

Eagle CAD Tutorial

Eagle0 R0.2

EE401
Design

Revision History

Rev.	Date	Contributor	
0.2	January 2004	L.Wyard-Scott	Format changes.
0.1	October 2003	L. Wyard-Scott	Format changes: no content change.
0.0	September 2003	Vincent Sieben, L. Wyard-Scott	Creation.

Contents

1	Overview	1
1.1	Creating a Project	1
2	Schematic Editor	3
2.1	Adding Schematic Components	3
2.1.1	Adding a Component from the Standard Libraries . .	4
2.1.2	Adding a Component from the EE401 or Other Libraries	5
2.1.3	Adding Power Sources	6
2.1.4	Adding a Frame	6
2.1.5	Adding Text	7
2.1.6	Changing Component Values	8
2.1.7	Move, Rotate, and Delete Commands	8
2.1.8	The Group Command	10
2.2	Adding Nets	10
2.2.1	Buses versus Nets	11
2.3	ERC - Electrical Rule Check	13
3	PCB (Printed Circuit Board) Editor	15
3.1	Placing Footprints	16
3.1.1	Part Placement Guidelines	17
3.2	Routing the Board	17
3.2.1	Routing Tips	17
3.2.2	EE 401 Milled PCB Routing Guidelines	18
3.2.3	Auto Routing	19
3.2.4	Ripping Up Tracks (Delete Tracks)	21
3.2.5	Manual Routing	23
3.3	DRC - Design Rule Check	24

4	Building a Custom Library	25
4.1	Update Schematic and PCB	29
5	Exporting Schematic and PCB Images	31
5.1	Exporting to Encapsulated Postscript	31
5.2	Exporting to Bitmap	33
	Bibliography	34

Chapter 1

Overview

In this tutorial, you will design a simple LED flasher PCB (Printed Circuit Board) using the Eagle Software package. This tutorial is intended to provide a basic introduction to the Eagle PCB design package and will guide you through the Eagle program's features in the proper design order. A project starts in the Eagle schematic Editor with the electrical schematic and then leads into the Eagle PCB Layout Editor where the board is designed and routed. By the end, you will have the basic tools necessary to start on a more serious design.

1.1 Creating a Project

An Eagle project encompasses all design work required in making a PCB. To create a project follow the instructions below.

1. Open Eagle
2. Select **File > New > Project** from the menu bar. Alternately, right-clicking on a project directory will bring up a context-sensitive menu with the same selections. If required, new or existing project directories can be added by selecting **Options > Directories** and appending a directory path to the "Projects" entry. New schematics and board layouts can also be created in this manner.
3. Name the project - **LED_Flasher** in the Control Panel Window.

Next a schematic document is to be created within the project directory. Follow the instructions below to create a schematic document.

1. Select the LED_Flasher Project Folder by left clicking on the dot to the right of the project name. The dot should light up **green** when the current project is selected, see Figure 1.1.
2. Select **File > New > Schematic** from the menu bar. A schematic editor window editor will open.
3. From the schematic window, save the file as **led.sch**. This is done by selecting **File > Save** (or **Save As...**) from the menu bar.

Finally, you have to create a new PCB document within the project named **led.brd**. This file will be the board layout. Figure 1.1 below is what the eagle control panel window should look like when these steps are fully completed. The project contains all necessary files and is ready to start adding parts to create the electrical schematic.

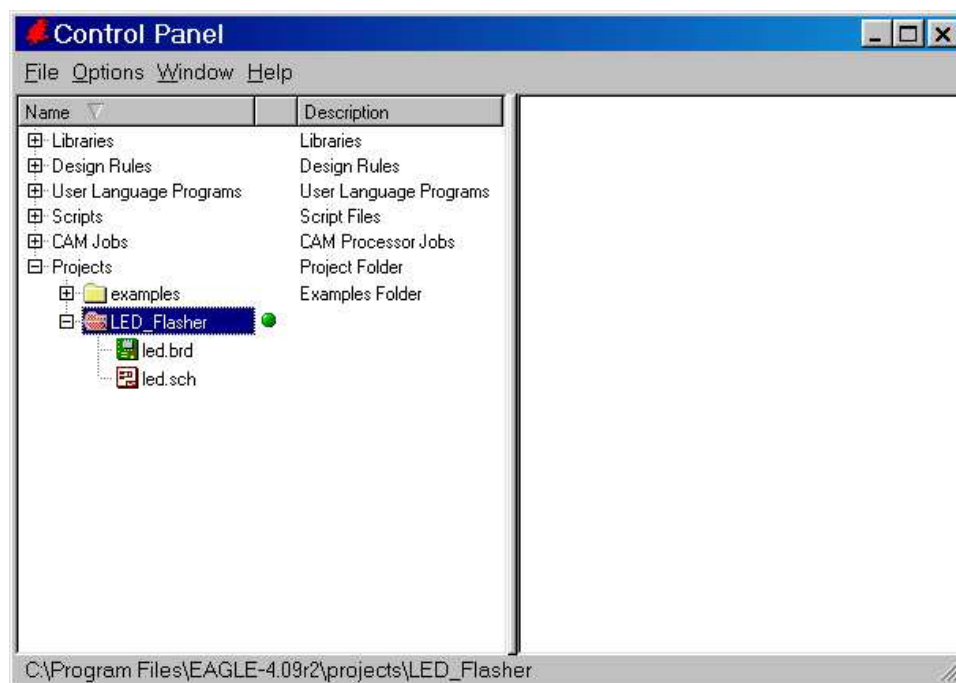


Figure 1.1: Successful project file creation.

Chapter 2

Schematic Editor

The first step in the design process for most electrical systems starts with a schematic. The schematic editor within the Eagle package allows the user to create a visual representation of the electrical system being designed. A good schematic will allow any electronic literate person to recreate the electrical system. This portion of the tutorial will guide you through creating a schematic for the example LED Flasher circuit. The procedure below provides you with the necessary tools to start on your own project schematic.

2.1 Adding Schematic Components

Components are added to the schematic diagram from part libraries. A library is composed of several components and each component within the library has a schematic and a PCB representation. The schematic representation in the library is an electrical symbol, where as the PCB representation of a component is known as a footprint. Figure 2.1 shows an example of a 555 timer component within the "Linear IC" library. Note in the middle window the schematic symbol is shown, and in the right window, the PCB footprint is shown.

