CircuitCREATOR™

Logic CREATOR: *Schematic Capture*

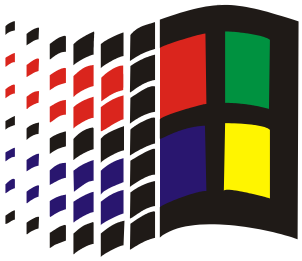
Symbol CREATOR: *Part/Symbol Editor*

Board CREATOR: *Printed Circuit Board Editor*

View CREATOR: *Photo-Plotter Viewer*

And

Route CREATOR: *PCB Auto-Router*



Microsoft Windows 95/98, NT, Windows 2000 and XP

ACKNOLEDGEMENT and COMPANY INFORMATION

The CIRCUIT CREATOR product is published and marketed by

Advanced Microcomputer Systems, Inc.
1377 S. Andrew Avenue,
Pompano Beach, Florida, 33069 USA.
Web Site: www.advancedmsinc.com
E-mail: info@advancedmsinc.com
(954) 784-0900
Fax (954) 784-0904

CIRCUIT CREATOR™
INDEX

INTRODUCTION

The purpose of this package is to help you evaluate the Schematic Capture, PC board layout, and Auto-Routing software of the CIRCUIT CREATOR electronic schematic and PC board layout system. This evaluation package is divided into several sections. The sections are arranged in order of increasing detail. The first section gives a general overview of the entire CIRCUIT CREATOR system. Other sections include specifications, software installation, and hands-on tutorials. The final section gives information on how to order the products mentioned in this package.

CONTENTS OF THE EVALUATION PACKAGE

This evaluation package contains special versions of the LOGIC CREATOR schematic editor, BOARD CREATOR printed circuit board editor, and the ROUTE CREATOR printed circuit board auto-router programs. The package is designed to give you hands-on experience using the software, show you how easy it is to use and, and demonstrate its capabilities. We think this package will convince you that this system can create and maintain electronic schematic designs easily and efficiently. Also, this is a very cost-effective system.

If, after purchasing the CIRCUIT CREATOR product, you are not satisfied with its performance, you may return it within 30 days for a full and prompt refund of the purchase price.

LIMITATIONS OF THE EVALUATION PACKAGE

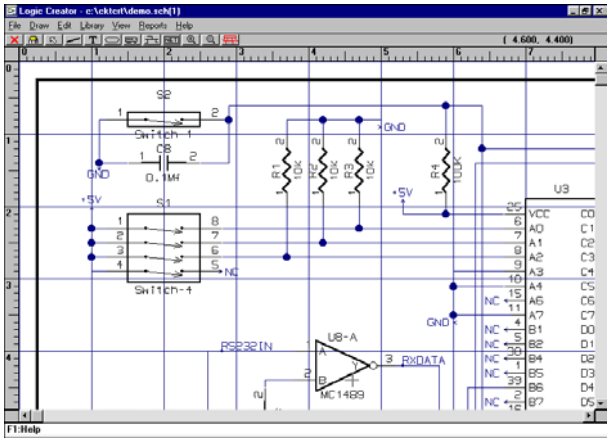
The program furnished in the evaluation package functions exactly like the real version except that you will not be able to save your designs to a disk file. Also, the library of parts and symbols in the evaluation package contain only the parts and symbols required in the demos and tutorials. The real version comes with complete libraries of over 25,000 part/symbol combinations.

CIRCUIT CREATOR SYSTEM: OVERVIEW

This section of the evaluation package is designed to give you an overview of the entire CIRCUIT CREATOR system. A brief description of each of the major components, and how they interact with each other is presented.

Logic Creator: Schematic System

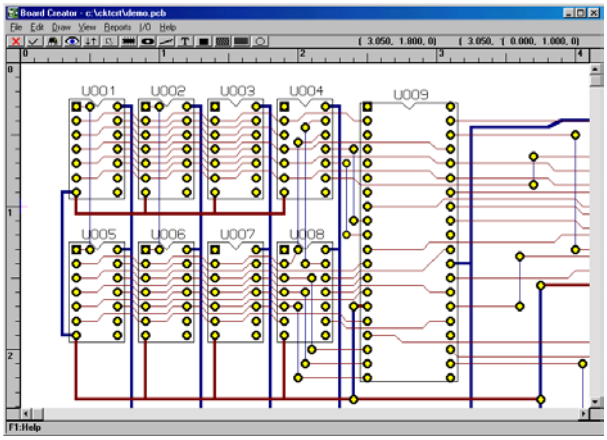
The LOGIC CREATOR schematic system is a high-end electronic schematic entry system. It is used to create and/or update electronic designs. It also is used to directly print schematics, generate electronic reports of designs, and to generate a data file that is used by the BOARD CREATOR PC board layout system to create the initial PC board for a design, or to update an existing design.



Logic Creator

Board Creator: PC Board Layout System

The BOARD CREATOR system is used to layout PC board designs. This system is used to create, edit, and maintain the layout masks for PC board designs, print the masks to printers, plotters, and photo-plotters, maintain the component footprint symbols, schedule routes for the auto-router, and to print net lists, part lists,

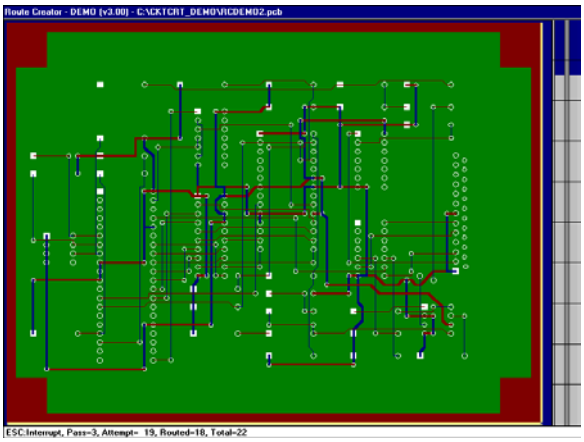


Board Creator

and other PC board specific reports. Also, the program can check a PC board layout for agreement with physical design rules. Such things as clearances of lines to pads, lines to lines, lines to vias, as well as numerous other checks are performed.

Route Creator: Auto-Router

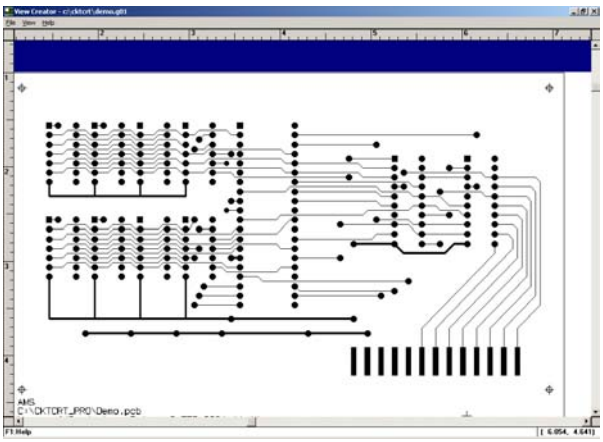
The ROUTE CREATOR program is used to automatically route the electrical connections of a PC board. The auto-router uses a method of artificial intelligence to automatically route the paths of electrical connections. The auto-router will preserve any nets that you wish to pre-route manually. Also, the auto-router allows for being interrupted, manually editing the board with the editor, and continuing the automatic routing.



Route Creator

View Creator: Gerber Viewer

The VIEW CREATOR program may be used to preview Gerber photo-plotter files on the computer screen, prior to actually printing the files.



View Creator

SPECIFICATIONS

The following is a summary of the specifications for the CIRCUIT CREATOR system.

SYSTEM WIDE FEATURES

- Runs on IBM PC, IBM PS/2 and compatible computers with WIN 95/98 2000 Operating System.
- State-of-the-art, object-oriented user interface with pull-down menus and pop-up dialog boxes. Dynamic point, click and drag editing.
- On-line and context-sensitive help.
- Automatic "check-pointing" of the work in progress. Protects against loss of work in the event of a power failure. Time between check-points is controlled by configuration.
- Complete user manuals with tutorials for all the major subsystems.
- Video support for EGA, VGA, and Super-VGA and many other display devices supported by WIN95/98 2000 System..
- Hard copy support for PostScript, Epson, Hewlett-Packard, IBM, Gerber, and many other printers/plotters supported by WIN95/98 2000 System..
- Parameters such as the default path for drawings, memory usage, default width of lines, display colors, etc. may all be configured.

SCHEMATIC ENTRY

- A complete electronic schematic working environment in a single program.
- True hierarchical designs. Each hierarchy level has independent net naming and pseudo-part reuse capability.
- Fully compatible with the BOARD CREATOR printed circuit board system.

- Up to 32 E-size sheets of logic in a single design, even more through the use of hierarchical designs.
- Comes with standard part and symbol libraries. Over 25,000 part/symbol combinations and can be expanded by the user.
- Symbols may be dynamically mirrored and/or rotated. A single symbol definition may be displayed in any of sixteen different orientations. No need to make separate symbol definitions for different views.
- De-Morgan equivalents and more. A single part may be represented by many different symbols. Rotation and mirroring of part symbols from a single definition. Symbols are based on true geometric definition, not a raster image.
- Automatic re-routing of connections when objects are moved. Not just "air-lines", but true schematic routing via use of "smart-lines".
- Bezier curves for true curved lines, not just arc segments.
- Electronic/printed design reports of parts-list, net-list, loading, design verification, etc.
- Direct design error checking. Errors are pointed out directly on the schematic with an explanation of exactly what the problem is.
- "Double-click" to update any object on the schematic.
- Supports hierarchical drawings and full hierarchical net naming conventions (inheritance).

PC BOARD LAYOUT

- A complete electronic printed circuit board layout working environment in a single program.
- Fully compatible with the LOGIC CREATOR schematic system.
- Imports other Schematic and PCB files.
- Boards up to 32 x 32 inches, 255 layers.

- Comes with a complete set of standard foot-print part symbols, including surface-mount devices.
- Footprint symbols may be dynamically rotated and/or mirrored to the back of the board. No need to make separate symbol definitions for different orientations or board sides.
- "Double-click" to update any object on the board.
- Support for blind and buried vias.
- Support for power and ground planes.
- "Copper pour" command for filling unused areas of a board with conductor, such as power or ground. Can be selected and removed as a single unit, or edited as individual lines.
- Direct design and clearance error checking. Errors are pointed out directly on the screen. Checks that all points that should be connected are actually connected to each other. Checks for nets improperly connected (shorts). Checks for pad-to-pad, pad-to-via, pad-to-line, via-to-line, and line-to-line clearances. Component, solder side, and inner layers may have different clearance rules.
- Lines 0.001 to 0.255 inches wide.
- Up to 25 different pad shapes and sizes per board. users may define their own pad shapes. Different layers of the same pad may have different shapes.
- Text in 16 combinations of rotation and mirroring. Eight degrees of boldness.
- Bezier curves for true curved lines, not just arc segments.
- Arcs and circles.
- Built-in report generator for net-list, part-list, drill-hole, etc.
- Built-in hard copy output allows queuing of up to ten plots.
- "Rat's nest" prompting to aid in optimal component placement and manual routing of nets.
- User specified layout grid.

- Users may work in mils, inches, or millimeters, and switch at any time.
- Can create/update an CIRCUIT CREATOR board from the following net/pin list formats: CIRCUIT CREATOR, Schema, Orcad, Futurenet, Pcad.
- Can create a net/pin list from an CIRCUIT CREATOR board in the following formats: CIRCUIT CREATOR, Schema, Orcad, Futurenet, Pcad, Applicon Bravo, Applicon Leap, Computervision, Calay, Cadnetix, Scicards, Spice, Telesis, Vectron, Multiwire, and Excellon drill list.

PC BOARD AUTO-ROUTER

- Fully compatible with the BOARD CREATOR printed circuit board system.
- Routes multi-layer boards, two layers at a time.
- Routes boards with different line widths, such as wide lines for power and ground, and normal lines for other nets.
- Router respects nets pre-routed (manually) by the user.
- Router may be interrupted, board edited with BOARD CREATOR, and restarted to finish the routing.
- User selectable routing parameters for line width and clearances.
- User selectable routing strategies.
- Visual display of the routing progress.
- Full support for routing blind/buried vias.
- Routing schedule may be generated automatically with BOARD CREATOR, by hand, or any combination of the two.
- Routing schedule may specify routing to a specific layer. This is required to route to edge connectors and SMD devices.
- Uses a proprietary, state-of-the-art, artificial intelligence algorithm to route the nets. Routing completion rates are matched only by mainframe and high-end work-stations routers costing much more and requiring very expensive equipment.

CUSTOMER SUPPORT

- Free Email and Fax support during normal business hours.
- If you sign up for our yearly update service, you will receive all scheduled program revisions as they become available.
- If you decide not to sign-up for the yearly update service, you can always get the latest revision of any CIRCUIT CREATOR program for a small update fee.

HARDWARE REQUIREMENTS

The following hardware is required to use the CIRCUIT CREATOR system on an IBM PC or compatible computer.

COMPUTER

An IBM PC, IBM PS/2, or compatible computer with a minimum of 4 MB of memory and a hard disk with WIN95/98 2000 System is required.

GRAPHICS

A VGA, EGA, Super VGA, or equivalent graphics card is required.

MOUSE

Microsoft, Logitech, Mouse Systems mouse, or a compatible mouse is recommended.

HARD-COPY

A printer/plotter is not required by the evaluation package. For the real system, a printer is recommended for hard copy printout of schematic and PC board designs, check plots and reports.

INSTALLATION OF SOFTWARE

A) CD-ROM Installation

If you received this software on a CD-ROM you can simply put the CD into your drive and the installation should start automatically. If it doesn't then run setup.exe off the CD.

B) Download from Web

If you downloaded the demo from the web then run the file "**cktcrt_demo.exe**".

In both cases the installation program with ask the name of the directory where you wish to install the demo. You can use the default (CKTCRT) or give it any name you wish. The same goes the for name of the Program (Start Menu) Group.

To start any of the programs go to CIRCUIT CREATOR DEMO program group (on the Start Menu) and click on the appropriate CIRCUIT CREATOR icon.

If you have any problems using the Start Menu, consult your *Microsoft WIN 95/98 Users Manual*.

Circuit Creator Tutorial

INTRODUCTION

The next few sections present tutorials for the LOGIC CREATOR schematic entry program, BOARD CREATOR PC board layout program, and the ROUTE CREATOR program. These tutorials will give you actual hands-on experience using the CIRCUIT CREATOR system. The tutorials will lead you through the creation of a simple schematic and its PC board layout.

NOTATION

The following notation is used throughout the tutorials.

The notation "*XXX*" indicates that the characters shown between the quotes are to be typed. For example, "XYZ" indicates that the characters *X, Y*, and *Z* are to be typed. Do not type the quotes.

The notation, *XXX*, indicates this single key, key combination, menu/command, or dialog-box button should be pressed. For example, **ENTER** means press the enter key while G indicates the G key is to be pressed. **FILE/OPEN** means to select the OPEN command from the FILE menu. **OK** means to select the **OK** button in a dialog-box. Unless otherwise noted, either upper or lower case may be used.

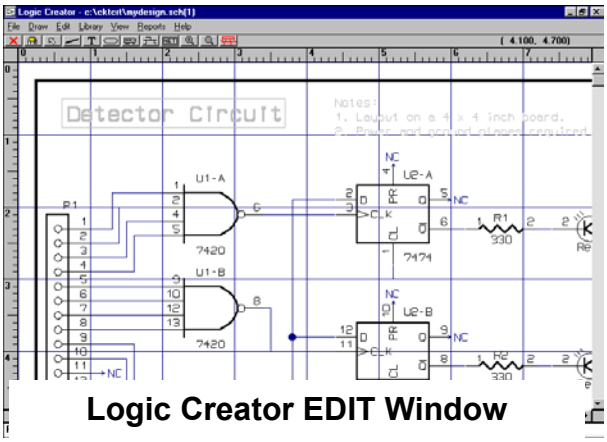
The notation, (*xxx, yyy*), indicates a coordinate. For example, (10.0,15.250) indicates the horizontal position is 10 inches and the vertical position is 15.250 inches.

The tutorial steps are listed in terms of the keyboard. If your system has a mouse, feel free to use it. The left button of the mouse is equivalent to the **ENTER** key, while the right button is equivalent to the **ESC** key.

Initially, great detail is given. As we proceed through each of the tutorials, and you learn the basics; less detail will be given. If you will be using a plotter connected to a serial port.

LOGIC CREATOR: SCHEMATIC TUTORIAL

This tutorial leads you, step by step, through the creation of a simple schematic.



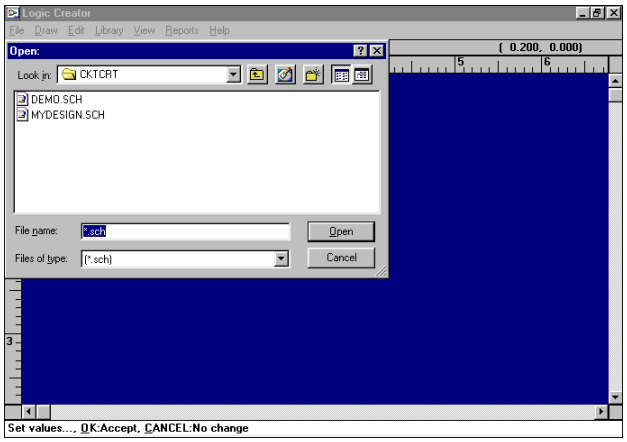
Logic Creator EDIT Window

STARTING LOGIC CREATOR

To run the LOGIC CREATOR program, double click on the CIRCUIT CREATOR program group (if it has been minimized) and then double click on the LOGIC CREATOR icon.

LOADING A SCHEMATIC

To get a feel for the system, we will first load a sample schematic file. We will then learn how to navigate about the schematic on the display. First, an explanation of how to select commands from the pull-down menus and dialog-boxes will be given.



Loading Schematic File Screen

If you are using the keyboard, a pull-down menu may be displayed (pulled-down) by holding the **ALT** key down and pressing the key

indicated by the underlined character in its name on the title-bar. Upper or lower case may be used.

When a pull-down menu is displayed, the → and ← arrow keys may be used to move to the next menu to the right or left. The **NUMLCK** key must be off. Use the ↑ and ↓ arrow keys to move the menu selector up and down. Move to the **FILE/OPEN** command and select it by pressing the **ENTER** key.

If you are using a mouse, you may pull-down a menu by moving the pointer to the menu name in the title-bar and pressing the left mouse button and releasing it. The menu will appear when the button is released. To select a menu item, move the pointer to the desired item and press the left mouse button and release it. The selection is made when the button is released. For quicker menu selections, move the pointer to the title and press the left mouse button and do not release the button yet. Now, move the pointer to the desired menu item and release the button to make the selection.

FILE SELECTION DIALOG BOX

After selecting the **FILE/OPEN** command, a dialog-box will appear which is prompting for the name of a schematic file. At this point, we can type the file name of a schematic file, or select one from the list that is displayed. Use the arrow keys to move the selector to the file in the list named "DEMO.SCH" and press **ENTER**. This will copy the file name from the list into the file field of the dialog-box. Now press **ENTER** again to directly accept this name, or press the Home key to move the selector to the **OK** button, then press **ENTER**.

When a dialog-box is displayed, clicking the left mouse button on the **OK** button, or pressing the ALT-O key combination will immediately accept the values in the dialog-box. Pressing the **HOME** key will move the selector to the **OK** button.

Clicking on the Cancel button, or pressing the **ALT-C** combination will immediately cancel the dialog-box. Pressing the **END** key or the **ESC** key will move the selector to the cancel button. Once the selector is on the cancel button, pressing **ESC** again will cancel the dialog-box. Any changes made to the values shown in a dialog-box do not

actually take effect until the **OK** button is pressed. If the **CANCEL** button is pressed, the changes made to values in the dialog-box are ignored.
For a file-selection or part-selection dialog-box, pressing **ENTER** twice while on the same name in the list implies that this is the name you want to accept immediately.

GETTING HELP

Press the **F1** key to pop-up a window of help information. Notice that the help information is specific to the current command being performed. The UP and DOWN buttons may be used to page the help text up and down in the window. You may also use the **NEXT, PREV**, or **GENERAL** buttons to view other help topics.

Now, select the **EXIT** button to close the help window. Help is always available with the **F1** key.

MOVING AROUND A SCHEMATIC

We will now learn how to change the view of a schematic displayed on the screen. Pull-down the View menu. Now, move down and select the Zoom Out command. The display will be redrawn at a smaller scale. You will notice that the **ZOOM OUT** menu item has **CTRL+PGDN** listed in the right margin next to it. This indicates that this command has a hot-key. This key combination may be used when the menu is not displayed to directly execute this command. Let's try it. Now, hold the Ctrl key down and press the **PGDN** key to zoom-in. Once you are more familiar with the program, using hot-keys will improve your productivity, especially if you do not have a mouse.

Zoom back to the original scale by pressing the **CTRL+PGUP** key twice.

The current coordinate of the cursor is displayed in inches in the upper-right corner of the screen. Use the cursor keys to move the cursor to coordinate **(2.000,1.000)**. Follow the track of the cursor on the screen, and watch the value in the upper-right corner change. Notice that the active cursor (*the little cross*) is not always the same as the mouse pointer position.

Holding the **SHIFT** key down and using the cursor keys will move the cursor one inch at a time. Hold the **SHIFT** key down and use the ← and ↑ arrows to move quickly to coordinate **(0.000,0.000)**, the upper-left corner of the sheet.
The display may be panned using the keyboard by bumping into one of the edit window edges. Let's pan right. Move the cursor with the **SHIFT** → key until it bumps the right edge of the window. The portion of the schematic that was out of view to the right of the screen is now displayed.

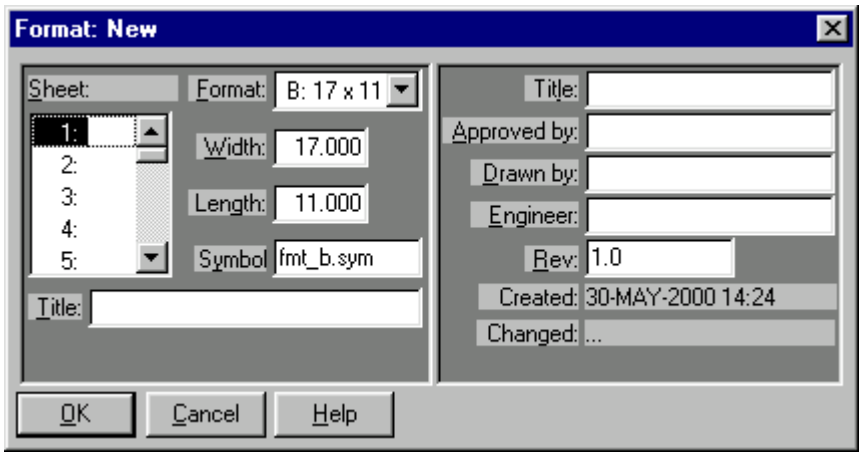
With a mouse, the view may be panned by using the vertical slide-bar at the right, or the horizontal slide bar at the bottom.

Display the upper-left corner of the sheet at normal scale by pressing **CTRL+HOME.**

REFERENCE GRID

The lines, normally green, that you see on the screen represent a reference grid. This grid is used to assist the user in drawing a schematic and lining up objects. It is not part of the schematic itself, and is not printed on hard-copy output. From the **VIEW** menu, select the **GRID** command. This dialog-box is used to control the display of the reference grid, and the step size of the cursor movements. Select the check-box entry for **DISPLAY** so the X check-mark is off. Now, select **OK**. Notice that the grid lines no longer appear in the display; however, the cursor-motion grid is still active.

A single **LOGIC CREATOR** design file may have up to thirty-two sheets of drawings. This demo drawing has two active sheets. From the **VIEW** menu, select the **SHEET**



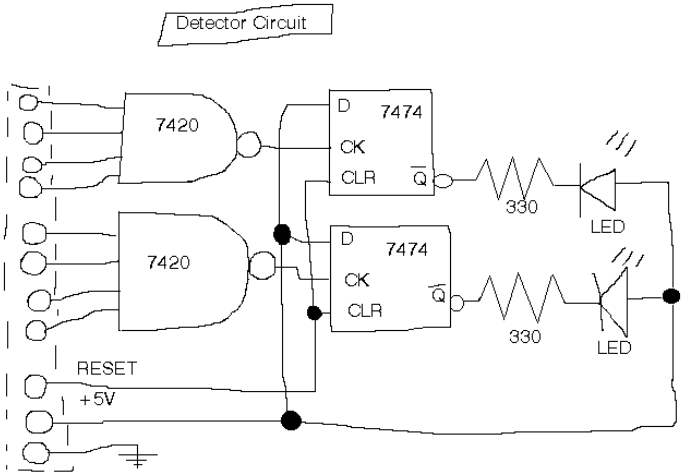
Design Format Dialog-Box

command. Now, from the list, select sheet number two. Sheet two should now be shown on the display.
This completes a summary of how to start **LOGIC CREATOR**, make menu selections, set the parameters in a dialog-box, load an existing schematic, view the schematic, and control the reference grid. Use the G hot-key and turn the grid back on.

CREATING A NEW SCHEMATIC

Our assignment is to create the schematic drawing for a small board design entirely from scratch. Once created, it will be checked for errors, with assistance from the program, and various reports about the design will be generated. This circuit is intentionally simple and small; however, it demonstrates the basic steps by which any circuit, regardless of size, is created. A rough, hand-drawn sketch of the circuit is given in the figure that follows. This is the way actual schematics normally start out, although an experienced designer may use LOGIC CREATOR to directly enter designs without the use of sketches.

From our rough sketch, we have all the information we need to begin entering our schematic. From the FILE menu, select the NEW command. If you have made any changes to the sample schematic we have been viewing, you will be prompted whether or not you want to save these changes. If prompted, type N for no.



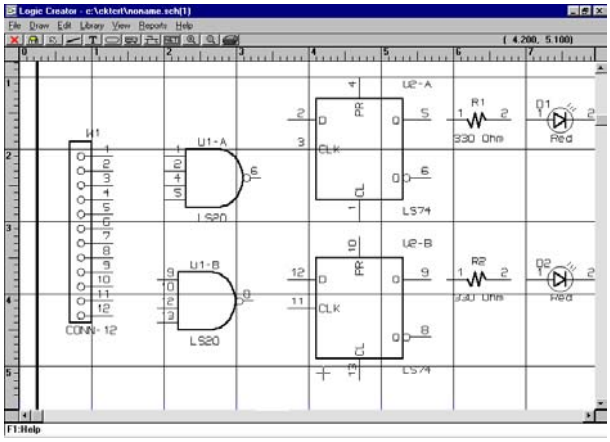
ROUGH SKETCH OF SCHEMATIC

We are presented with the FORMAT dialog-box. For the overall title, enter "Tutorial Demo Lesson"; for the title of sheet number 1, enter "Detector Circuit". The default settings for the remainder of the parameters will do. Anyway, we can use the EDIT/FORMAT command later if we want to change any of these values. Select the **OK** button to accept the settings. We now have a blank sheet, sheet number one, to work on.

PLACING THE PARTS

We are now ready to place the parts of our circuit on this sheet. Pull-down the **LIBRARY** menu and select the IC's library entry. We will first place the four-input NAND gates on the sheet. From the list of parts that is displayed, locate and select the **7420** part name. The outline of this part is now attached to our cursor. Move the cursor to coordinate **(2.0,1.5)**. **ENTER** to place the symbol at this point. This particular part has two gates in it. Notice that the pin numbers for the gate corresponding to the 1st section of this part have been placed. Also, the reference number has automatically been filled in as U1-A.

Now, while we still have the symbol for this part on our cursor, let's add the other gate. Move the cursor to (2.0,2.8) and **ENTER**. Notice that this gate has pin numbers corresponding to the 2nd gate in the part, and the reference number is U1-B.



PARTS PLACED ON SCHEMATIC

Now we will add the flip-flops. From the library menu, select the **IC's** library. This time, locate and select the **7474** part. Move to **(4.4,1.4)** and **ENTER** to place the first flip-flop. This part has been labeled as *U2-A*. Now, move to **(4.4,3.3)** and place the other half of the flip-flop. Isn't this much easier than drawing the parts every time?

