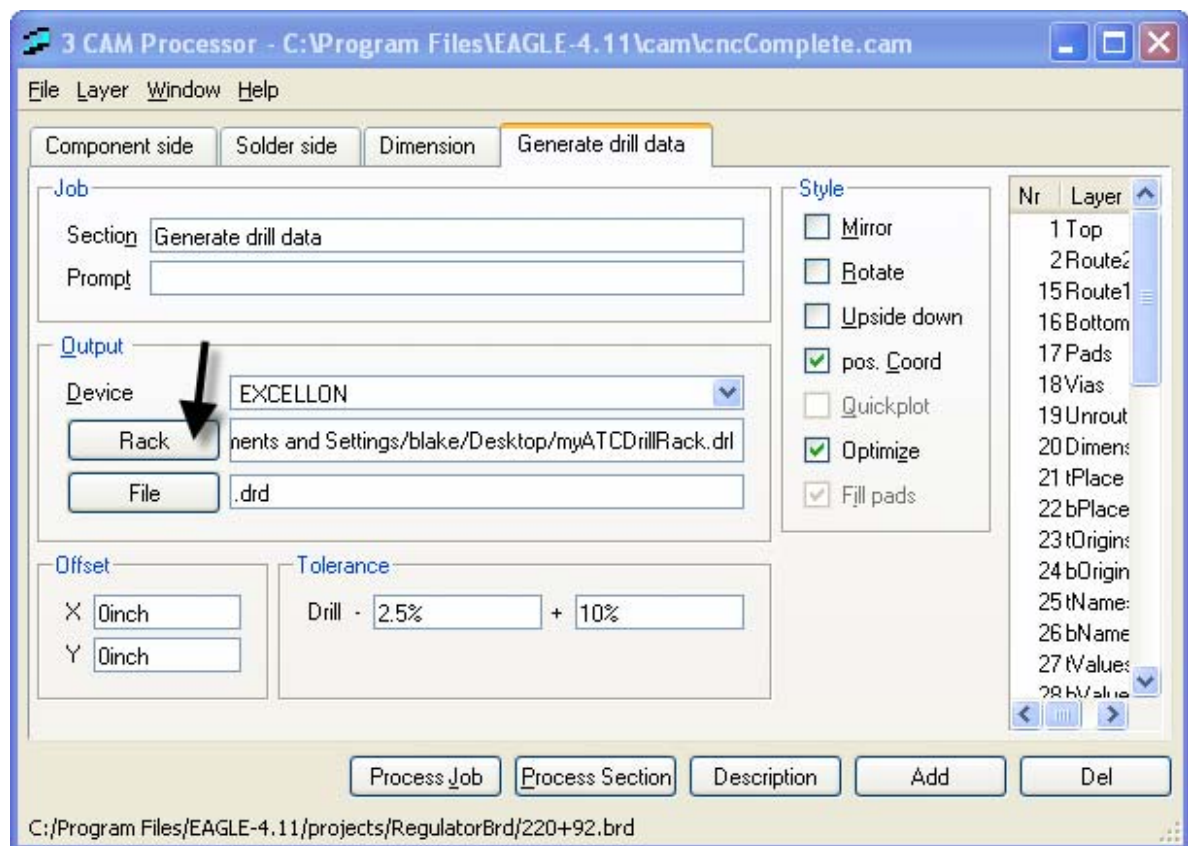
In the following sections you will configure IsoCam and RoutePro to use the same drill rack. Since the drill diameter information isn't stored in the Excellon file it is imperative that the drill setups be consistent.



There are some Excellon drilling formats that also include the drill diameter (in addition to everything else normally included) which makes it unnecessary to have a separate drill rack. Unfortunately, those formats aren't as standard as the convention using two separate files.

STEP 2: SETUP CAM FILE

1. Copy the 'cncComplete.cam' file to the 'CAM' directory which Eagle is installed in (typically c:\Program Files\Eagle v4.x\)
2. Save the file to the CAM directory in the directory where eagle is installed.
3. Open the CAM Processor and load the job 'cncComplete.cam'.
4. Choose the tab labeled **Generate Drill Data**
5. Click on the button labeled “Rack” and select the drill rack file you created in the previous step.



6. Save the modified CAM file by going to **File->Save Job**. When prompted save the Job as something like “myCustomGerber.cam.”

Note: The cncComplete.cam is a job file that will generate the top and bottom layers, the board dimension layer and the Excellon drilling information. It's preconfigured properly for use with IsoCam and it exports all the necessary information for CNC milling in one step which saves several steps.

Configuring IsoCam

1. Open IsoCam
2. Click **Edit->Edit Gerber/drill startup info**
3. Make sure that all the settings match the ones that appear in the figure below.

The screenshot shows a dialog box titled "Startup info gerber/apertures/drill/drill tools". It contains two main sections: "Gerber parameters" and "Drill parameters".

Gerber parameters:

- Number format:
 - Digits 1: 2
 - Digits 2: 3
 - Units: inch
 - Zero suppression: Leading
 - Number format: Integer
- Apertures:
 - Skip lines: 9
 - Units: mils
 - DCode column: 1
 - Object type column: Round
 - X size column: 2
 - Y size column: Not used

Drill parameters:

- Number format:
 - Digits 1: 2
 - Digits 2: 3
 - Units: inch
 - Zero suppression: Leading
 - Number format: Integer
- Drill tools:
 - Skip lines: 0
 - Maximum line nr: 23
 - Units: mm
 - Tool column: 1
 - Diameter column: 2

At the bottom of each section are buttons: "Load current gerber values" and "Load current drill values". On the right side of the dialog are buttons: "OK", "Cancel", and "Help".

Figure 10: Parameters for retrieving data from the drill and aperture files for compatibility with Eagle PCB Software.

Note: There is currently a bug in the IsoCam software which causes it to not save the drill tools automatically. The workaround is to load the drill rack file each time you load IsoCam which is explained fully in the procedure for using IsoCam. Every effort has been made to simplify the procedure in this guide. Therefore, as soon as the bug has been corrected we will update this section of the guide. If you would like to be notified when the guide is updated please send an email to blaker@umd.edu.

Information on the parameters which may be applicable to users who aren't using Eagle

Skip Lines: Some PCB software packages put a header on certain files. The Skip lines parameter is to skip header information that some CAD software exports in addition to the drill tool info. In the sample file that's part of the IsoCam tutorial there are several lines given

Tool Column: this is the column where the tool number is located

Diameter Column: this is the column where the drill's diameter is located.

Preparing Isolation Milling Files

In this section you will learn how to use IsoCam to create the milling data which then can be loaded into RoutePro 2000 to mill your PCB.

From PCB CAD software package a standard file format called Gerber can be exported. This common format allows you to use any PCB software package you want as long as you can export Gerber files.

IsoCam takes the Gerber files and creates the milling data by isolating the copper traces so they're not shorted. After all the milling data has been generated, it creates HPGL formatted files which is the format used by the Bungard Machine (and photo plotters).

Files generated by IsoCam are ready to be processed by Bungard's proprietary software RoutePro.

Section Requirements



It will take about thirty minutes to do this the first time and about 5 minutes once you become proficient working with the software.

- Completion of the previous section “Configuring Software.”

Board Layout Considerations for Isolation Milled PCB

THROUGH-HOLE-PLATING CONSIDERATIONS

Typically, boards created with a CNC milling machine do not have the luxury of through hole plating. When through-hole-plating is available through-hole components can be soldered on the bottom side (solderside) and still have traces routed on the top side (component side) because the top and bottom layers are electrically connected. Unless other equipment has been purchased, you will need to make sure that routing isn't done on the top layer.

In Eagle we do this using the tRestrict layer. The tRestrict layer keeps the autorouter from placing tracing on the topside of the board where you have specified. This is done by selecting the rectangle command in Eagle and then setting the layer to tRestrict. Then create rectangles over any component which cannot be soldered on the top side. Such as pcb connectors or IC's if a socket is going to be used. Hand picking which components can't be soldered is good because it makes the number of vias necessary to route the board smaller. However, it leads to potential problems with board assembly if you forget that a particular component needs to be soldered on the top side.

Alternatively there is a ULP (user language program) which can be downloaded off CadSoft's website (<http://www.cadsoft.de>) named restrict.ulp which will automatically added tRestrict rectangles over all the components on the topside.

DESIGN RULE CHECK

Boards created using isolation milling are harder to solder then boards created with an etching process since the copper that is left behind can lead to shorts.

To minimize the problems caused by this a special design rule check file has been created and can be downloaded at <http://www.blakerobertson.com/guides/pcb/bungard>. If you design passes the design rule checks then it will be much easier to solder and trace widths are safe for the accuracy of the machine.