Eagle comes with many standard libraries that include a large variety of parts. However, if a component is not listed in the standard libraries, an Internet search might provide the library needed, or else you can create the component representations yourself, as described in Chapter 4. An "EE401" library has also been created, contains contains many of the common components used in the course. To add a component in the schematic editor follow the procedure below.

2.1.1 Adding a Component from the Standard Libraries

1. In the schematic window select **Edit > Add** from the menu bar. Alternatively, there is a tool bar to the left of the schematic window which has several shortcuts. If the mouse is hovered over the toolbar, a comment will appear indicating the icons command function. Either way, a new window will pop up displaying available libraries as shown in Figure 2.1.
2. In the search edit box, type ***555*** and hit **Enter**. The search results are displayed in the upper left scroll box. The ***** serves as a wild card, so that any component in the available libraries containing the text "555" will show up in the search results. To start a new search simply clear the edit box, hit Enter once and the search criteria will be reset.
3. Expand the Linear Library and then the **"*555"** heading. To expand the general headings click on the **"+"** sign.
4. Find the LM555N in the upper left scroll box and select it by clicking on it. It is important to note the details in the right windows when a component is selected, particularly note the schematic symbol and the PCB footprint package. There are many types of 555 timers in the linear library with the main difference being the PCB footprint. The "footprint" package of interest to us is an 8 pin DIP (Dual In-line Package), which corresponds to the LM555N. See Figure 2.1.

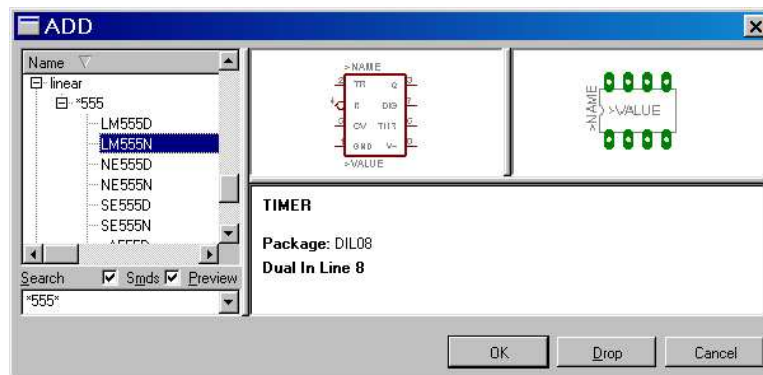


Figure 2.1: Edit box while adding a 555 timer.

Whenever possible it is best to select the same component package that you will be using on the PCB. In the situation where you are not certain of the package type, it is possible (and necessary) to change the package type at a later time.

5. Ensure the LM555N is selected and hit **OK**.
6. The program will now return to the schematic window and the component is placed with the mouse. Click the left mouse button when you want to add the component, and click the right mouse button to rotate the component. Mirroring of the symbol can be performed only after the component is placed.
7. Press **ESC** on the keyboard when you are done placing a single 555 timer.
8. Now you should be back in the “Add” component window. Press **ESC** or click **Cancel** to get back to the schematic editor; however, if you want to add more components you can do so as mention in the steps above.

To complete the LED flasher circuit the rest of the components must be added. For resistors and capacitors, the custom EE401 library will be used as shown in the following section.

2.1.2 Adding a Component from the EE401 or Other Libraries

The first step involved in adding components from an external library is to include or “USE” the library file. Library files have the extension *.lbr. Follow the directions below to add resistors from the EE401 library.

1. Select **Library > Use** from the menu bar in the schematic editor window.
2. Locate the directory where the desired library file (EE401.lbr) is stored in.
3. Select the EE401.lbr and click **OPEN**. The library is now included as part of the standard libraries for this schematic file when adding a component. To add the component use the add command as if EE401 is part of the standard libraries using the procedure above.

4. Add the component **R3** under the EE401 library. Add 3 of these resistors.

The parts added above are a few required to completed the LED flasher circuit that will be constructed. Below is a list of parts that will be required to complete the design. Ensure all the components have been added before proceeding to the next section.

- 555 Timer Dip Package (**Search: LM555N**)
- 3 Resistors (**Search: R3**), **EE401 Library**
- 5mm LED - Light Emitting Diode (**Search: LED5MM**)
- Capacitor (**Search: CAPP1L**)
- A 2 Pin Power Header (**Search: PINHD-1X2**)

2.1.3 Adding Power Sources

Power Sources such as ground, twelve volts, and five volts are added in the same manner as components. There are several libraries that contain supplies, but use the EE401 library. If special voltages are required, library **supply1** and library **supply2** contain other supply symbols for the schematic.

For the LED Flasher project add:

- 4 - GND supplies (**Search: GND**),**EE401 Library**
- 4 - 12V supplies (**Search: 12V**),**EE401 Library**

2.1.4 Adding a Frame

A frame is mandatory for any schematic. A frame provides the reader with a description of important information regarding the schematic, such as: what the schematic represents, who designed it, and how many revisions it has gone through. Frames are added in the same manner as a component is added, as the procedure below illustrates.

1. Select **Edit > Add** from the menu bar in the schematic editor window.
2. Search for **LETTER_P** in the search edit box. There are many frames that can be used from the “Frames” library; to view all frames do a search for ***frame***.

3. Select the **LETTER_P** click **OK**. Add the frame such that it encompasses the schematic diagram. Press **ESC** twice to return to the schematic editor.

2.1.5 Adding Text

After adding the frame, it is necessary to place text to clarify the schematic information.

1. Select **Draw > Text** from the menu bar in the schematic editor window.
2. Enter the desired text. For the frame, start with the text: “LED FLASHER, BY: Your Name”.
3. Click **OK**. The text will now appear on the mouse cursor; place the text in the top frame box. Similarly add the revision number.