Now, we will add the 330 ohm resistors to the schematic. Select the **RESISTORS** library from the library menu. Select the resistor part **R330**. Place one resistor at coordinate **(6.2,2.0)** and a second one at **(6.2,3.9)**. They will be labeled as *R1* and *R2*.

Now, we will add the light-emitting diodes to the schematic. In this case, we will intentionally make a mistake, and recover from it. Select the part LED-RED from the MISC library. Place the first LED at **(7.4,2.0)** and the second one at **(7.4,3.9)**. Press the **ESC** key to end the part placement command. Look at the orientation of the LEDs on the screen. You will notice that the cathode end of the LED is on the incorrect (right) side. This is not what we want! If we had noticed this while the part was on the cursor, we could have used the **F9** or **F10** keys to set the desired mirroring and orientation of the symbol. Since our diodes are already placed, we will update them with new values.

Move the cursor to any point within the area of one of the diodes and press **ENTER** twice. This will select the diode, which is now displayed in the select color (*usually red*). Once selected, an item may be moved, deleted, or its parameter settings changed. Press **U** to select the **UPDATE** command. This command may also be selected by pressing the **ENTER** key again, without moving the cursor.

Change the value for **MIRROR** from **NONE** to **RIGHT-LEFT**. This is done by moving the selector to this field and pressing **ENTER**. Select the right-to-left from the pop-up menu and select the **OK** button. Notice that the diode pin numbers and symbol shape have flipped ends. This method may also be used to change the rotation of the part symbol, change the reference value, and to select alternate part symbols.

Now, using the technique just learned, change the mirror parameter for the other diode so that it looks like this one.

Next, we will complete the placement of our part symbols by adding the connector. From the library menu, use the **MISC** library to select the **CONN-12** connector part. Place the connector at location **(0.4,1.8)**.

WIRING THE CIRCUIT

Since all of our parts are placed, we are now ready to wire them together. First, let's wire the connection from U1 pin 6 to U2 pin 3. From the DRAW menu, select the WIRE command, or just type W. Move the cursor to (3.4,2.1). **ENTER** to begin the wire line. Move to (4.4,2.1) and **ENTER** to draw a segment of the wire. Press **ESC** to stop drawing this net's wires.

Now, let's make a similar connection from U1 pin 8 to U2 pin 11. Move to (3.4,3.4) and **ENTER** to start the net line. Move to **(3.5,3.4)**, **ENTER**; to **(3.5,4.0)**, **ENTER**; to **(4.4,4.0**), **ENTER**, and **ESC**.

Got the hang of it? In actual practice, the wiring process is even easier because you are drawing the lines by looking at the points that need connecting on the screen instead of working from a list of points as we are doing here. If you add a wire in the wrong place, simply press the F2 key to undo it, and then draw the right wire.

Now, connect the following points with wires. Remember to use **ESC** to stop drawing one net before starting the next net wire.

U2-6, R1-1	(6.0,2.3) to (6.2,2.3).
D1-2, R1-2	(7.2,2.3) to (7.4,2.3).
U2-8, R2-1	(6.0,4.2) to (6.2,4.2).
D2-2, R2-2	(7.2,4.2) to (7.4,4.2).
D1-1, D2-1	(8.4,2.3) to (8.6,2.3) to (8.6,4.2) to (8.3,4.2).

Now, move the cursor to (8.6,4.2). At this point, we want to add a connect dot and continue the net. **ENTER** to start the line. Now, without moving the cursor, **ENTER** again. This will add a connect dot. From this point, move to (8.5,5.1) and **ENTER**. Continue this net line to (3.8,5.1), **ENTER**; then to (3.8,3.8), **ENTER**, and into U2 pin 12 at (4.4,3.8). Now **ESC** and move back to (3.8,3.8). Add a connect dot here, and continue the net to (3.8,1.9) and end at U2 pin 2 (4.4,1.9). Move to (3.8,5.1) and add a connect dot. Continue the net to (1.6,5.1), to (1.6,4.1), and into pin 10 of the connector at (1.0,4.1).

By referring to the rough sketch of our circuit, see if you can complete the wiring of this circuit.

ADDING NET NAMES

Now, we will add the net names for the nets that have names. Select the NET NAME command from the DRAW menu. Move to location (2.1,5.1) and **ENTER**. In the dialog-box, set the net name to "+5V" and the location to ABOVE. Select **OK** to complete the adding of the net name.

Pin 12 of the connector is the ground net. We will show this by using the ground symbol. First, use the DRAW/WIRE command to extend the net line from connector pin 12 to (1.2,4.5) and down to (1.2,5.0). Now, select the DRAW/NET NAME command again. Move to location (1.2,5.0) and **ENTER**. This time, select the button in the dialog-box labeled GND. Internally, and on reports, the name of this net will be referred to as GND; however, on the schematic, the graphic symbol for standard ground is shown. This same technique may be used to show analog-ground and chassis-ground. Select **OK** to complete the command.

UNUSED PINS

The unused pins of all components on the schematic should be labeled with a net name of "NC" (no connect). Move to location (1.0,4.3) and add a net name of "NC" with a position value of RIGHT. Notice how the arrow for a RIGHT type net name label is automatically displayed for you.

Add "NC" net names for the following locations:

U2-4 (top)
U2-5 (right)
U2-10 (top)
U2-9 (right)

TEXT/LINES

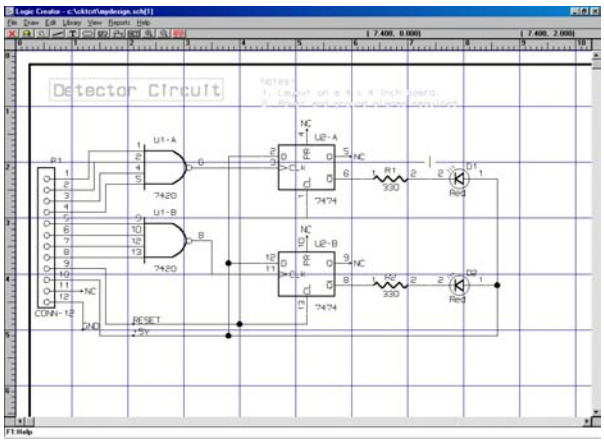
Now, we will complete our schematic by adding some notational text. Select the DRAW/TEXT command. Move to (4.4,0.6) and **ENTER**. Set the text string to "Notes:" and select **OK**. In the same manner, move to location (4.4,0.8) and **ENTER** "1. Layout on a 4 x 4 inch board.", and at (4.4,1.0) **ENTER** "2. Power and ground planes required.". Now, move to (0.7,0.8) and **ENTER**. Enter "Detector Circuit" for the text, but this time select BOLD and set the text size to 0.188.

Now, select the DRAW/LINE command and, just like the wires were drawn, draw a box around the text that was just added.

Please note the difference between wires and lines. Wires are used to electrically connect components together lines are for graphic effect only.

SAVING THE SCHEMATIC

This completes the tutorial schematic. Let's save it to disk. Select the FILE/SAVE AS command. Enter "mydesign" for the name and select **OK**. The evaluation version of the program will not actually save the drawing.



SAVE SCHEMATIC FILE

CHECKING THE DESIGN

To check our layout work, select the DESIGN CHECK command from the REPORTS menu. If you have followed all the instructions carefully, you should get a report of "No errors detected." Otherwise, the errors will be pointed out to you on the screen.

If no errors were detected, just to show you how the error reporting works, move to location (6.1,1.9) and select the "NC" net name label. Press the Del key to delete the

label. Run the check again. This time an error should be detected at this location. You can use the F2 key, UNDO, to repair your schematic.

If errors were detected, see if you can correct them, then run the check again.

GENERATING REPORTS

Now that we have checked the design for errors, let's generate the formal reports for this design. From the REPORTS menu, select the FORMAL command. Check the PIN LIST, NET LIST, and BILL OF MATERIAL reports. Select **OK** to start making the reports. The reports are written to files on the disk. The pin list report will be in "MYDESIGN.PIN", the net list will be in "MYDESIGN.NET", and the bill of materials will be in "MYDESIGN.BOM". The files are in ASCII format, and may be printed on your.

EXITING THE PROGRAM

We are finished with this schematic for now. Select **EXIT** command on the FILE menu or just type ALT+X to exit the program.

SUMMARY OF SCHEMATIC ENTRY

Let us emphasize that the entry of the tutorial schematic may seem a little tedious. This is because we have stepped you through each of the steps at a very detailed level. In actual practice, you are choosing your own locations for parts, paths for the nets, and the order in which you wish to layout the circuit. You will have a much greater sense of freedom. Just follow the principles you have learned here and it will flow very smoothly. At this point, you may want to re-start the program and "just play around" for a while.

BOARD CREATOR: PC BOARD TUTORIAL

This tutorial leads you, step by step, through the creation of a simple PC board.

STARTING BOARD CREATOR

To run the BOARD CREATOR program, double click on the CIRCUIT CREATOR program group (if it has been minimized) and then double click on the BOARD CREATOR icon.

LOADING A BOARD

To get a feel for the system, we will first load a sample PC board file. We will then learn how to navigate about the board on the display. First, an explanation of how to select commands from the pull-down menus and dialog-boxes will be given.

If you are using the keyboard a pull-down menu displayed (pulled-down) by holding the Alt key down and pressing the key indicated by the underlined character in its name on the Title-Bar. Upper or lower case may be used.

When a pull-down menu is displayed, the ← and → arrow keys may be used to move to the next menu to the right or left. The NUMLCK key must be off. Use the ↑ and ↓ arrow keys to move the menu selector up and down. Move to the **FILE/OPEN** command and select it by pressing the **ENTER** key.

If you are using a mouse, you may pull-down a menu by moving the pointer to the menu name in the title-bar and pressing the left mouse button and releasing it. The menu will appear when the button is released. To select a menu item, move the pointer to the desired item and press the left mouse button and release it. The selection is made when the button is released. For quicker menu selections, move the pointer to the title and press the left mouse button and do not release the button yet. Now, move the pointer to the desired menu item and release the button to make the selection.

After selecting the **FILE/OPEN** command, a dialog-box will appear which is prompting for the name of a PC board file. At this point, we can type the file name of

a file, or select one from the list that is displayed. Use the arrow keys to move the selector to the file in the list named "DEMO.PCB" and press **ENTER**. This will copy the file name from the list into the file field of the dialog-box. Now press **ENTER** again to directly accept this name, or press the HOME key to move the selector to the **OK** button, then press **ENTER**.

When a dialog-box is displayed, clicking the left mouse button on the **OK** button, or pressing the ALT-O key combination will immediately accept the values in the dialog-box. Pressing the Home key will move the selector to the **OK** button.

Clicking on the **CANCEL** button, or pressing the ALT-C combination will immediately cancel the dialog-box. Pressing the **END** key or the **ESC** key will move the selector to the cancel button. Once the selector is on the cancel button, pressing **ESC** again will cancel the dialog-box. Any changes made to the values shown in a dialog-box do not actually take effect until the **OK** button is pressed. If the cancel button is pressed, the changes made to values in the dialog-box are ignored.

For a file-selection or part-selection dialog-box, pressing **ENTER** twice while on the same name in the list implies that this is the name you want to accept immediately.

GETTING HELP

Press the **F1** key to pop-up a window of help information. Notice that the help information is specific to the current command being performed. The **UP** and **DOWN** buttons may be used to page the help text up and down in the window. You may also use the **NEXT, PREV,** or GENERAL buttons to view other help topics.

Now, select the **EXIT** button to close the help window. Help is always available with the **F1** key.

NAVIGATION

We will now learn how to change the view of a PC board displayed on the screen. Pull-down the VIEW menu. Now, move down and select the **ZOOM OUT** command. The display will be redrawn at a smaller scale. You will notice that the **ZOOM OUT**

menu item has CTRL+PGDN listed in the right margin next to it. This indicates that this command has a Hot+Key. This key combination may be used when the menu is not displayed to directly execute this command. Let's try it. Now, hold the CTRL key down and press the PGDN key to zoom-out. Once you are more familiar with the program, using hot-keys will improve your productivity, especially if you do not have a mouse.

Zoom back to the original scale by pressing the CTRL+PGUP key twice.

It should be noted that if you pick the pan and zoom commands directly, you do not need to wait for the redraw to complete before entering another pan or zoom command. In this case, the current redraw will be stopped, and the next redraw will begin directly.

The current coordinate of the cursor is displayed in the upper-right corner of the screen. Use the cursor keys to move the cursor to coordinate (2.000,1.000). Follow the track of the cursor on the screen, and watch the value in the upper-right corner change. Notice that the active cursor (the little cross) is not always the same as the mouse pointer position.

Holding the SHIFT key down and using the cursor keys will move the cursor one inch at a time. Hold the SHIFT key down and use the arrow key to move quickly to coordinate (0.000,0.000), the upper-left corner of the sheet.

The display may be panned using the keyboard by bumping into one of the edit window edges. First, zoom in with the CTRL+PGUP key. Now, let's pan right. Move the cursor with the SHIFT □ key until it bumps the right edge of the window. The portion of the board that was out of view to the right of the screen is now displayed.

With a mouse, the view may be panned by using the vertical slide-bar at the right, or the horizontal slide-bar at the bottom.

Display the upper-left corner of the board at normal scale by pressing CTRL+HOME.

We will now explore how to select the layers of information that are displayed on the screen. Execute the LAYER/MASKS command on the VIEW menu.

You will notice that each layer of the board is listed along with its type and current conductor color. If there is not a check-mark by a layer, then items on that layer will not be displayed. The actual colors and number of colors available will depend on your particular hardware. The display state of each layer is changed by moving the focus indicator to that layer and pressing **ENTER** to turn it on or off.

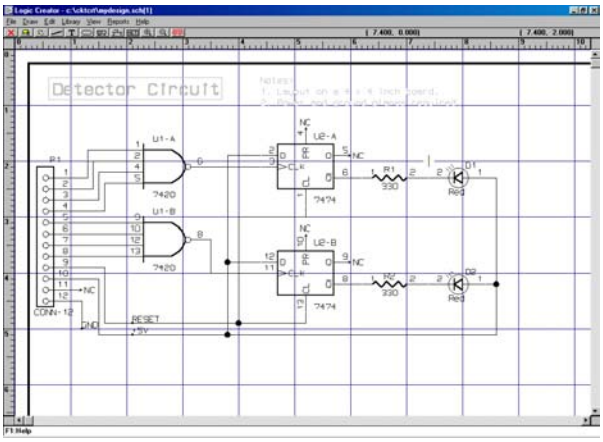
Also, this dialog-box allows you to select which masks will be displayed.

Turn off all layers except layer 1 and all masks except the silk-screen mask. Press the **OK** button. Notice that only items which are on the silk-screen mask for layer 1 of the board are displayed.

Now, let's display just the solder-side layer of the board. Select the VIEW/LAYERS/MASKS command again. Enable the "all" layer, and layer 4 (solder-side, last layer). Enable all the masks. Press **OK**. Notice that only the items that belong to the solder-side, or go through the board, are displayed.

CURSOR GRID

The cursor grid is set to 50 mils by default. You have probably noticed that the cursor steps 50 units each time the arrow keys are pressed. Select the GRID/CURSOR command on the VIEW menu. Set the cursor grid X and Y grid steps to "0.025" and press **OK**. This sets both the cursor grid step values to 25 mils. Move the cursor about the screen with the arrow keys. Notice that the cursor now moves in steps of 25 mils.



VIWE CURSOR GRID

This completes a summary of how to start the BOARD CREATOR editor, load an existing board, view the board, choose which layers and masks are to be displayed, and how to set the cursor grid step values.

How to Design PCB Using PTH Components Tutorial

Our assignment is to create a PC board which implements the design shown in the following figure. Once created, it will be checked for errors, with assistance from the program. Masks and reports about the design will be generated. This circuit is intentionally simple and small; however, it demonstrates the basic steps by which any circuit, regardless of size, is created. This circuit corresponds to the tutorial lesson used in the companion LOGIC CREATOR schematic system.

The overall board specification is as follows:

1. Board size is 3.00" x 2.00".
2. Board has 2 signal routing layers, route 10 mil lines.
3. Power and ground provided by power planes.
4. Conductor masks required.
5. Solder mask required.
6. Silk-screen required.
7. Drill Drawing required.

Bill of Material:

Count	Part-Name	Description
1	7420	Dual 4-Input Nand Gates
1	7474	Dual D-Type Flip-Flop
1	CONN-12	12 Pin Connector
2	LED-RED	Light Emitting Diode, Red

```
       2     R330       330 Ohm Resistor
```

Since this board corresponds to a schematic drawn with the LOGIC CREATOR schematic system, we will use the "update-list" produced by LOGIC CREATOR to create out initial PC board. We will begin the new board by selecting the CREATE FROM LOGIC CREATOR command on the I/O menu. Since we already have a board loaded, we will be questioned as to whether or not we wish to save the changes made to the current board. Answer NO when prompted.

The following is the complete LOGIC CREATOR update-list for the "MYDESIGN" tutorial lesson. It defines each part, part-footprint, and net assignment for every pin on the board. This is all the information that is required to create the initial PC board for a design.

Board Creator Update List:

```
UPDATE_LISTv

PART 7420 U1 DIP-14
PAD U1 1 #0007
PAD U1 2 #0008
PAD U1 3 NC
PAD U1 4 #0009
PAD U1 5 #0010
PAD U1 6 #0001
PAD U1 7 GND
PAD U1 8 #0002
PAD U1 9 #0011
PAD U1 10 #0012
PAD U1 11 NC
PAD U1 12 #0013
PAD U1 13 #0014
PAD U1 14 +5V
PART 7474 U2 DIP-14
PAD U2 1 RESET
PAD U2 2 +5V
PAD U2 3 #0001
PAD U2 4 NC
PAD U2 5 NC
PAD U2 6 #0003
PAD U2 7 GND
PAD U2 8 #0005
PAD U2 9 NC
PAD U2 10 NC
PAD U2 11 #0002
PAD U2 12 +5V
PAD U2 13 RESET
PAD U2 14 +5V
PART R330 R1
DISC-2
PAD R1 1 #0003
PAD R1 2 #0004
PART R330 R2
DISC-2
PAD R2 1 #0005
PAD R2 2 #0006
PART LED-RED D1
DISC-2
PAD D1 1 +5V
PAD D1 2 #0004
PART LED-RED D2
DISC-2
PAD D2 1 +5V
PAD D2 2 #0006
PART CONN-12 W1
CONN12
PAD W1 1 #0007
PAD W1 2 #0008
PAD W1 3 #0009
PAD W1 4 #0010
PAD W1 5 #0011
PAD W1 6 #0012
PAD W1 7 #0013
PAD W1 8 #0014
PAD W1 9 RESET
PAD W1 10 +5V
PAD W1 11 NC
PAD W1 12 GND
END_UPDATE_LIST
```

We are now prompted for the name of the update-list file to be used. Select the "MYDESIGN.UPD" file from the list. BOARD CREATOR will now read the part and

net assignment information from the update-list file and create a board with these parts on it.

COMPONENT PLACEMENT

We now have a PC board with all the required component parts. The next step is to move each part to its actual location on the board, and then we will adjust the board to its true size and number of layers. Although moving the parts is a manual process, the program will assist us. It will show us how the parts connect to one another. To do this we will need a routing schedule.

Run the CHECK/SCHEDULE command from the REPORTS menu. Use a default routing line width of 0.010" (10 mils). This will generate a routing schedule for us. The schedule "rats-nest" lines are visible on the screen when the schedule mask is enabled, or when a component is selected. Since these lines show us exactly the pin-to-pin connections that must be routed, we will want to place the parts so the length of these lines are as short as is reasonable.

Before we begin placing parts, we will setup the select options for doing component placement. Execute the SELECT command on the EDIT menu and press the O key to pop-up the select command options. Set the select layer to ALL. Enable COMPONENT and TOUCHING modes. Enable all the other check-boxes except the SCHEDULE mask. By not enabling the schedule mask for selection, we will not accidentally select schedule lines that are "just passing through" our select area. Selecting a component will still select the schedule lines that belong to that particular component. Press **OK**.

We are now ready to begin placing the parts. Move the cursor to location (0.000,0.000) and zoom in twice (CTRL+PGUP). This is a good working scale for placing this board.

For this tutorial lesson, we will move all the components by selecting pin one of each component. This is done to establish the coordinate numbers listed in this tutorial. When you are working on your own, you may move a component by selecting anywhere on the component.

Move the cursor to location (0.200,1.300) and press **ENTER** twice, or click with the left mouse button. This will select component U2. Now, press **ENTER**, or press the left mouse button and continue to hold it down. This will "grab" the component for moving. Move the part with the arrow keys, or by moving the mouse. Notice how the schedule lines stretch to show you how this part connects to other parts. Move until you are at location (2.100,0.650). Press **ENTER**, or release the left mouse button, to place the part at this location. If you make a mistake, press F2 (undo), try again.
You do not need to un-select the component, it will be un-selected when the next component is selected. Using the procedure just established, place the following components:

Select U1 at (0.200,0.300). Move it to (1.250,0.650).
Select U2 at (0.200,1.000). Move it to (2.100,0.650).
Select R1 at (0.200,2.300). Move it to (2.100,1.500).
Select R2 at (0.200,2.700). Move it to (1.100,1.500).
Select D2 at (0.200,3.100). Move it to (1.100,1.700).
Select D1 at (0.200,3.500). Move it to (2.100,1.700).

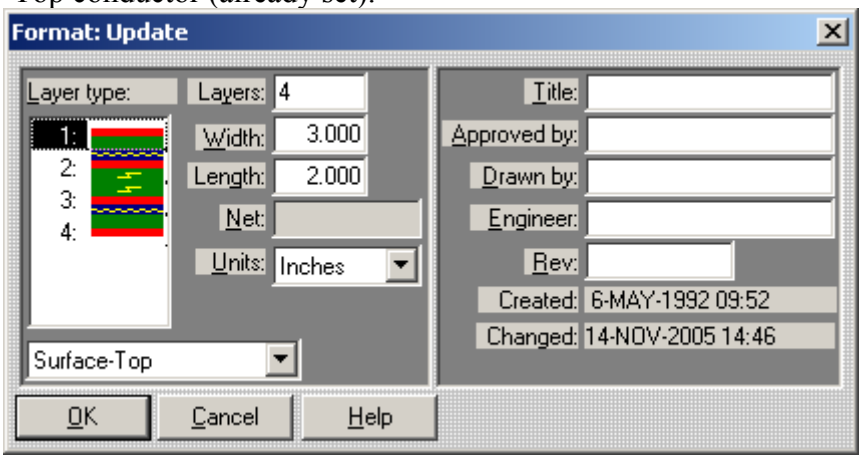
Now we have placed all the components except the edge connector. In addition to moving it, it also needs to be rotated. Select the connector, W1, at (0.300,4.100). Grab the connector and begin to move it just like the other components. While the connector is on the cursor, we may press the O key (or F10) to rotate the part 90 degrees. Press the O key three times to rotate the part a total of 270 degrees. Move to (0.150,0.450) and place the part. Press the Z key to unselect the component.

Now all the components are placed in a reasonable location for routing the electrical connections.

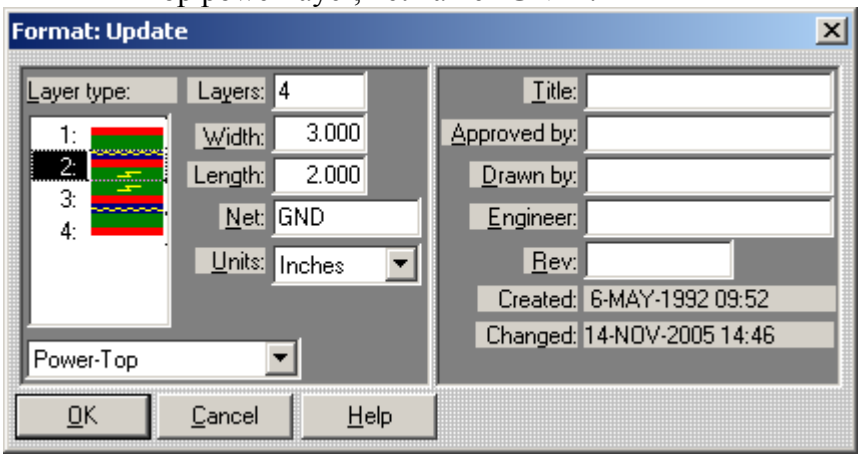
SET TRUE BOARD SIZE/LAYERS

Execute the BOARD FORMAT command on the EDIT menu. Set the number of layers to 4, the width to 3.000", and the length to 2.000". To set a layer type, move the layer focus indicator to the layer number and press **ENTER** to pop-up a menu. Select the layer type from the menu. Set the layer types as follows:

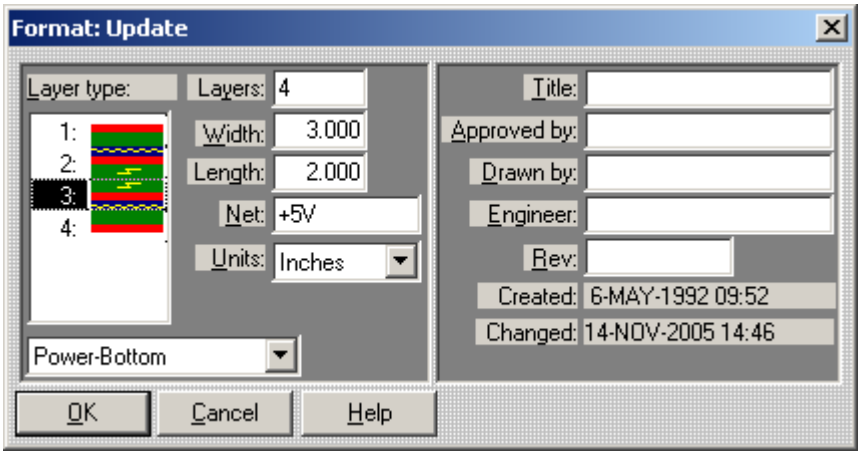
1 - Top conductor (already set).



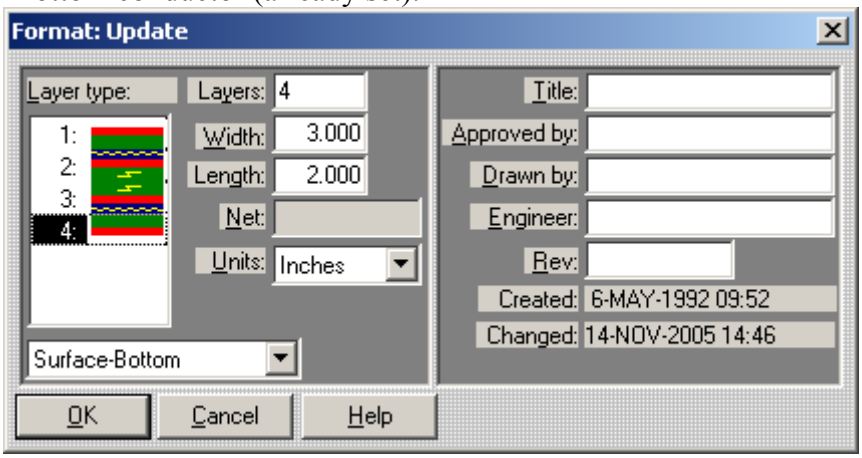
2 - Top power layer, net name "GND".



3 - Bottom power layer, net name "+5V".



4 - Bottom conductor (already set).



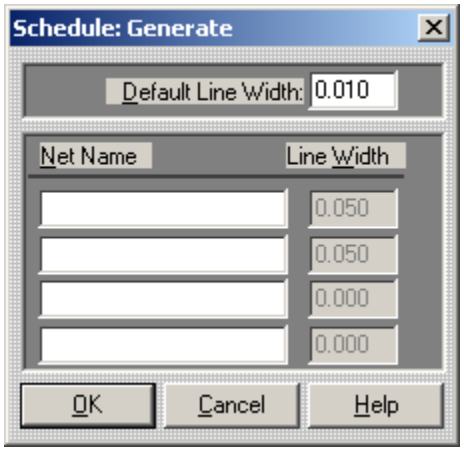
To set the net names, move the layer focus indicator to one of the power layers. Since this is a power layer, the net name field will now become enabled. If you use the keyboard, type ALT+N to pop-up the net name field. Type the net name into the net field. Do this for both of the power layers.