Exporting Gerber & Drill Files from PCB CAD Software

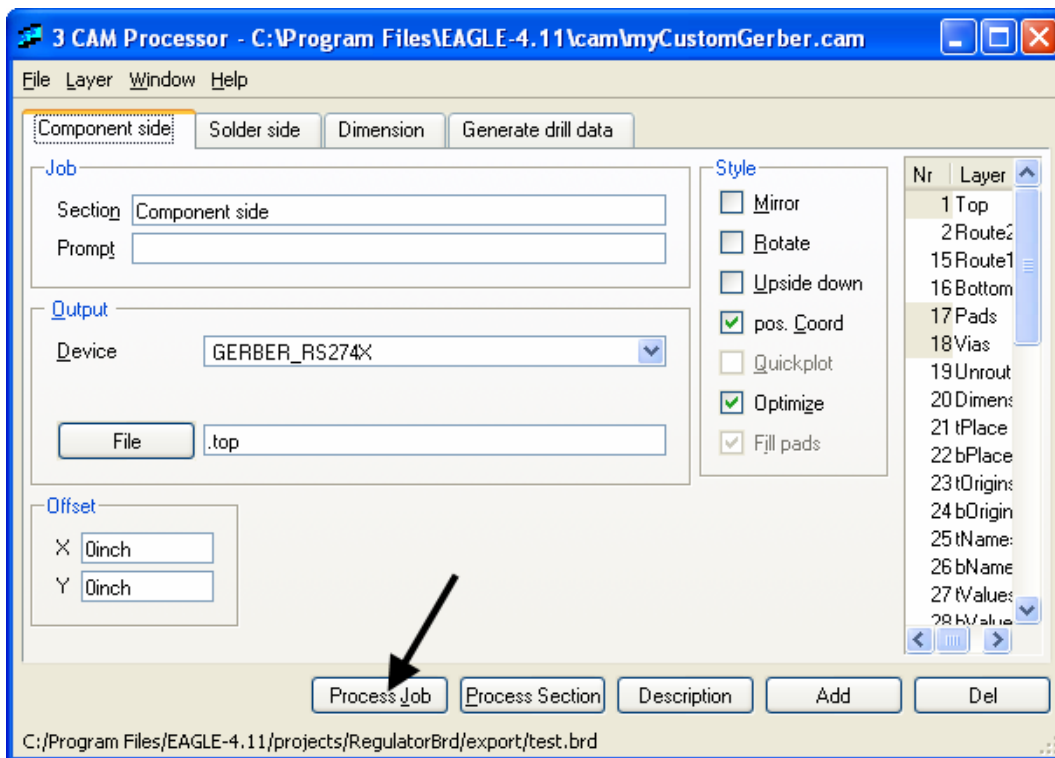
After completing the PCB layout in the PCB CAD software, you will have to export your design to the Gerber format. The Gerber format is the standard format used in the industry for PCB designs. These are necessary since each PCB Design software package uses proprietary formats for the files that are saved onto your computer. It's analogous to documents created in Microsoft Word not being compatible with other word processing software.

After exporting your design to the Gerber format, you will have a number of files which represent the different layers of a PCB design. There are separate layers for the copper traces and pads, the solder masks, the silk-screening, the drills, etc.

Each PCB software package has a different process for exporting Gerber files. If you are using a different PCB design package other than Eagle you will have to consult the documentation for your particular software.

Procedure:

1. Open the job you created in the section “Configuring Eagle PCB”. If you haven’t already done so, go back and complete this section now.
2. Click **Process Job**



Generating the Milling Data using IsoCam

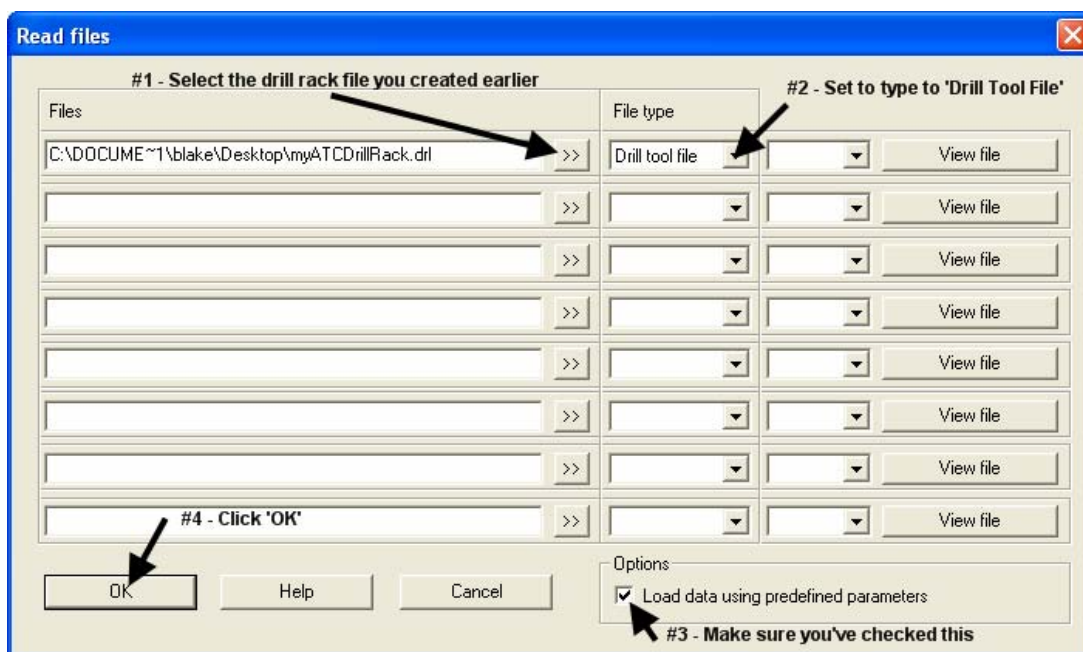
If you follow the steps in this section you will be able to quickly generate your milling files. This also gives you a global overview of the most important features of IsoCam.

All the files that are used in this section are provided (see section on obtaining files). They are located in the \Examples\IsoCamGuide\ folder. If you use the Gerber files in that directory, you can confirm that you are doing everything properly by looking at the screen shots.

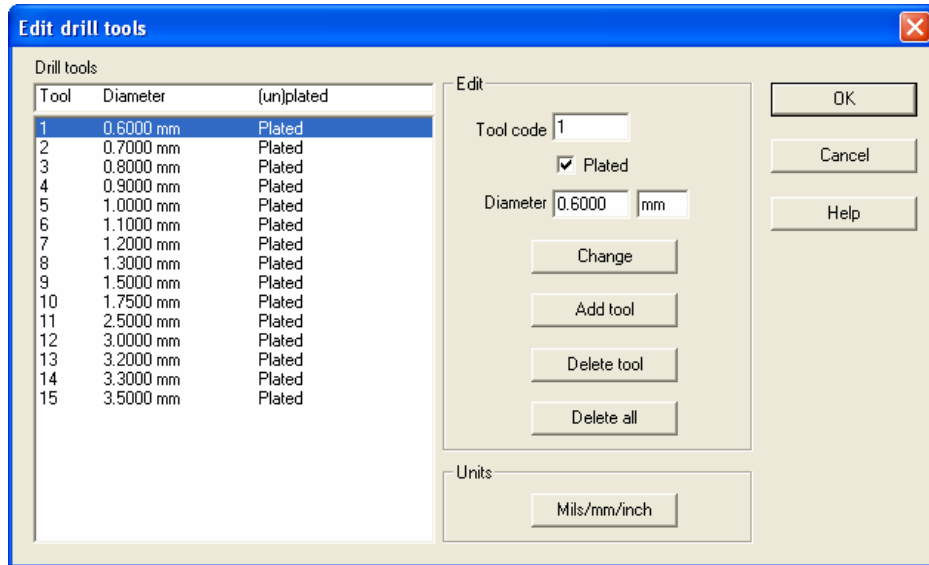
NOTE: Your files may have a slightly different path (the place where the files are stored on your computer) and filename then the ones used in the screenshots. When comparing certain things, look at the file extension if appropriate.

Procedure:

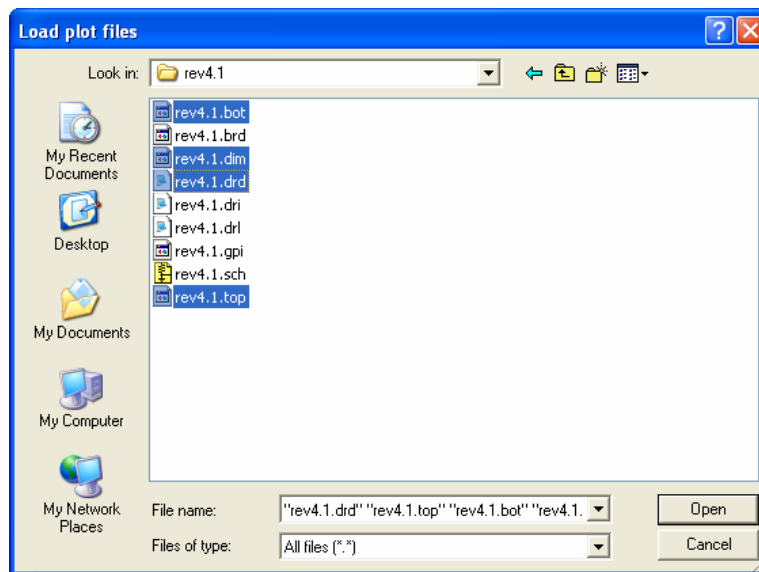
1. Load drill rack file:
 - a. Select **File** -> **load files** then locate your drill rack file and press **OK**.
 - b. A window like the one below will appear. Set the file type to drill tool file. Then, ensure the **load data using predefined parameters** box is checked. Finally, click **OK** and your drill rack files will load properly.



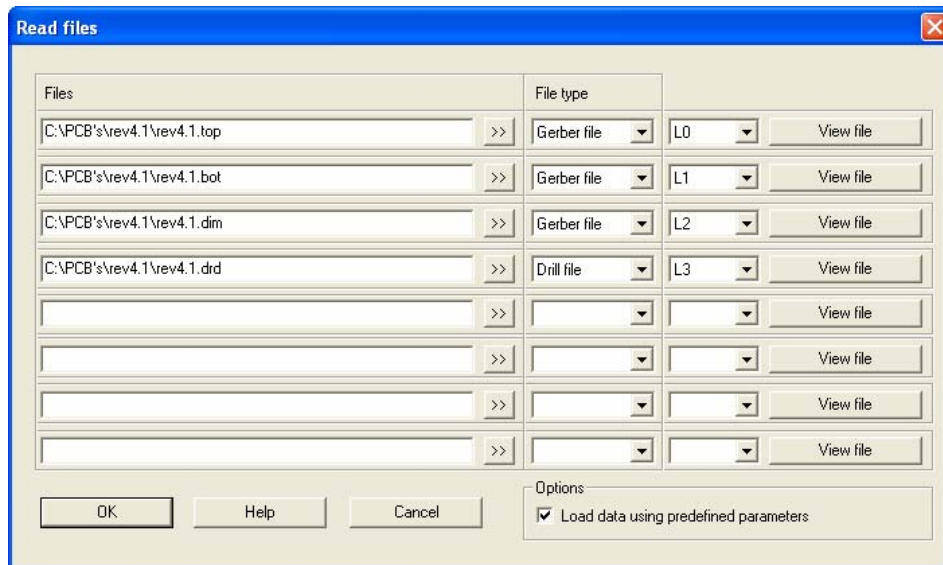
2. To verify the drills loaded properly:
 - a. Select **Edit** -> **edit global drill tools**
 - b. A window like the one below will appear. If the drill rack loaded properly each drill bit that was in your rack file will appear in the pane on the left. Press **OK** to continue.