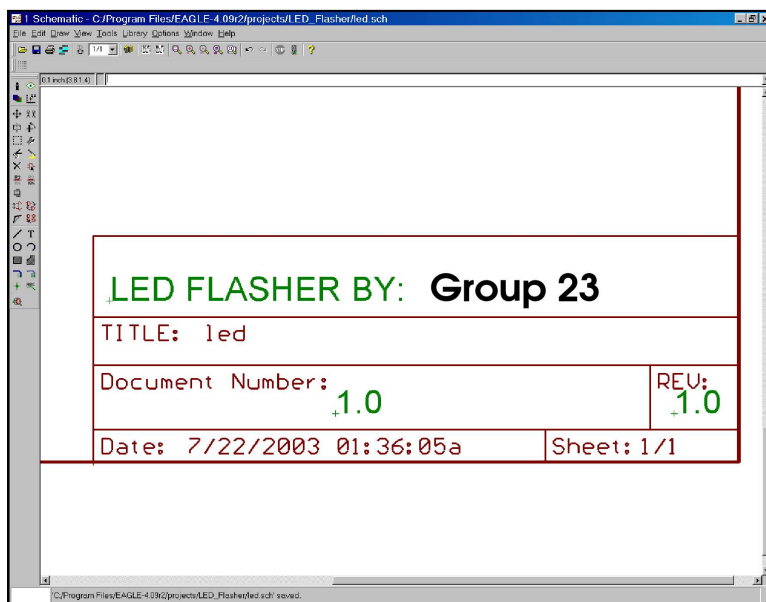


Figure 2.2: A very simple frame box with pertinent information.

2.1.6 Changing Component Values

One of the final steps to completing a schematic is to add labels and values to give each component the required information.

1. Select **Edit > Value** from the menu bar in the schematic editor window.
2. Select the component R3 by placing the mouse cursor on R3's cross hairs and **Left Click**. A new window will pop up requesting the value.
3. Enter the value for the part; for R3 enter a value of **470**.
4. Click **OK**. The value will now appear as text beside the schematic symbol.

Label the remaining components with the appropriate values.

- LED: **Red**
- C1: **10 μ F (10 uf)**
- R1 and R2: **10k Ω (10k)**

2.1.7 Move, Rotate, and Delete Commands

Below is how the move command is used, but the procedure is similar for other Eagle commands.

1. Select **Edit > Move** from the menu bar.
2. Place the cursor over the object's cross hairs (hot spot).
3. Left click once. (For the Rotate and Delete commands, this will institute the action.)
4. Move the object with the mouse to the desired location, and Left click again to release.

By using the Move, Rotate, and Delete commands, each of the components can be organized so that wiring the nets is efficient.

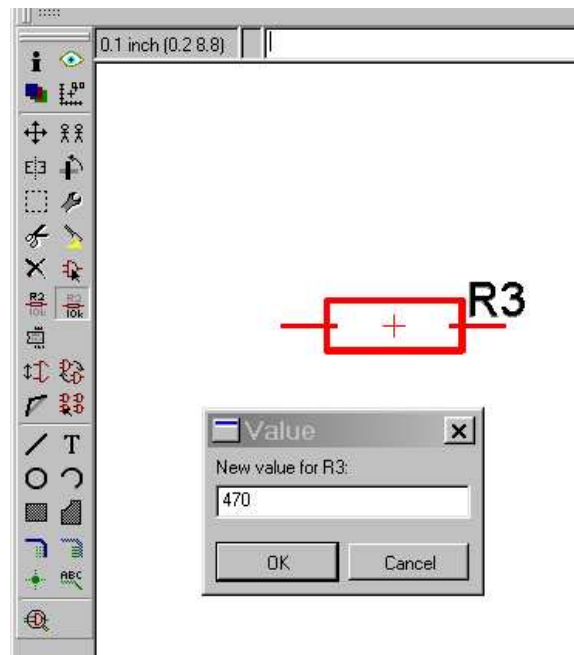


Figure 2.3: Adding a value to a resistor.

2.1.8 The Group Command

The Group command is a special command that can save time. It is used in conjunction with another command such as rotate, move, or delete. Grouping objects will allow the additional command to be issued to several objects at once. The procedure below is an example of using the group and move commands together to move several objects at once.

1. Select **Edit > Group** from the menu bar.
2. Group the desired objects with a rectangular box or a polygon. The objects will highlight when successfully grouped.
 - Rectangle selection box: Left click and hold to start the rectangle. Drag the mouse cursor to create a rectangular box around the objects which are to be grouped. Release the left mouse button to complete.
 - Polygon selection box: Left click and release where the start of the polygon begins. Draw lines by moving the mouse to each point and left click at each bend to form the polygon shape. Click the right mouse button when the polygon is finished.
3. Select **Edit > Move** from the menu bar. At this point, another command can be substituted for the move command.
4. **RIGHT CLICK** once on any object in the group and this will cause the group to be commanded. This functionality is described in the status bar at the bottom of the application window.
5. Drag the group to the new location with the mouse and **LEFT CLICK** once to release.

2.2 Adding Nets

Nets are connections that link various component pins. Below is the procedure for adding a net with a label.

Do not use the WIRE command to create logical connections, use the NET command. Using NET sets internal connection information that is required for routing a PCB. The WIRE command does not allow as flexible control over this feature.

1. Select **Draw > Net** from the menu bar in the schematic editor window.
2. Position the mouse cursor over one of the nodes which will be connected and Left Click once without holding.
3. Drag the wire to the desired positions, each time that you wish to change direction, click the left mouse button. To toggle between **orthogonal and diagonal** draw modes **click the right** mouse button. The current mode is displayed in the menubar at the top of the application.
4. Complete the net at the final node.
5. Edit the Net Name and add a Net Label. To edit the Net name select **Edit > Name** and then click on the net line. When using the PCB layout editor later, meaningfully net names help identify connection paths. To draw a Net Label which shows the net name in the schematic editor, select **Draw > Label** and click on the appropriate net.

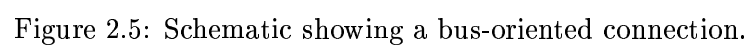
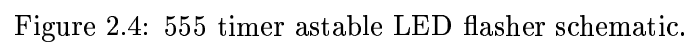
Make all the required nets for the schematic shown below. When adding nets to power supplies, **START AT THE POWER SUPPLY FIRST** and then connect to the other nodes. This ensures that the power supplies net name is persevered for the ERC (Electrical Rule Check) discussed later.

When joining one net to another, Eagle will prompt for the resulting overall net name. Ensure you select the correct option.

2.2.1 Buses versus Nets

In this section, the difference between buses and nets is discussed. A bus is a collection of several nets visually grouped into one line to simplify the appearance of the schematic diagram. Buses are just drawing elements. Logical connections can only be made with the NET command. A simple bus will be added to the schematic to illustrate how buses are used and created. The schematic will look as shown in Figure 2.5 after completing the following steps. This bus serves no purpose electronically and is included only for demonstration purposes.

