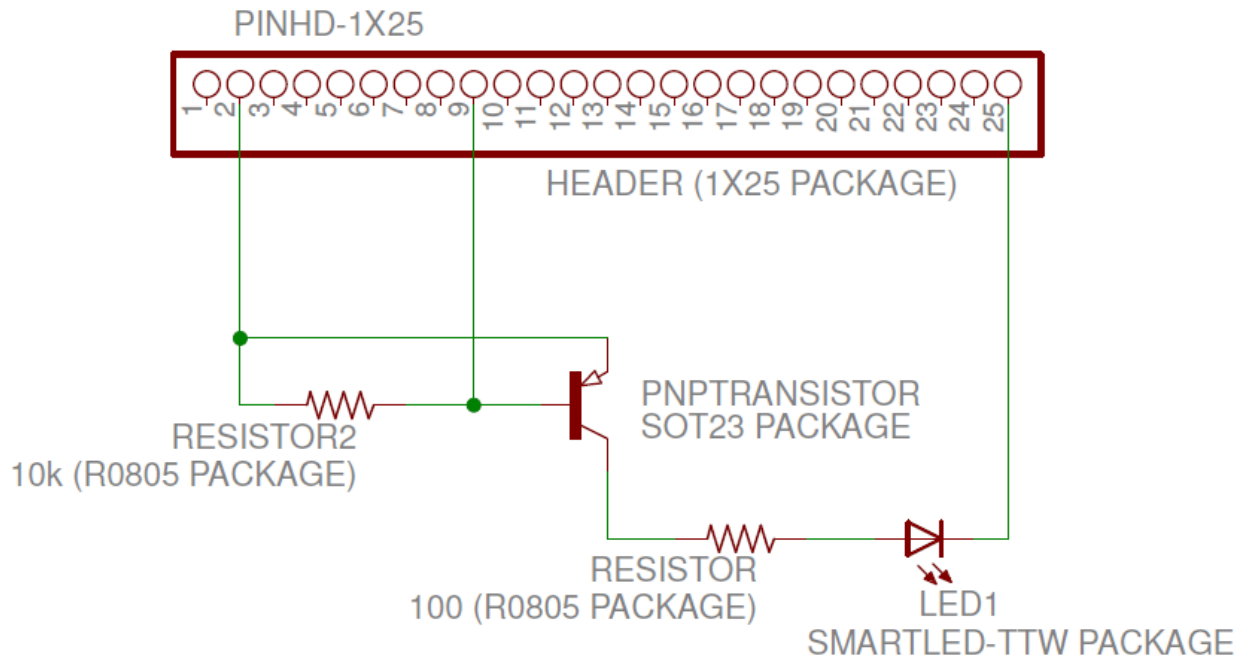


## Getting to know Eagle

1. Download (<http://www.cadsoft.de/freeware.htm>) and install Eagle.
2. We are going to design in Eagle a board to test the kicker board. Here is the circuit that we will work with:



The rectangle with 25 pins at the top is a set of header pins similar to the header pins on each board that plugs into the motherboard on the robot. Pin 2 is a 5V supply pin, pin 9 is called the DONE pin, and pin 25 is the ground (0V) pin. This circuit is supposed to turn on the LED when the kicker board finishes charging the capacitors. Before the capacitors are done charging, pin 9 is at 5V. After the capacitors are done charging, pin 9 goes down to 0V, which turns on the PNP transistor. The PNP transistor acts like a switch. When the left terminal is at 5V, the switch is off. When the left terminal is at 0V, the switch is on, which allows current to flow in the direction of the arrow, from the top terminal to the bottom terminal. In our case, current flows from pin 2, through the PNP transistor, through the LED, to pin 25. This turns on the LED when the capacitors are done charging.

In this project, we will create this circuit in Eagle and design a printed circuit board (just like the boards that go into the robots right now).