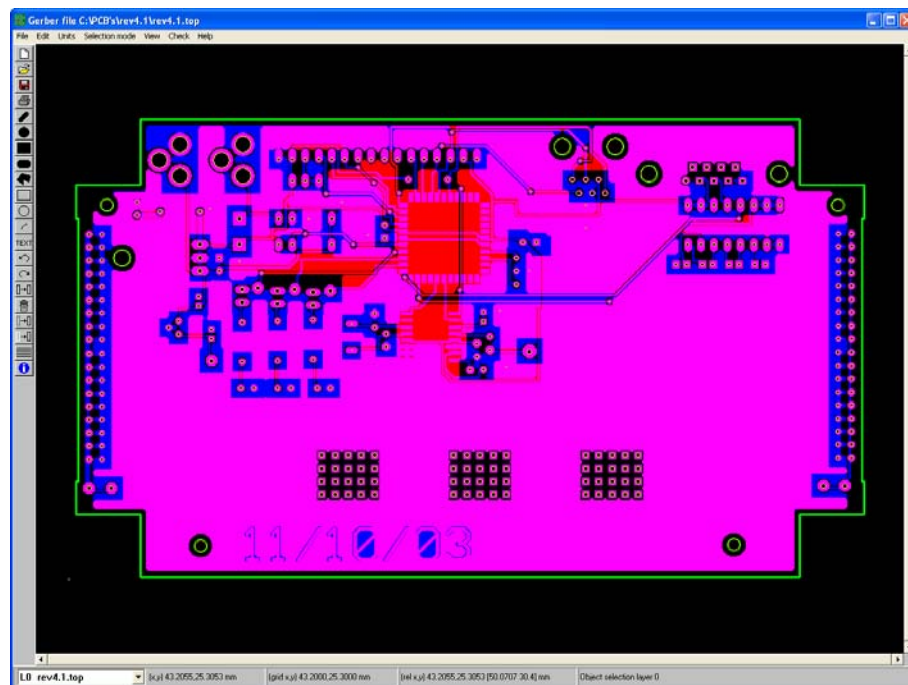
3. Load the Gerber/Excellon files with extensions: .top, .bot, .drd, and .dim.
 - a. When the **Load plot files** window like the one below appears, press and hold <CTRL> and select the files with extensions .top, .bot, .drd, and .dim.



- b. When the **Read files** window appears make sure the file types match their respective extension like the one in the figure below. Make sure the **Load data using predefined parameters box** is checked. Then click **OK**.



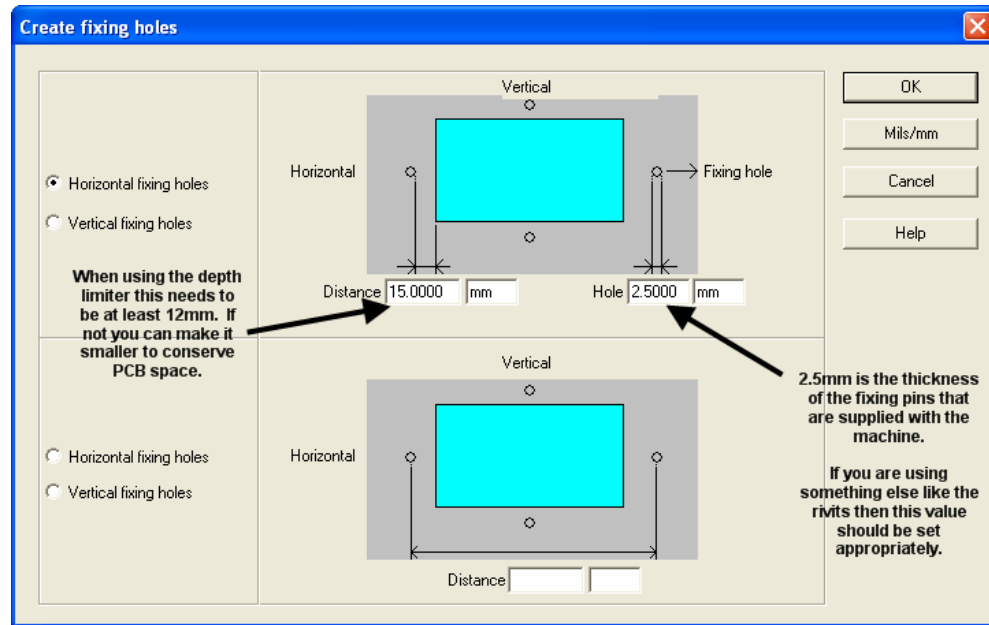
c. After you finish loading the files your screen should look like this.



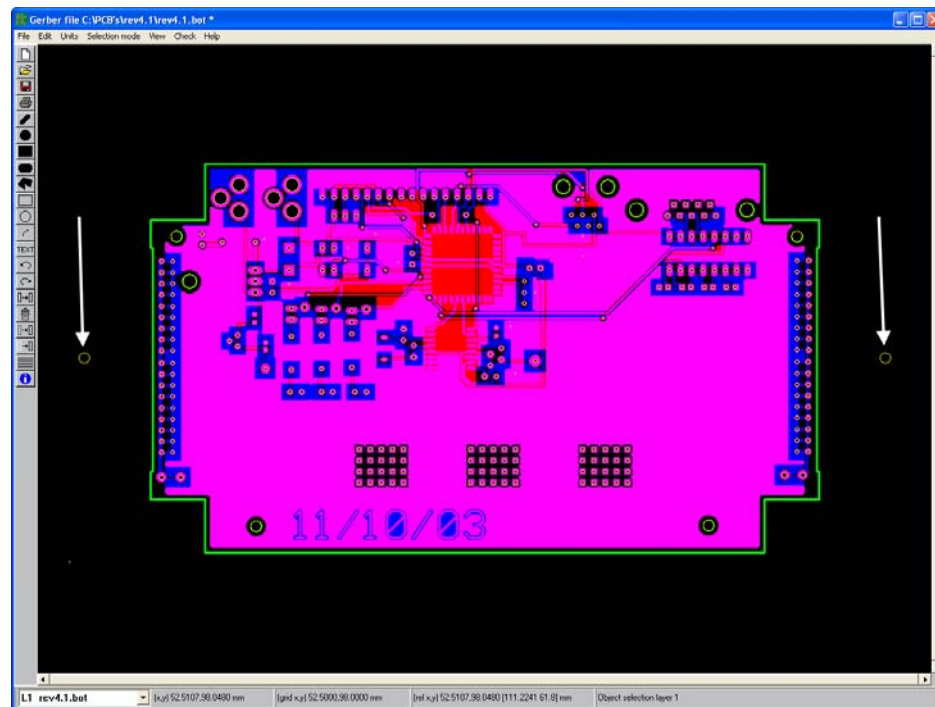
4. Add fixing holes

a. Select **Edit** -> **add fixing holes**

- b. A window will appear like the one below. See the image for information on how to set the distance and hole size. For more information click the help button. When finished press **OK**.



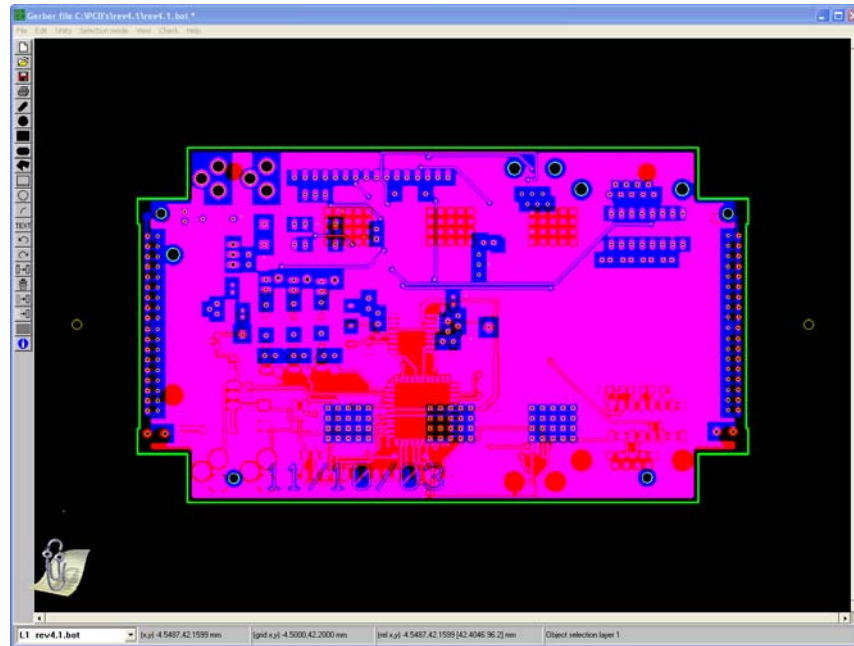
- c. You should then see fixing holes appear on your design. (See screenshot below)



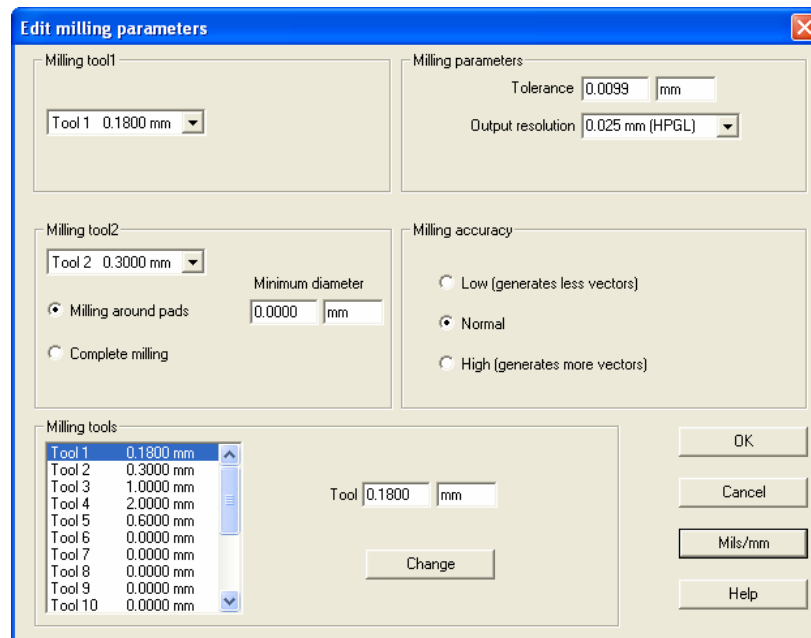
5. Mirror the bottom layer

- a. Change the layer to the bottom layer (the one with the .bot extension).