1. Open Eagle and Open the schematic file for the 555 LED Flasher Circuit designed above.



2. Add two headers, (**Search: PINHD-1X4**). Place them as shown in Figure 2.5.
3. **Create a bus.** To create a bus, select **Draw > Bus**, and then draw the bus as shown in Figure 2.5.
4. **Name the bus.** A bus name must be the names of all net signals that it represents. To name a Bus, select **Edit > Name** and click on the bus to be named. Enter the bus name with the following syntax:
(**ARRAY[0..2],SINGLES**) Arrays are created with the ArrayName[Start..End] syntax and a single net is just the net name. Any number of nets can be added, but Nets must be comma separated.



Figure 2.6: Editing bus names.

5. **Label the bus.** Add a label to the bus.
6. **Connect the NETS to the BUS.** Use the NET command and link all the pins on each pin header to the bus. *Start from the bus*, which will bring up a list of available nets; choose the appropriate net and then connect it to the pin header. Ensure that each net connects to the bus with a “bus entry”, or diagonal line as shown in Figure 2.7. Use the right mouse button to alternate between orthogonal and diagonal modes of drawing NETS.
7. **Add net names** to all nets entering and leaving the bus. This is required in order to make all information about the connections visible to the viewer of the schematic. Skipping this step has no effect on the operation of Eagle, but makes the schematic impossible to read for humans.

2.3 ERC - Electrical Rule Check

The ERC command is a tool that is used to test the schematic for electrical errors. The ERC will only return possible errors and warnings, but it is up

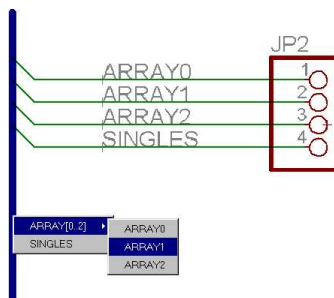


Figure 2.7: Popup menu showing bus names.

to you to ensure there are no electrical violations. To run the ERC, select **Tools > ERC** from the menu bar. When the ERC is run on the 555 Flasher circuit, 1 error and 1 warning should result.

- The warning is due to a naming difference between VCC and V+ for the 555 timer chip which is acceptable.
- The error is due to an unconnected input pin, which is part of the design in this case.

As mentioned before, it is up to the user to interpret the ERC errors and warnings correctly. The ERC can help to save debugging time.

Chapter 3

PCB (Printed Circuit Board) Editor

Almost every piece of modern electronic equipment contains at least one PCB (Printed Circuit Board). A printed circuit board is a fibreglass sheet that provides support for components and connections. Copper traces that are present on the top and bottom surfaces act as wires to create connectivity between components. Eagle's Board Layout Editor provides a means of organizing components on a PCB and creating the connections by drawing the copper traces. Complete the following steps before proceeding to the next section.

1. Open the LED Flasher schematic file.
2. Select **File > Switch to board** from within the schematic editor. This will bring up the board layout editor with the components placed in an unorganized manner. Note the “airwires” that connect the pins of the various components. Airwires indicate the net connections created in the schematic editor. Later, the components will be organized in a bid to minimize the overall length of the airwires.

It is important to always have both the board file and its corresponding schematic open at the same time. This allows annotation to occur wherein changes to the schematic will automatically update the PCB (and vice-versa). Failing to have both files open can lead to an inconsistency between them: a difficult situation to rectify.

3.1 Placing Footprints

A footprint is a collection of pads that represent how a component mounts to a PCB.

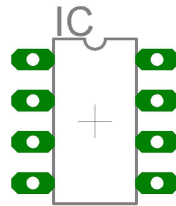


Figure 3.1: Example footprint for an 8-pin DIP.

When organizing footprints use the **MOVE** and **ROTATE** commands. Connection nets are shown by “airwires” which are yellow lines that indicate a connection between component’s pins. Try to minimize the length of these lines by placing like nodes in a common region. After moving footprints, use the RatsNest command to redraw the airwires to make sure the footprint placement is effective in minimizing the length travelled. Figure 3.2 is a suggested parts layout.

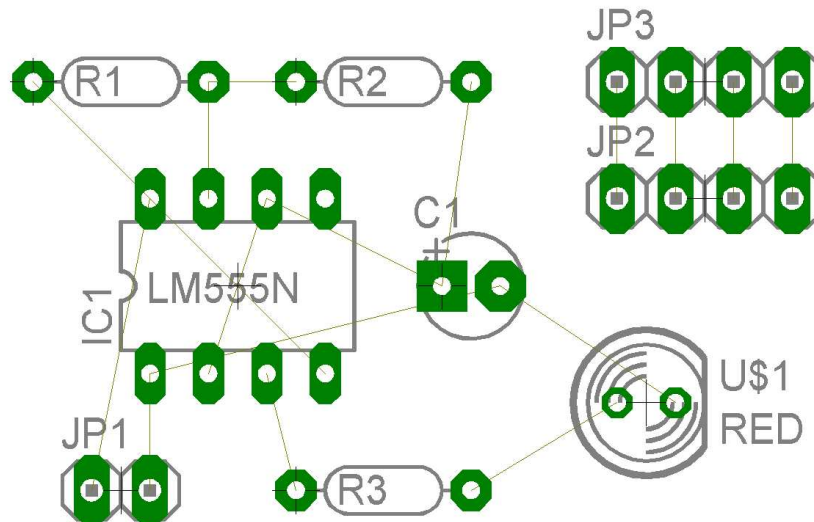


Figure 3.2: Example arrangement of footprints.

3.1.1 Part Placement Guidelines

For your design, remember the following tips and guidelines regarding the placement of footprints.

- Try to place high pin count components in the centre of the board. This provides more surface area to route nets.
- The light version of Eagle limits the physical board dimensions to 3 by 4 inches.
- It is important to include **mounting holes** as the board will be mounted in a mechanical structure at some point.
- Use a coarse grid setting (say 100 mil or even 500 mil) for positioning of parts. This makes it easier to line-up components. The grid can be reduced as needed for fine-tuning.
- Place connectors to off-board equipment at the edge of the PCB.
- Traces should be at least 20 mil from the edge of the PCB.