Set the title, engineer, and other "accounting" fields to appropriate values. These values will be printed on reports generated for this board.

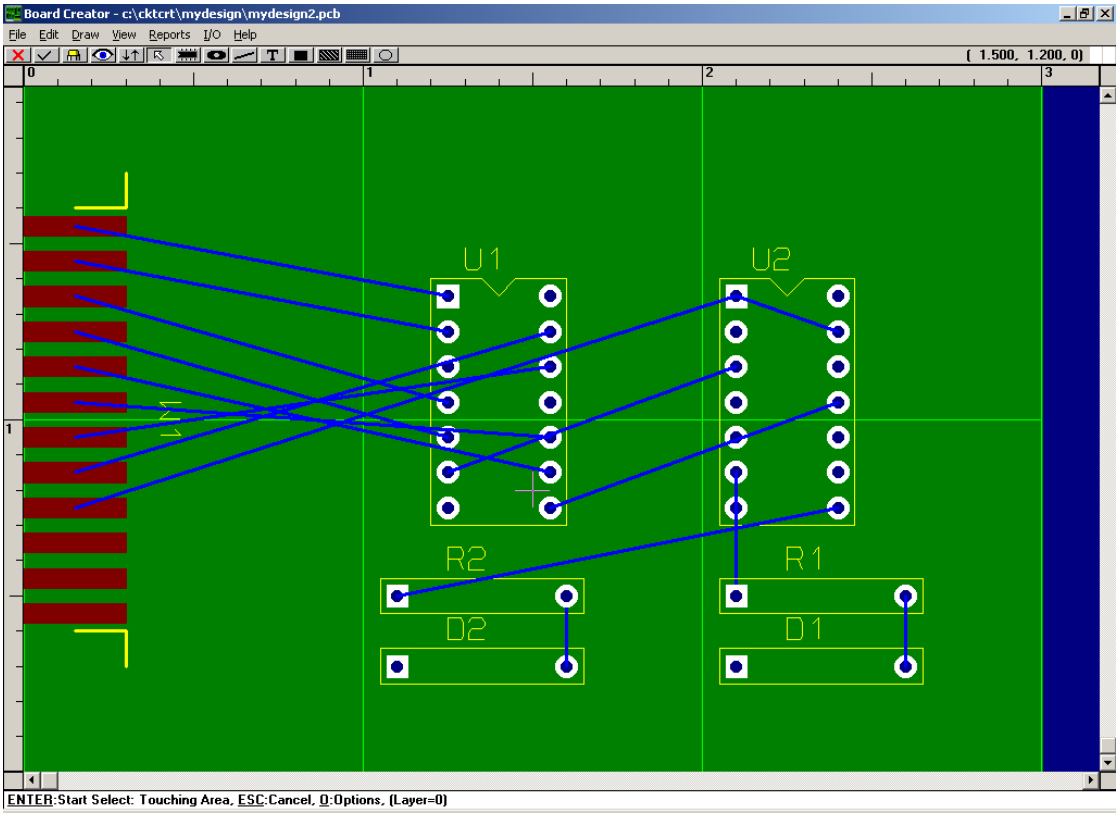
Press **OK** to accept the new board definition.

ROUTING THE BOARD

We are ready to route the conductor lines that make the desired circuit. Before we begin routing the board, we will run the CHECK/SCHEDULE command on the REPORTS menu again. This is done for two reasons. First, we have just done a major placement of the board. With the new component locations, it may be possible to generate a more efficient routing schedule. Also, we have just added a power and ground layer to the board. There is no longer a need to schedule lines for the "GND" and "+5V" nets. Pads assigned to these nets will connect directly to the power and ground layers.



Now we will begin routing the board. Execute the LINE/CURVE command on the DRAW menu. From our specification, the width of the routing lines is 10 mils. The current line width is displayed in the bottom status line. If this is not 0.010", press the O key, or click on the status line with the mouse pointer, to display the line drawing options. Set the line width to 0.010" (10 mils).



For this board, we will the use the component-side for the mostly horizontal routes, and the solder-side for the mostly vertical routes.

From our procedures given in Chapter 2 for routing a board, we will route the shortest lines first. Since we can see from the routing schedule lines shown on the screen which pads need to be connected together, look for some short ones.

Move to coordinate (1.600,1.500) and press **ENTER**, or click with the left mouse button. The schedule line from this pad is now high-lighted, showing us the connection that needs to be made. Since this route will be mostly vertical, we will route this line on layer 4, the solder side of the board. Notice that the status line indicates the current edit layer is layer number 1. Press F9 to "toggle" to routing layer to layer 4. Now move to (1.600,1.700) and press **ENTER**, or click with the left mouse

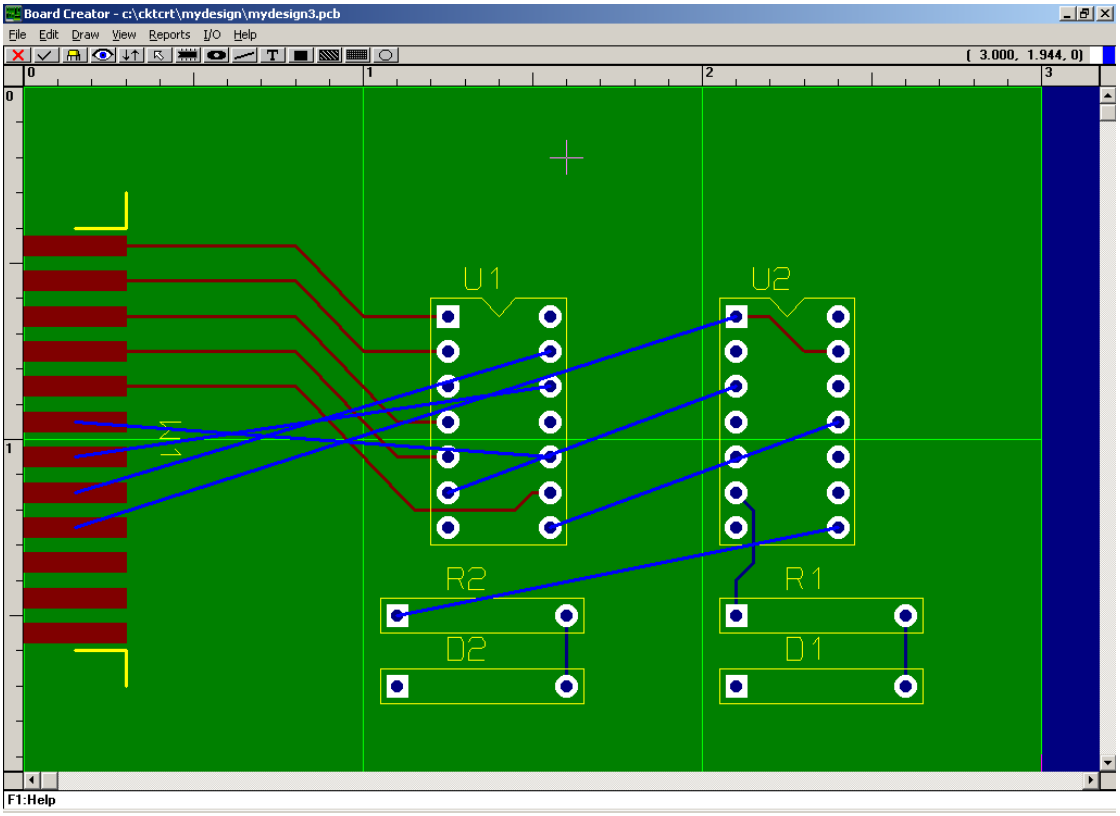
button. This will complete the first route. Notice that the schedule line is now gone, indicating that this route is complete. Press **ESC**, or click the right mouse button to stop drawing this line. If you make a mistake while still drawing a route, press BACKSPACE to "backup" (instead of UNDO).

Now let's do another route. Move to (2.600,1.500), **ENTER**, move to (2.600,1.700), **ENTER**, and **ESC**.

Got the hang of it? Routing lines on your own will actually be much easier. You will work directly from the screen instead of reading coordinates from a book. Now let's route a line with some bends in it. Move to (2.100,1.500), **ENTER**, move to (2.100,1.400), **ENTER**, move to (2.150,1.350), **ENTER**, move to (2.150,1.200), **ENTER**, move to (2.100,1.150), **ENTER**, and **ESC**.

Now, press F9 to toggle back to layer 1. Route lines between the following points. More instruction will be given when we add a via for the first time.

```
Layer 1: (2.100,0.650), (2.200,0.650), (2.300,0.750), (2.400,0.750).
Layer 1: (0.150,0.450), (0.800,0.450), (1.000,0.650), (1.250,0.650).
Layer 1: (0.150,0.550), (0.800,0.550), (1.000,0.750), (1.250,0.750).
Layer 1: (0.150,0.650), (0.800,0.650), (1.100,0.950), (1.250,0.950).
Layer 1: (0.150,0.750), (0.800,0.750), (1.100,1.050), (1.250,1.050).
Layer 1: (0.150,0.850), (0.800,0.850), (1.150,1.200), (1.450,1.200),
         (1.500, 1.150), (1.550, 1.150).
```



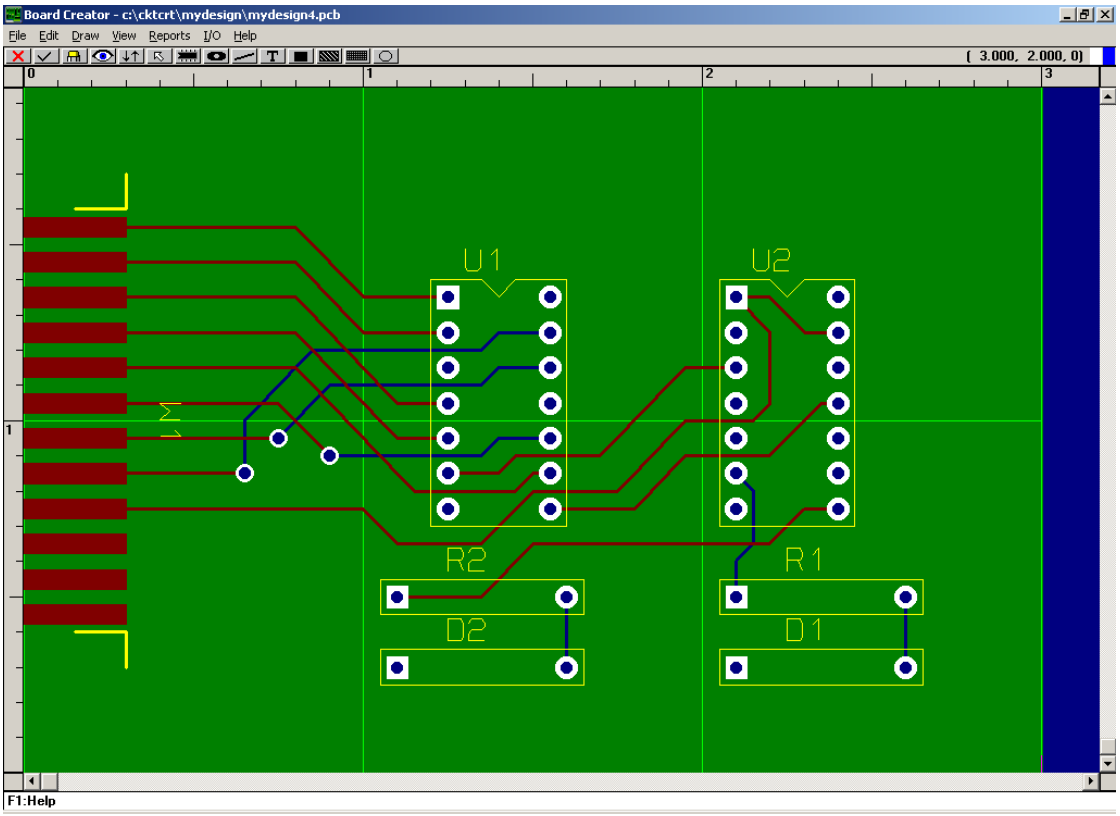
The next line will require a via. Route the line as before, up to the point where a via is noted. At the point just **ENTER** again, or click the left mouse button. This is the same as drawing a line segment with no length. This will add a via at this point, and toggle the routing layer. Continue drawing the remainder of the line on the other layer.

```
Layer 1:  (0.150,0.950), (0.750,0.950), (0.900,1.100), VIA,
          (1.350,1.100), (1.400,1.050), (1.550,1.050).
```

Remember to start each line on the proper layer.

```
Layer 1:  (0.150,1.050), (0.750,1.050), VIA, (0.900,0.900),
          (1.350,0.900), (1.400,0.850), (1.550,0.850).
```

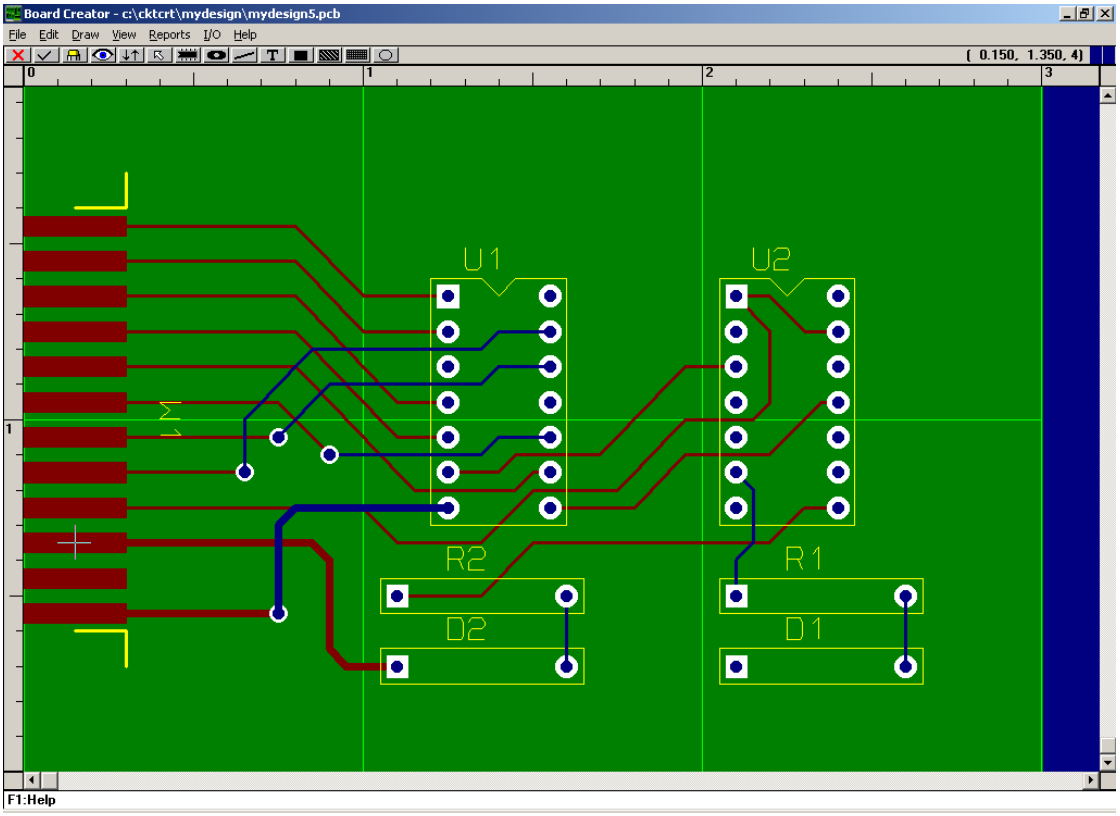
```
Layer 1:  (0.150,1.150), (0.650,1.150), VIA, (0.650,1.000),
          (0.850,0.800), (1.350,0.800), (1.400,0.750), (1.550,0.750).

Layer 1:  (1.550,1.250), (1.800,1.250), (1.950,1.100), (2.200,1.100),
          (2.350,0.950), (2.400,0.950).

Layer 1:  (1.250,1.150), (1.400,1.150), (1.450,1.100), (1.700,1.100),
          (1.950,0.850), (2.100,0.850).

Layer 1:  (1.100,1.500), (1.350,1.500), (1.500,1.350), (2.200,1.350),
          (2.300,1.250), (2.400,1.250).

Layer 1:  (0.150,1.250), (1.000,1.250), (1.100,1.350), (1.350,1.350),
          (1.500,1.200), (1.750,1.200), (1.950,1.000), (2.150,1.000),
          (2.200,0.950), (2.200,0.750), (2.100,0.650).
```



The remaining two routes deserve some special attention. Even though this board has power and ground planes, the planes themselves must receive power from some external connection. For this board, one pin on the edge connector is assigned to net "GND", and another one is assigned to net "+5V". Since these pins do not have holes, they do not connect to the power planes directly, and must be routed to a point that does. Otherwise the board will not be powered.

Power lines are normally routed with a wider than normal line width. Set the line width to 0.025" (use the O key to pop-up the line dialog-box). Now, route the following power lines:

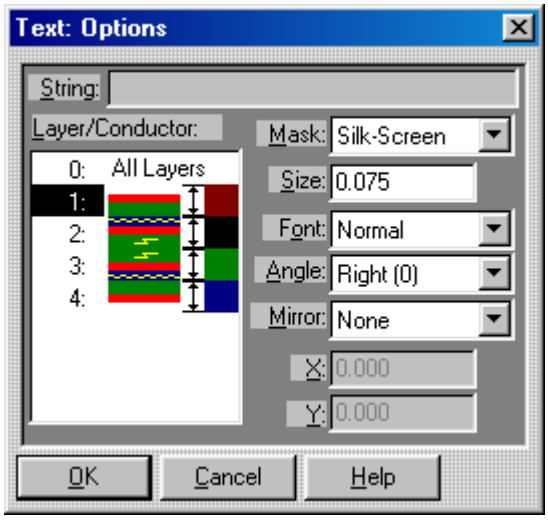
```
Layer 1:  (0.150,1.350), (0.850,1.350), (0.900,1.400), (0.900,1.650),
          (0.950,1.700), (1.100,1.700).

Layer 1:  (0.150,1.550), (0.750,1.550), VIA, (0.750,1.300),
          (0.800,1.250), (1.250,1.250).
```



This completes the routing of the board. Notice that there was never a need to refer directly to the update-list, or any other lists. All the information necessary to place and route the board was presented directly on the screen where you were working.

ADDING TEXT

Next, we will learn how to add notational text strings to the board. Text is often added to directly identify the layer of the mask, the company name, copyright notice, etc. Be careful when adding text to a conductor layer. The text will be etched in copper on the actual board. Do not place the text where it will touch any conductor lines.



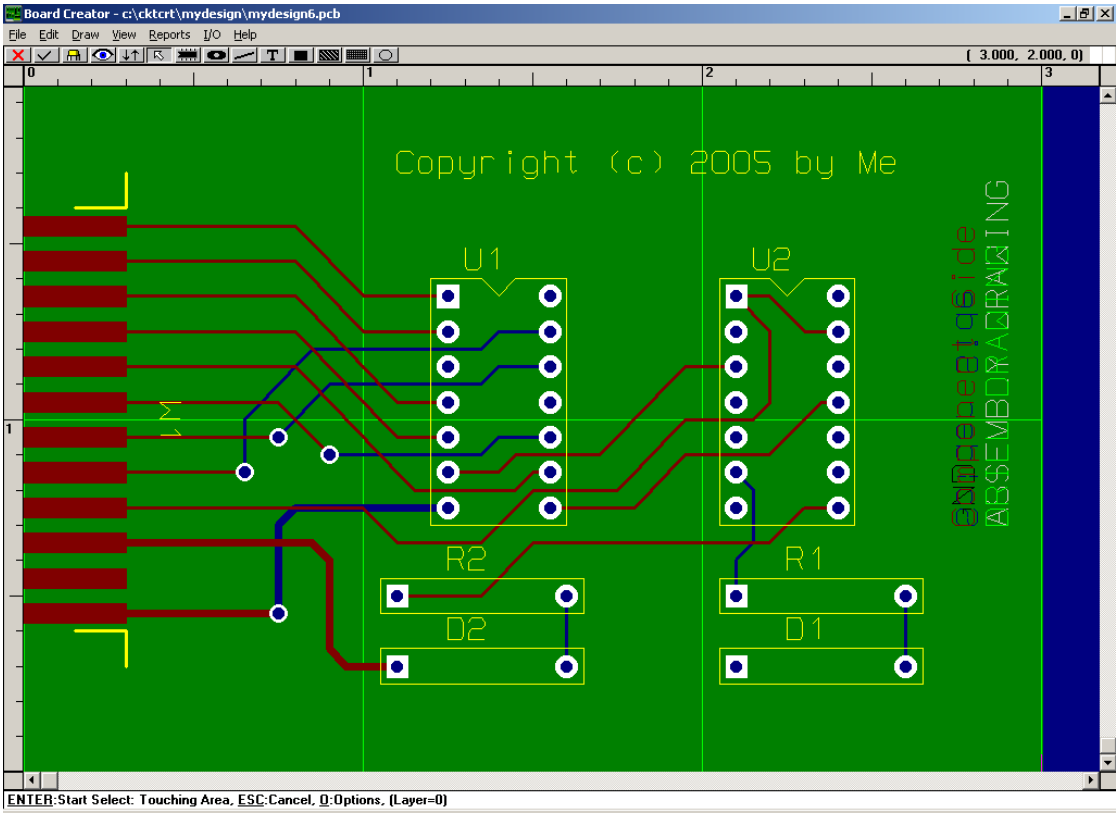
Select the DRAW/TEXT command. Move to (1.000,0.300) and **ENTER** to begin adding a text string. In the dialog-box, enter "Copyright (c) 2005 by Me" for the text, layer 1, mask SILK-SCREEN, size 0.062", angle 0, font NORMAL, and mirroring NONE. Press **OK** to add the text. The text will appear on the board at the location of the cursor and have attributes, which match these settings.

Move to location (2.800,1.300). Press **ENTER** and add the string "Component Side" to the CONDUCTOR mask with an angle of UP. Put "GND" on power layer 2. Notice how the display adjusted to show a power layer. Put "+5V" on power layer 3.

Move to (2.750,1.300) and add "Solder Side" on layer 4 with an angle of DOWN, and mirroring set to RIGHT-TO-LEFT. This will make this text readable when it is viewed from the back-side of the board.

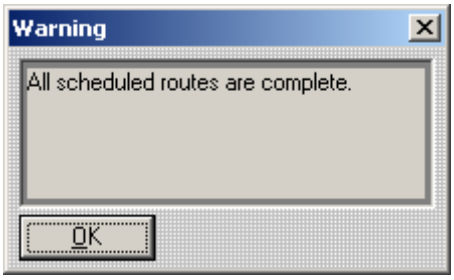
Move to (2.900,1.300) and add the string "ASSEMBLY DRAWING" on the ASSEMBLY DRAWING mask of layer 1 (angle 0, no mirroring). Likewise, add the string "DRILL DRAWING" to the DRILL DRAWING mask of layer 1.

You may add any additional text as desired. Set the angle, size, and mirror factors as desired in the dialog-box.

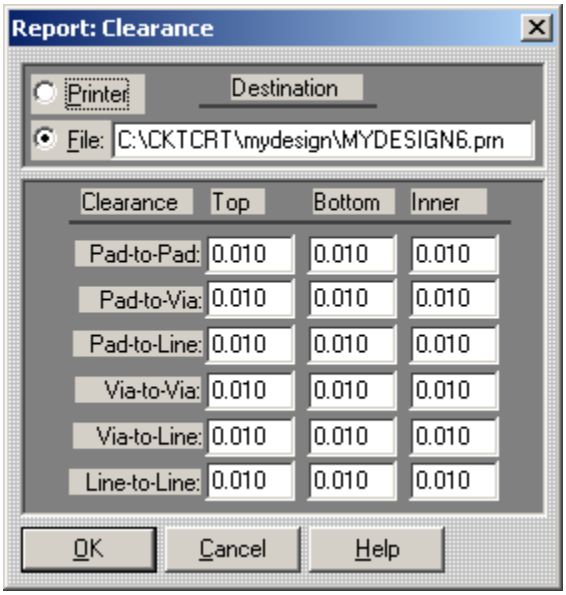


CHECK THE LAYOUT

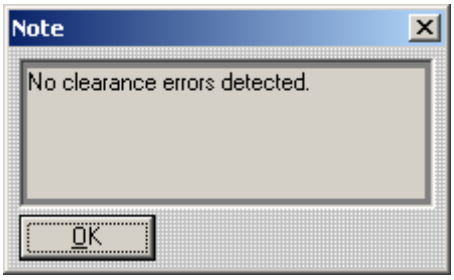
To check our board for routing completeness, run the CHECK-SCHEDULE command from the REPORTS menu. If you have completed the board correctly, a message "All scheduled routes complete" will appear; otherwise, the schedule lines for any uncompleted routes will be generated. If you have routed two nets together, creating a "short", the location of the problem will be displayed directly on the screen.



Also, run the CHECK CLEARANCE command from the REPORT menu. This command will check that all lines, pads, vias, etc. have at least the specified clearance distance from each other. If errors are detected, they will be pointed out directly on the screen.



If any errors are detected, try to correct the problem and re-run the checks until there are no errors.

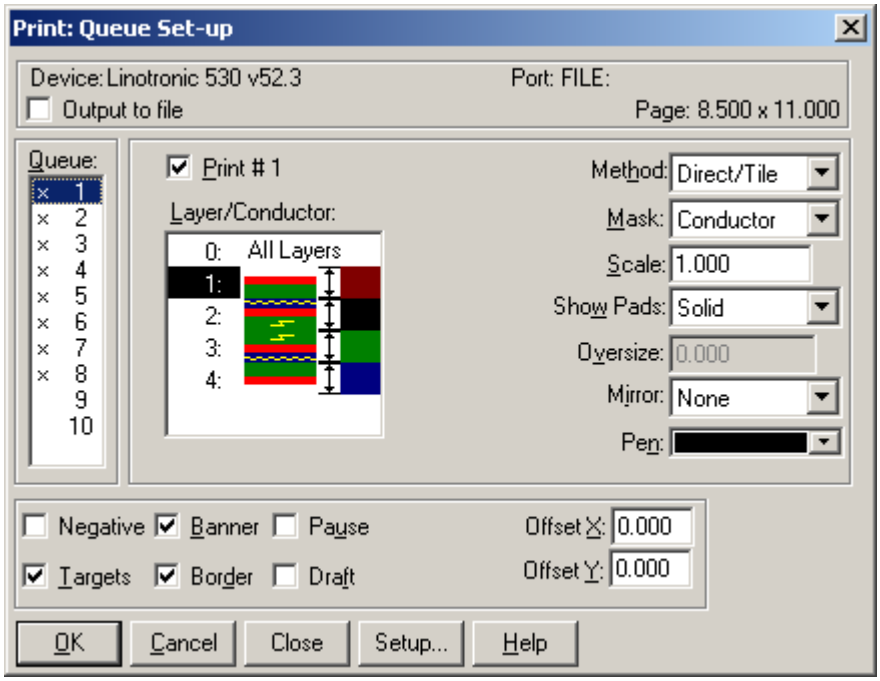


PRINTING THE MASKS

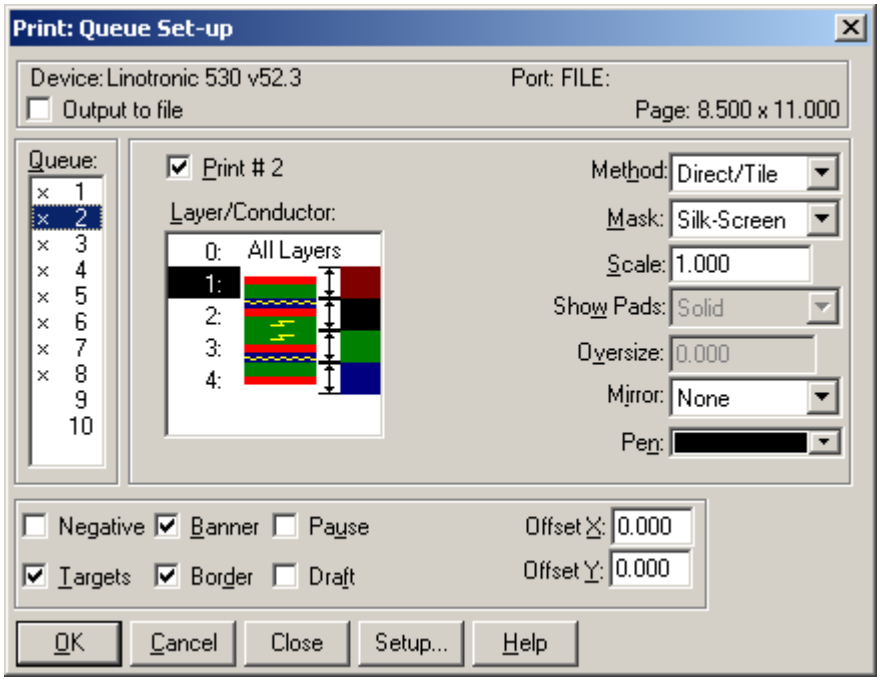
Now that the board is complete, and has passed all the error checks, it is time to print the masks for this board. If you have not yet selected the printer device, use the SET-UP PRINTER command on the FILE menu to select your printer device and set the basic printing options.