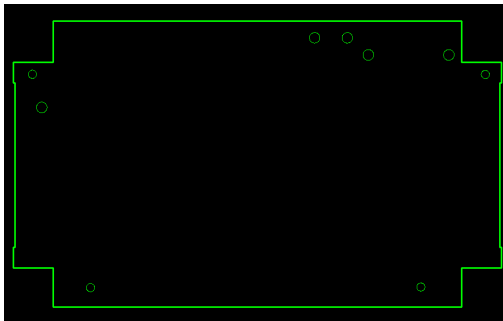
- b. Right click some where on your design and choose **mirror over fixing holes**.
- c. Once you've done this you can see the bottom layer has been mirrored. Compare the image above and the image below.



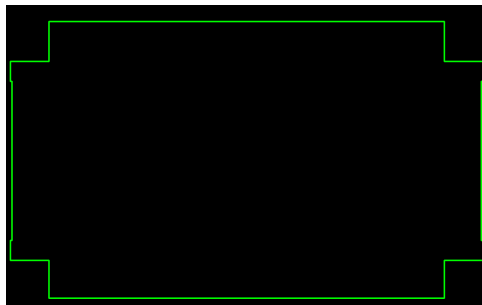
6. Create Milling Data for the bottom layer.
 - a. Select **File -> create milling data**
 - b. When the **edit milling parameters** window appears set the options so they appear as they do in the window below.



- c. After the Milling Data is created it will ask you if you want to check the milled data.
 - d. Then save the milling data by selecting **File -> save milling data**. When it asks you for a file name, name it 'bot.plt'. (a .plt file is the type of hpgl file that the Bungard machine uses) If another screen appears that says "Save Gerber" then you accidentally selected **File->save as** instead of **File->save milling data**. Close the window that appeared and save the milling data.
7. Create Milling Data for top layer.
- a. Select the top layer.
 - b. Follow the same procedure as you did for creating the bottom milling data except save it as top.plt since it's the top layer.
8. Create the board cutout.
- a. Select the Dimension layer (the one with the .dim extension)
 - b. Hide all the layers except for the dimension layer.



- c. If any holes show up inside the PCB, delete them by selecting them and pressing the delete key.



- d. Select the board outline.
- e. Right click anywhere on the layout and select **create board cutout**.

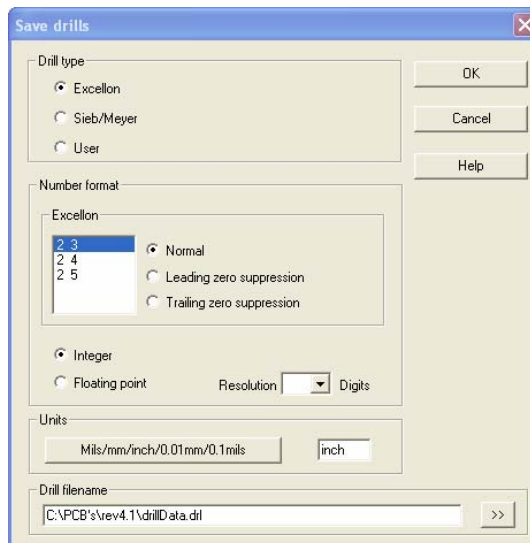
- f. Select the board cutout options you want. Choose a suitable routing bit for the cutout and enter its diameter into the board thickness line.
- g. Once you click okay the board cutout milling data will be created and the screen will look like the one below.



- h. Then save the milling data (File -> 'save milling data') and name it dim.plt.

9. Save the drilling data

- a. Select drill data layer (the one with extension .drl).
- b. Select **File->Save As**. Type in a file name such as 'drillData.drl.' (IsoCam doesn't append an extension so include one in the file name).
- c. Select the drill format as shown in the image below. Specifically, ensure that the number format is 2,3. Then click **OK**.



10. Save the fixing hole data

- a. Select the fixing layer 'fixing.drl'.

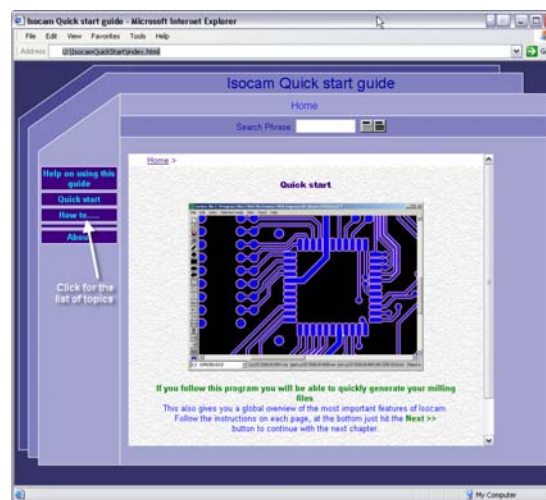
- b. Follow the same procedure as you did for creating the drill information except save it as 'fixing.plt' since it's the fixing layer.

Advanced Features Using IsoCam

This section explains some of the features of IsoCam that can be utilized after becoming comfortable with the basic milling process.

The advanced features in IsoCam allow you to do a lot more with your design. A very good quick start tutorial already exists that explains the advanced features IsoCam offers. Therefore, these advanced features will not be discussed in this guide. The quick start tutorial is included on the CDROM accompanying this manual or can be downloaded (see section “Obtaining Files Used in this Manual”).

1. Open the “IsocamQuickstart” folder and then double click on the index.html file.
2. Once a window appears like the one below, click on the “How to...” link



3. This page lists all the topics included in the quick start guide. Click on the feature you would like to learn more about.

PRINCIPAL TOPICS INCLUDED IN THE TUTORIAL:

1. “Create a rubout” - Allows you to remove all the copper from an entire area.
2. “Convert objects into milling data” - explains how to convert the text from your PCB design to milling data so it’s treated as text only and not isolated. This feature makes your text much easier to read and allows you to use a smaller font for the text. Additionally, this makes routing much faster.
3. “Create milling objects” - allows you to create lines, shapes, or text which will be milled.
4. “Create board cuts” - includes additional information on board cuts such as segmentation cutting and internal PCB cutouts.
5. “Manipulating objects” - shows how you can manipulate a selected object.

Milling a Printed Circuit Board

This chapter will explain the process of milling a printed circuit board using the Bungard CCD.

This section describes basic drilling and isolation routing procedures. See the next section for more advance milling options such as routing depth limiting and creating multiple circuit boards from one blank board.

Milling Requirements



It will take about thirty minutes to complete the milling



- Drilling and routing files created using the procedure outlined in the previous section.
- 1 blank circuit board.
- 1 base board.
- Sufficient drill and routing bits to meet your design specifications.
- Adhesive tape
- 4mm hex wrench
- 2 fixing pins
- Metric ruler

Securing Base Board and Blank Circuit Board to the Milling Table

1. Open RoutePro 2000 and click the “To Y Max” button. This will move the spindle to the farthest Y position to leave an open workspace on the milling table.



CAUTION!!! Stand clear of the milling table after clicking “Y to Max”. Moving parts may cause personal injury or damage to the machine if they are touched.

2. Place the base board (included) onto the milling table and secure it by pressing the plastic and metal disks against sides of backing board and tightening the screws in eccentric mounting holes. See Figure 2.
3. Place the blank circuit board onto the backing board at least 1/2 inch from X and Y table axis.
4. Secure the circuit board to the backing board using adhesive tape.
5. In the RoutePro window, enter the thickness of the base board and circuit board into their corresponding locations. See Figure 1. These settings must be accurate and must remain the same throughout the milling process. The thickness of each may be verified with a metric ruler or caliper.

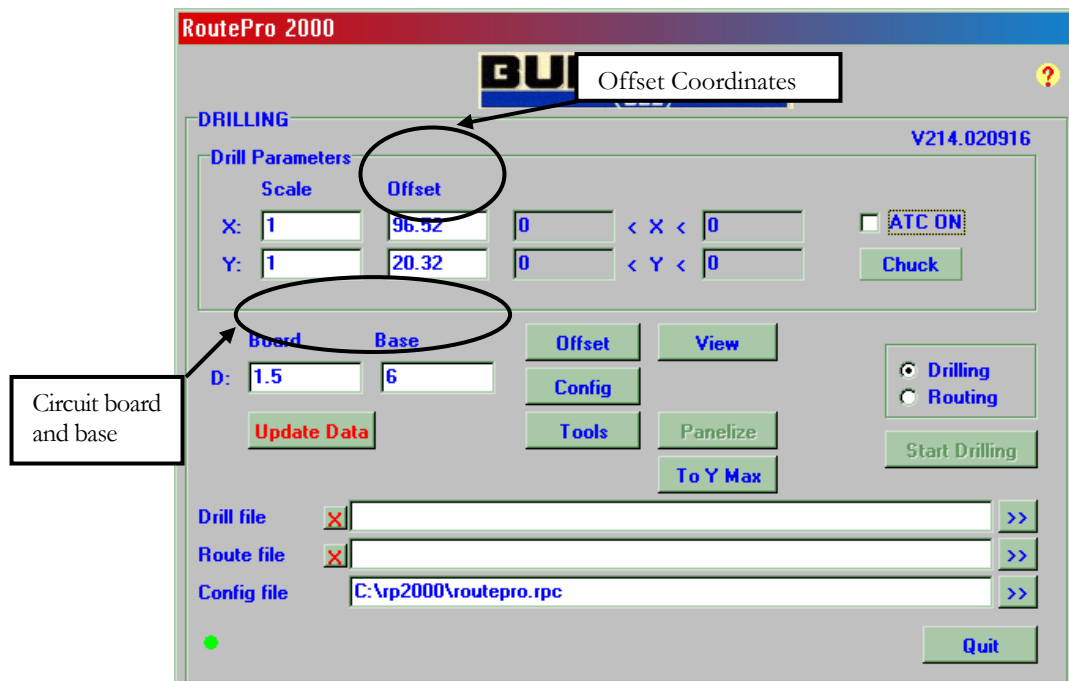


Figure 1 – Main drilling window.