3.2 Routing the Board

Routing a board involves drawing the locations of the resulting copper traces between components to complete the net connections made in the schematic editor. Below are some basic rules to keep in mind when routing a PCB.

3.2.1 Routing Tips

- Try to keep your inside trace angles greater than 90 degrees as sharp angles tend to collect conductive debris and have other high-frequency effects. Two consecutive 45-degree angles redirect the trace 90 degrees. Use “T” trace intersections if you have to. Avoid “Y” trace intersections.
- Utilize the area under components such as ICs or resistors to route.
- Have a set of callipers or a ruler on hand for measuring package dimensions and lead sizes.
- Part specification sheets typically include package information that will aid in determining pad placement.

- All polarized components (including ICs) should be placed with consistent orientation.
- Hole sizes should be at least 10 mil larger than the component lead.
- Pads should be at least 15 mil larger than the hole size.
- Place a PCB revision number and name on the top layer.
- Components should be at least 200 mil from the edge of the PCB.
- Octagonal pads provide more surface area to solder to.

3.2.2 EE 401 Milled PCB Routing Guidelines

In order to help ensure that your board is milled with minimal trouble, please adhere to the following guidelines. Most of these guidelines are present due to the nature of the PCB milling process used in the lab.

- Design only single-sided boards: only the bottom layer will have copper traces. Double-sided boards may be permitted for unique situations, but only at the approval of the course technician.
- Milled board outlines can only be rectangular.
- Milled boards do not allow traces to be placed between pads spaced at 100 mil (this includes most DIP (DIL) packages).
- Milled boards are single sided, which means only the bottom layer can contain tracks.
- With DB9 connectors and other parts where the pads are not on a simple grid pattern, to connect to the pads use the **ROUTE** button to first connect from the off-grid pad to the next point on your board.
- Minimum trace width is 40 mil.
- Minimum clearance for any disconnected traces or pads is 13 mil. This clearance, however, can lead to difficult soldering tasks. The clearances used later in this document will lead to a PCB that will be simpler to solder.
- Minimum pad size for jumpers, vias, and components is 80 mil.

- Only three drill sizes are supported: 32 mil, 40 mil, and 125 mil. Depending on the technician's workload, however, these boards are often drilled with only one of the smaller bits, meaning the larger holes will need to be "opened up" using a drill. The following table is a quick guide to helping you select drill sizes:

Drill Size	Use
32 mil	Most components.
40 mil	Rectifier diodes, power transistors, and pin headers.
125 mil	Mounting holes.

- You can run up to two traces easily out the end of a 0.3 inch wide DIP package.
- Place the dimension outline 50 mil outside your desired board size. The default PCB panel size is 4x6 inches. If no outline is provided, you are likely to get a rough 4x6 panel with two jig holes in your circuit layout.

3.2.3 Auto Routing

When routing a PCB it is common to combine auto routing and manual routing. The auto router can route the majority of airwires/nets quickly, and then special or difficult nets can be manually routed. Auto routing is a method where the computer routes connections based on a set of rules set by the DRC and the Auto router setup. The auto router will not always finish the board the way you want, but it can save time. Provided setup is correct, an auto router will route the board nets according to the design rules and schematic connections. The following procedures will setup the auto router and then route the connections.

Before auto routing, complete the procedure below to configure the DRC to conform to the milled board specifications.

1. Select **Tools > DRC** from the menu bar within the layout editor. A window will pop up that contains several configuration tabs.
2. Set Clearances. Click on the **Clearance** tab, and set the **wire to wire** clearance to 40mil and the **wire to pad** clearance to 30 mil. For information purposes there is a picture shown to the left of each tab window to depict what the setting configures and at the bottom of the window there is a description of what is being configured. See Figure 3.3.

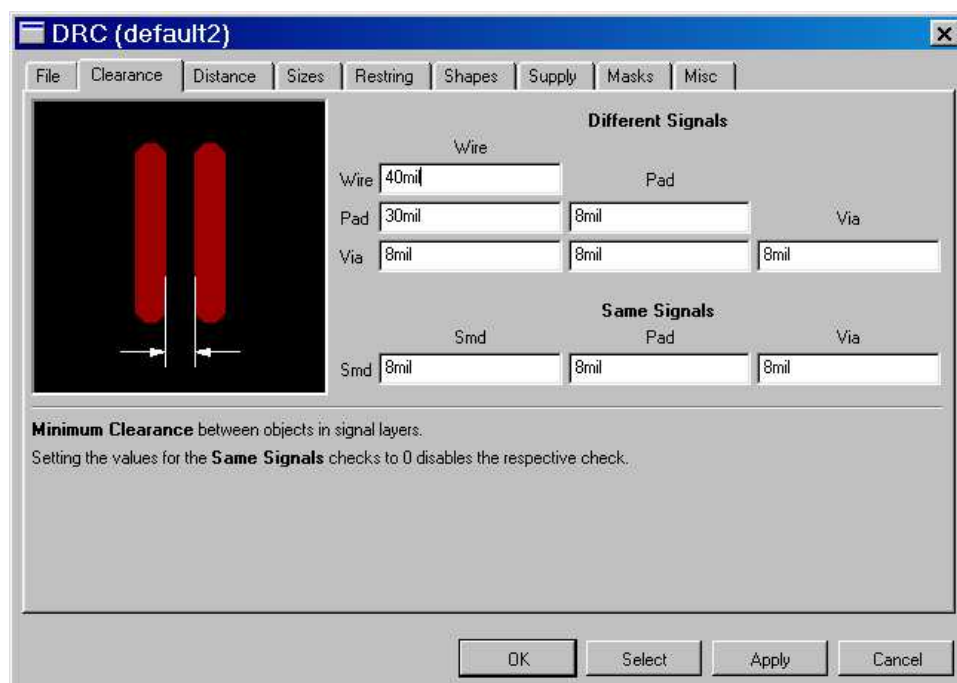


Figure 3.3: DRC configuration: the clearance tab.

3. Set minimum track width. Click on the **Sizes** tab, and set the **Minimum Width** to 40 mil and the **Minimum Drill** to **32 mil**.
4. Set the minimum PAD size by clicking on the **Restring** tab, and set all the minimums to 80 mil.
5. Set the shape of the pads by clicking on the **Shapes** tab, and set the Top and Bottom shape to **octagon**.
6. To save the DRC settings made, Click on the **File** tab, Click the *Save As* button, and name the file.
7. Click OK. Clicking OK will close the DRC setup and it will also run the DRC.

