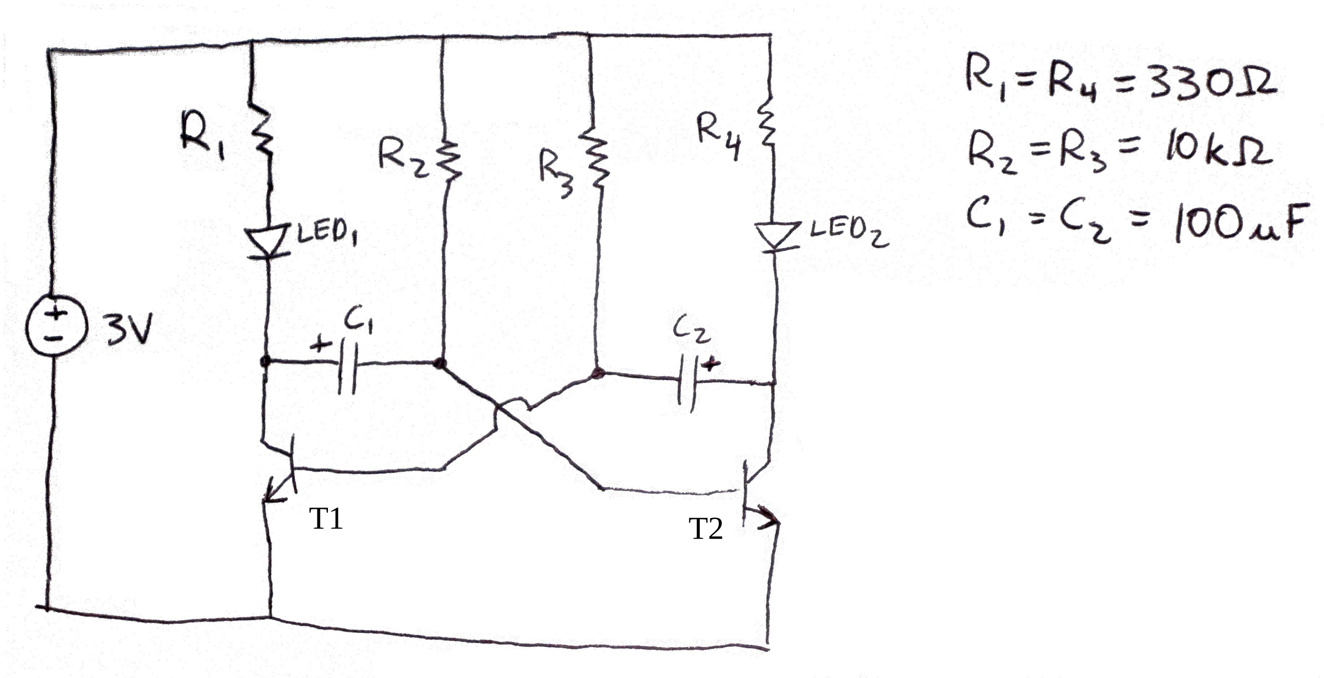
Advance TE Session: Eagle Cad Tutorial

Circuit – LED Blinker



<table>
<tr><th>Part</th><th>Library</th><th>Device</th><th>Package</th><th>Value</th></tr>
<tr><td>R1, R4</td><td rowspan="3">rcl</td><td rowspan="2">R-US_0207/10</td><td rowspan="2">0207/10</td><td>330 Ω</td></tr>
<tr><td>R2, R3</td><td>10 kΩ</td></tr>
<tr><td>C1, C2</td><td>CPOL-USE2-5</td><td>E2-5</td><td>100 uF</td></tr>
<tr><td>LED1, LED2</td><td>led</td><td>LED3MM</td><td>LED3MM</td><td>-</td></tr>
<tr><td>T1, T2</td><td>transistor</td><td>2N3904</td><td>TO92</td><td>-</td></tr>
<tr><td>Battery</td><td>custom</td><td>custom</td><td>custom</td><td>-</td></tr>
</table>

Schematic Capture

All of this section is in the schematic window

1. Open Eagle and create a new schematic
    - Before you save, make sure to create a new board. This will keep the schematic and board in sync.
    - Save your schematic as *training_board.sch*
    - Note: if you get a warning that says "No forward-/backannotation will be performed!", you did something wrong, so start over.
2. Add parts to the schematic
    - The "Add" tool can be found in the toolbar to the left of the screen or in the "Edit" menu.