### Circuit Schematic

- a. Open up Eagle and select File -> Open -> New Schematic
- b. Add all of the parts in this circuit to the schematic using the **Add** tool (looks like a “D”) on the left toolbar. Each part in the diagram above has a part name and a package name. In the

**Add** tool you should search for the part name and then select the part that has the same package.

- a. After you have found the part, you may double click on it and it will show up in the schematic. Click to place the part where you want it. You can rotate the part before placing it down by right clicking.
- b. At any time, you may press Escape to exit the tool.
- c. To move any item after it has been placed, use the **Move** tool. To rotate any item, first select it using the **Move** tool and right click to rotate it 90 degrees.
  - a. Each item may be selected using any tool by clicking on the item's associated crosshair. Notice that each item has a crosshair. Don't just click on the item itself, because while most times the crosshair for the item is in the center of the item, many times it is not.
  - b. If you click too close to more than one crosshair, Eagle does not know which item you selected. In this case, Eagle will highlight one of the items it thinks you selected. If that is the item you want, left click again. If not, right click and Eagle will highlight another item it thinks you selected.
- d. You can find and edit the name/value of any item by using the **Info** tool (looks like an "i") and clicking on the item.
  - a. After clicking on any item, in the **Properties** window, you can select the **Smashed** option to separate the item and its name/value as separate items. Then, you may move the name or value of the item independently of the item itself (note that the name and value will have individual crosshairs).
- e. You may duplicate an item by using the **Copy** tool and clicking on the item you want to duplicate. Note that you should use the **Info** tool to change the name/value of the duplicated item to something unique.
- f. Add wires as shown in the diagram using the **Wire** (looks like a diagonal line) tool. At any time, you may press Escape to exit the tool and stop drawing more wires.
  - a. The wires should connect appropriately. To check that two wires are connected, make sure they have the same names using the **Info** tool.
  - b. You can force two intersecting wires to be connected by using the **Junction** tool (looks like a green dot). In the diagram, I have placed two green dots at two different wire intersections to force a connection.

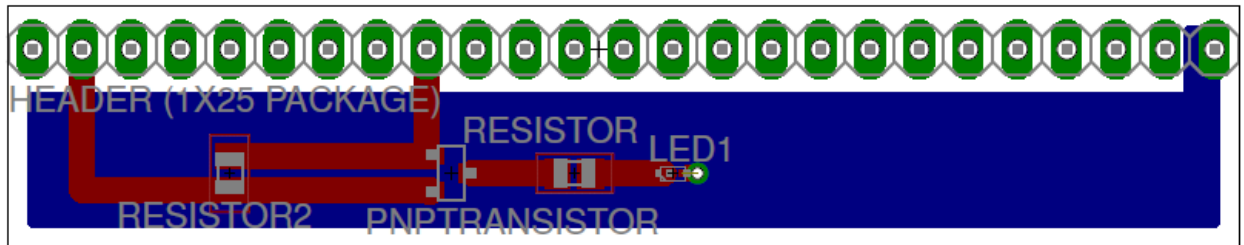
## Board Layout

After creating the circuit schematic, click on the **Board** button at the top to switch to board view. (Create a new board if Eagle asks you to.) You will see all the parts that you placed in your circuit in the board view also, except:

- 1) They are not connected
- 2) They are arranged in a mess

We want to lay out a printed circuit board that neatly arranges the parts and connects them together using *traces* (printed wires). Our board will be .5in tall by 2.5in long and have two

layers. The top layer will consist of the traces that connect our parts, and the bottom layer will be entirely a ground layer. Below is a diagram of the completed board.