Now the auto router will follow the DRC configuration when routing the board. To setup and run the auto router:

1. Select **Tools > Auto** from the menu bar within the layout editor. A window will pop up that contains several configuration tabs.
2. Under the **General** tab, set **Top** to **N/A** and set **Bottom** to *****. This will make a single-sided PCB. Also set the grid size to 50 mil.
3. Click OK to start the auto router.

The auto router may not completely route all connections. In such a case, the DRC configuration and auto router setup can be manipulated to completely route the design. If manipulating settings becomes tedious, then manual routing can be attempted as discussed below.

3.2.4 Ripping Up Tracks (Delete Tracks)

It may be necessary to delete a single track, multiple tracks, or maybe even all tracks. The RIPUP command is useful when shifting components to optimize boards. The **RIPUP** command can be accessed by selecting **Edit > Ripup** from the menu bar.

1. To rip up an individual track:
Use the **RIPUP** command and *LEFT* click on the individual track to delete.

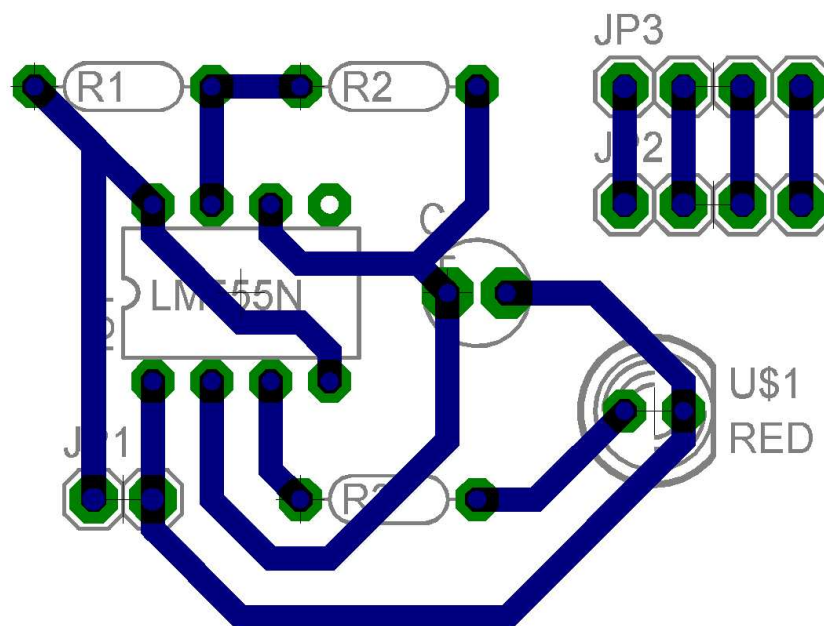


Figure 3.4: Example of a completely routed board.

2. To rip up a signal:
Use the **RIPUP** command and type the signal name in the command box located above the main window. For Example GND.
3. To rip up everything but a few signals:
Use the **RIPUP** command and type the signal name in the command box preceded by a "!". For example: ! **GND VCC**.
4. To rip up a group of tracks:
Use the **GROUP** command and place a box around the tracks to be ripped up.
Use the **RIPUP** command and *RIGHT* click on one of the selected tracks. All tracks within the group will be deleted.
5. To rip up all tracks:
Use the **RIPUP** command and then click the **Traffic Light** on the top toolbar. Click yes to the confirmation.

If you want, RIPUP the entire design and Auto route it again.

3.2.5 Manual Routing

Manual routing involves routing nets by manually placing the copper tracks similar to drawing lines. This involves physically drawing the track from pin to pin and can be time consuming but necessary. The auto router is not a perfect tool and it sometimes can make a board more complicated than is necessary. The auto router also can get “stuck” where it may not be able to route a board. Therefore, it will be necessary to do some, if not all, of the routing manually. Lets try some manual routing by completing the 4-pin headers again, but this time manually. Route the 4-Pin headers similarly to the example shown in Figure 3.5.

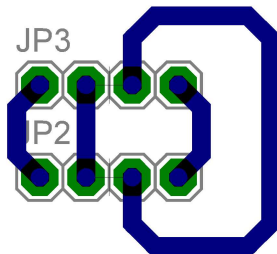


Figure 3.5: Manually routed 4-pin headers.

Start by ensuring the two 4-Pin headers only have airwires, by ripping up connections, if necessary.

To manually route:

1. Select **Edit > Route** from the main menu bar.
2. A toolbar will appear below the menubar that allows the track options to be set. Select a track width of 40mil (**Width: 0.04**) using this toolbar.
3. Click on the first pad where you want to start the track and draw the track while using the airwire to guide you to the other pads until the connection is complete. **Remember** when drawing lines, a right click will switch modes between orthogonal and diagonal. A left click will create a bend in the line.
4. After completely routing an airwire, the ROUTE command will automatically break the line indicating a successful connection. The ROUTE command stays active and is ready to start another airwire.

As with automatic routing, the design must be checked using the DRC to ensure that no errors will occur in manufacturing the board, as outlined in the next section.

3.3 DRC - Design Rule Check

The PCB layout editor adds an additional check: the design rule check. It provides a method for ensuring that the board is designed to the specifications of a manufacturer. In the case of EE401, the routing guidelines previously outlined are the requirements to manufacture a board using the milling method. The DRC must be setup correctly, and at the completion of a design the it must return no warnings or errors. To configure the DRC consult the section on Auto Routing.

As a simple additional check, turn off all layers but “UNROUTED”. Refresh the screen and make certain there are no unrouted traces.

Chapter 4

Building a Custom Library

If the component to be added is a specialized component and an Internet search for the library file has not been successful, a custom library file can be constructed (or an existing part library can be modified). Instructions below will guide you through creating a component, footprint, and schematic symbol within a custom library. The component that is created as an example is a 2 PIN terminal block.

1. *Open Eagle and create a new library file.* Select **File > New > Library** from the menu bar. Save the library as “mylib.lbr” in your directory.
2. *Add a description of your library.* Select **Library > Description** from the menu bar. Type in text that describes the library. For example, “This is my custom library”.
3. *Create a new device.* Select **Library > Device** from the menu bar. In the edit box labelled *new* type **2PINTerm**. Click yes to the confirmation.
4. *Create a new package.* Select **Library > Package** from the menu bar. In the edit box labelled *new* type **2PINTerm**. Click yes to the confirmation.
5. *Create a new symbol.* Select **Library > Symbol** from the menu bar. In the edit box labelled *new* type **2PINTerm**. Click yes to the confirmation.