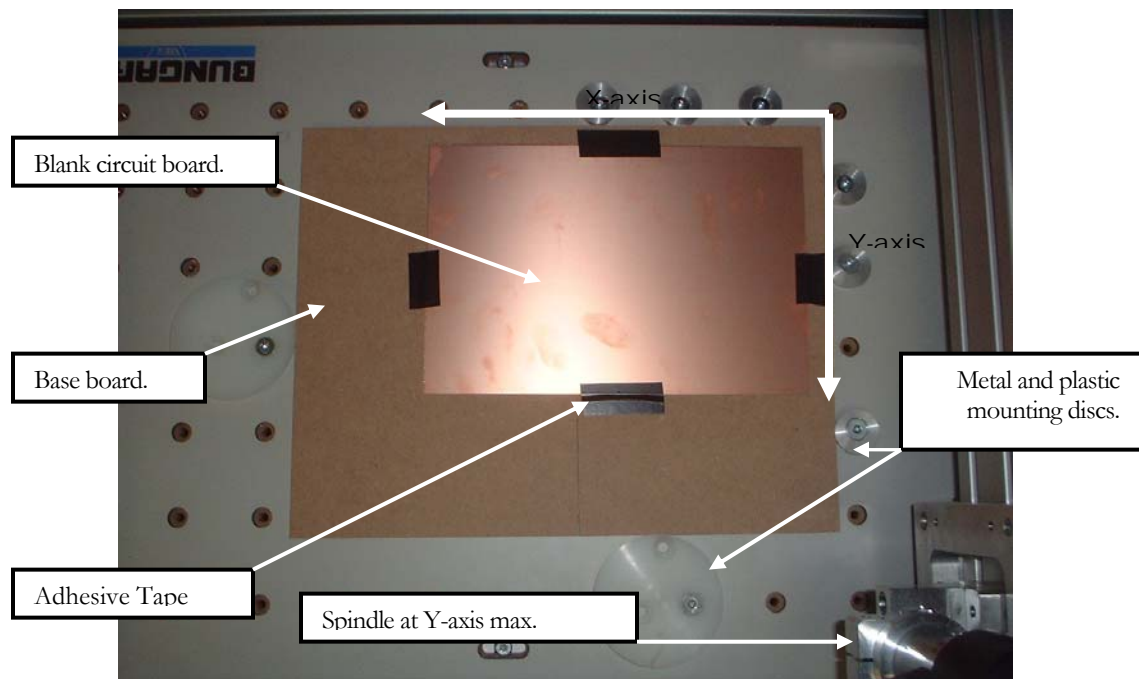


Figure 2 – Blank circuit board secured to base board.

Defining a Milling Offset Position

It is necessary to define an offset position for the machine so that it will start milling at the correct location on the circuit board. It is very important to not change the offset once it has been set until all drilling and routing is complete for a circuit board.

1. Click the “Offset” button in the main window of RoutePro. This will open a new window and cause the spindle to move to its home position ($X=Y=0$)



Warning: Stand clear of the milling table after clicking **Offset**. Moving parts may cause person injury or damage to the machine if they are touched.

2. Using the arrow buttons, move the spindle in the X-Y directions until the spindle is directly above the point on the circuit board where you wish to begin milling. This will mark the corner of the board nearest to the table origin ($X=Y=0$). See Figure 3.
3. To verify the correct position, you may lower the spindle until it is just above the board using the button with the finger pointing down icon. See Figure 3

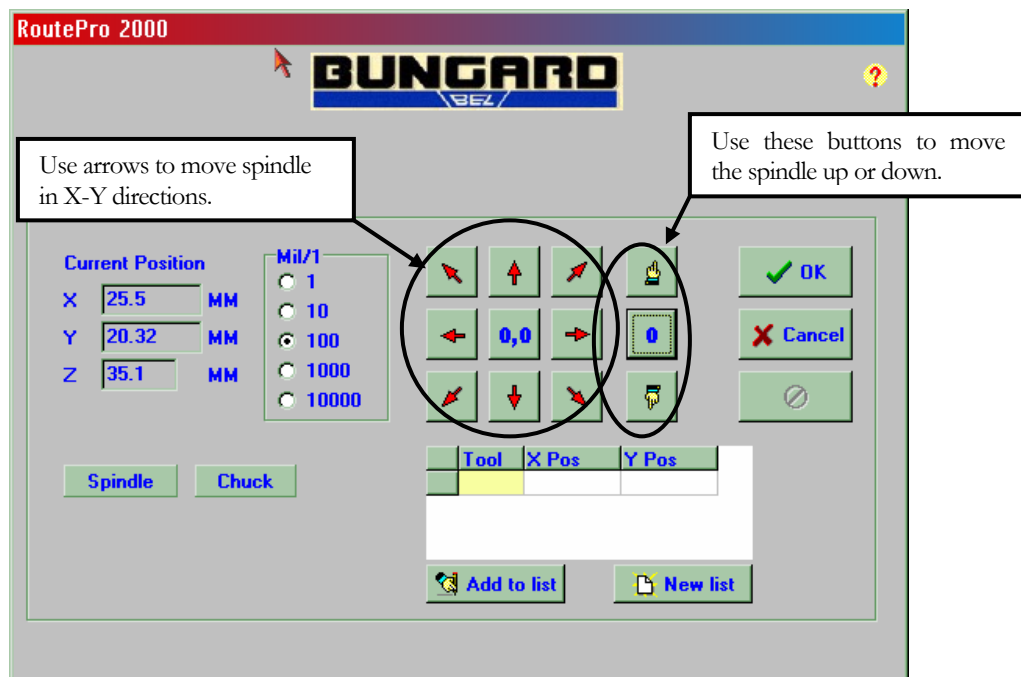


Figure 3 – Offset window.

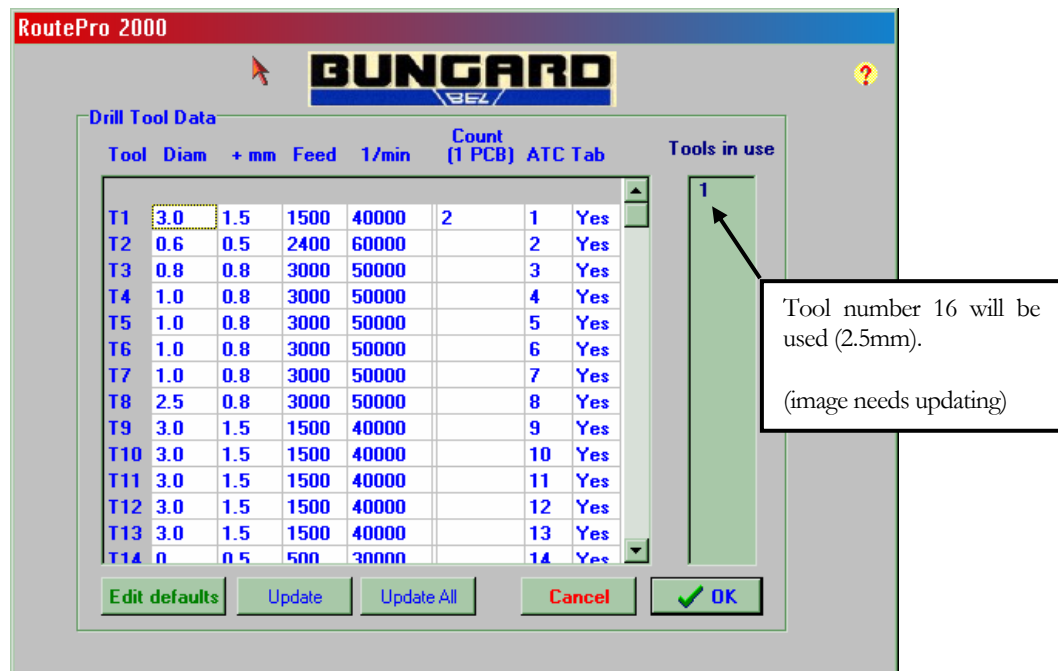
4. After the position is verified, return the spindle to its zero position by clicking the “0” button.
5. Click “OK” to exit the offset window and return to the main RoutePro window.

6. In the main window, click the “Update Data” button.
7. Write down the offset coordinates from main window to use for the remainder of the milling process. See Figure 1.

Drilling Fixing Holes

Fixing holes must be drilled so that the circuit board will remain in its exact position relative the spindle for the remainder of the milling process. The holes are drilled through the circuit board and the base board at once. Fixing pins are then inserted through the holes to secure the boards together.

1. Load drilling data for fixing holes into RoutePro. See Figure 5.
2. Ensure “Drilling” radio button is selected in main window.
3. Click the “Tools” button to open the tool configuration window. On the right side of the window the column labeled “Tools in Use” lists which tools will be used for drilling fixing holes. See Figure 4.



4. Figure 4 – Drill tool configuration window.
5. Ensure that the tool being used has the same diameter as the fixing pins. The diameter of the included fixing pins is 2.5mm.
6. Check the “+mm” field and ensure that it is 1mm less than the base board thickness. This is necessary so that the fixing holes will be drilled deeply enough into the base board to secure the circuit board firmly.
7. Click “OK” to exit the “Tools” window and return to the main window.