Execute the FILE/PRINT command. The dialog-box for printing allows you to queue up to ten masks to be printed. Queue the following masks:

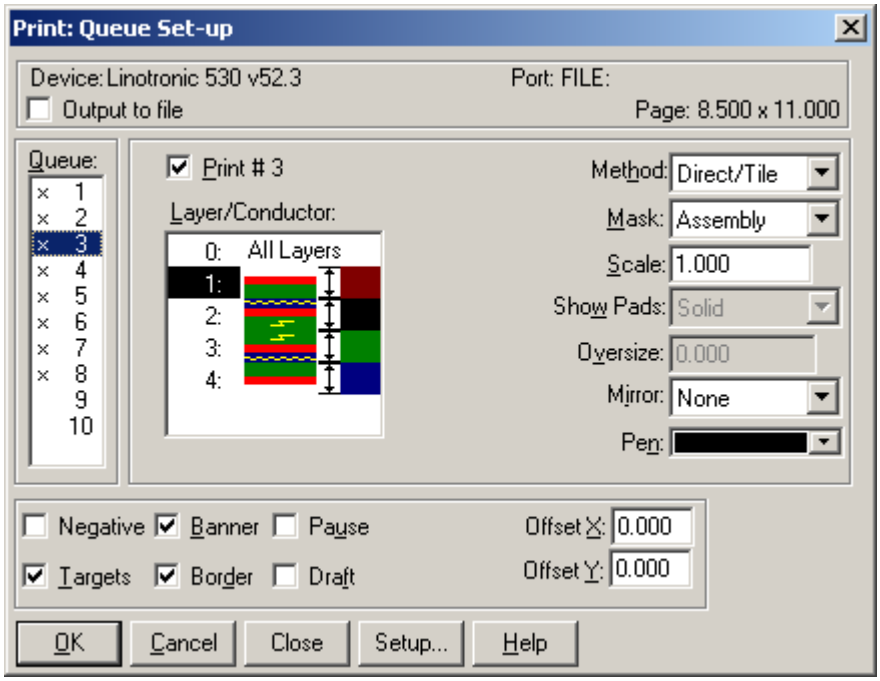
1. Layer 1, Conductor mask. (Top Side Conductor)



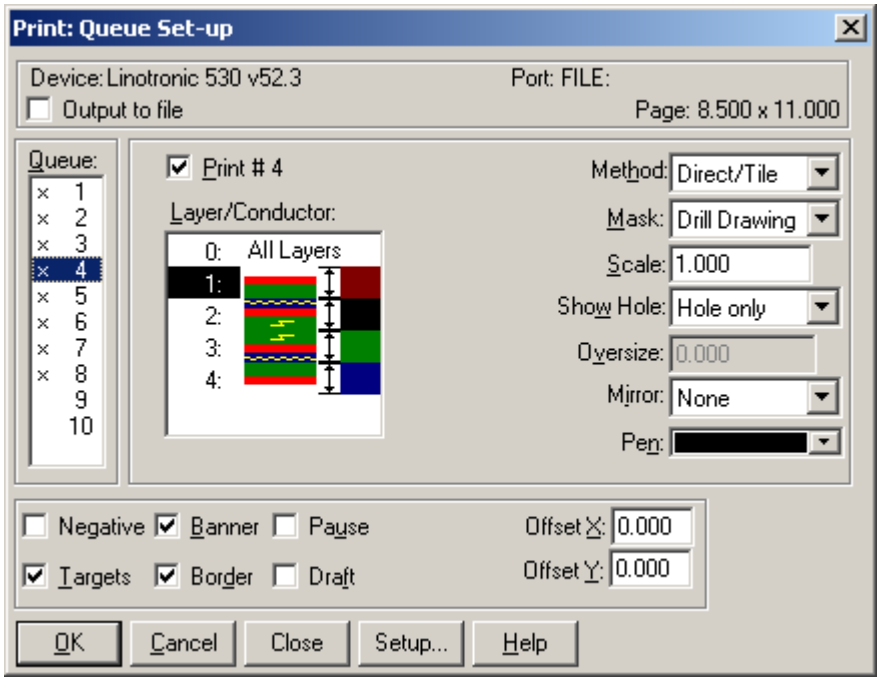
2. Layer 1, Silk-screen mask. (Top Side Silk Screen)



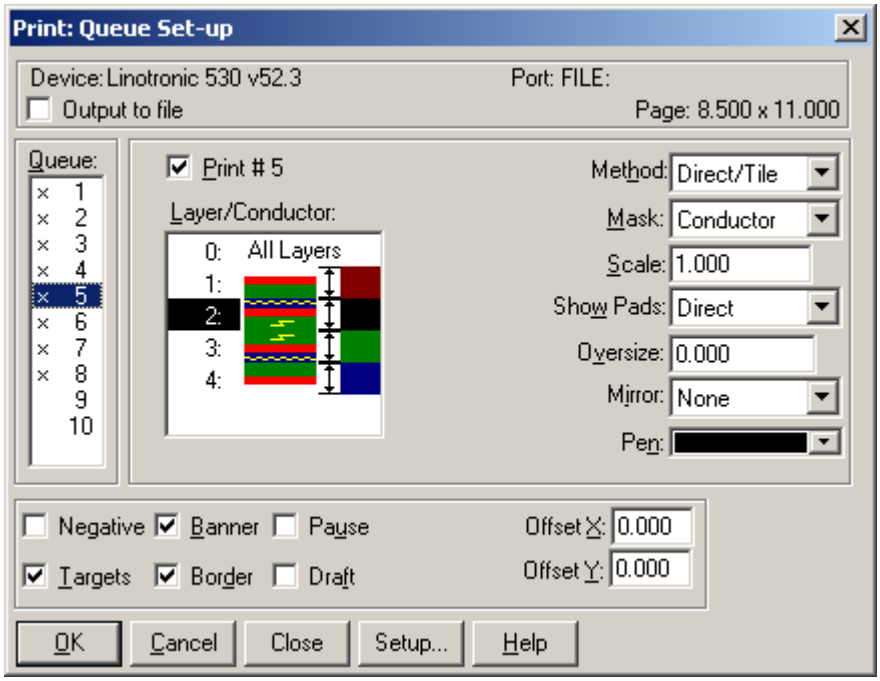
3. Layer 1, Assembly drawing. (Top Side Assembly)



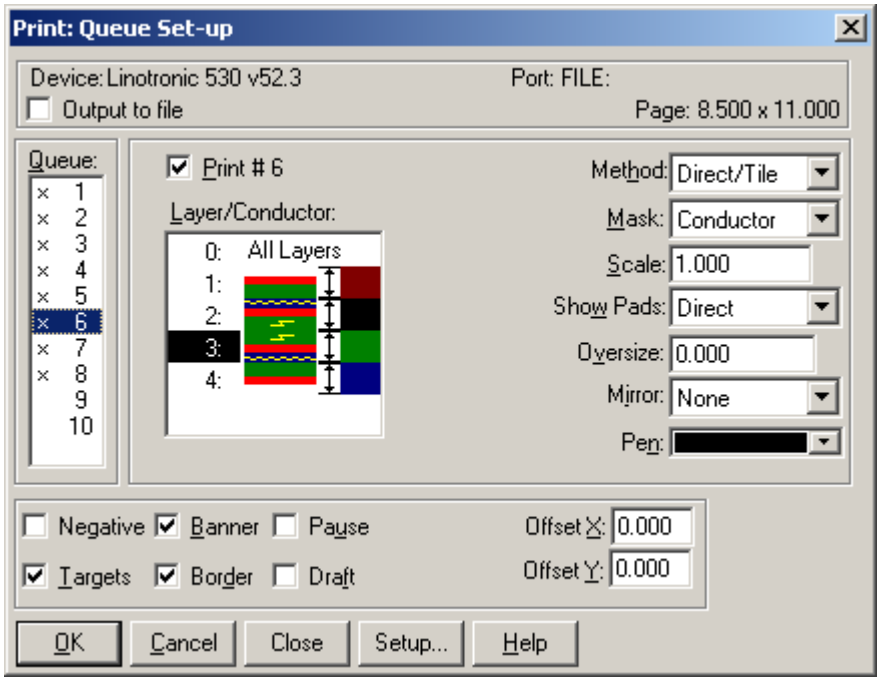
4. Layer 1, Drill Drawing. (Top Side Drill Drawing)



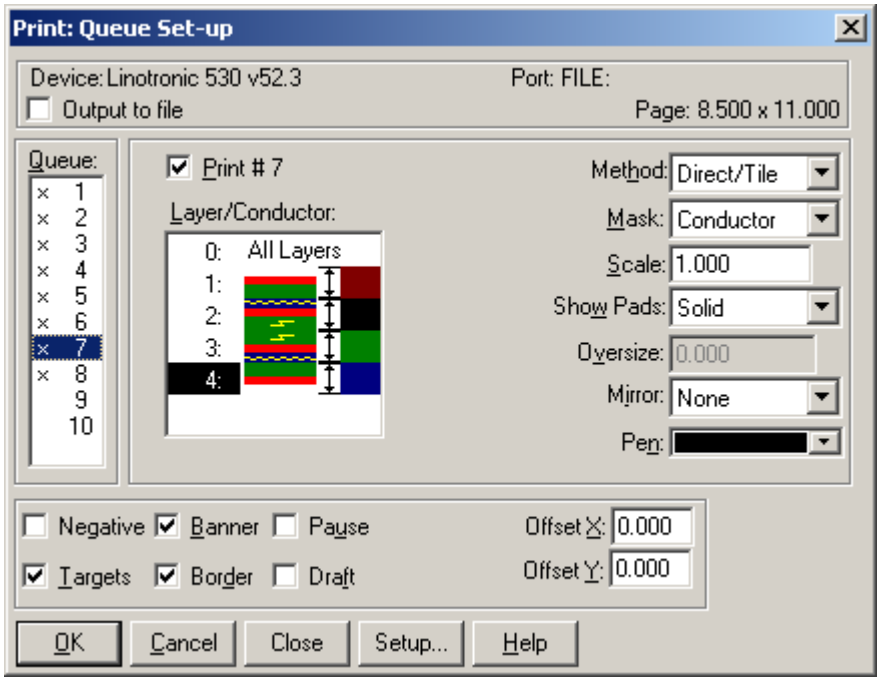
5. Layer 2, Conductor mask. (Inner-Top Side Conductor)



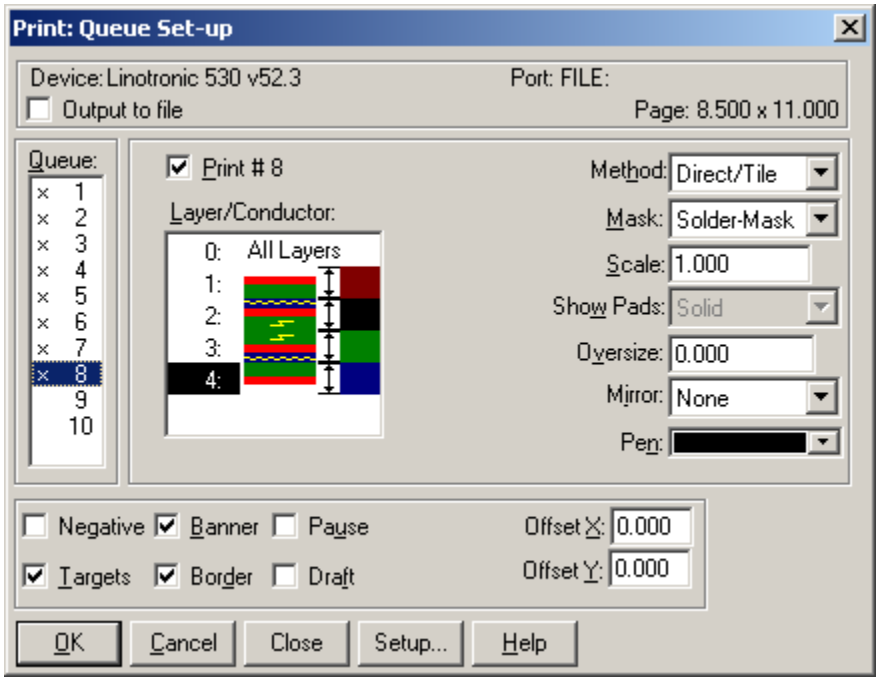
6. Layer 3, Conductor mask. (Inner-Bottom Conductor)



7. Layer 4, Conductor mask. (Bottom Side Conductor)



8. Layer 4, Solder mask. (Bottom Side Solder Mask)

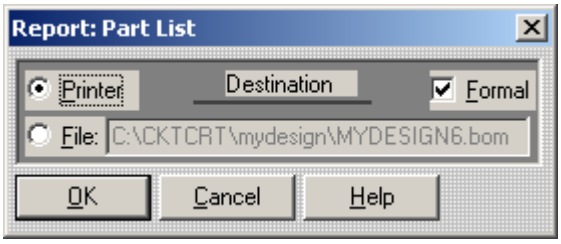


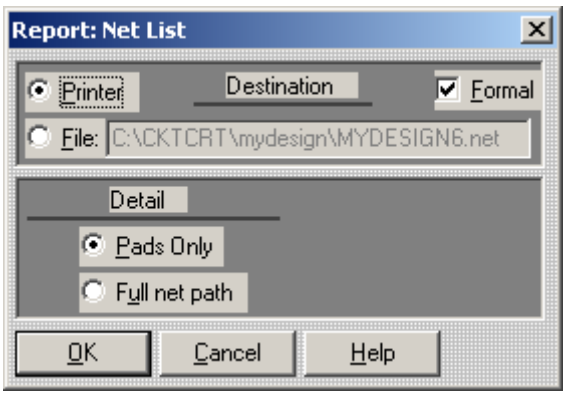
Enable TARGETS, BANNER, and BORDER. Set the method to PANELS to save paper. Set the other options as desired. If your printing device requires you to change paper between each print (as required by some plotters), enable the PAUSE option. Review the settings and make sure a check mark, indicating that the queue slot is enabled for printing, appears next to each of the first seven queue slots. Press **OK** to begin printing the masks.

For most boards, it is necessary to set-up the printing parameters for a board only once. The printing set-up parameters are saved with the board.

REPORTS

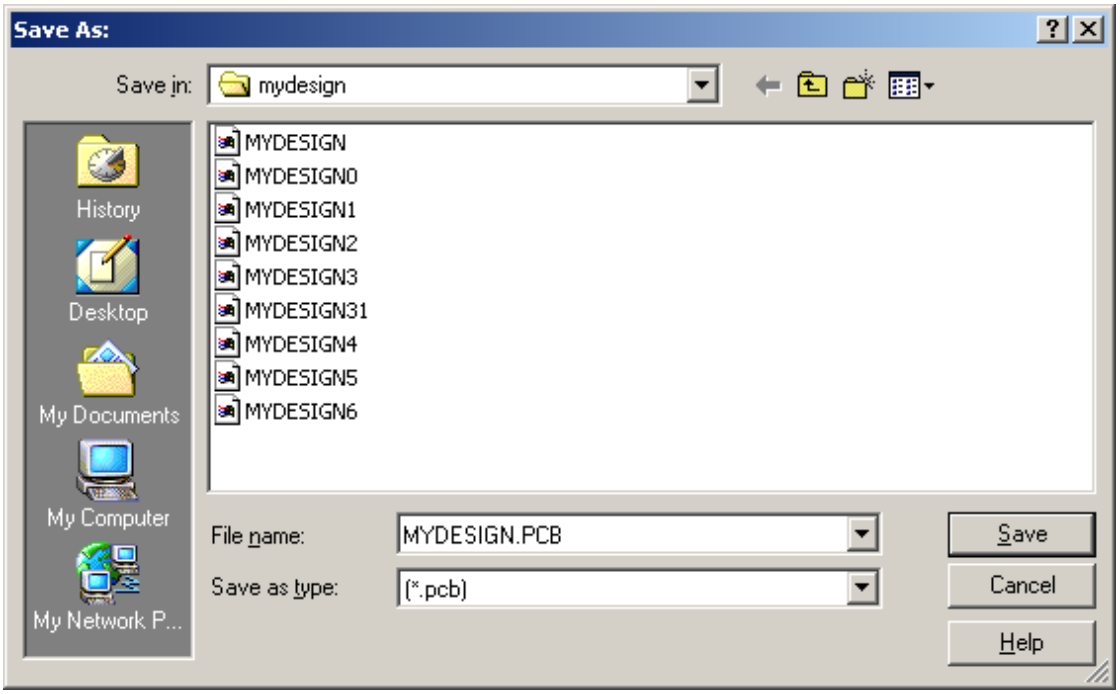
From the commands on the REPORT menu, generate a PARTS LIST, a NET LIST (pads only option), and a SUMMARY report. If you have a printer, you may direct the reports directly to the printer. Compare these reports with the schematic net and parts list. They should agree.





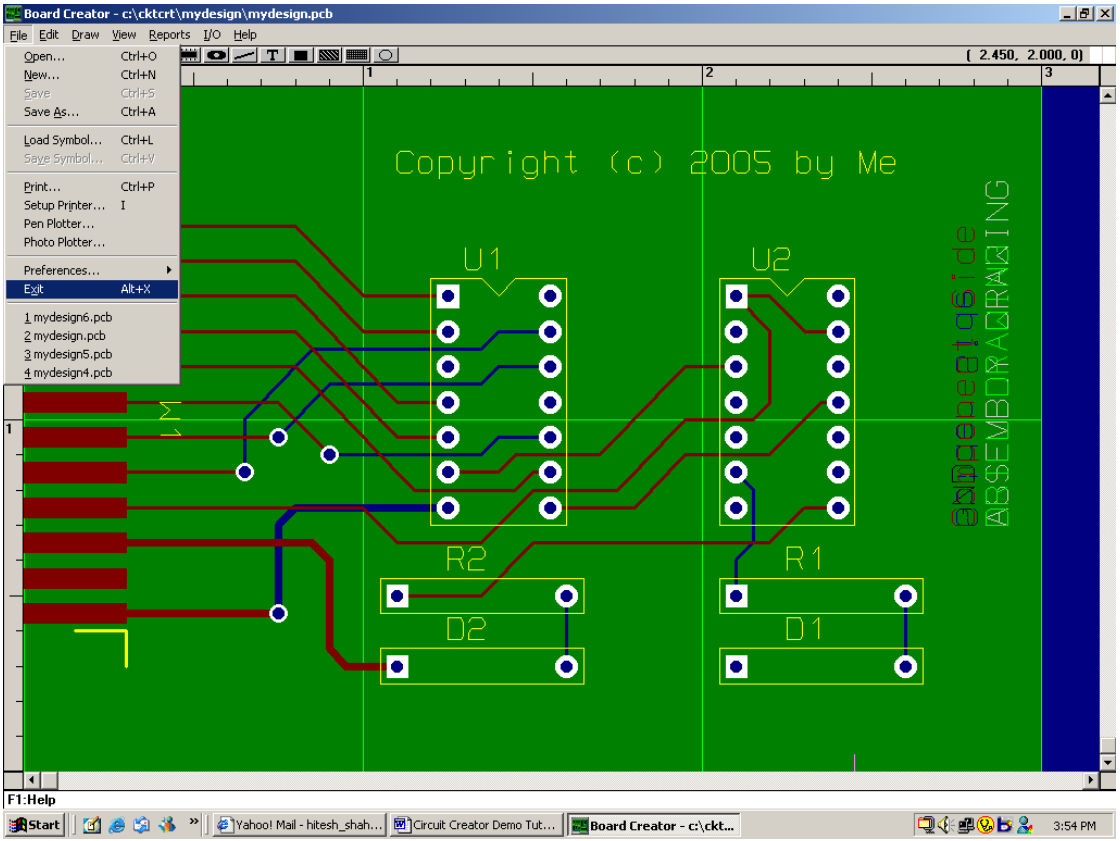
SAVING THE BOARD

Now we will save our board to a file. Select SAVE AS command from the FILE menu. Enter "MYDESIGN" for the file name and press **OK**. This will save the board into the file "MYDESIGN.PCB".



EXIT THE PROGRAM

We are finished with the board for now. Select the **EXIT** command on the FILE menu, or just press ALT-X, to exit the program.



ADDITIONAL TRAINING

The tutorial you have just completed is designed to familiarize you with the basic steps in using BOARD CREATOR to create a board. To become a more advanced user of BOARD CREATOR, and to use its features to your best advantage, the following self-training procedure is suggested.

Start the BOARD CREATOR program and use the FILE/NEW command to create a new board (any reasonable size). Go through each of the commands on the menus, one at a time. For each command, display the help information with the **F1** key. Study the options for each command and practice each of these operations on the board. It is not necessary to construct a real or complete board. Just use the board as a scratch area to

practice the commands. For example, practice the DRAW/TEXT command by placing text of various sizes, angles, mirroring, layers, etc. on the board. Move a text string. Move and rotate an area of text strings. Update a text string. Delete a text string. Do this for each command and you will quickly become a "power user" of BOARD CREATOR.

In the tutorial just completed, you were instructed how to perform each step in the construction of a simple board. Therefore, there was little or no opportunity to make changes to the board during the tutorial. In the real world, you will need to make changes, move things around, and correct mistakes. Changes are made with the SELECT command. In fact, it is the default command. Just press **ESC** to exit the current command, and you are running the SELECT command. Before any object can be changed, it must first be selected. For more information on the select command, execute the SELECT command and then press **F1** for detailed help about the command and its usage.

ROUTE CREATOR: AUTO-ROUTER TUTORIAL

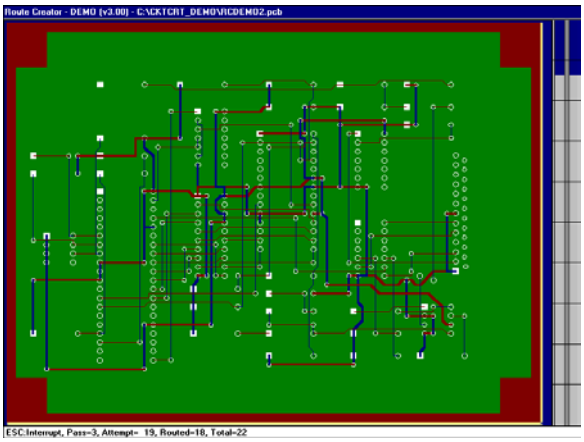
This chapter shows you how to run ROUTE CREATOR demos. In actual practice, the ROUTE CREATOR auto-router is used to automatically route the connections for a board. Although the router cannot route every board to 100% completion, it does perform the vast majority of the board layout.

Before a board is routed, the board is first "scheduled" for routing. The routing schedule is generated with the BOARD CREATOR editor. The schedule gets a board ready for routing by determining which points on the board need to be connected. The sample boards used in the demos have already been scheduled.

Once a board is prepared, the auto-router uses artificial intelligence to route the connections of the board. The auto-router will preserve any nets that you have pre-routed. Also, the auto-router may be interrupted while it is running. At this point, the BOARD CREATOR editor may be used to manually make adjustments to the board. The board may then be resubmitted to the auto-router to continue the remainder of the routing.

STARTING ROUTE CREATOR

To run the ROUTE CREATOR program, open the Start Menu and go to the CIRCUIT CREATOR program group and then click on the ROUTE CREATOR icon.



ROUTE CREATOR SCREEN

ROUTE CREATOR begins by prompting you for the name of the PC board file that is to be routed. Select one of the demo boards, "RCDEMO1" or "RCDEMO2". RCDEMO1 demonstrates routing a board with no pre-routed nets. RCDEMO2 demonstrates routing a board which has the power and ground lines pre-routed.

The default routing options for the board will be displayed. Observe the option settings, but do not change any of them at this time.

Press the **OK**, BEGIN ROUTING button to begin routing the demo board.

The program will first display the board in its current state. Next, the program will begin routing the board. This board will take a few minutes to route. This is a real board. It corresponds to the sample "voice synthesizer" circuit used in the LOGIC CREATOR schematic editor. When the routing is complete, a message box will appear, indicating the status of the routing. Press the **OK** button to continue.

How to Design PCB Using SMT Components Tutorial

This tutorial leads you step by step through the creation of a simple printed circuit board. Initially, great detail is given. As we proceed through the tutorial, and you learn the basics; less detail will be given.

Starting BOARD CREATOR

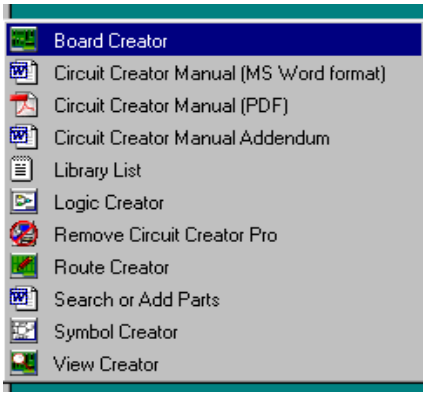
The BOARD CREATOR program can be run from the CIRCUIT CREATOR supervisor, or executed directly from the DOS command line. If you are using the CIRCUIT CREATOR supervisor, do the following:

Click on START button on Windows Desktop

Select Programs and Click on Circuit Creator

To start the BOARD CREATOR program from the DESKTOP, do the following:

Click on START button on Windows Desktop
Select Programs and Click on Circuit Creator
Click on Board Creator

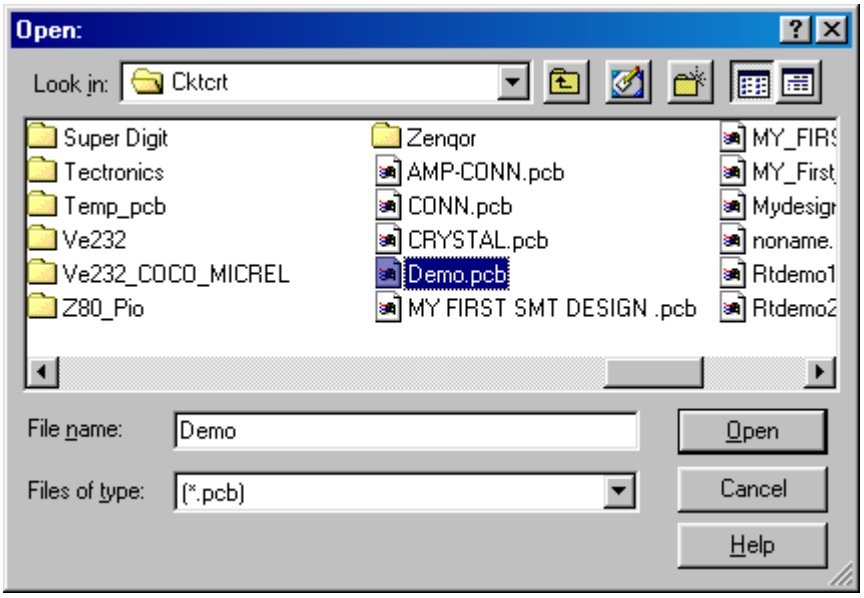


Loading a Board

To get a feel for the system, we will first load a sample PC board file. We will then learn how to navigate about the board on the display. First, an explanation of how to select commands from the pull-down menus and dialog-boxes will be given.

If you are using the keyboard, a pull-down menu may be displayed (pulled-down) by holding the Alt key down and pressing the key indicated by the underlined character in its name on the title-bar. Upper or lower case may be used.