All of the above three creation steps bring up the same configuration window. The window allows the user to create, but also to view all devices, packages, and symbols within a library.

A library file is composed of several other files. For instance, each device within the library has its own file (*.dev). Similarly each package and symbol have a file within the library (*.pac and *.sym respectively). A device combines a package (footprint) and symbol (schematic) to create a component. Figure 4.1 illustrates the library file hierarchy.

A lot of work can potentially be saved by copying packages or symbols from existing libraries. To do this, open the source library and select the package or symbol. Turn all layers on, select everything using the **GROUP** command, and **CUT** the information. Use **PASTE** to place this information into the destination package or symbol.

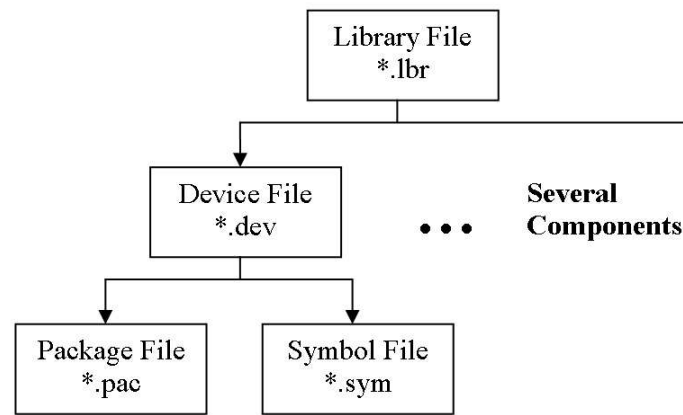


Figure 4.1: Library hierarchy.

6. Draw the schematic symbol as shown in Figure 4.3.
 - (a) *Open the symbol file.* Select **Library** > **Symbol** from the menu bar. Highlight 2PINTerm and click OK.
 - (b) *Create 2 PINS.* Select **Draw** > **Pin** from the menu bar. Notice the tool bar that appears below the main menu bar; the properties of the pin can be set with this tool bar. The important property is the direction of the pin because it is used in the ERC. For the Power Header configure the tool bar as shown in Figure 4.2. Add

two pins, to stop adding pins click the STOP sign on the top tool bar.

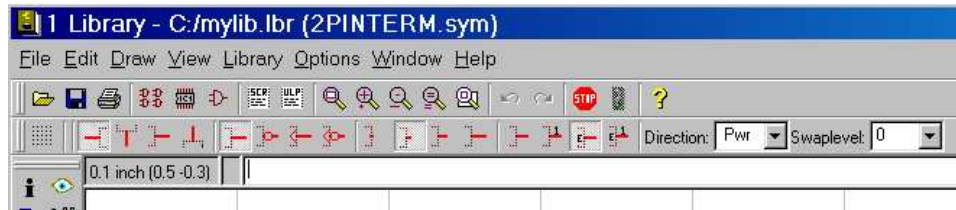


Figure 4.2: Configuration for power pins.

- (c) *Name the PINS.* Select **Edit** > **Name** from the menu bar, and click on the Pin to be named. Name one of the pins POS(+) and the other NEG(-).
 - (d) *Draw the SYMBOL shape in Figure 4.3.* This will require using the wire and circle draw commands, but it will also require a change in the Grid size, Select **View** > **Grid** from the menu bar and enter a size of 0.01 inch.
 - (e) *Add in ">NAME" and ">VALUE"* using text command and place the text strings on the Name and Value layers respectively. These strings will allow dynamic naming of the part when it is used in the schematic editor. One way to change the layer is to: first place the text, Select **Edit** > **Change** > **Layer** from the menu bar, select the appropriate layer, and finally click on the text to be changed.
 - (f) *Add a centre mark.* Select **View** > **Mark** and place the cross hair at the end of one of the pins. This will be the "hot-spot" when the part is inserted into a schematic.
 - (g) *Save.*
7. Draw the PCB package as shown in Figure 4.5
- (a) *Open the package file.* Select **Library** > **Package** from the menu bar. Highlight 2PINTerm and click OK.
 - (b) *Create 2 PADS.* The Drill and Pad sizes are configured based on the components wire leads but must also conform to manufacturing standards. For these two pads, configure as shown in Figure 4.4.

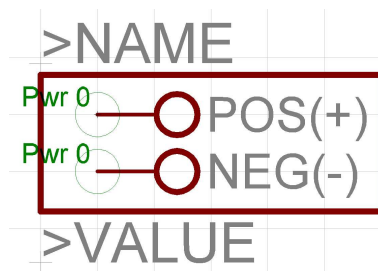


Figure 4.3: Final 2 pin terminal block schematic symbol.

When drawing the package, it is best if you have the actual part in front of you, along with callipers, a micrometer, or a ruler. This will allow you to measure the size of leads (for establishing hole size), lead pitch, and other package sizes.

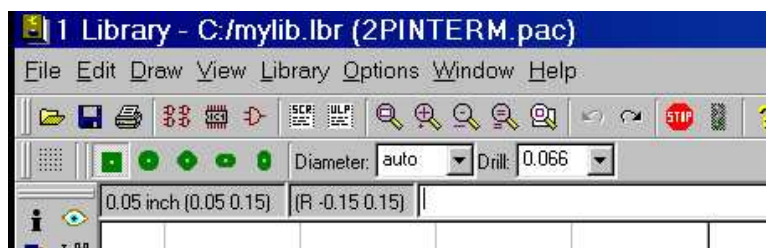


Figure 4.4: Configuration for pads.

- (c) *Name the PADS.* Name one POS and the other NEG.
- (d) *Draw the PACKAGE* or Footprint on the Top Solder Mask Layer (tPlace). When drawing the Solder mask layer, ensure the component's mechanical clearances are met so that the component does not interfere with other components when mounted to the PCB. These measurements, including pad spacing, are specified by the component's data sheet. When creating this package refer to the 100mil grid settings shown in Figure 4.5.
- (e) *Add in ">NAME" and ">VALUE"* using text command and place on the tName and tValue layers respectively. This is done for the same reason as with the device symbol.
- (f) *Save.*

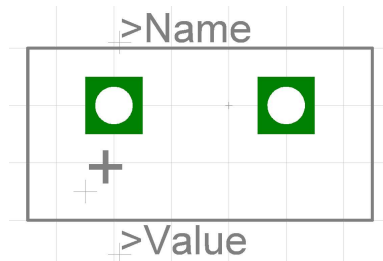


Figure 4.5: Final 2 pin terminal block PCB package.