8. In the main window, check to make sure the board and base thickness are correct and click the “Update Data” button to ensure that all data is updated.
9. Toward the top of the main RoutePro window, minimum and maximum values are listed for X and Y directions. The circuit board must be within these areas for proper milling. Using a metric ruler measure from the X and Y axis to the edges of the circuit board and verify that it is location within these limits. See Figure 5.

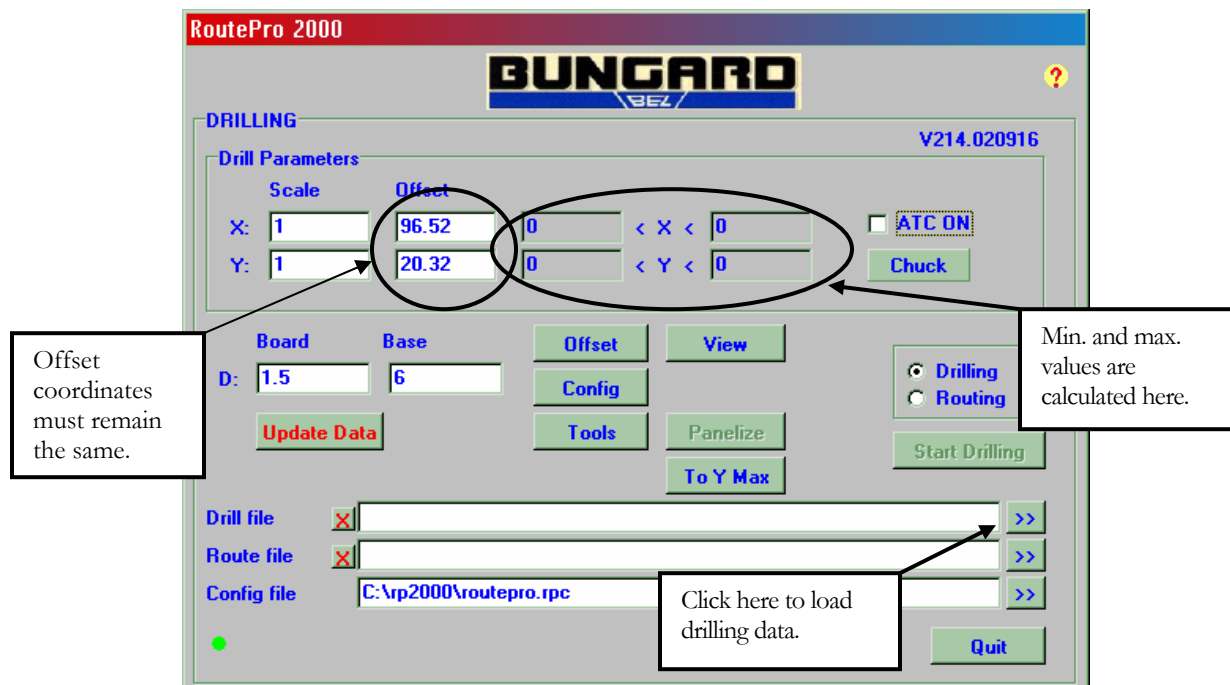


Figure 5 – Main drilling window

10. If using automatic tool change, ensure that the ATC check box is selected.
11. Before beginning to drill check to make sure the test pin is loaded into the spindle chuck. To load the test pin into the chuck, click the “Chuck” button once. Insert the test pin into the chuck and press the “Chuck” button once more.
12. Click “Start Drilling”
13. The board view along with a control toolbar window will appear. Press the arrow button to begin drilling. See below.

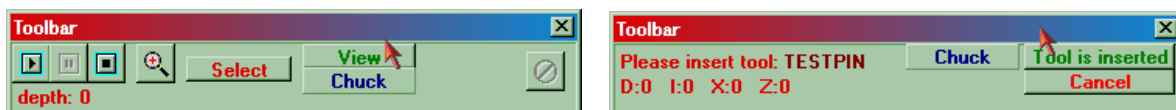


Figure 6 – Control toolbar and prompt.

14. Follow the instructions from the control toolbar window prompt to complete drilling.



CAUTION!!! Stand clear of the milling table after drilling begins. Moving parts may cause personal injury or damage to the machine if they are touched. An emergency stop button is located to the far right of the control toolbar window. This should be used in the event that the machine causes any safety hazard.

15. When the drilling is complete, insert the test pin back into the spindle chuck.

16. Insert fixing pins into the holes that were drilled through the circuit board and base board. These will secure the board through the remaining drilling and routing processes. The board may not be moved from this location until all milling is completed.

Drilling Holes for Circuit Components

Holes must now be drilled so that circuit component leads may be fed through them and secured to the circuit board.

1. Load drilling data for circuit components. See Figure 5.
2. Ensure “Drilling Data” button is selected in the main window.
3. Verify that the offset coordinates in the main window match the coordinates used in the fixing hole drilling procedure. If they are not the same, change them manually by clicking the fields and entering the coordinates. See Figure 5.
4. Ensure that the board and base thickness are set correctly in the main window.
5. Click “Update Data”
6. The X and Y minimum and maximum values will now be recalculated. As was done when drilling the fixing holes, use a ruler measure from the X and Y-axis to the edges of the circuit board and verify that its location is within these limits.
7. Press “Start Drilling” and follow the toolbar prompt as was done when drilling the fixing holes.



CAUTION!!! Stand clear of the milling table after drilling begins. Moving parts may cause personal injury or damage to the machine if they are touched. An emergency stop button is located to the far right of the control toolbar window. This should be used in the event that the machine causes any safety hazard.

Routing the Top Side of the Circuit Board

1. The first side of the board to be routed will be referred to as the “top side”. This does not necessary refer to the top side of the board in any specific design.
2. Load top side routing data and drilling data from into RoutePro at once. See Figure 7.
3. Ensure that the “Routing” radio button is selected in the main window.
4. Click the “View” button to view the routing design and drilling layout at once. Make sure that the drilling and routing data are aligned as desired. If they are not aligned properly, then one or more of your files may be corrupt. See the troubleshooting section of this manual.

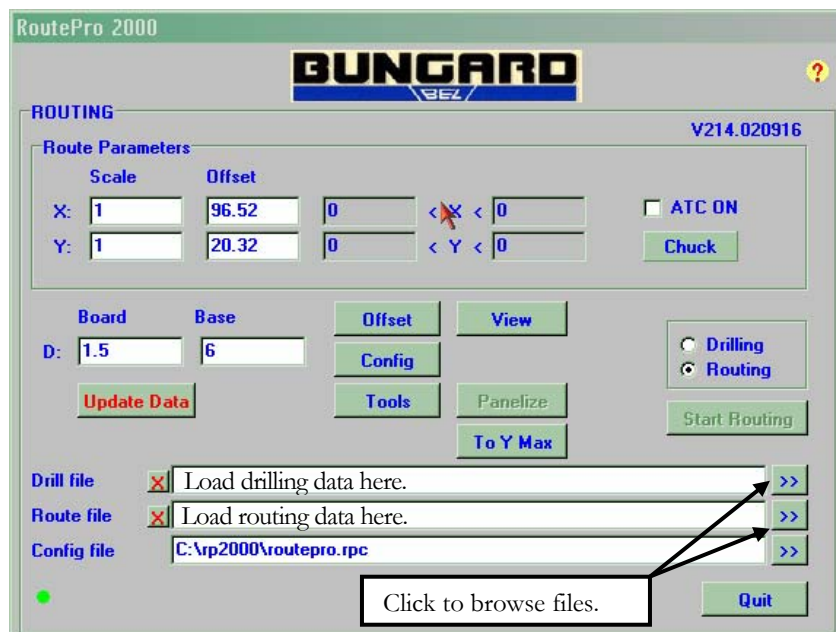


Figure 7 – Main routing window.

5. Set the offset coordinates to the same that were used for drilling.
6. Ensure the base board and circuit board thickness are correct.
7. Click “Update Data”
8. The X and Y minimum and maximum values will again be recalculated. As was done in previous steps verify that the circuit board’s location is within these limits.
9. Click “Start Routing” and then the arrow key in the control toolbar window.



CAUTION!!! Stand clear of the milling table after drilling begins. Moving parts may cause personal injury or damage to the machine if they are touched. An emergency stop button is

located to the far right of the control toolbar window. This should be used in the event that the machine causes any safety hazard.

10. If your board design is single sided, you are now finished milling. Using a pair of pliers to remove the fixing pins and lift the completed circuit board from the base board. Vacuum excess dust from the milling table and reinsert the test pin for the next use.

Routing the Bottom Side of the Circuit Board for Double Sided Designs

1. When the top side routing is complete, load the bottom side routing file.
2. Click the “Y to Max” button.



Warning!!! Stand clear of the milling table after clicking **Y to Max**. Moving parts may cause injury or damage to the machine if they are touched.

3. Using a pair of pliers, remove the fixing pins and circuit board from the baseboard.
4. Turn the circuit board over to the other side by flipping it toward the X-axis and about the axis created by the fixing holes. The blank side of the circuit board should be facing upward.
5. Place the circuit board back onto the base board and insert the fixing pins through the circuit board into the same fixing holes on the base board.
6. Again, ensure that the offset coordinates match those from top side routing and drilling steps.
7. Check to make sure the base board and circuit board thickness are correct.
8. Click “Update Data”
9. Once more, the X and Y minimum and maximum values will be recalculated. As was done in previous steps verify the location of the circuit board.
10. Press “Start Routing” and the arrow button to begin routing.



Warning!!! Stand clear of the milling table after drilling begins. Moving parts may cause personal injury or damage to the machine if they are touched. An emergency stop button is located to the far right of the control toolbar window. This should be used in the event that the machine causes any safety hazard.

11. Once routing has finished, reinsert the test pin into the spindle chuck.
12. Vacuum excess dust from the work area and remove the finished circuit board. All milling is now complete.

Advanced Milling Options

The following options should be utilized only after becoming comfortable with the basic milling process. These options will be used to create more accurate designs and increase productivity for making larger quantities of identical circuit boards.