- For each part in the table above, use the “Add” tool and find the library for the part and then find the correct device. Double-click on the part and add however many parts you need to the schematic. Press escape on your keyboard to return to the “Add” tool once you've added enough parts.
- Note: **don't** click on the “Drop” button in the “Add” tool, or it will get rid of the library you have selected. If you do it by accident, you can add libraries back in the control panel.
- Note: Don't worry about the battery for now. Eagle doesn't have a part for it, but we will make one later.

3. Arrange the parts
   - Arrange the parts like in the schematic above (or however you want them) using the “Move” tool.
   - If you right click while using the move tool, you can rotate parts.
   - You can mirror parts either with the “Move” tool using the toobar at the top of the screen or with the “Mirror” tool.
4. Set the part values
   - Set the values to the parts like in the table above. A “-” means that part doesn't have a value.
   - You can either use the “Value” tool or edit the part properties directly. You can get to the part properties using the “Info” tool or by right clicking on the part and then clicking on “Properties”.
   - Note: You can also rename the parts in a similar way to this.
5. Route wires between the parts
   - Use the “Wire” tool to connect all of the parts like in the schematic above.
   - Add junctions where multiple wires intersect using the “Junction” tool. This isn't strictly necessary, but it makes your schematics more clear.
   - Note: you can change the shape of wire using the toolbar at the top of the screen.
   - Note: if you mess up, you can undo or use the “Delete” tool.
6. Run an error check
   - In the “Tools” menu, click on “Erc”
   - fix any important errors

Board Layout

All of this section is in the layout window

1. Turn on the grid
   - Press the “Grid button on the toolbar at the top of the screen, then select “On” for Display.
2. Change the size of the board
   - There should already be a square board drawn. If there isn't, draw one using the “Wire” tool; make sure to set the layer of the board as “20 Dimension” using the drop-down box at the top of the screen.
   - Resize the board outline so it is 1.5 inches by 2 inches. You can either use the “Move” tool or edit the position values in the properties of the outline.
3. Move all of your components into the board outline
   - Just like in the schematic, use the “Move” tool.
   - Note: you can move multiple parts by selecting a group of parts using the “Group” tool and then with the move tool, right click and then click on “Move: Group”. You can use other tools on groups, too.
4. Arrange parts on the board
   - You can either arrange the parts like they are arranged in the schematic or arrange them however you want.
   - Note: make sure none of the parts overlap. The layout defines how the board will be physically made, so spacing is important.
   - Note: don't forget to leave some space for the battery.
5. Route wires between the parts
   - First, click on the “Ratsnest” button in the toolbar on the left side of the screen. This reroutes all of the airwires so they take the shortest path
   - Use the “Route” tool to connect pins that are linked with airwires. If you don't have enough space to route wires on the top of the board, you can also route them on the bottom. Change the layer of the wire using the drop-down box on the toolbar at the top of the screen from “Top” to “Bottom”.
   - Note: you may want to reorient parts to make it easier to route wires. Just make sure you click “Ratsnest” if you move a part very far, that way you can see the shortest paths between connected pins.
   - Note: again, be careful with spacing. If you route wires across each other in the same layer, they will be physically connected.
   - Note: it is a good design practice to route wires so that they stay on the grid. Stick to 90°