- a. Use the **Move** tool to move the sides of the white box so that the box has dimensions .5in by 2.5in. This white box is the extent of our board. You may not place anything outside this box – it will not be considered when the design is fabricated!
- b. Arrange the parts inside the white box as shown in the diagram. Remember that you can rotate any part by using the **Move** tool and right clicking.
  - a. There will be a bunch of yellow lines between the parts. These are where connections will be made, as indicated by your circuit schematic. Rotate and place each part so that none of the yellow lines cross (that way, your connections will not cross).
- c. Now we will add the red traces. These traces are all on the top layer.
  - a. Click on the **Route** tool (looks like a red trace). Notice that on the top bar the leftmost drop-down menu currently has “1 Top” selected. So we are working with the top layer.
  - b. On that bar, there is another dropdown menu called **Width**. That’s how wide we want our traces to be. In this case, we want all our traces to be width 0.05.
  - c. Connect all of the parts according to the yellow lines by clicking on one end of the yellow line and then clicking on the other end. You can change how the traces bend by clicking the appropriate button on the top bar.
  - d. For now, do not connect the LED to the ground pin. We will use a different type of connection for that one.
- d. Now we will add in the second layer (ground layer). Big ground layers are helpful for at least two reasons:
  - 1) Things are more stable when there is a lot of ground
  - 2) It makes laying out our board easier, because if we have anything that we need to connect to ground, instead of routing a long trace from there to a ground pin, we can just punch a hole through the board and access the ground layer, as we will do later.

We will create the ground layer using the **Polygon** tool.

- a. With the **Polygon** tool selected, on the top bar change the “1 Top” layer to the “16 Bottom” layer.
- b. Trace out the boundaries of the ground layer. Notice that the boundaries include the rightmost header pin. In the robot, this pin (actually, the rightmost three pins) connects to ground.

- c. Click on the **Ratsnest** tool (looks like a green X). The polygon will be filled in blue.
  - d. Next, we need to define that the polygon we created is actually ground. Back in the circuit schematic, use the **Info** tool to find the name of any wire that connects to the ground pin. We must change the name of the polygon to that name. (In general, everything that is connected together will have the same name).
  - e. In the board view, click the **Name** tool (has the writing “R2 10k” on it). Then click on the polygon and change the name. When you click on any item with the **Name** tool, Eagle will highlight all items that are connected to it. Click on the **Ratsnest** tool again to update your change.
- e. Now we will connect the LED to ground by punching a hole between the top layer and bottom layer. In the actual created board, these holes will be lined with metal so anything touching the hole on the top layer will be connected with anything touching the hole on the bottom layer. These holes are called *vias*.
- a. Click on the **Via** tool, and select a round via on the top bar (middle option). Then click to place the via connected to one pad of the LED.
  - b. We need to define the name of the via so that Eagle can know to connect the via to ground. As you did in the previous step, use the **Name** tool to change the name of the via to the correct name. Click the **Ratsnest** tool to update your change.
- f. At this point, there should be no more yellow lines. Everything should be arranged and connected correctly. A couple notes:
- a. All of the words describing the names/values of the parts in board view will be printed out on the actual fabricated board. Thus, you want to be careful what and where these words are. Generally, you want to include abbreviated names of the parts (like “R2” for resistor 2) and values of the parts (like 100 or 10uF) and place these labels on the board where they won’t get covered up by parts you solder on. We call these labels *silkscreen labels*. (Notably, for descriptive reasons, the silkscreen labels in my board layout example are not good style because they are too long and there are no part value labels.) To change the labels of an item in board view, use the **Info** tool and click on the item to bring up the **Properties** window. You can also check the **Smashed** box in the **Properties** window to separate the labels from the item so you may move or delete them independently. To delete a label, use the **Delete** tool (looks like a black X) and click on the label. Notice that if you change a label, the corresponding label in the circuit schematic also changes.
  - b. You cannot delete any items except labels using the **Delete** tool in board view. If you do so, an error message will show saying “Cannot backannotate this operation!” You can only delete parts in the circuit schematic. As for traces and polygons, you may use the **Ripup** tool (to the right of the **Route** tool) to click on and delete the connection. After you do so, a yellow line showing a needed connection will reappear.
3. After you are finished with your circuit schematic and board layout, save your project and email it to me ([rjin@mit.edu](mailto:rjin@mit.edu)) so I can take a look at it. Thanks and good luck!