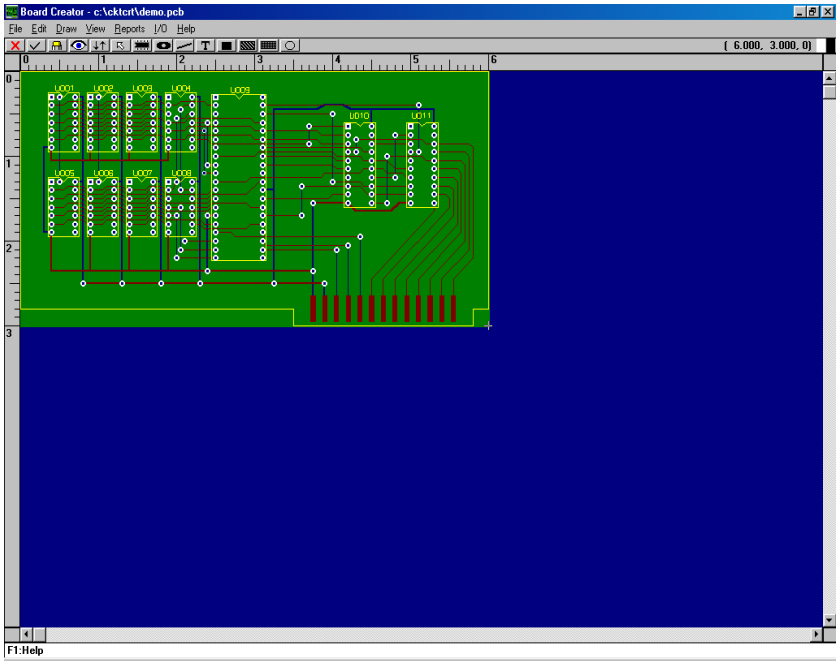
When a pull-down menu is displayed, the → and ← arrow keys may be used to move to the next menu to the right or left. The NumLock key must be off if you are using the keyboard's keypad arrow keys. Use the ↑ and ↓ arrow keys to move the menu selector up and down. Move to the Open command in the File pull-down menu and select it by pressing the Enter key.



If you are using a mouse, you may pull-down a menu by moving the pointer to the menu name in the title-bar and pressing the left mouse button and releasing it. The menu will appear when the button is released. To select a menu item, move the pointer to the desired item and press the left mouse button and release it. The

selection is made when the button is released. For quicker menu selections, move the pointer to a pull-down menu heading and press the left mouse button and do not release the button yet. Now, move the pointer to the desired menu item and release the button to make the selection.

After selecting Open from the File menu a dialog-box will appear which is prompting for the name of a PC board file. At this point, we can type the name of a file, or select one from the list that is displayed. Use the arrow keys to move the selector to the file in the list named "DEMO.PCB" and press Enter. This will copy the file name from the list into the file field of the dialog-box. Now press Enter again to directly accept this name, or press the Home key to move the selector to the OK button, then press Enter.



When a dialog-box is displayed, clicking the left mouse button on the OK button, or pressing the Alt-O key combination will immediately accept the values in the dialog-box. Pressing the Home key will move the selector to the OK button.

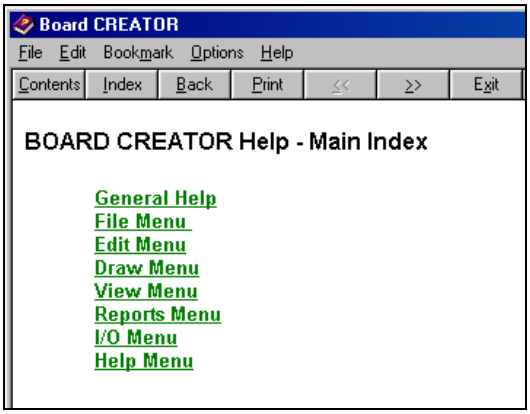
Clicking on the CANCEL button, or pressing the Alt-C combination will immediately cancel the dialog-box. Pressing the End key or the Esc key will move the selector to the CANCEL button. Once the selector is on the CANCEL button, pressing Esc again will cancel the dialog-box. Any changes made to the values shown in a dialog-box do not actually take effect until the OK button is pressed. If the CANCEL button is pressed, the changes made to values in the dialog-box are ignored.

For a file-selection or part-selection dialog-box, pressing Enter twice or double clicking with the mouse pointer while on the same name in the list implies that this is the name you want to accept immediately.

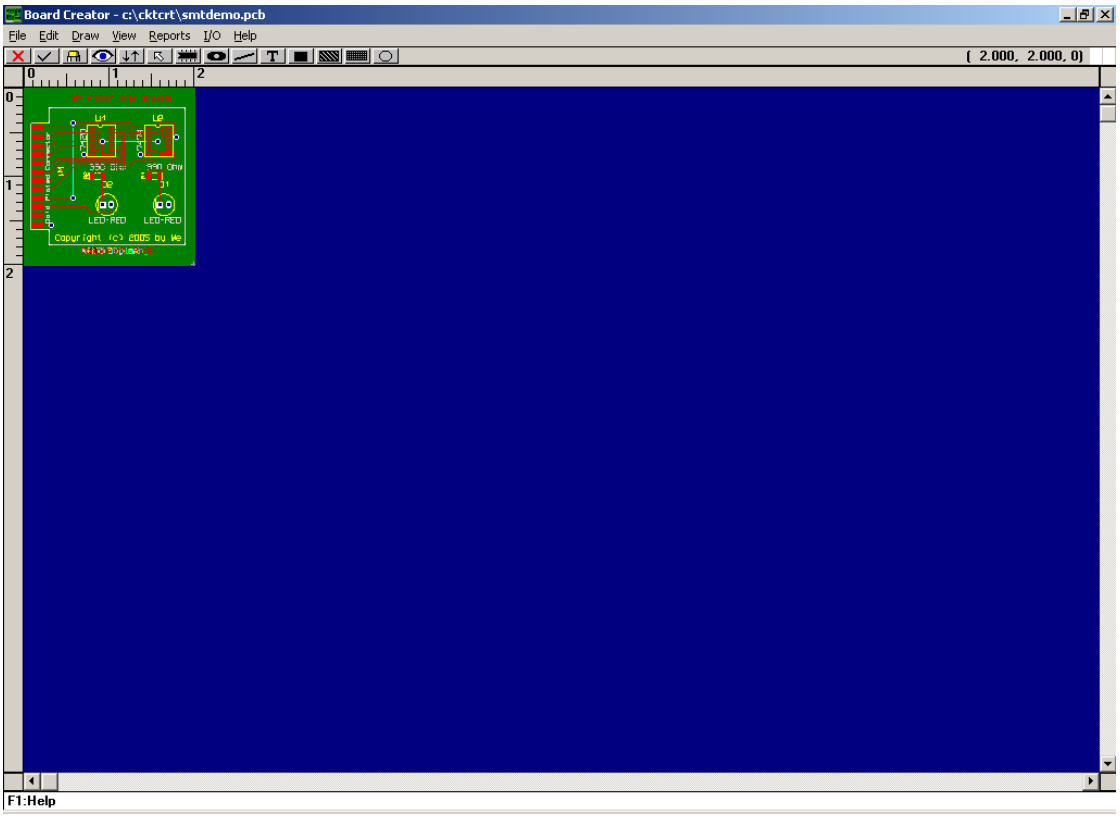
Getting Help

Go to HELP menu or just Press the F1 key to pop-up a window of help information. Notice that the help information is specific to the current command being performed. The UP and DOWN buttons may be used to page the help text up and down in the window. You may also use the NEXT, PREV, or GENERAL buttons to view other help topics.

Now, select the EXIT button to close the help window. Help is always available with the F1 key.

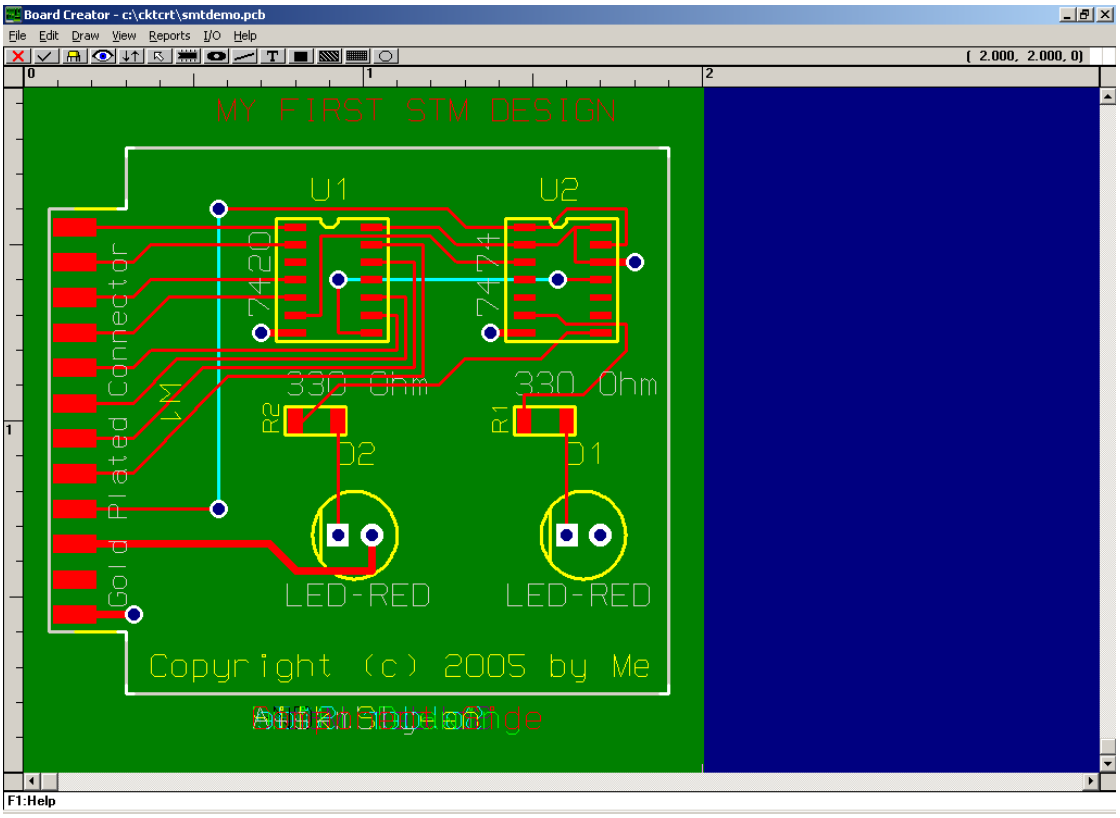


Navigation



We will now learn how to change the view of a PC board displayed on the screen. Pull-down the View menu. Now, move down the menu and select the Zoom Out command. The display will be redrawn at a smaller scale. You will notice that the Zoom Out command has Ctrl + PgDn listed in the right margin next to it. This indicates that this command has a hot-key. This key combination may be used when the menu is not displayed to directly execute this command. Let's try it. Now, hold the Ctrl key down and press the PgDn key to zoom-out. Once you are more familiar with the program, using hot-keys will improve your productivity, especially if you do not have a mouse.

Zoom back to the original scale by pressing the Ctrl + PgUp key twice.



It should be noted that if you pick the pan and zoom commands directly, you do not need to wait for the redraw to complete before entering another pan or zoom command. In this case, the current redraw will be stopped, and the next redraw will begin directly.

The current coordinate of the cursor is displayed in the upper-right corner of the screen. Use the arrow keys to move the cursor to coordinate (2.000,1.000). Follow the track of the cursor on the screen, and watch the value in the upper-right corner change. Each time the arrow key is pressed, the cursor will move in 50 mil (0.050") increments (this is the current grid setting). Notice that the active cursor (the little cross) is not always in the same position as the mouse pointer.

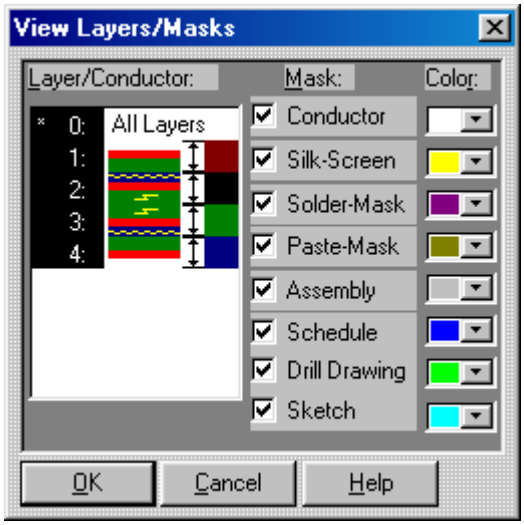
Holding the Shift key down and using the arrow keys will cause the cursor to move ten times the current grid setting. Hold the Shift key down and use the ← and ↑ arrows to move quickly to coordinate (0.000,0.000), the upper-left corner of the sheet. Notice that the cursor is now moving in 500 mil (0.500") increments each time the arrow key is pressed. On some keyboards, you will need to turn off the NumLock key and use the keypad's arrow keys.

The display may be panned using the keyboard by bumping into one of the edit window edges. First, zoom in with the Ctrl + PgUp key. Now, let's pan right. Move the cursor with the Shift → key until it bumps the right edge of the window. The portion of the board that was out of view to the right of the screen is now displayed.

With a mouse, the view may be panned by using the vertical slide-bar at the right, or the horizontal slide-bar at the bottom.

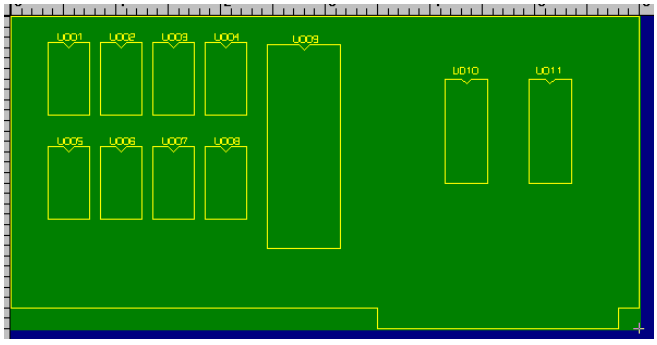
Display the upper-left corner of the board at normal scale by pressing Ctrl + Home.

We will now explore how to select the layers of information that are displayed on the screen. Execute the Layer/Masks command from the View pull-down menu.



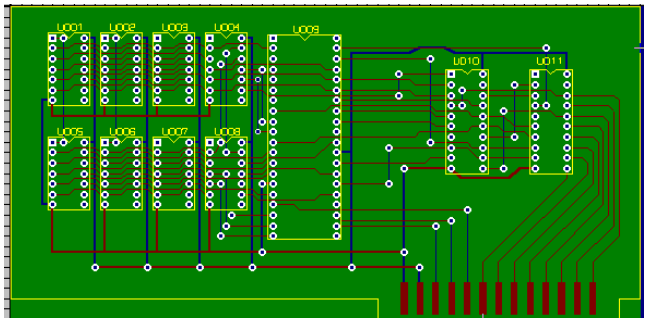
You will notice that each layer of the board is listed along with its type and current conductor color. If there is not a check-mark by a layer, then items on that layer will not be displayed. The actual colors and number of colors available will depend on your particular hardware. The display state of each layer is changed by moving the focus indicator to that layer and pressing ENTER to turn it on or off (removing the check mark next to a particular layer).

Also, this dialog-box allows you to select which masks will be displayed.



Turn off all layers except layer 1 and all masks except the silk-screen mask. Press the OK button. Notice that only items which are on the silk-screen mask for layer 1 of the board are displayed.

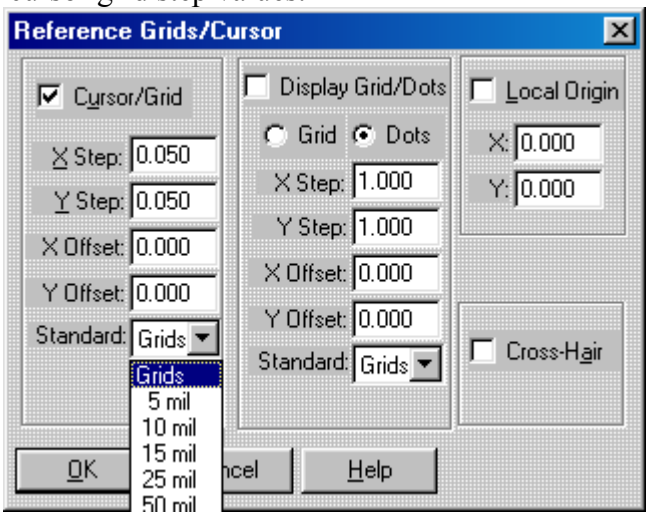
Now, let's display just the solder-side layer of the board. Select the VIEW/LAYERS/MASKS command again. Enable the "All" layer, and layer 4 (solder-side, last layer). Enable all the masks. Press OK. Notice that only the items that belong to the solder-side, or items that go through "all" layers, are displayed. Items that go through "all" layers are pads and vias. If a line is drawn on the "all" layer (layer 0), it will appear on every conductor layer that is printed or plotted.



Cursor Grid

The cursor grid is set to 50 mils by default. You have probably noticed that the cursor steps 50 units each time the arrow keys are pressed. Select the GRID/CURSOR command from the VIEW menu. Set the cursor grid X and Y grid steps to "0.025" and press OK. This sets both the cursor grid step values to 25 mils. Move the cursor about the screen with the arrow keys. Notice that the cursor now moves in steps of 25 mils.

This completes a summary of how to start the BOARD CREATOR editor, load an existing board, view the board, choose which layers and masks are to be displayed, and how to set the cursor grid step values.



Creating a New Surface Mount PC Board

Our assignment is to create a PC board which implements the design shown in the following figure. Once created, it will be checked for errors, with assistance from the program. Masks and reports about the design will be generated. This circuit is intentionally simple and small; however, it demonstrates the basic steps by which any circuit, regardless of size, is created. This circuit corresponds to the tutorial lesson used in the companion LOGIC CREATOR schematic system.

The overall board specification is as follows:

1. Board size is 2.00" x 2.00".
2. Board has 2 signal routing layers, route 10 mil lines.
3. Power and ground provided by power planes.
4. Conductor masks required.
5. Solder mask required.
6. Silk-screen required.
7. Drill Drawing required.

Bill of Materials:

Count	Part-Name	Description
1	7420	Dual 4-Input Nand Gates
1	7474	Dual D-Type Flip-Flop
1	CONN-12	12 Pin Connector
2	LED-RED	Light Emitting Diode, Red
2	R330	330 Ohm Resistor

Since this board corresponds to a schematic drawn with the LOGIC CREATOR schematic system, we will use the update-list file produced by LOGIC CREATOR to create our initial PC board. We will begin the new board by selecting the CREATE FROM LOGIC CREATOR command from the I/O menu. Since we already have a board loaded, we will be questioned as to whether or not we wish to save the changes made to the current board. Answer NO when prompted.

The following is the complete LOGIC CREATOR update-list for the "MYDESIGN" tutorial lesson. It defines each part, part-footprint, and net assignment for every pin on the board. This is all the information that is required to create the initial PC board for a design. This information is included to show you the data that is contained in an update list file. We will not have to refer to this data directly.

Board Creator Updated List:

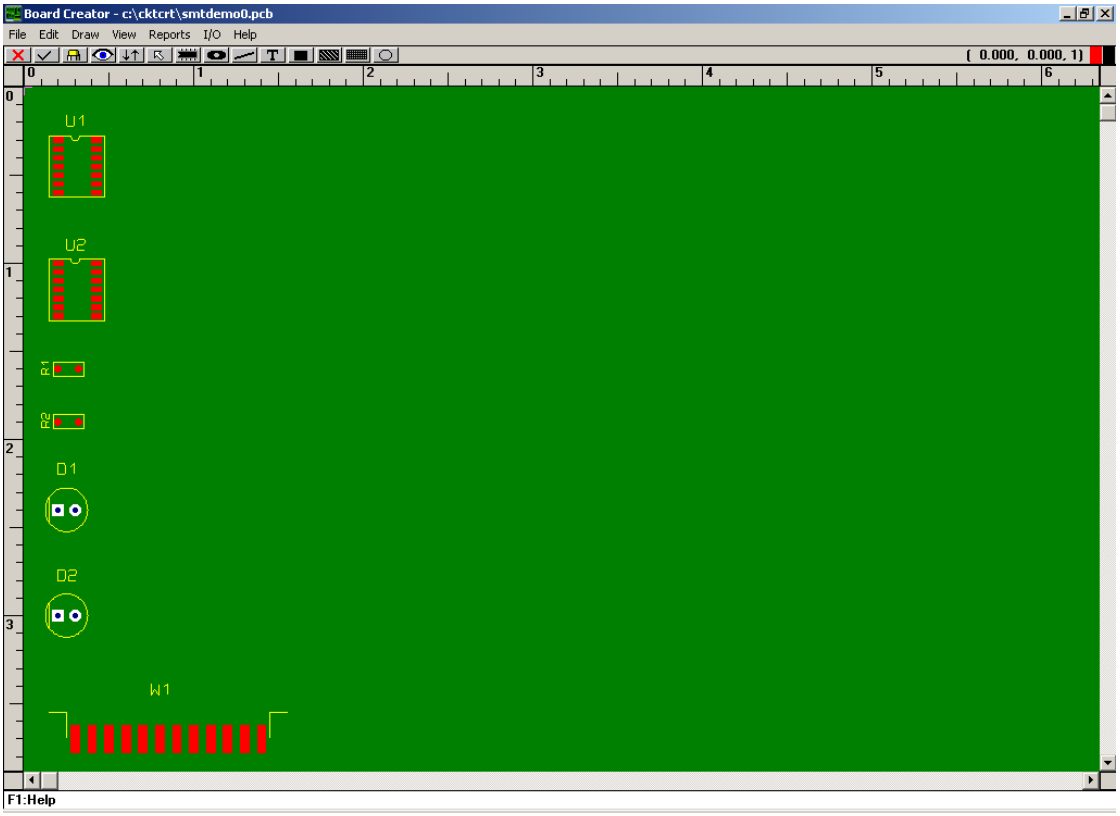
```
UPDATE_LIST
PART 7420 U1 SMT-14V
PAD U1 1 #0007
PAD U1 2 #0008
PAD U1 3 NC
PAD U1 4 #0009
PAD U1 5 #0010
PAD U1 6 #0001
PAD U1 7 GND
PAD U1 8 #0002
PAD U1 9 #0011
PAD U1 10 #0012
PAD U1 11 NC
PAD U1 12 #0013
PAD U1 13 #0014
PAD U1 14 +5V
PART 7474 U2 SMT-14V
PAD U2 1 RESET
PAD U2 2 +5V
PAD U2 3 #0001
PAD U2 4 NC
PAD U2 5 NC
PAD U2 6 #0003
PAD U2 7 GND
PAD U2 8 #0005
PAD U2 9 NC
PAD U2 10 NC
PAD U2 11 #0002
PAD U2 12 +5V
PAD U2 13 RESET
PAD U2 14 +5V
```

```
PART R330 R1 SM-1206H
PAD R1 1 #0003
PAD R1 2 #0004
PART R330 R2 SM-1206H
PAD R2 1 #0005
PAD R2 2 #0006
PART LED-RED D1 LED2
PAD D1 1 +5V
PAD D1 2 #0004
PART LED-RED D2 LED2
PAD D2 1 +5V
PAD D2 2 #0006
PART CONN-12 W1 CONN12
PAD W1 1 #0007
PAD W1 2 #0008
PAD W1 3 #0009
PAD W1 4 #0010
PAD W1 5 #0011
PAD W1 6 #0012
PAD W1 7 #0013
PAD W1 8 #0014
PAD W1 9 RESET
PAD W1 10 +5V
PAD W1 11 NC
PAD W1 12 GND
END_UPDATE_LIST
```

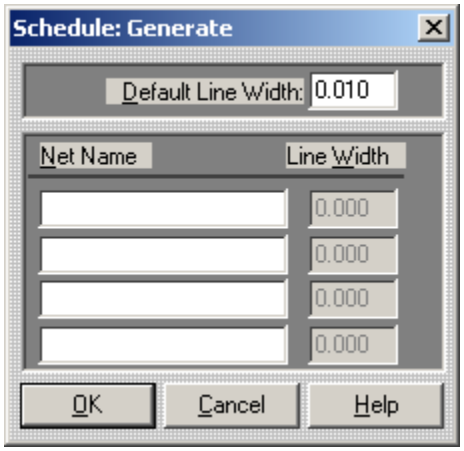
We are now prompted for the name of the update-list file to be used. Select the "SMTDEMO.UPD" file from the list. BOARD CREATOR will now read the part and net assignment information from the update-list file and create a board with these parts on it.

Component Placement

We now have a PC board with all the required component parts. The next step is to move each part to its actual location on the board, then we will adjust the board to its true size and number of layers. Although moving the parts is a manual process, we will be assisted by the program. It will show us how the parts connect to one another. To do this we will need a routing schedule.



Run the CHECK/SCHEDULE command from the REPORTS menu. Use a default routing line width of 0.010" (10 mils). This will generate a routing schedule for us. The schedule rats-nest lines are visible on the screen when the schedule mask is enabled, or when a component is selected. Since these lines show us exactly the pin-to-pin connections that must be routed, we will want to place the parts so the length of these lines are as short as is reasonable.



Before we begin placing parts, we will set-up the select options for doing component placement. Execute the SELECT command from the EDIT menu and press the O key to pop-up the select command options. Set the select layer to ALL. Enable COMPONENT and TOUCHING modes. Enable all the other check boxes except the SCHEDULE mask. By not enabling the schedule mask for selection, we will not accidentally select schedule lines that are just passing through our select area. Selecting a component will still select the schedule lines that belong to that particular component. Press OK.

We are now ready to begin placing the parts. Move the cursor to location (0.000,0.000) and zoom in twice (CTRL+PGUP). This is a good working scale for placing this board.

For this tutorial lesson, we will move all the components by selecting pin number one of each component. This is done to establish the coordinate numbers listed in this tutorial. When you are working on your own, you may move a component by selecting anywhere on the component.

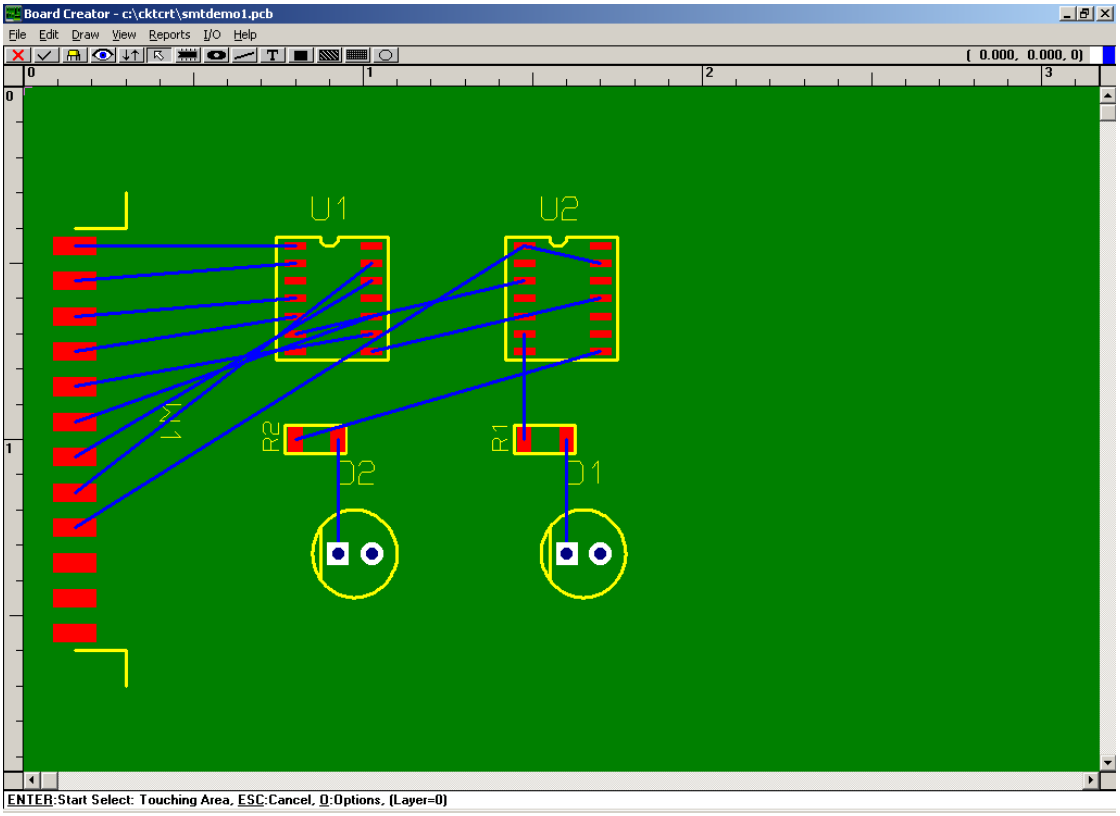
Move the cursor to location (0.200,1.000) and press ENTER twice, or click once with the left mouse button. This will select component U2. Now, press ENTER, or press the left mouse button and continue to hold it down. This will grab the component for moving. Move the part with the arrow keys, or by moving the mouse. Notice how the schedule lines stretch to show you how this part connects to other parts. Move until

you are at location (1.475,0.450). Press ENTER, or release the left mouse button, to place the part at this location. If you make a mistake, press F2 (undo), and try again.