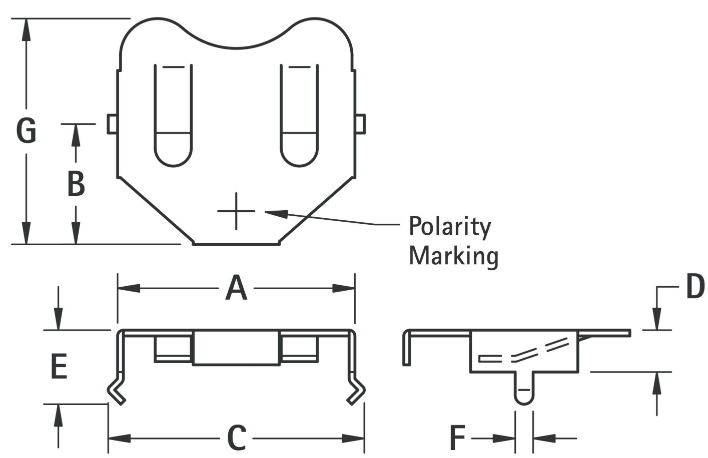
and 45° degree turns; wires at odd angles may not get machined correctly.

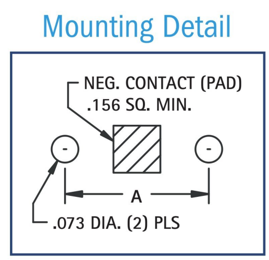
  - Note: if the layout is getting to cluttered and you can't see, you can hide layers to make it easier to see. Use the “Display” tool on the toolbar to the left of the screen to select which layers are visible.

6. Autorouter – not all that great
  - The “Auto” tool will automatically route wires for you, but the results it gives are usually much worse than what you can do by hand.

1. Run a Design Rule Check
  - In the “Tools” menu, click on “Drc”, then click “Check”.
  - Fix any major problems
  - Note: You can change all sorts of variables in the DRC to make sure that you board is correct. The default values will usually work, but if you want to be sure, you should use the specifications given by the board house you are going to have make you board.

Custom Part





There isn't a part in the Eagle libraries that we can use for the battery clip I chose, so we will have to make a custom part. There are three things that define a part: the symbol (what you will see on the schematic), the package (what you will see on the layout), and the device (defines how the symbol and package are connected). The image above is from the datasheet for this part and will help us make the part's package. For our part, A = 0.831”, B = 0.416”, C = 0.890”, D = 0.156”, E = 0.256”, F = 0.062”, and G = 0.782”.

1. From the control panel, create a new library
  - Save it as “keystone.lbr”, since Keystone is the name of the company that makes the part. Save it in the folder with your board and schematic files, not in the default Eagle library

folder. This way, if you ever move your files, you will still have the library.

2. Create a new symbol
    - Click the “Symbol” button on the toolbar at the top of the screen.
    - Name the symbol “BATTERY” and click okay
    - Using the “Wire” tool draw a battery symbol. You don't have to use the same symbol as using in the schematic above; it just needs to represent a battery.
    - Add pins to the symbol using the “Pin” tool. This part has two pads which are connect to the positive terminal of the battery and one which is connected to the negative terminal. Since Eagle can't connect multiple pins to one pad, we have to add three pins: two for positive and one for negative.
    - Name the pins descriptively (ex. POS1, POS2, and NEG1).
3. Create a new package
    - Click the “Package” button on the toolbar at the top of the screen.
    - Name the package 3003, which is the name of the package in the datasheet, and click next.
    - Add two pads for the positive contacts using the “Pad” tool. As in the figure above, make these pads round, have a Drill of 0.073, and be 0.831” apart from each other.
    - Make a negative contact using the “Smd”. As in the figure, the contact should be a square with a edge of 0.156” minimum. The negative contact should be halfway between the positive pads.
    - Draw an outline for the part using the “Wire” tool with it set on the “tPlace” layer. You can make the outline of varying complexity, but it should at least show what area the part covers on the layout. The simplest outline you could do would be a box that surrounds the positive and negative pads and is > 0.782” (G) wide.
    - If you want, you can also mark where the part name and part value will be drawn on the layout using the “Text” tool. If you write “$NAME” or “$VALUE”, Eagle will automatically fill in the name or value at that point on the layout.
    - Name the pads and contact descriptively (ex. POS1, POS2, and NEG1).
    - Note: it is easier to draw these parts if you center the part at (0,0).
4. Create a new device
    - Click the “Device” button on the toolbar at the top of the screen.
    - Name the device 3003 and click next.