Milling Requirements



It will take about thirty minutes to complete the milling



- Drilling and routing files created using the procedure outlined in the previous section.
- 1 blank circuit board.
- 1 base board.
- Sufficient drill and routing bits to meet your design specifications.
- Adhesive tape
- 4mm hex wrench
- 2 fixing pins
- Metric ruler

Using the Depth Limiter

The depth limiter is used during routing to allow for regulation of the routing depth in order to compensate for uneven board surfaces. The depth limiter is designed to work with a G30N type routing bit. This type of bit has a conical tip and comes to a fine point (see Figure 9). If using this option, fixing pins may not be located within 10mm of the nearest area to be routed as they will interfere with the depth limiting device. The depth limiter is used for routing ONLY. Using the depth limiter for drilling will cause damage to the machine.

INSTALLING A DEPTH LIMITING DEVICE ONTO THE SPINDLE



Figure 9 – G30N type routing bit.

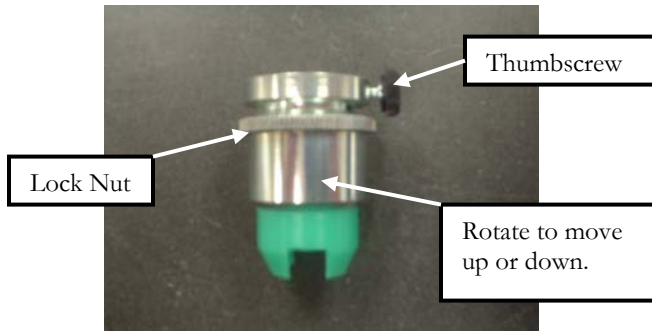
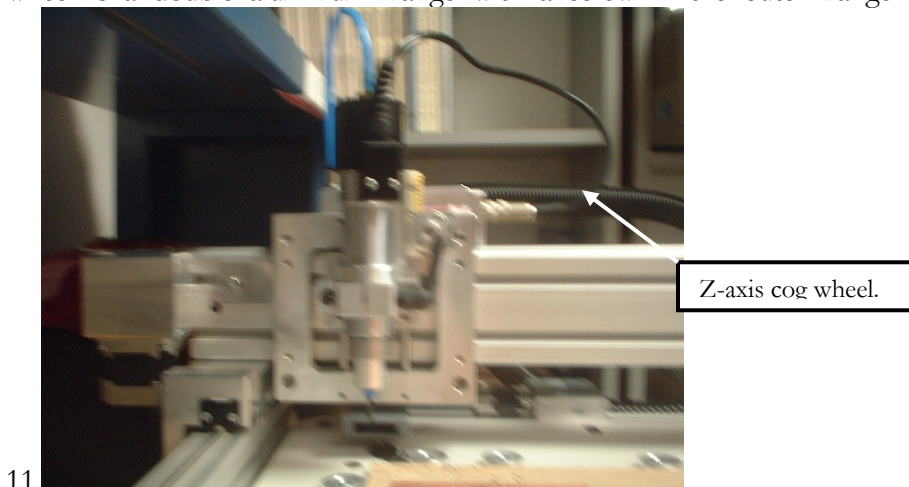


Figure 10 – Depth limiter

1. Unscrew the small thumbscrew located on the depth limiter. See Figure 10.
2. Slide the depth limiter over the bottom of the spindle shaft and push upward until it hits the upper flange on the spindle.
3. Tighten the thumbscrew to secure the depth limiter in place.
4. Find the Z-axis cog wheel located near the Z axis slider to the right of the spindle. The wheel is a double aluminum flange with a screw in the outer flange. See Figure



11.

Figure 11 – Spindle and Z-axis Cog Wheel

5. Remove the screw from the outer flange of the cog wheel. You should now be able to move the spindle up and down about 2mm without the Z axis motor turning.
6. Place a G30N bit into the spindle chuck

SETTING THE DEPTH LIMITER TO A DESIRED ROUTING DEPTH

1. Turn the bottom part of the depth limiter clockwise so that the green conical piece is below above the tip of the routing bit.
2. Click the “Offset” button to open the offset window
3. Use the arrows to position the spindle over the circuit board surface in an area that will not be used.
4. Select the “1” radio button in the “Mil/1” area of the offset window. This will allow for more precision movements of the spindle.
5. Click on the “Spindle” button turn on the spindle and allow it to run at the lowest speed.



Warning: Be extremely careful when working around the rotating spindle. Contact with the router bit will result in personal injury. The spindle must be rotating at all times when the router bit is in contact with the circuit board surface. Failure to do this will result in permanent damage to the router bit.

6. Lower the spindle until the tip of the routing bit just touches the circuit board surface.
7. Observe the router depth and lower the spindle until the router is at the desired depth.
8. To test the router depth, move the spindle in the X or Y direction for a distance of about 1 cm. Observe the routed area and adjust the depth as necessary.
9. Carefully screw the bottom of the depth limiter counter clockwise so that it moves downward and makes contact with the board.
10. At this point, stop the spindle from rotating by clicking the “Spindle” button once more.
11. Turn the threaded lock nut on the depth limiter counter clockwise until it hits the lower part of the depth limiter and secures it in place.
12. Click “OK” to exit the offset window and return to the main window.
13. The depth limiter is now set and requires no additional adjustments. You may now follow the basic routing process with the depth limiter in place. It is not necessary to readjust the depth limiter for future milling processes if the same routing depth is desired.

Panelizing Circuit Boards

Panelizing allows for multiple identical boards to be made from one large blank circuit board. The following panelizing steps must be followed for each part of the milling process.

1. Click the “Panelize” button in the main RoutePro window. This will open the panelizing window.
2. Using a ruler, measure the length and width of the circuit board that will be used.
3. Enter these dimensions (in mm) into the “Board size X” and “Board size Y” fields in the panelize window. See figure 12

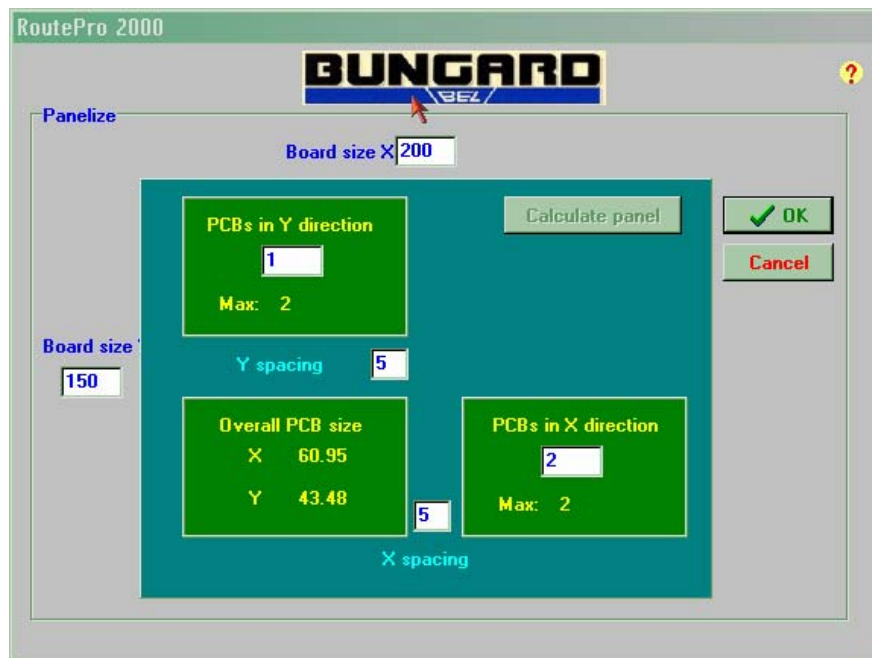


Figure 12 – Panelize Window

4. Click “Calculate Panel”. Based on the size of the design that is currently loaded, the total number of boards in each direction will be calculated.
5. Enter the number of boards to be milled in each direction.
6. The “X spacing” and “Y spacing” fields determine how far apart adjacent boards will be located. Enter the desired spacing value in each direction.
7. Click “OK” to return to the main window.
8. Proceed with basic drilling and routing processes. Ensure to recalculate the panel after each new file is loaded into RoutePro.

Desk References

This section contains reference materials that are designed to be printed out and be placed nearby while you are working on PCB layouts.

Preparing Isolation Milling Files

Once you've already completed a few PCBs you will remember exactly how to do each of these steps. It's easy to forget the order in which everything should be done. The list below lists the steps involved in preparing isolation milling files in sequential order.

1. Load drill rack file. Remember to set the File Type to "Drill Tool File"
2. Load the Gerber/Excellon files with extensions: .top, .bot, .drl, and .dim.
3. Add fixing holes
4. Mirror the bottom layer. Change the layer to the bottom layer (the one with the .bot extension). Right click some where on your design and choose 'mirror over fixing holes.
5. Create Milling Data for the bottom layer. Select 'File' -> 'create milling data'. Then save the milling data by selecting 'File' -> 'save milling data'.
6. Create Milling Data for top layer. Select the top layer. Follow the same procedure as you did for creating the bottom milling data except save it as top.plt since it's the top layer.
7. Create the board cutout. Select the Dimension layer (the one with the .dim extension). Hide all the layers except for the dimension layer. If any holes show up inside the PCB, delete them by selecting them and pressing the delete key. Then save the milling data (File -> 'save milling data') and name it dim.plt.
8. Save the drilling data. Select the fixing layer 'fixing.drl'. Use Save As to save drill data.
9. Save the fixing hole data. Select the fixing layer 'fixing.drl'. Use Save As to save drill data.

Troubleshooting

In this section you'll find answers to common problems.

For most of the problems we've encountered while making PCB's we have provided solutions below.

Problem: When I view my files in RoutePro 2000, the drilling and routing data isn't synchronized.