8. Form the Device. Combine the symbol and package into the device to form the 2 Pin Power Header.
 - (a) *Open the device file.* Select **Library** > **Device** from the menu bar. Highlight 2PINTerm and click OK.
 - (b) *Add the Symbol.* Select **Edit** > **Add** from the menu bar. Highlight 2PINTerm and click OK. Place the symbol at the location of the cross hairs in the left window panel. Click the Stop Sign.
 - (c) *Add the Package.* Select **Edit** > **Package** from the menu bar. Highlight 2PINTerm and click OK. Different packages can be associated with the schematic symbol at this juncture, if required.
 - (d) *Connect PINs to PADs.* Select **Edit** > **Connect** from the menu bar. Match the appropriate pins to pads, click connect, and finally click OK. A green check mark indicates successful connections.
 - (e) *Add a Prefix.* This adds prefix text to the component name when the component is added to schematics and PCBs. For example a resistor prefix is R; therefore, resistors are named R1, R2, etc. Set this prefix to **JP**.
 - (f) *Save.* Save and close the Library. The component is complete.

4.1 Update Schematic and PCB

Replace this power header with the one used in the 555 timer circuit designed previously. Open the schematic file and the PCB layout file. If only one is opened, the changes made in one of the files will not reflect in the other Perform forward annotation: replace the schematic power header, and

then route the design in the PCB Layout editor. If the component was constructed correctly, airwires will indicate the connections to be made in the PCB layout editor.

Chapter 5

Exporting Schematic and PCB Images

This chapter describes how to export the schematic diagram and PCB layout to an encapsulated postscript or bitmap file for use within a document. After completing a design, documentation usually follows. Encapsulated postscript is the recommended export format because of one main advantage: they are vector-based and scalable. When the schematic image is included in a document the resulting visual is less degraded and more crisp. Below, two procedures are outlined: one to export in encapsulated postscript, and another to create bitmaps.

5.1 Exporting to Encapsulated Postscript

Exporting to postscript format is done through the **CAM Processor** within Eagle. The following procedure will guide you through creating what is known as a “CAM job” which will in turn allow you to easily export your schematics and PCB layouts to postscript format.

1. Open Eagle with the file you wish to export to postscript.
2. Select **File > CAM Processor** from the menu bar within the files window.
3. Configure the CAM Processor with the following attributes:
 - Output Device = EPS, which stands for Encapsulated Postscript

- File = filename.eps, this is where you want the output EPS file to be placed.
- Ensure that “pos.Coord” is checked.
- Select the layers that are to be exported to the right of the window.

Figure 5.1 below is the CAM Processor configured for EPS output.

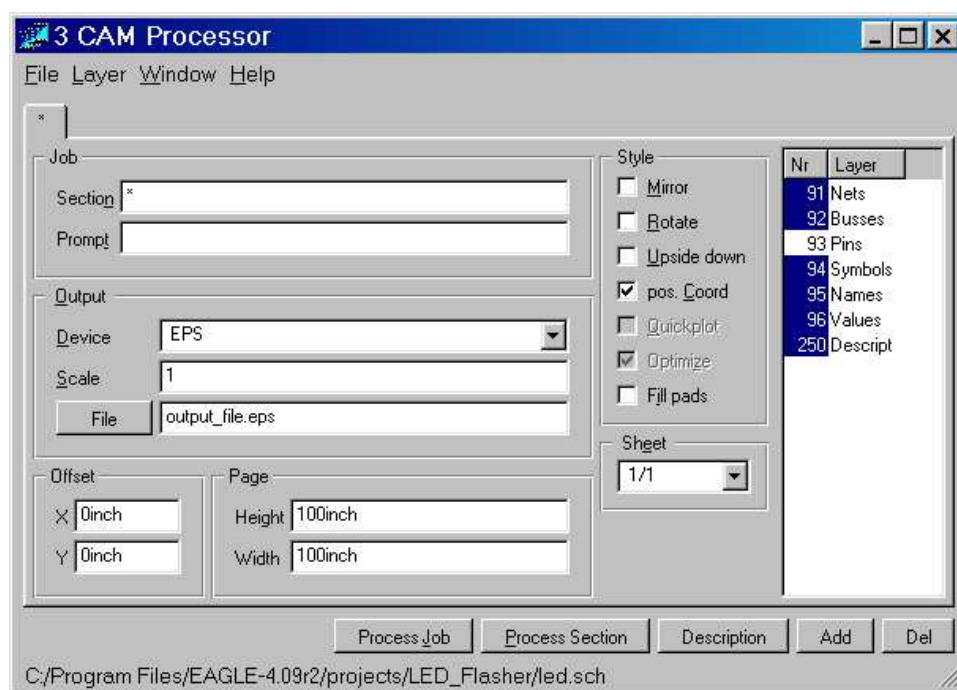


Figure 5.1: The CAM processor configured for encapsulated postscript output.

4. Click on **Process Job**

If you will be exporting to postscript often, the CAM Processor state can be saved. Select **File > Save As** within the CAM Processor window and select an appropriate name.

When including an encapsulated postscript image into a document, many word processors, notably MS Word, will not show the image contents. It is possible to add functionality to Word to overcome this problem. Regardless, the image will be present when printed on a postscript printer. Alternately, “Alladin Ghostscript” and “Alladin Ghostview” are MS windows-based applications that are used to view (E)PS files.

5.2 Exporting to Bitmap

Exporting to Bitmap or Graphic format is done through the Export option within Eagle. This method of exporting is similar to many windows programs and is thus rather straightforward. Below is the procedure.

1. Open Eagle and open the file you wish to export.
2. Select **File > Export** from the menu bar within the files window.
3. Another menu will pop up select **Image**.
4. Choose the appropriate output file and options.
5. Click OK.

Bibliography

- [1] CadSoft Computer Inc. Schematic-layout-autorouter tutorial. Technical report. <http://www.cadsoftusa.com/info.htm>.