- Use the “Add” tool to add a BATTERY symbol to the device.
- Click “New” at the bottom of the screen and add the 3003 package to the device
- Click “Connect” at the bottom of the screen and then connect the pins (on the symbol) to the corresponding pads (on the package).

5. Tell Eagle to use the library you created
   - In the schematic, in the “Library” menu click the “Use” button and then select your library
6. Add the battery to your design
   - Add the battery to the schematic and then connect it in both the schematic and in the layout.

Finish the Layout

1. Draw ground pours
   - In the schematic, rename the wire (referred to as a net in this context) connected to the negative terminal of the battery to “GND”. You can do this either with the “Name” tool or by directly changing the properties of the wire.
   - On the layout, use the “Poly” to draw a boarder around your board on the “Top” layer.
   - Use the “Name tool to connect they boarder to “GND”
   - Use the “Copy” tool to create a second boarder. Edit the properties of the second boarder so it is on the “Bottom” layer.
   - Click on “Ratsnest” and watch the pours fill in.
   - Note: ground pours are a good idea for most boards because they can reduce the amount of noise in your circuit. Since PCBs are made by etching away the layers of metal, not by building them up, some board houses charge by how much copper they have to *remove*, so adding ground pours might make your board a little cheaper.
   - Note: you can also use pours for other things such as heat sinks or if you need an oddly shaped trace.
2. Draw silkscreen
   - Use the “Text” tool or the “Wire” tool to draw the silkscreen, and remember to put it on the correct layer. There is no fixed rule on which layers are included on the silkscreen, but there are already layers for part outlines (“tPlace” and “bPlace”), part names (“tNames” and “bNames”), and part values (“tValues” and “bValues”), which can all be included on the silkscreen. I personally like to create another set of layers (called “tSilk” and “bSilk”) that I use for anything I want on the silkscreen that isn't in the other layers. You can add new layers in the “Display” tool on the toolbar to the left of the screen.

- Note: you usually do the silkscreen last, after you know where all of the parts will be.
- Note: make sure the silkscreen doesn't cover any of the pads, vias, or holes; otherwise, that part of the silkscreen will be removed.
- Note: Common things to use silkscreen for are component outlines and part numbers (these should already be on your layout) as well as board revision labels and signatures.
- Note: it normally isn't necessary to include the part values on the silkscreen. Some people don't put part numbers on the silkscreen, either.

3. Final check
    - Run an error check on the schematic and a DRC on the layout.
    - Look for anything that may be out of place or that you forgot.

Ordering (for reference)

You won't be ordering the board as a part of this tutorial, but this will tell you how to do it.

1. Parts selection and ordering
    - If possible, it is better if you order your parts before you make you PCB, so you test the circuit first.
    - There are many different suppliers of electronics components such as Digikey and Mouser. Just shop around for the best prices.
2. Find a board house
    - It is best if you pick your board house before you start the layout so you can know what specifications you board must meet.
    - There are too many board houses to list, so just look around. Just be careful, because many board house have minimum quantities for orders and large shipping costs that can make them look cheaper than they actually are.
    - One of the best deals for a student is the $33 each deal from Advanced PCB. You can get two layer boards up to 60 square inches that for $33 each plus $15 shipping. If you are a student, there is no minimum quantity.
3. CAM processor
    - You use the CAM processor to generate files to get your board made. You can add layers to each CAM job and it will output files in a specific format. You will run a job for each of the copper layers, for the silkscreen, and for the drills. Each board house wants different files, so just refer to their website to find what they want.