You do not need to unselect the component; it will be unselected when the next component is selected. Using the procedure just established, place the following components:

Select U1 at (0.200,0.300). Move it to (0.800, 0.450).
Select U2 at (0.200,1.000). Move it to (1.475, 0.450).
Select R1 at (0.200,1.600). Move it to (1.475, 1.000).
Select R2 at (0.200,1.900). Move it to (0.800, 1.000).
Select D1 at (0.200,2.400). Move it to (1.600, 1.325).
Select D2 at (0.200,3.000). Move it to (0.925, 1.325).

Now we have placed all the components except the edge connector. In addition to moving it, it also needs to be rotated. Select the connector, W1, at (0.300, 3.700). Grab the connector and begin to move it just like the other components. While the connector is on the cursor, we may press the O key (or F10) to rotate the part 90 degrees. Press the O key three times to rotate the part a total of 270 degrees. Move to (0.150, 0.450) and place the part. Press the Z key to unselect the component.

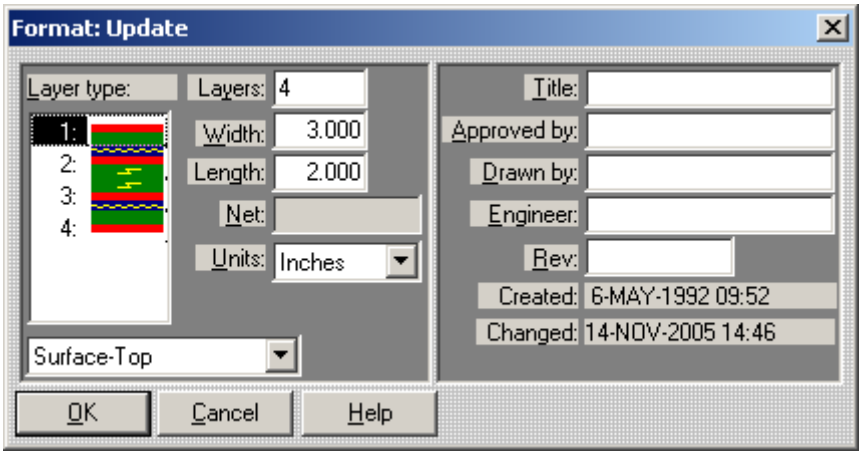


Now all the components are placed in a reasonable location for routing the electrical connections.

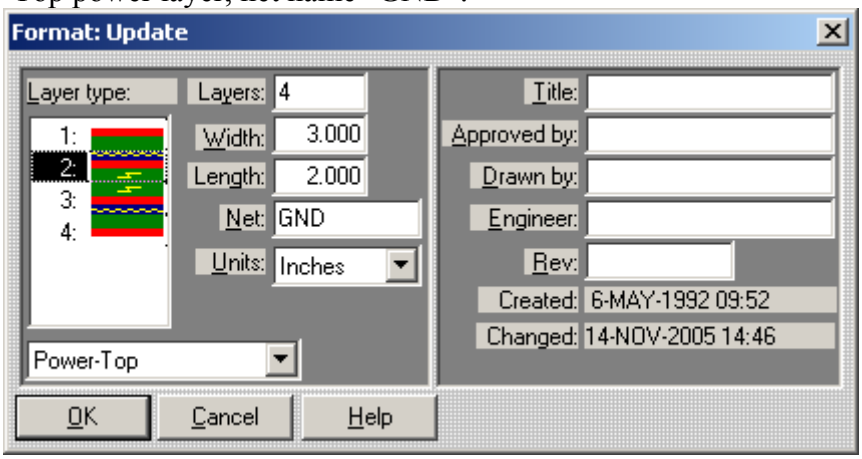
Set True Board Size/Layers

Execute the BOARD FORMAT command from the EDIT menu. Set the number of layers to 4, the width to 2.000", and the height to 2.000". To set a layer type, move the layer focus indicator to the layer number and press ENTER to pop-up a menu. Select the layer type from the menu. Set the layer types as follows:

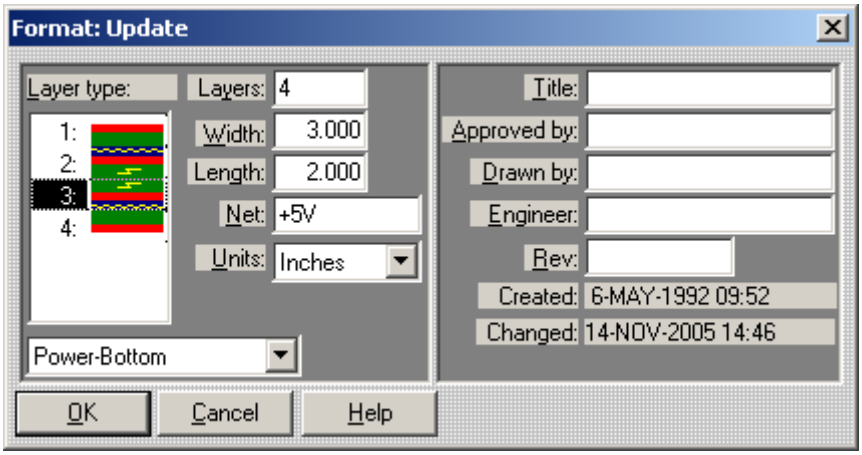
1 - Top conductor (already set).



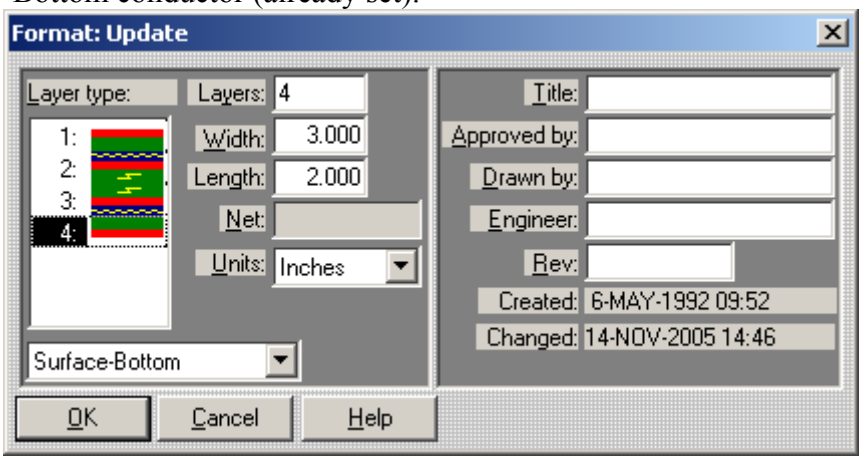
2 - Top power layer, net name "GND".



3 - Bottom power layer, net name "+5V".



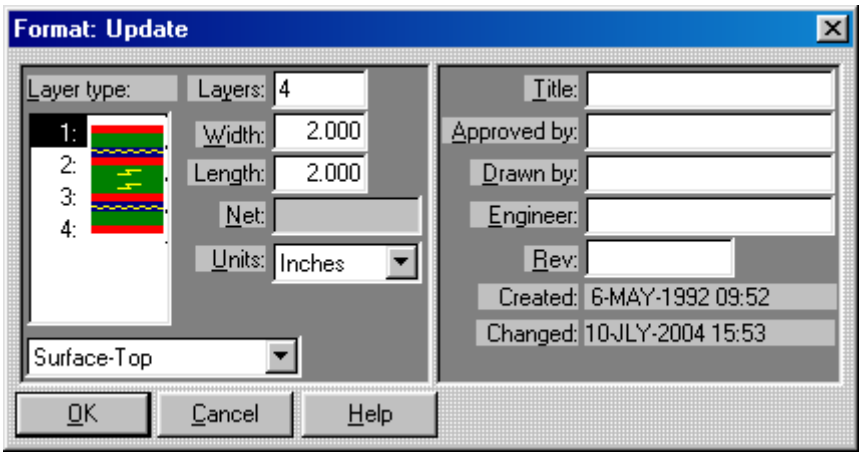
4 - Bottom conductor (already set).



To set the net names, move the layer focus indicator to one of the power layers. Since this is a power layer, the net name field will now become enabled. If you use the keyboard, type Alt + N to pop-up the net name field. Type the net name into the net field. Do this for both of the power layers.

Set the title, engineer, and other accounting fields to appropriate values. These values will be printed on reports generated for this board.

Press OK to accept the new board definition.

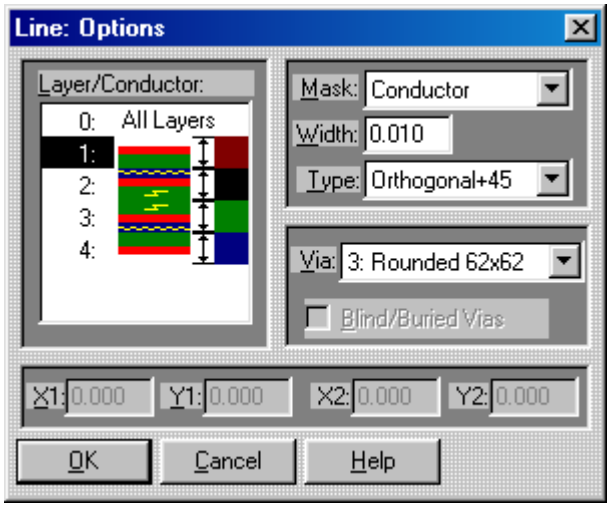


Routing the Board

We are ready to route the conductor lines that make the desired circuit. Before we begin routing the board, we will run the CHECK/SCHEDULE command from the REPORTS menu again. This is done for two reasons. First, we have just done a major placement of the board. With the new component locations, it may be possible to generate a more efficient routing schedule. Also, we have just added a power and ground layer to the board. There may not be a need to schedule lines for the "GND" and "+5V" nets. Through-hole pads assigned to these nets will connect directly to the power and ground layers. A message will appear after the board has been scheduled telling us the number of routes we must make to complete this particular design.

Now we will begin routing the board

Execute the LINE/CURVE command from the DRAW menu. From our specification, the width of the routing lines is 10 mils. The current line width is displayed in the bottom status line. If this is not 0.010", press the O key, or click on the status line with the mouse pointer, to display the line drawing options dialog-box. Set the line width to 0.010" (10 mils).



For this board, we will the use the component-side for most of the routes because of the surface mount footprints. In some instances, it may not be possible to route on the component-side, therefore, a via will be used and the routing will be continued on the bottom solder-side.

From our procedures suggested in Chapter 1 for routing a board, we will route the shortest lines first. Since we can see from the routing schedule lines shown on the screen, which pads need to be connected together, look for some short ones.

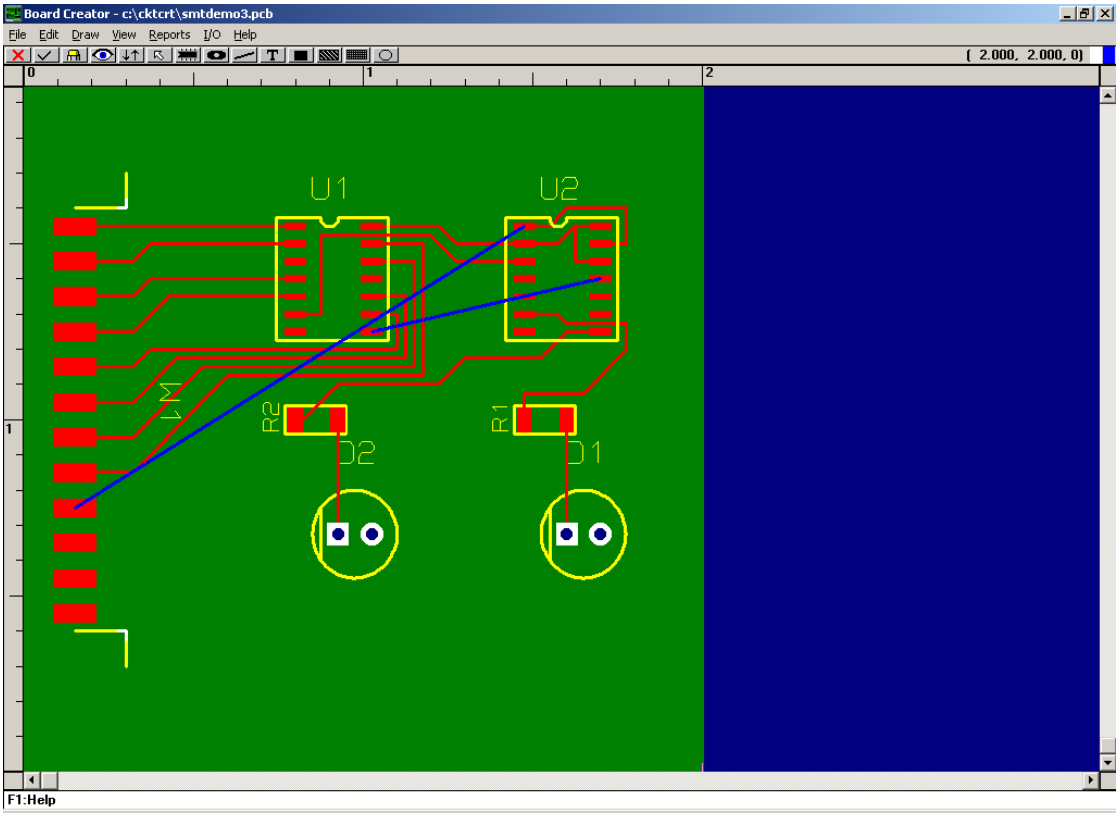
Move to coordinate (1.600,1.325) and press ENTER, or click with the left mouse button. The schedule line from this pad is now highlighted, showing us the connection that needs to be made. Since this route is going to a surface mount pad, we will route this line on layer 1, the top component-side of the board. Notice that the status line indicates the current edit layer is layer number 1. Now move to (1.600,1.000) and press ENTER, or click with the left mouse button. This will complete the first route. (Note: in order for a route to be complete, you must route lines to the exact center of pad or component's pin) Notice that the schedule line is now gone, indicating that this route is complete. Press ESC, or click the right mouse button to stop drawing this line. If you make a mistake while still drawing a route, press BACKSPACE to backup (instead of UNDO).

Now let's do another route. Move to (0.925,1.325), ENTER move to (0.925,1.025), ENTER, and ESC.

Got the hang of it? Routing lines on your own will actually be much easier. You will work directly from the screen instead of reading coordinates from a book. Now let's route a line with some bends in it. Move to (0.800,1.000), ENTER, move to (0.825,1.000), ENTER, move to (0.925,0.900), ENTER, move to (1.225,0.900), ENTER, move to (1.300,0.825), ENTER, move to (1.525,0.825), ENTER, move to (1.600,0.750), ENTER, move to (1.700,0.750), ENTER, and ESC.

Route lines between the following points. More instruction will be given when we add a via for the first time.

```
Layer 1: (0.150,0.450), (0.800,0.450).
Layer 1: (0.150,0.550), (0.325,0.550), (0.375,0.500), (0.800,0.500).
Layer 1: (0.150,0.650), (0.325,0.650), (0.375,0.600), (0.800,0.600).
Layer 1: (0.150,0.750), (0.325,0.750), (0.425,0.650), (0.800,0.650).
Layer 1: (0.150,0.850), (0.325,0.850), (0.375,0.800),
         (1.100,0.800), (1.100,0.700), (1.025,0.700),
Layer 1: (0.150,0.950), (0.325,0.950), (0.450,0.825),
         (1.125,0.825), (1.125,0.650), (1.025,0.650).
Layer 1: (0.150,1.050), (0.325,1.050), (0.525,0.850),
         (1.150,0.800), (1.150,0.550), (1.025,0.550).
Layer 1: (0.150,1.150), (0.325,1.150), (0.600,0.875),
         (1.175,0.875), (1.175,0.500), (1.025,0.500).
Layer 1: (0.800,0.700), (0.875,0.700), (0.875,0.475),
         (1.200,0.475), (1.275,0.550), (1.475,0.550).
Layer 1: (1.025,0.450), (1.225,0.450), (1.275,0.500), (1.475,0.500).
Layer 1: (1.700,0.550), (1.625,0.550), (1.625,0.450), (1.700,0.450).
Layer 1: (1.475,0.500), (1.575,0.500), (1.625,0.450), (1.700,0.450).
Layer 1: (1.700,0.500), (1.775,0.500), (1.775,0.400),
         (1.600,0.400), (1.550,0.450), (1.475,0.450).
```



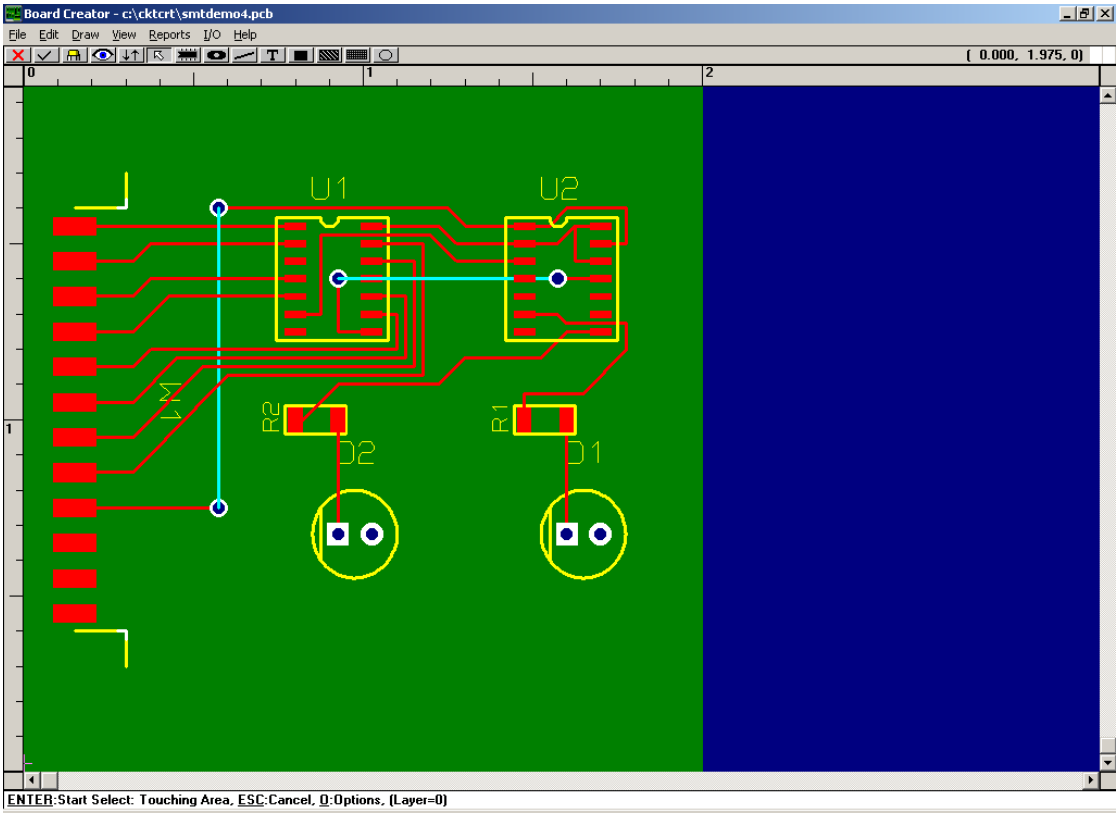
The next line will require a via. Route the line as before, up to the point where a via is noted. At the point just ENTER again, or click the left mouse button. This is the same as drawing a line segment with no length. This will add a via at this point and toggle the routing layer. Continue drawing the remainder of the line on the other layer.

```
Layer 1: (1.700,0.600), (1.575,0.600),VIA,(0.925,0.600),
         VIA, (0.925,0.750), (1.025,0.750).
```

Remember to start each line on the proper layer.

```
Layer 1: (1.475,0.450), (1.300,0.450), (1.250,0.400),
         (0.575,0.400), VIA,(0.575,1.250),VIA,(0.150,1.250).
Layer 1: (1.475,0.700),(1.300,0.700),VIA,(1.300,1.000),
         VIA,(1.475,1.000).
```

The remaining routes deserve some special attention. Even though this board has power and ground planes, the planes themselves must receive power from some external connection. For this board, pin #10 on the edge connector is assigned to net "GND", and pin #12 is assigned to net "+5V". Since these pins do not have holes (the pads that are used for the connector appear only on the top component layer), they do not connect to the power planes directly, and must be routed to a point that does. Otherwise the board will not be powered.

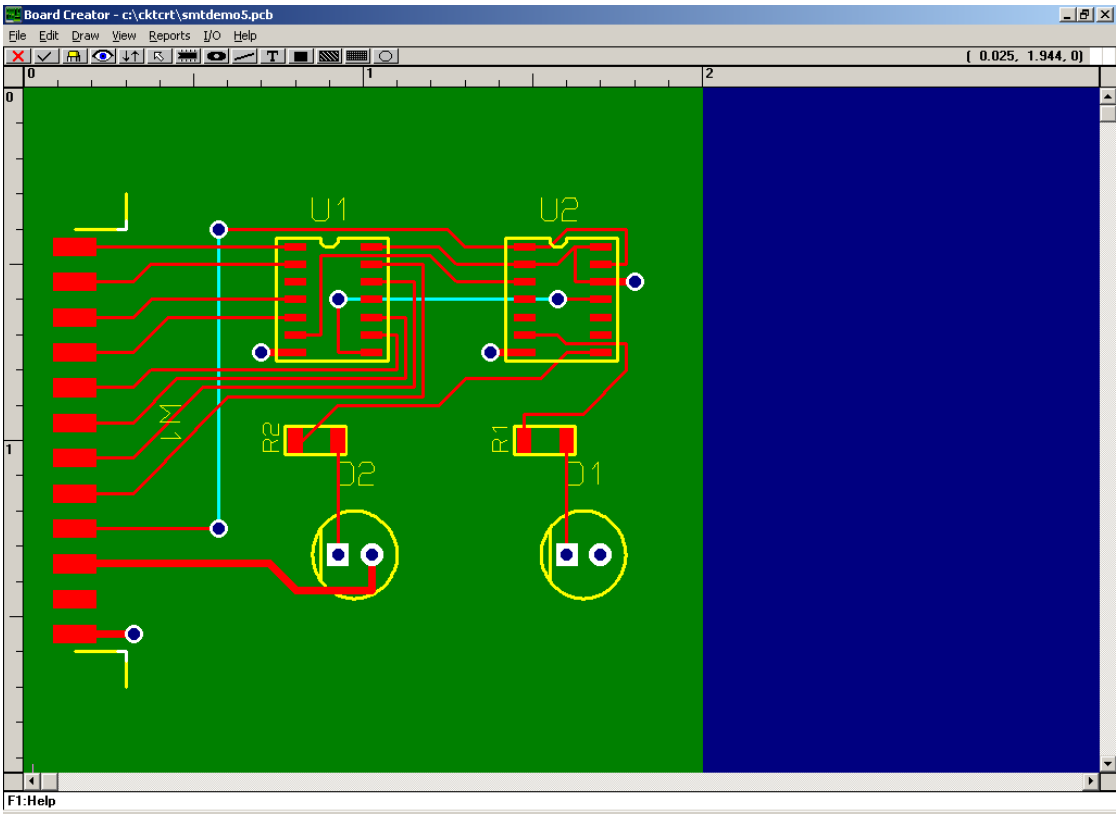


Power lines are normally routed with a wider than normal line width. Set the line width to 0.250" (use the O key to pop-up the line dialog-box). Now, route the following power lines:

```
Layer 1: (0.150,1.350), (0.725,1.350), (0.800,1.425),
         (1.025,1.425), (1.025,1.325).
```

In the last route that was placed, power is received from the edge connector to the power plane through the power pin (or pad) of D2. Now the ground plane must be connected to the edge connector. To accomplish this, move to (0.150,1.550) and ENTER, move to (0.325,1.550) and place a via. The via will automatically connect to the proper power plane when the board is rescheduled. Connect power and ground to the IC's (U1 and U2) by routing the following lines:

```
Layer 1: (0.800,0.750),(0.700,0.750),VIA.
Layer 1: (1.475,0.750),(1.375,0.750),VIA.
Layer 1: (1.700,0.550),(1.800,0.550),VIA.
```

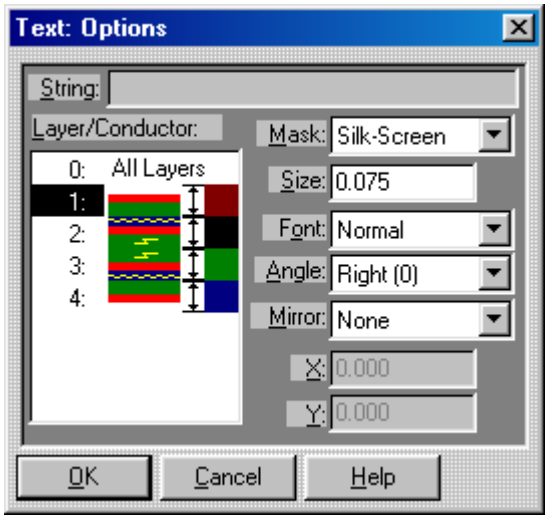


This completes the routing of the board. Notice that there was never a need to refer directly to the update-list, or any other lists. All the information necessary to place and route the board was presented directly on the screen where you were working. Run CHECK/SCHEDULE from the REPORTS menu to confirm that "all routes are complete".

Adding Text

Next, we will learn how to add notational text strings to the board. Text is often added to directly identify the layer of the mask, the company name, copyright notice, etc. Be careful when adding text to a conductor layer. The text will be etched in copper on the actual board. Do not place the text where it will touch any conductor lines.

Select the DRAW/TEXT command. Move to (0.350, 1.725) and ENTER to begin adding a text string. In the dialog-box, enter "Copyright (c) 2005 by Me" for the text, layer 1, mask SILK-SCREEN, size 0.062", angle 0, font NORMAL, and mirroring NONE. Press OK to add the text. The text will appear on the board at the location of the cursor and have attributes, which match these settings.