Case 1: Routing data is larger then drilling data

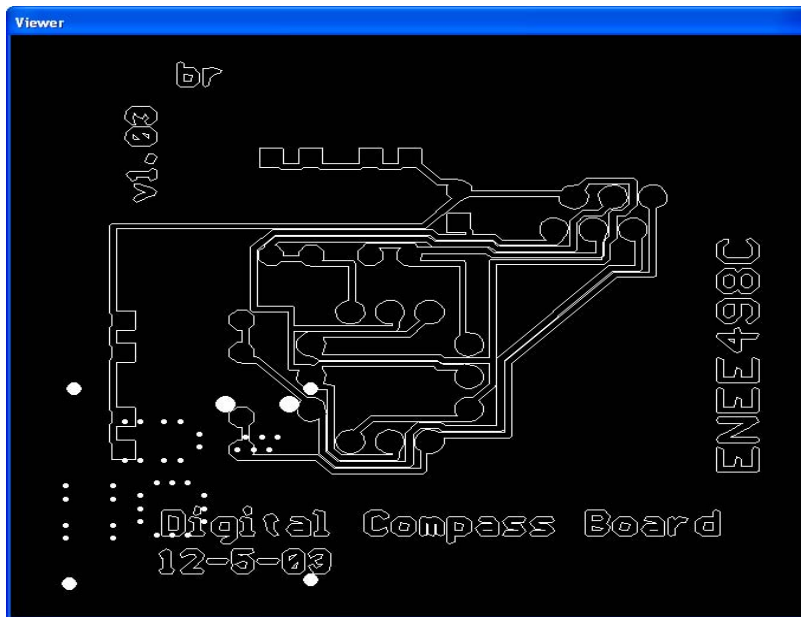


Figure 11: Case 1 Example

Solution:

Check that in IsoCam you haven't accidentally changed the 'output resolution.' It should be set to 0.025mm (HPGL).

Case 2: Drilling and Routing Data are slightly offset.

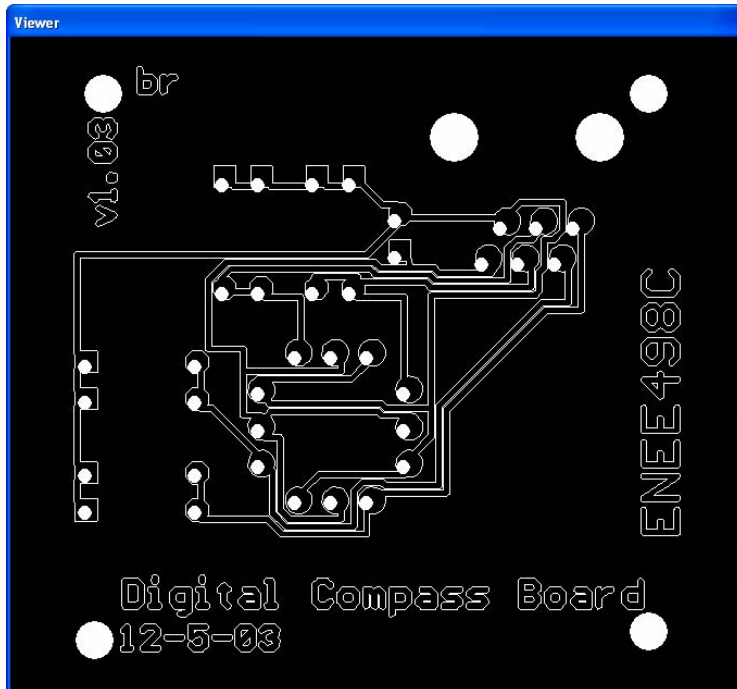


Figure 12: Case 2 Example

Solution:

- Check that the settings in route pro under comfit are set properly.
- Check offset used

Case 3: Drilling data is much larger then the routing data.

Solution:

Most likely when saving the drill files the '2 4' drill format was selected rather than the '2 3'. If this doesn't remedy the problem check the settings in Route Pro under **config**.

Future Revisions

This section lists some of planned additions to the manual.

The revisions outlined here couldn't be included in the initial release. Some revisions that are due, are necessary because of software bugs that are being researched by company. Some revisions are necessary because isolation milling is a huge topic to know everything about. Amendments will be made to the guide as my research continues with the University of Maryland.

This guide will be used primarily by other universities and therefore the guide's primary media will be in a digital form.

If you would like to be notified when the guide is updated please send an email to blaker@umd.edu.

- After some usability testing, some format changes will be made.
- Add routing bit information to Recommendations section.
- Improve cncComplete.cam to remove unnecessary .dri and .gpi files and create a separate directory for the exported files.
- Expand Troubleshooting section.
- How to configure RoutePro 2000.
- Add information on changing layers and hiding and showing layers in IsoCam.
- How to deal with board conflicts when creating milling data.
- Guide on what the different routing bits are and used for end-mill, down-cut, up-cut.
- Add note pointing out the tolerances on the screenshot in configuring Eagle.
- Add vacuum related assembly info.
- Create screenshot video for each of the board export sections.
- Desk Reference needs to be improved. Create reference for milling operations.

Appendix

The appendix contains references which provide additional information about the topics discussed in this manual.

Appendix A: Glossary of Terms:

The terms below were written by ThinkTink a leader in PCB prototyping and consulting.

Computer Numerical Control (CNC) - Controlling the motion of an object using electrical or hydraulic actuators directed by a computer. In CNC machining, a computer directs a set of drive motor to move a cutting tool (or the object being cut) in a precise pattern often consisting of joined lines, arcs and circles.

Router - A cylindrical rotary cutting tool with "teeth" ground in a spiral along the entire EFL. Tools of this sort are commonly used on composites consisting of a very hard material (glass, granite) embedded in a much softer matrix (PVC, acrylic, epoxy, phenolic).

Spindle - Any motor driven shaft fitted with a drive motor on one end and a tool holder on the other. Spindles are available with belt drives, direct drive, gear drives and air turbines

Tolerance - The acceptable deviation from a given specification.

Appendix B: Technical Definition of Gerber & NC Drill Files

Definition courtesy of PCBInc.com

The gerber files are only X Y coordinates. They direct the laser where to go when making the film, that is all; the real key to the gerber plot is the Aperture list or Decode chart.

The Aperture list tells the plotter what to plot when it goes to a given coordinate. There are two types of Apertures, FLASH and DRAW.

FLASH Apertures are flashed in a given place per the coordinates from the gerber file. After each coordinate or pair of coordinates, if it is a draw command, there is a decode, i.e. D100, D10, D51, etc...The plotter plots whatever the decode list has in that value at the precise point called out in the gerber file. It may plot a 50 mil circle, a 20 mil square, a 50 x 100 mil rectangle, or what ever Aperture was called out by the Aperture list.

The second type of Aperture is a draw. This is what will direct the plotter to drag (draw) a line with the width being the Aperture value from the first point in the gerber file to the second point of the pair listed. The given Aperture. i.e. apertures may be 20, 50, 10 etc. each depicting a given trace width.

There are many formats for gerber files, 2.3; 2.4; 3.3; and so on... This defines the number of digits to the left and right of the decimal point. 2.3 format is 2 digits to the left, and 3 digits to the right. Also, zeros can be suppressed. This function will suppress either the leading or trailing zeros. All newer types of photo-plotters and CAM software (used by board shops and plotting companies) can handle any of the above formats with unlimited Aperture styles. The only requirement is that flash Apertures must be smaller than .500" in diameter.

Most CAD systems generate gerber files and aperture lists. You just have to make certain to send the Aperture list, as it is equally as important as the gerber file.

There are some newer and more advanced CAD systems that generate the Aperture list within the gerber files. This process helps lesson errors in plotting and is typically used in ORCAD products and referred to as 274X format.

Finally, if you are generating gerber files for producing a board, you will also need a NC DRILL file. Systems that produce gerber files also produce NC DRILL files. The standard file is an Excellion drill file. Excellon is the industrial leader in CNC drills for the PC board market.

Following is an example of a drill file (X Y Coordinate list) and a tool file (A list of the tool, drill, sizes), i.e. T01 = .031", T02 = .040", etc... Once again, the base file is a coordinate list and the tool file guides the drill as to which size once it gets there.

The drill file format is a little more critical than the gerber format. 2.3 or 2.4 are the industry standard. There is no punctuation or spaces in either drill or gerber files, the decimal place is assumed by the format. Excellon drills read data from the left to the right. Therefore, leading zeros must be present, if they are suppressed, the coordinates will not be correct.

Example, the number 3.500" would be X03500. If the leading zeros were suppressed, the machine would think the coordinate was 35.000 inches.

Trailing zeros are normally suppressed to reduce the file size. The coordinates X03500 and X035 are both the same. If a coordinate, either X or Y remain the same from one hole to the next, they need not be repeated. i.e. X05000Y01000 followed by Y02000 would be two holes, the 1st is x 5.0" y 1.0" and the second is x 5.0" y 2.0".

The commands m00 and m30 in a drill file simply mean end of tool size or end of file.

The % indicates the start of a file. There are also repeat commands and other more advanced settings that your system may use. **JUST MAKE SURE TO HAVE LEADING ZEROS PRESENT IN YOUR DRILL FILES.**

For producing slots in drill files, holes are normally repeated .005" apart to create the slot. It is important to indicate this clearly on a print as the programmer at the board shop may program slots in routing or milling stages, depending upon the type of machinery available and the slot size.

In addition to providing a drill file, it is also advisable to provide a drill drawing or plot showing board dimensions and hole sizes. This is a good tool to provide as it allows the board shop to have a means of inspection and verification when checking the product.

The drill file is used to produce the product, but consider how difficult it would be to check hole sizes against. Hopefully this information will give you a clear understanding of Gerber files and the requirements of PC Board Manufacturers.

Bibliography

http://www.bungard.com/downloads/ccd_ane2.pdf.

Original Bungard CCD manual which the safety instructions were taken from.

<http://www.ee.ualberta.ca/~ee401/reference/PCBlayout.pdf>

A guide for laying out a PCB for isolation milling prepared by University of Alberta.

<http://claymore.engineer.gvsu.edu/~steriana/Publications/ASterian.ASEE2001big.pdf>

A paper comparing the differences between commercial PCB milling and isolation milling.

<http://www.pcboardsinc.com/emt/gerber.html>

Provided technical definition of NCI drill files

Markel, Mike. (2003). Technical Communication. Boston : Bedford/St.Martin's.

Technical reference for writing manuals. Formatting conventions were used from a Dell manual.