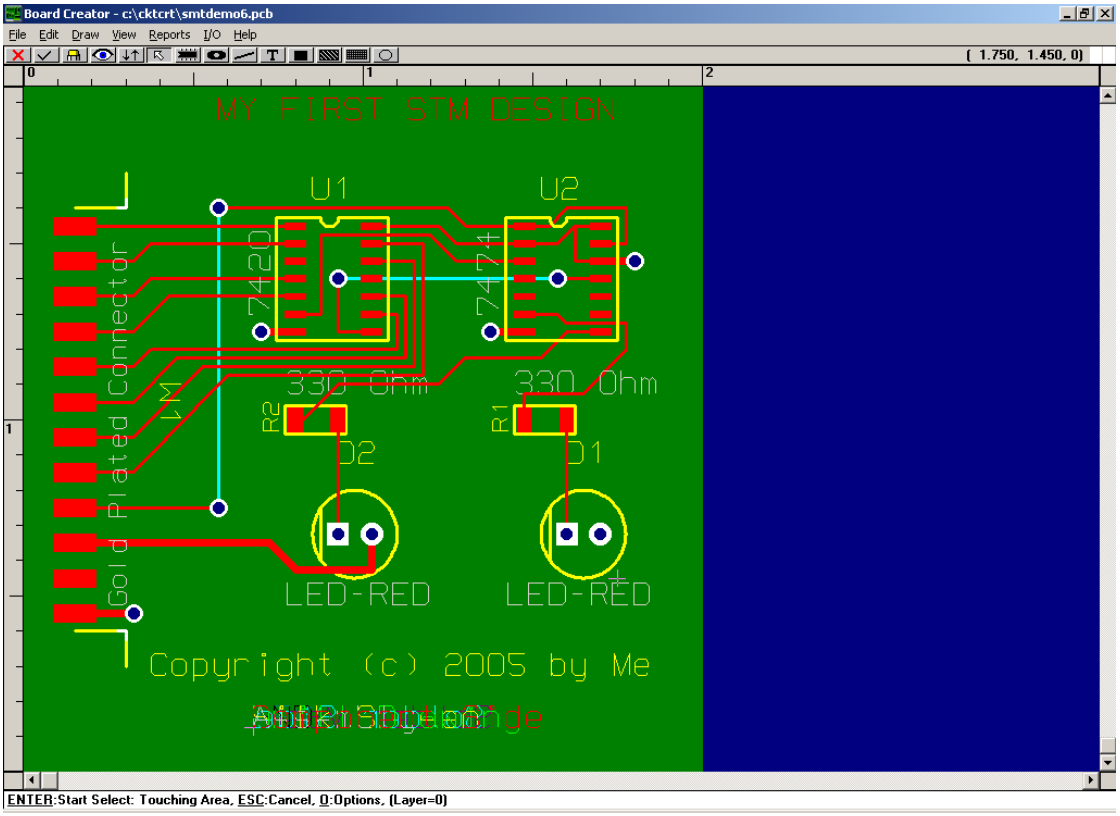


Move to location (0.675,1.875). Press ENTER and add the string "Component Side" to the CONDUCTOR mask with an angle 0. Remain at the same location and put "GND" on power layer 2. Notice how the display adjusted to show a power layer. Put "+5V" on power layer 3.

Add the string "Assembly" on the ASSEMBLY DRAWING mask of layer 1 (angle 0, no mirroring). Likewise, add the string "Drill Drawing" to the DRILL DRAWING mask of layer 1.

Move to (1.350,1.875) and add "Solder Side" on layer 4 with an angle of 0, and mirroring set to RIGHT-TO-LEFT. This will make this text readable when it is viewed from the back-side of the board.

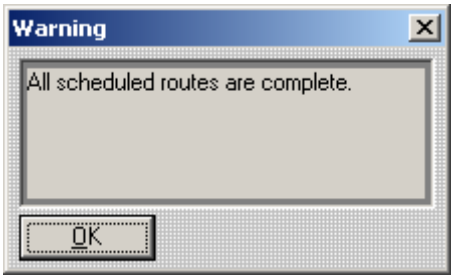
Move to (0.575, 0.150) and add "MY FIRST SMT DESIGN" on layer 1 with an angle of 0, Mirror None. This will make this text readable when it is viewed from the top-side of the board.



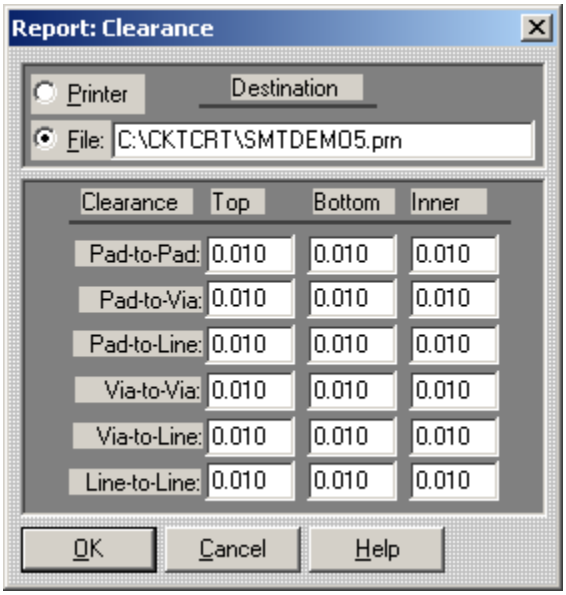
You may add any additional text as desired. Set the angle, size, and mirror factors as desired in the dialog-box.

Check the Layout

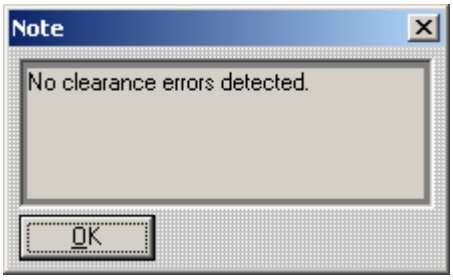
To check our board for routing completeness, run the CHECK-SCHEDULE command from the REPORTS menu. If you have completed the board correctly, a message "All scheduled routes complete" will appear; otherwise, the schedule lines for any uncompleted routes will be generated. If you have routed two nets together, creating a "short", the location of the problem will be displayed directly on the screen.



Also, run the CHECK CLEARANCE command from the REPORT menu. This command will check that all lines, pads, vias, etc. have at least the specified clearance distance from each other. If errors are detected, they will be pointed out directly on the screen.



If any errors are detected, try to correct the problem and re-run the checks until there are no errors.



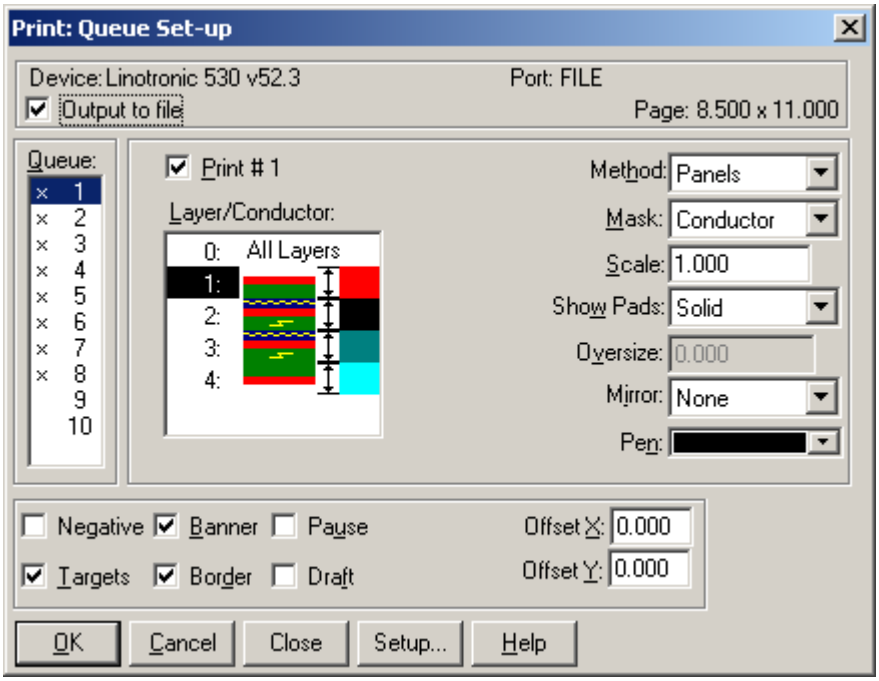
Printing the Masks

Now that the board is complete, and has passed all the error checks, it is time to print the masks for this board. If you have not yet selected the printer device, use the SET-UP PRINTER command from the FILE menu to select your printer device and set the basic printing options.

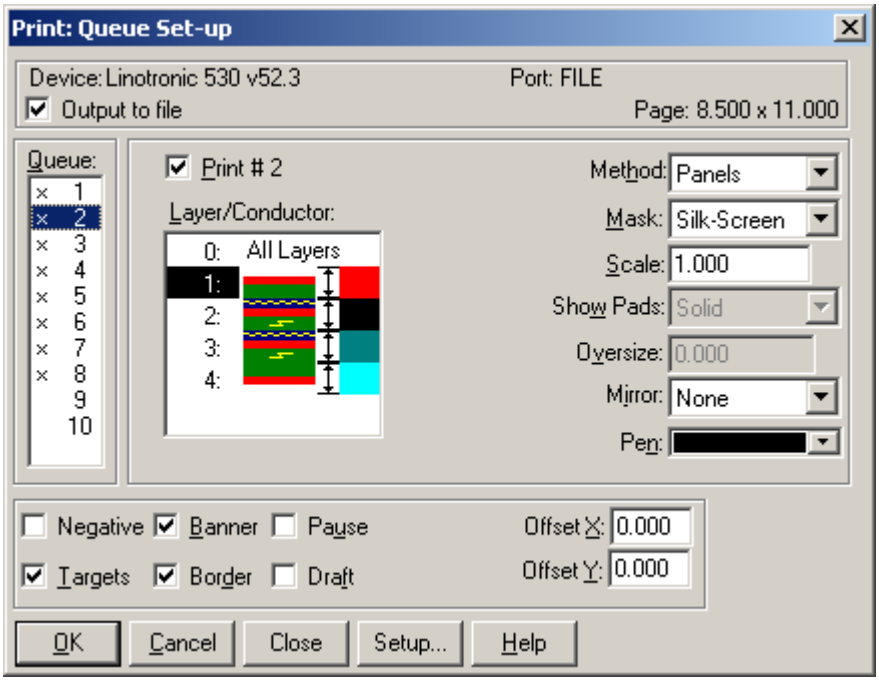
Execute the FILE/PRINT (or press CTRL+P) command. The dialog-box for printing allows you to queue up to ten layers or masks to be printed. To queue a layer or mask to be printed or plotted, select one of the queue numbers (1-10). This queue is enabled when a check mark appears next to the queue number. Each time the queue is selected, the check mark will appear and disappear. In the figure to the right, queue number one is highlighted with the focus bar and enabled with a check mark. All other items in this dialog box are referenced to the queue that is highlighted. For example, in the dialog box to the right, we have only one layer or mask queued to be plotted. The layer selected for queue number one is layer 1. The mask is conductor. Pads will be printed as solid. Targets, Banner, Border, and Pause after each page are all enabled for this first plot queue. If we had more than one layer or mask that we wanted to queue for printing, we could then select queue 2 and choose the options for this second plot.

Queue the following masks for the tutorial board:

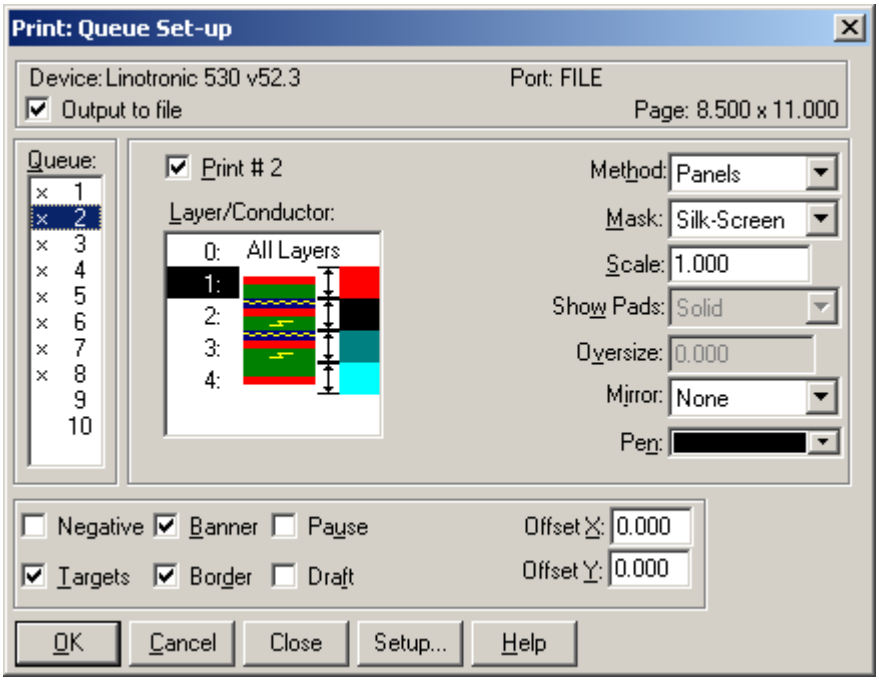
1. Layer 1, Conductor mask. (Top Side Conductor)



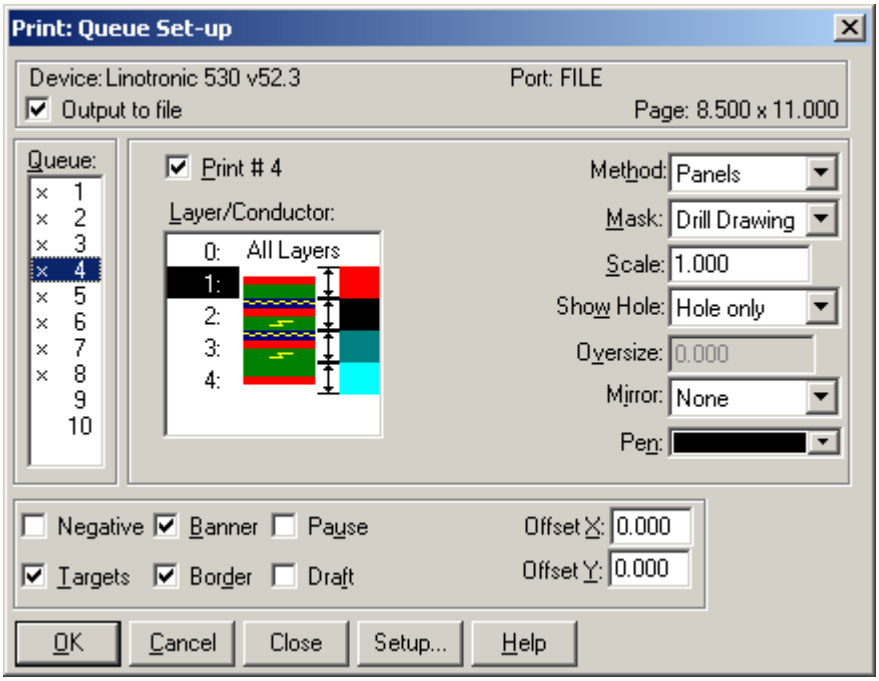
2. Layer 1, Silk-screen mask. (Top Side Silk Screen)



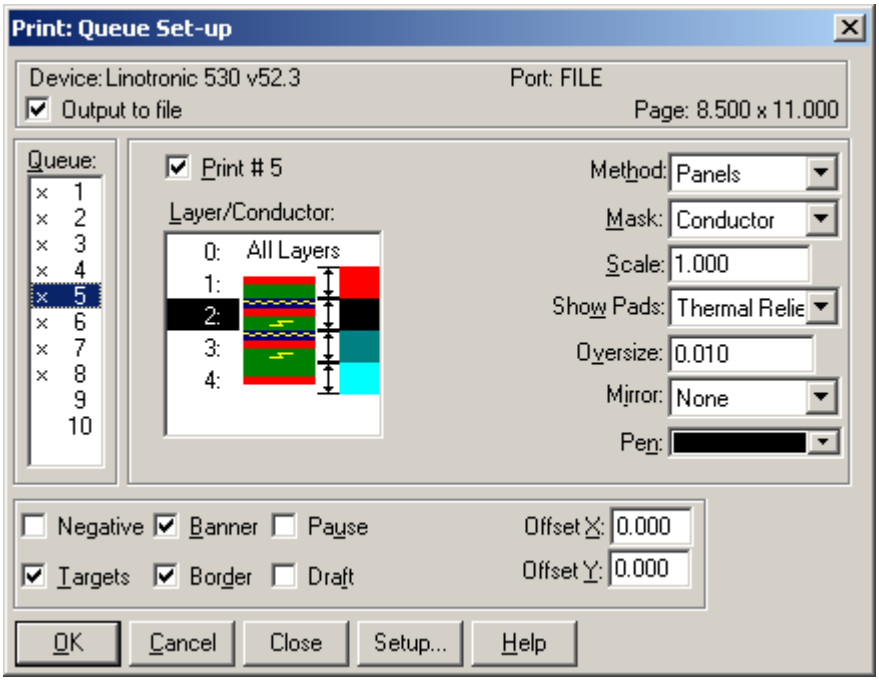
3. Layer 1, Assembly drawing. (Top Side Assembly)



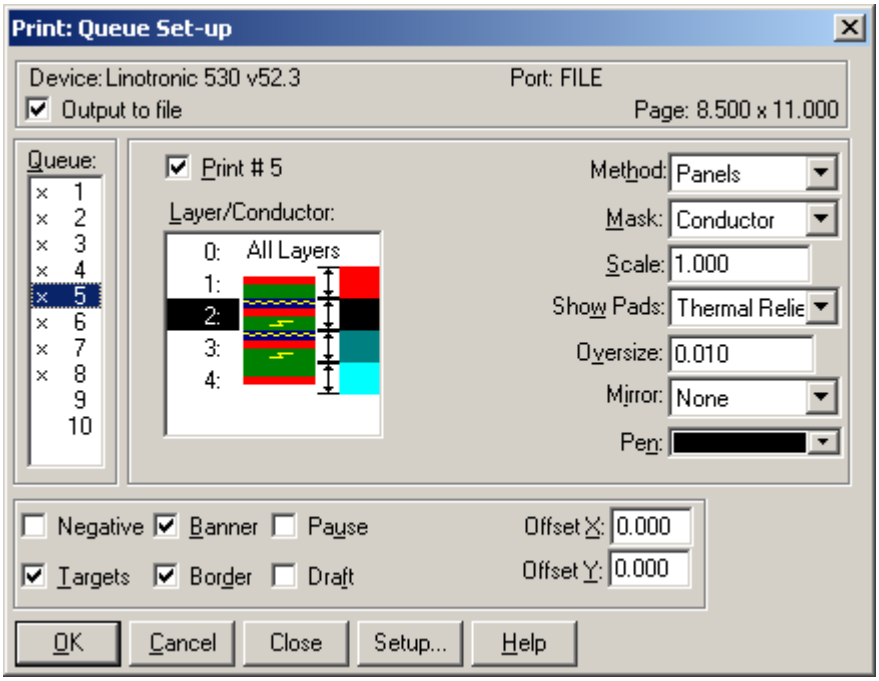
4. Layer 1, Drill Drawing. (Top Side Drill Drawing)



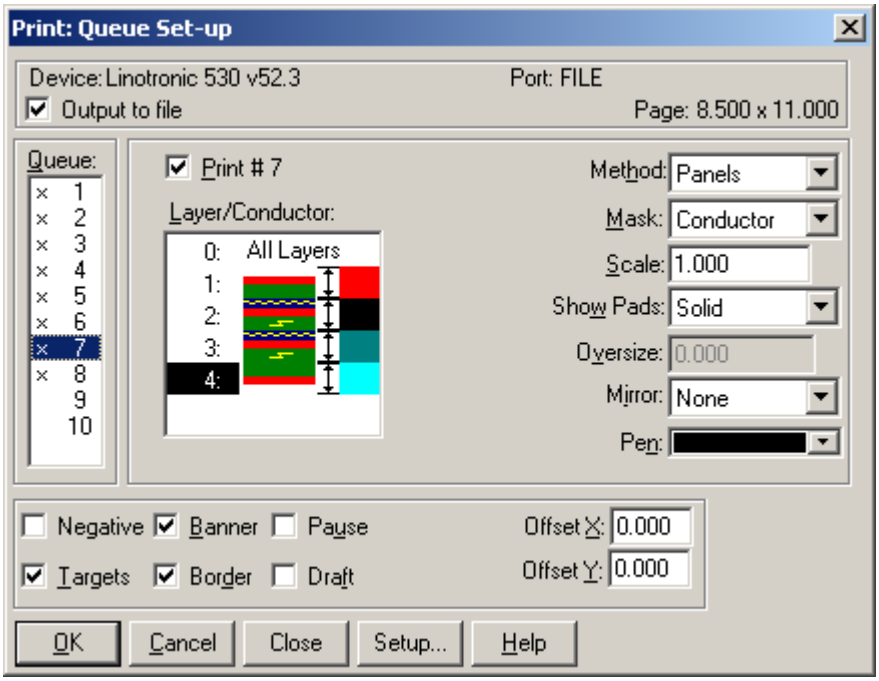
5. Layer 2, Conductor mask. (Inner-Top Side Conductor)



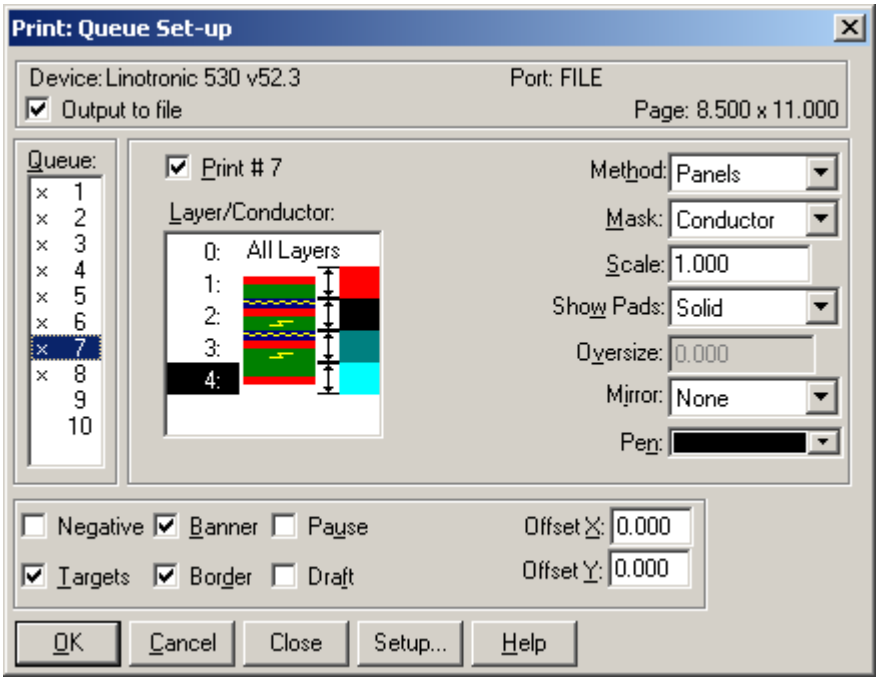
6. Layer 3, Conductor mask. (Inner-Bottom Conductor)



7. Layer 4, Conductor mask. (Bottom Side Conductor)



8. Layer 4, Solder mask. (Bottom Side Solder Mask)

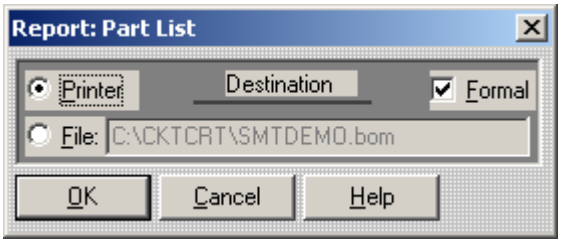


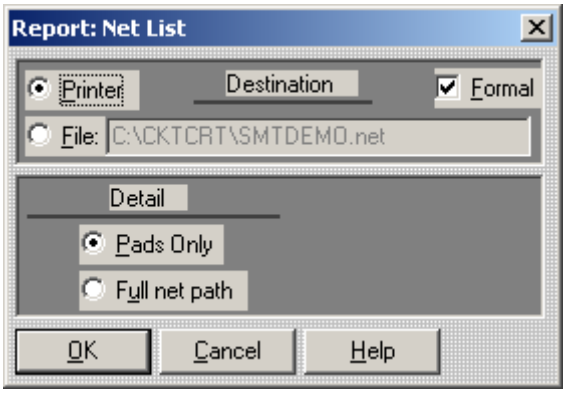
Enable TARGETS, BANNER, and BORDER. Set the other options as desired. If your printing device requires you to change paper between each print (as required by some plotters), enable the PAUSE option. Review the settings and make sure a check mark, indicating that the queue slot is enabled for printing, appears next to each of the first seven queue slots. Press OK to begin printing the masks.

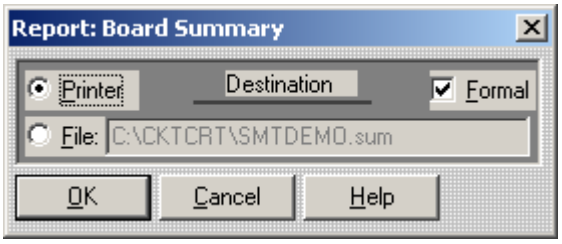
For most boards, it is necessary to set-up the printing parameters for a board only once. The printing set-up parameters are saved with the board.

Reports

From the commands on the REPORT menu, generate a PARTS LIST, a NET LIST (pads only option), and a SUMMARY report. If you have a printer, you may direct the reports directly to the printer. Compare these reports with the schematic net and parts list. They should agree.

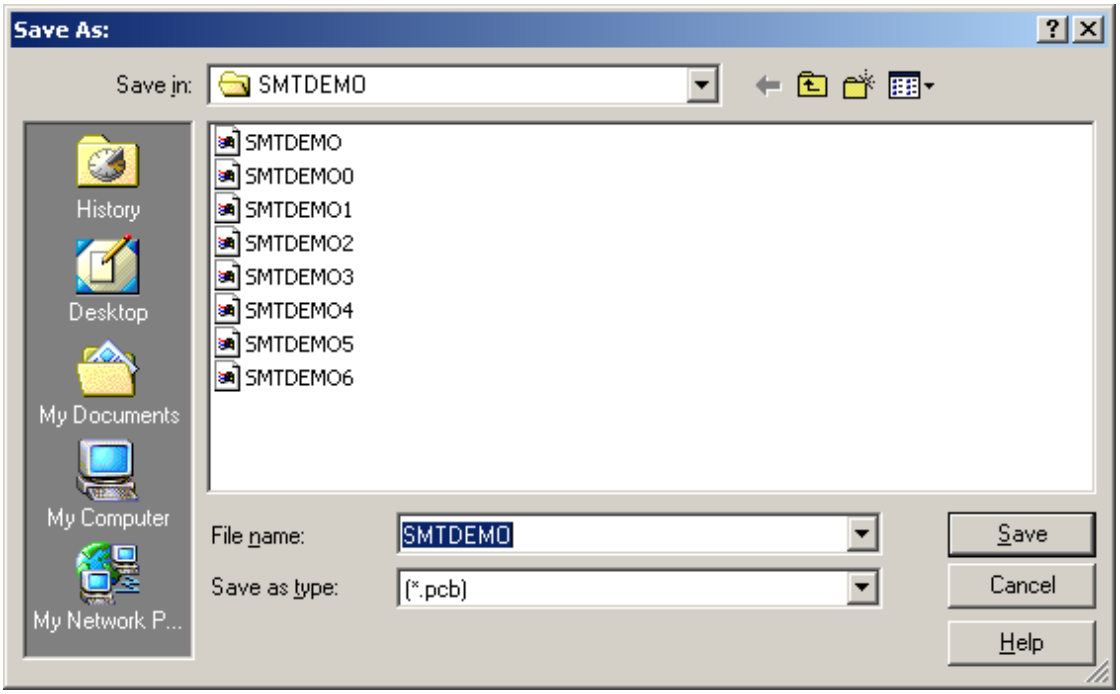






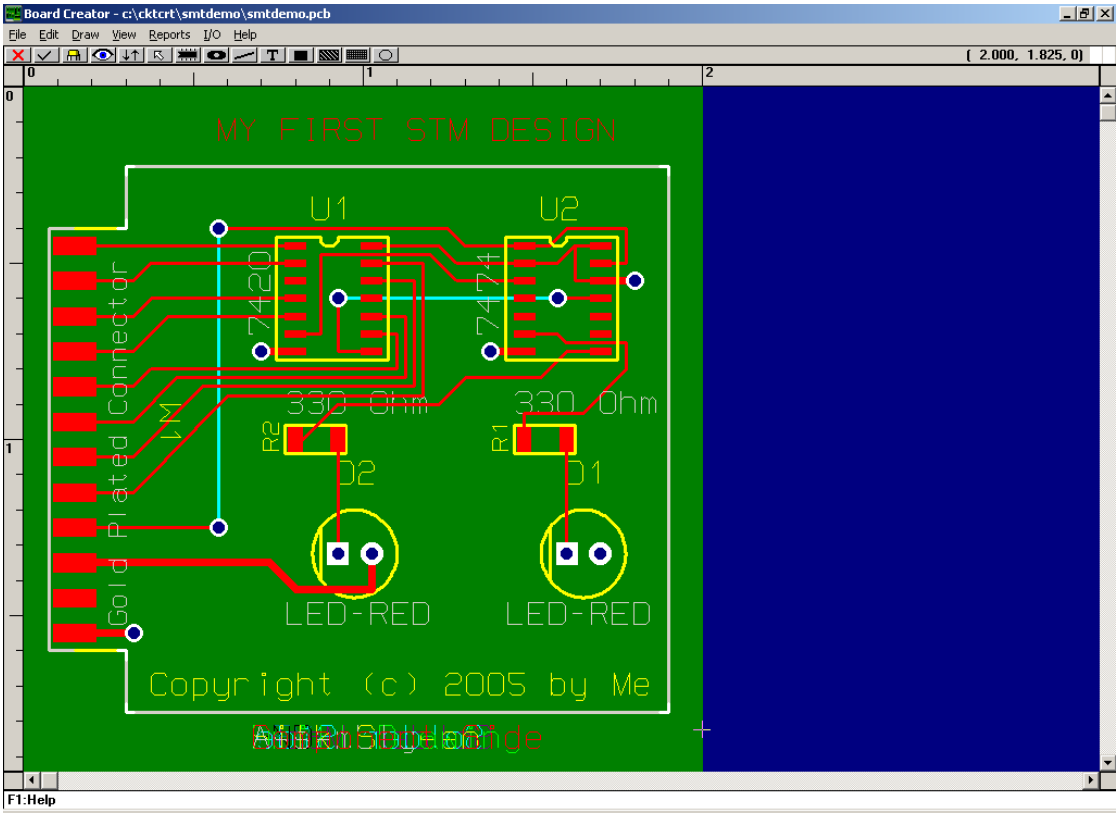
Saving the Board

Now we will save our board to a file. Select SAVE AS command from the FILE menu. Enter "SMTDEMO" for the file name and press OK. This will save the board into the file "SMTDEMO.PCB".



Exit the Program

We are finished with the board for now. Select the EXIT command from the FILE menu, or just press ALT-X, to exit the program.


Board Creator - c:\cktcrt\smtdemo\smtdemo.pcb
File Edit Draw View Reports I/O Help
(2.000, 1.825, 0)
MY FIRST STM DESIGN
U1
U2
7420
7474
330 Ohm
330 Ohm
R2
R1
D2
D1
LED-RED
LED-RED
Gold Plated Connector
VM
Copyright (c) 2005 by Me
F1:Help

Circuit Creator "Hot Keys"

Logic Creator Hot-Keys

The FILE Menu

OPEN	**CNTRL + O**
New	**CNTRL + N**
Save	**CNTRL + S**
Save As	**CNTRL + A**
Hierachy	**Push +**
Hierachy	**Pop -**
Print	**CNTRL + P**
Setup Printer	**I**
About	**H**
Exit	**ALT + X**

The DRAW Menu

Wire	**W**
Net Name	**N**
Port	**P**
Bus	**B**
Line	**L**
Text	**T**

The EDIT Menu

Format	**E**
Undo	**F2**
Cut	**SHIFT + DEL**
Copy	**C**
Paste	**SHIFT + INS**

Delete	**DEL**
Update	**U**
Select	**S**
Unselect	**Z**
Select Trace	**R**
Break Line	**CTRL + B**

The Library Menu

Freshen	**F**
Add Part by Library	**A**
Add Part by Name	**J**

The VIEW Menu

Sheet	**V**
Grid	**G**
Pan Up	**PgUp**
Pan Down	**PgDn**
Pan Right	**>**
Pan Left	**<**
Pan Center	**SHIFT + NUM 5**
Zoom In	**CTRL + PgUp**
Zoom Out	**CTRL + PgDn**
Zoom Home	**CTRL + Home**
Redraw	**CTRL + R**

The REPORTS Menu

Status	**Q**
Design Check	**D**
Formal	**Y**
Back Annotate	**K**

Board Creator “Hot Keys”

The FILE Menu

Open	**CNTRL + O**
New	**CNTRL + N**
Save	**CNTRL + S**
Save As	**CNTRL + A**
Load Symbol	**CNTRL + L**
Save Symbol	**CNTRL + V**
Print	**CNTRL + P**
Setup Printer	**I**
About	**H**
Exit	**ALT + X**

The DRAW Menu

Component Ref	**N**
Pad	**P**
Line/Curve	**L**
Text	**T**
Solid Area	**D**
Void Area	**V**
Region Pour	**R**
Arc	**A**

The EDIT Menu

Board Format	**E**
Break Line	**B**
Undo	**F2**
Cut	**SHIFT + DEL**
Copy	**C**
Paste	**SHIFT + INS**

Delete	**DEL**
Update	**U**
Select	**S**
Unselect	**Z**
Toggle Layer	**F9**
Component/Flat	**SPACE BAR**

The View Menu

Layer/Masks	**M**
Grid/Cursor	**G**
Pan Up	**PgUp**
Pan Down	**PgDn**
Pan Right	**>**
Pan Left	**<**
Point	**F4**
Center	**SHIFT + NUM 5**
Locate	**K**
Snap to Pad	**F**
Zoom Home	**CNTRL + Home**
Zoom In	**CNTRL + PgUp**
Zoom Out	**CNTRL + Pgdn**
Redraw	**CNTRL + R**

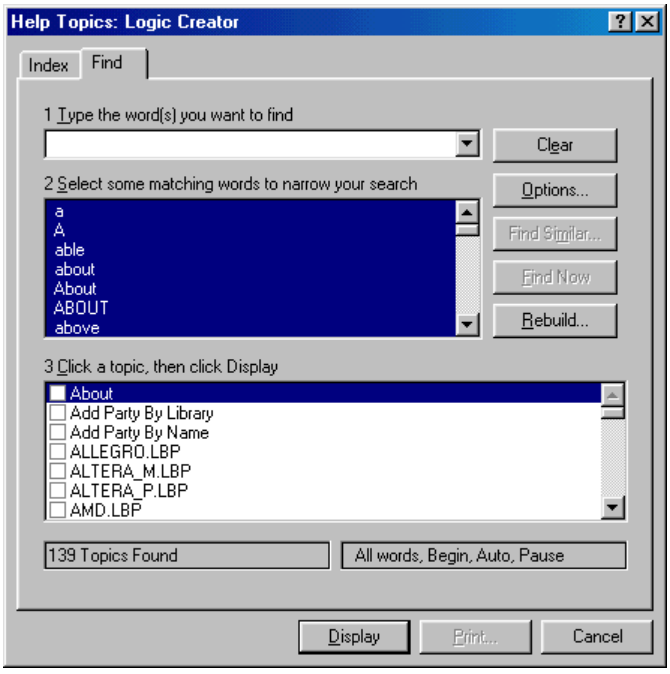
Frequently Asked Questions

LOGIC CREATOR

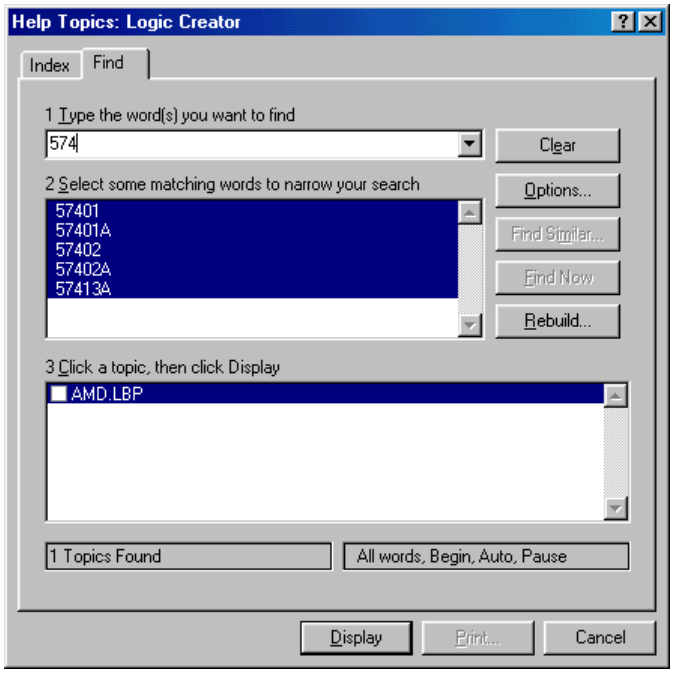
How to find library symbol:

All schematic symbols are categarized into various Library file with the extensipn LBP. Most part library are self explanetory for example 74LS or Intel or Motorola. But some part number, for example 57402A, _____may not lead to specific group. The following procedure will make it easy for you to find these type of unique parts.

First go to Help pull down menu and select Contex and Index. Then click on Index then press Find and the following screen will show.



Now type in the number you are looking for, in our case 57402 and the bottom half of the screen will show AMD.LBP. It tells us that our part is in AMD library.



If you do not find the part you are looking for this way use the following method.

There two files text files provided on the CD called CKT_LIBLIST.TXT and CKT_LIBLIST.DOC.

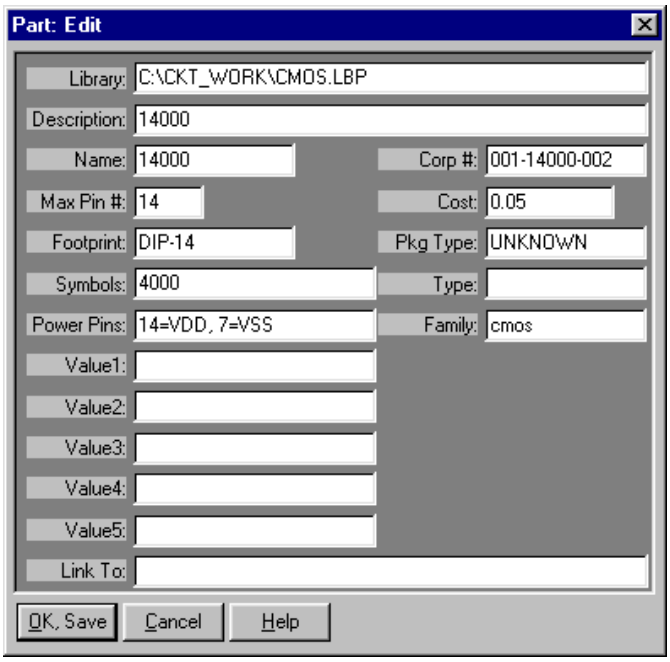
These files shold be in your CKTCRT directory. If you don't have these files, you can download the file from our web site : www.advancedmsinc.com/download

TXT file is simple Text file that can be opened by NOTEPAD or WORDPAD. DOC file is Microsoft WORD compatible file. Open one of these file and using Find function you can look for the desired component.

Copy Part / Symbol from one library to another library

Copy Part

Select Library under Part. Select Part. You will see following dialog box.



Enter new library name (including Path name if different) where you want to move the part to in section Library.

Press OK,Save

Copy Symbol

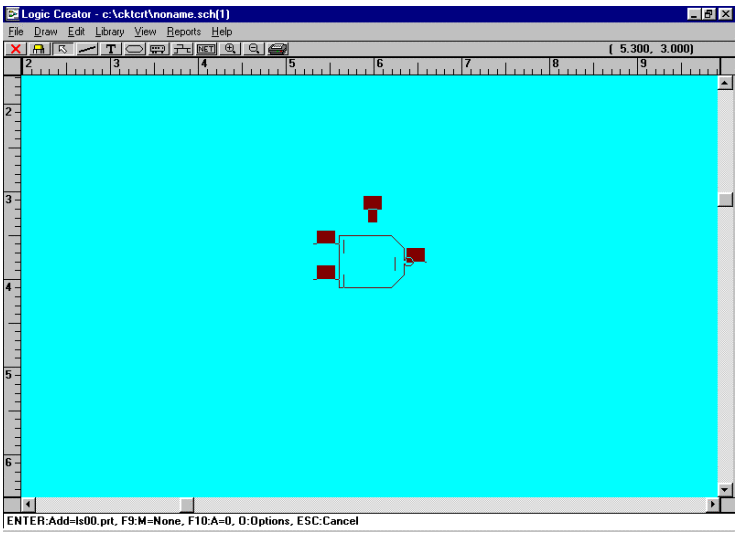
Now select symbol library and pick symbol you want to move. While the part is on your screen, select new library whwer you want to move this part. Then Select SAVE under SYMBOL.

How to Rotate Components:

When you are placing a component from symbol library on to schematic sheet, pick the component from the symbol library. Move the part at a desired location but do not click Mouse or press Enter. As it shows on the following figure the component does not show the pin number and components reference number clearly. Also notice the status line on the bottom of the screen. It shows

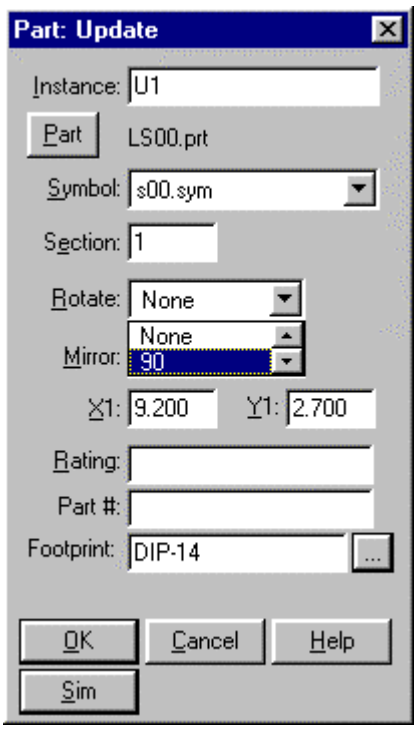
ENTER: Add=ls00.prt, F9=M=None, F10:-A=0, O=Options, ESC: Cancel

F9 function key allows to mirror the part and F10 function key allows to rotate the part in 90 degree increment. Once you have the desored location and desired direction of the part you are placing, Press Enter or Right Click Mouse. Now by pressing F10 you can rotate the part.



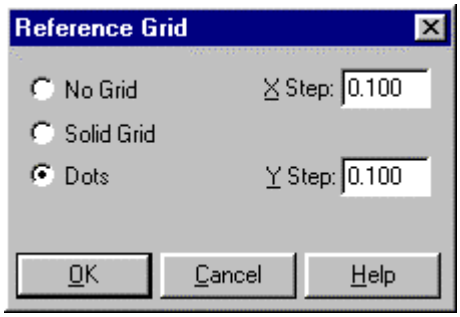
Rotate the Part after Placing on the Drawing Sheet

Once you have placed the part, double click the component. You will see the following screen. Select the correct option on the Rotate menu. Here you have four selections. None, 90, 180 and 270 Degrees. Select the deisired direction and Press OK.

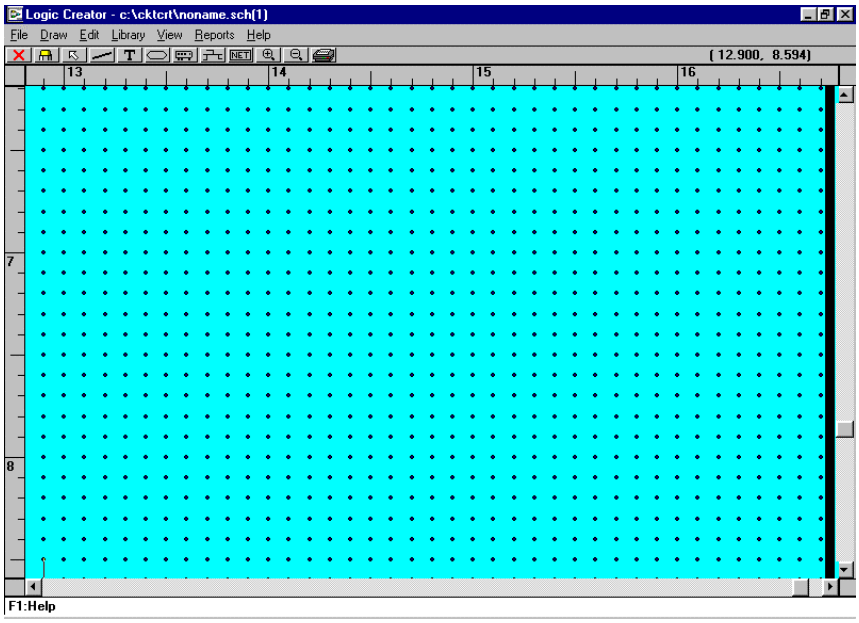


Grid : There are three options for the Grid selection as shown on the following figure.

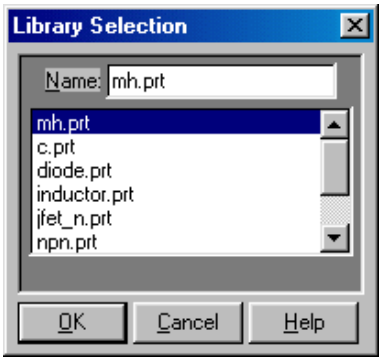
No grid, Solid Grid and Dots. Solid Grides and Dots have option for X and X Step.



The following screen shows Grid with DOTS selection.



Adding or Placing Discrete components:

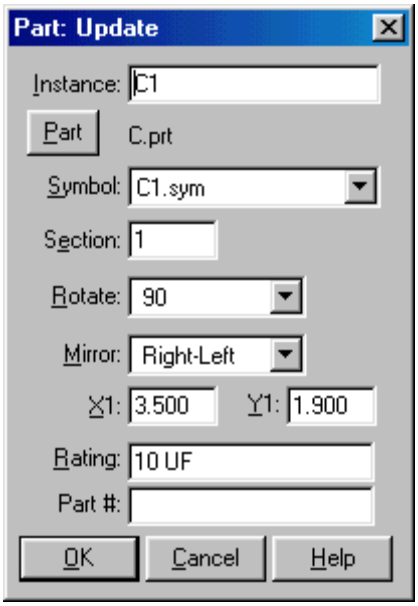


Adding Discrete components in Logic Creator has been made much easier in the latest version in compare to previous version as well as some of competitor version. In previous version, if you want to add a resistor, first you will need to create specific

graphical representation Symbol and part detail. Now you will be able to add all the discrete parts such as resistor, capacitor, inductor, crystal etc in the following EASY way.

First of all make sure you have configured " Discrete Components " as a one of the configured Library. To add a part go to LIBRARY Pull-down menu and select Discrete Components. You will see following Dialog Box.

Select C.PRT (to add a capacitor) from the available selections, and place symbol at the desired place. If you need to mirror the part press F9 and F10 to Rotate 90 degrees. Now Double click on the Capacitor you just placed on the sheet and the following Dialog box will appear.



Enter the desired capacitor value in Rating selection and Corp part number in Part # selection. Select desired Footprint in Footprint Selection. If you are not sure about the Footprint, turn on Preview option. Press Ok and you are done. The same procedure can be use to add crystal, inductor, NPN or PNP transistor or FET.

Q. If you initially create a PCB board file from LOGIC CREATOR and then make changes to the schematic, how do I update my BOARD CREATOR PCB file without losing the work already done on my board ?

A. When you have finished making changes to your schematic in LOGIC CREATOR, select the Formal/Export command from the REPORTS menu and generate a new PCB Update list. This will create an update list file with the extension upd. The update list contains part and net list information about the schematic. Next, under the IO pull down menu in BOARD CREATOR there is an option to Update from LOGIC CREATOR where you select the update list file generated from LOGIC CREATOR. This will update the your PCB design respecting component placement and any routes you may have completed.

Q. How do I delete a connect dot in LOGIC CREATOR ?

A. Selecting objects in LOGIC CREATOR is very similar to selecting objects in BOARD CREATOR. You have several SELECT options:

Bound by - means that only the objects completely *bound by* the marked area will be selected.

Touching - means that any object *touching* the marked area will be selected.

Therefore, in order to delete a connect dot, choose the *Bound By* option as your select mode option and *mark* an area around the connect dot. Once the connect dot has been selected, press the DEL key.

Board Creator

Q. How to Rotate Object on the PC Board

A. Board Creator allows you to Rotate the Part (Object, or Footprint) in the increment of 90 Degrees. First select the Part and then without moving the mouse, Press and Hold the left mouse button to pick up the part, but don't move the part. Continue to hold the left mouse button and Press F10 or Key O. Keep pressing F10 or key O until you get your desired position and orientation.

Q. Does the BOARD CREATOR have DXF output ?

A. Yes, Under the IO pull down menu in BOARD CREATOR select Export ASCII and DXF will be one of the output formats.

Q. How can I add or change the pen widths defined in my pen carousel and pen speed for my particular pen plotter ?

A. In order to add or change any printer/plotter options such as init strings (including pen speed), draw commands (including pen widths), you must edit the printer configuration file (or PCF file). The PCF files are in ASCII format and can be edited using any text editor. To change the pen speed of a pen plotter, you will need to modify the init string:

init "IN;PU;SP 0;VS 20;\13\\10\"

Where VS 20 sets the pen speed of the plotter. Refer to the reference manual included with your pen plotter for recommended pen speed values. To add pen width definitions (as in the case of a pen carousel), add the following similar information to the printer/plotter configuration file (PCF):

WIDTH x "SP y;"

Where x is the width of the pen (in mils) and y is the position of the pen in the pen carousel.

Q. **Is there any way to modify a regio**n pour (i.e. break lines or update individual lines) after the copper has been put in place ?

A. The region pour is placed as one component, which makes it very easy to select and remove if you need to make modifications on your PCB design. However, you can remove individual lines of the region pour if needed.

There are several options that can be used when selecting objects: touching, bound by, flat, or component. To configure these options, choose the Select command from the EDIT menu and then press the letter O for options.

Touching - when an area is selected, any object touching that "marked" area will also become selected.

Bound by - only the objects completely bound by the "marked" area will become selected.

Component - the entire component (or *footprint*) will be selected when any part of the component is selected.

Flat - the *Flat* mode is very unique in that it allows you to edit, update, or move individual objects that belong to a component (or *footprint*) without selecting the entire component.

Q. **The region pour feature in BOARD CREATOR should fill as a solid area and not cross hatched.**

A. If you require that the region pour fills as a solid area, simply set the line width of the line used in the copper pour slightly greater than the *x* and *y* step. This will cause the lines to overlap giving the appearance of a solid area of copper.

Q. How do I relate footprints in BOARD CREATOR to the pins on my symbols in LOGIC CREATOR ?

A. The symbols in LOGIC CREATOR contain the physical pin numbers on the symbol itself. Similarly, the *footprints* in BOARD CREATOR have pin numbers assigned to each pad used on the *footprint*.

Q. How do I create a new footprint in BOARD CREATOR ?

A. Footprints are created using the board editor (BOARD CREATOR). First select a New board under the FILE pull down menu. The default board size and settings will do. Next, select Pad from the DRAW menu and choose a pad (1-25) with the dimensions that you wish to use on your *footprint*. To display the pad options, press the letter O on the keyboard after selecting the Pad command. If you do not find a pad with the size and shape that you require, re-define one of the pad's dimensions by selecting Edit System Pad Table from the SYSTEM menu. (Note: pad types 1, 3, 7 and 9 are commonly used in all the DIP, DB Connectors, and discrete component footprints, so we suggest that you do not redefine these pads.) When you have selected a pad type, you may begin placing the pads on the board (with the proper pad spacing as required for your footprint) by simply *clicking* the right mouse button. The first pad that is placed will be assigned pin #1, the second will be pin #2, and so on. The pin number will be automatically incremented as you place the pads, therefore, you will want to place the pads in the order that they are numbered for your footprint. When you have finished placing pads, you will need to add a component reference by selecting Component Ref from the DRAW pull down menu. (Add U as a component reference to an IC, R to a resistor, C to a capacitor, and so on.) You may finish your footprint by outlining its shape by selecting Line/Curve from the DRAW menu and place these lines on the silkscreen mask. Finally, you will need to *select* the entire footprint that you have just created and select Save Symbol from the FILE menu and name your footprint. This footprint will be saved to a file with an extension PCS.

Q. When I try to do Create from LOGIC CREATOR in BOARD CREATOR I get the error message "cannot open .log" ?

A. This error message will occur when the path names configured for BOARD CREATOR are incomplete. Under the SYSTEM menu choose Paths/Options and check the path name configured for Drawing, Scratch, and Check Point. This path name must be complete. A valid path is as follows:

C:\BOARD CREATOR\PCB\

The path name must also be terminated with a backslash (\).

Q. I get the error "part not recognized" on a part that I created when I try to generate reports in LOGIC CREATOR. I know my part and symbol are in the libraries. What should I do ?

A. Check the part definition for your part using Symbol Creator. You may not have defined enough pins for your part in the *MaxPin#* field. The number you enter in this field must be greater than or equal to the largest pin number on your symbol. For instance, the part PHONEJK (a RJ-11 phone jack) uses a symbol that has four pins (pins 2, 3, 4, and 5). The part definition will need to show that this part has five pins since the largest pin number on the symbol is five. When you generate an UPDATE list with the PHONEJK part on your schematic, five pins will be listed for this part (however, pin #1 will be assigned as a NC pin).

Q. When I run Check/Schedule in BOARD CREATOR I get the following error message:

```
        Note!
====================
Routes not complete
132
```

What am I doing wrong ?

A. This is not an error message. This message is telling you that based on your net list, you (or the auto-router) will need to connect 132 pairs of pads together in order to complete this design. A route is *complete* when two pads are connected (or joined together) by placing lines on a conductor mask. These routed lines result in a copper track on the finished PC board.

Q. How do I move a group of items from one layer to another in BOARD CREATOR?

A. This is accomplished using the Cut and Paste options. First *select* the items you wish to move. Next, select Cut from the EDIT pull down menu. These items will be stored in the "paste buffer". Now select Paste from the EDIT pull down menu and a box will appear showing you the area of the objects to be pasted. Before you "click" the left

mouse button (or hit <ENTER>) to place the objects, press the letter O on the keyboard and you will be able to select the layer/mask that the objects will be "pasted" to.

Q. Every once in a while I see a message appear that says "saving to checkpoint" file. What is this ?

A. LOGIC CREATOR and BOARD CREATOR have an auto-backup feature that periodically saves your current drawing or PC board to a check-point or backup file in case of a power loss. You can specify how often this backup occurs by selecting Paths/Options from the SYSTEM menu in both LOGIC CREATOR and BOARD CREATOR. The number entered for the checkpoint is in seconds. Entering a zero (0) will disable this feature. If a power loss is encountered, when you try to load your work again, you will be prompted that a checkpoint file exists and if you want to load this *backup* file instead.

Q. When plotting Gerber in BOARD CREATOR, it always says "making sheet 1 of 2". I know that I have selected a large enough film size to fit my board on. What am I doing wrong ?

A. You never want to make two or more "plot" files for a single layer/mask when creating Gerber plots. First, check the "actual" size of your board. This is determined by selecting Board Format in the EDIT menu. The "width" (which is the horizontal measurement) and the "length" (which is the vertical measurement) are entered in this dialog box. This is your board size. The second problem that occurs while plotting Gerber is that a one inch border is added to all the sides of your board. Therefore, you will need to add two inches to your board's dimensions when selecting a film size. Finally, the film sizes defined in the Gerber configuration file (PCF) are oriented (L x W) whereas the dimensions of your board are (W x L). For example, if your board's dimensions are 6" x 9", you will need to select a film size that is larger than 11" x 8".

Q. How do I rotate a footprint ?

A. To rotate a footprint, it must first be selected. Once the footprint is selected, place the cursor in the selected area and press and hold the left mouse button. Continue holding the left mouse button and with the other hand press the F10 key or the letter O to rotate the footprint in ninety degree increments.

Rectangular pads of a footprint will only rotate when there is a pad defined in the PAD TABLE that has the opposite dimensions of the pad used on the footprint. For example,

let's say a footprint is using a pad type # 9 (which is a 20 x 60 rectangular pad with a 0.031 diameter hole). In order to rotate this footprint and also have the pads to appear as if they had rotated, there would need to be a pad (let's say Pad # 10) defined as 60 x 20 rectangular with a 0.031 diameter hole. When the footprint is rotated, BOARD CREATOR automatically replaces the pad with a another pad having the exact opposite dimension. If such a pad does not exist, the pad in the footprint will appear to have not been rotated.