

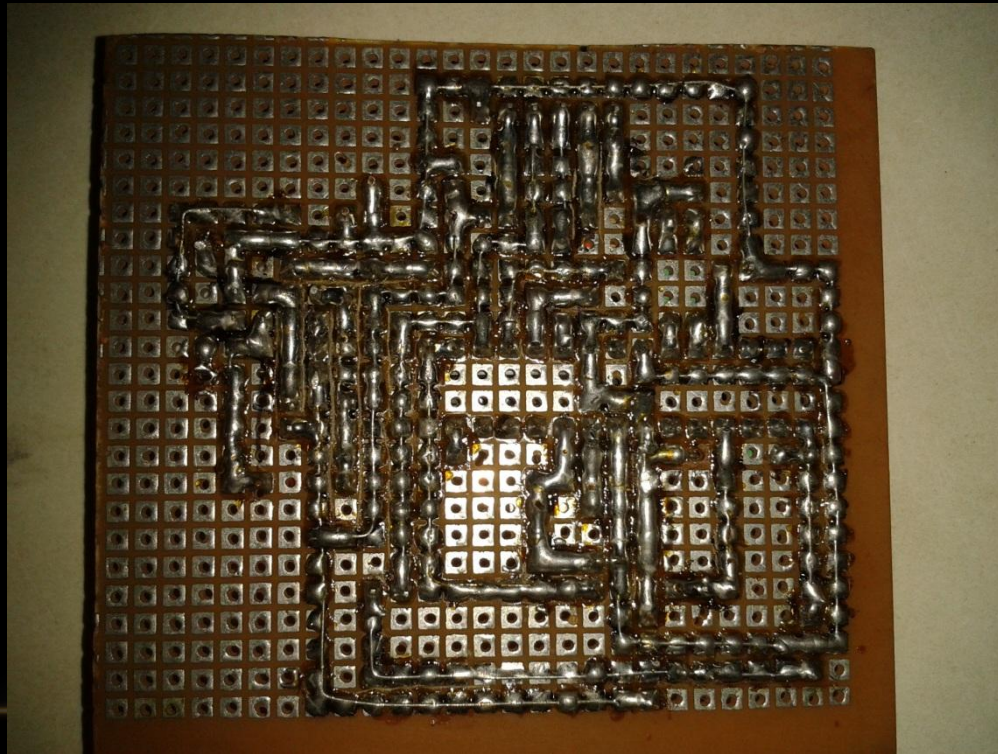
EAGLE BASICS

For The Winter Workshops, 2011

By The Electronics Club, IITB

EAGLE, THE CIRCUIT LAYOUT SOFTWARE

Hand drawn circuits get messy (& frustrating) after a point. This took me 3 days to implement & plan. & I'll need 3 more days if I need to make another one!



EAGLE, THE CIRCUIT LAYOUT SOFTWARE

EAGLE Has 3 Modes,

- The Schematic Editor

This is where you make the Circuit Diagram. There aren't any checks to see if your wiring is correct, so make sure you know how the circuit should look like.

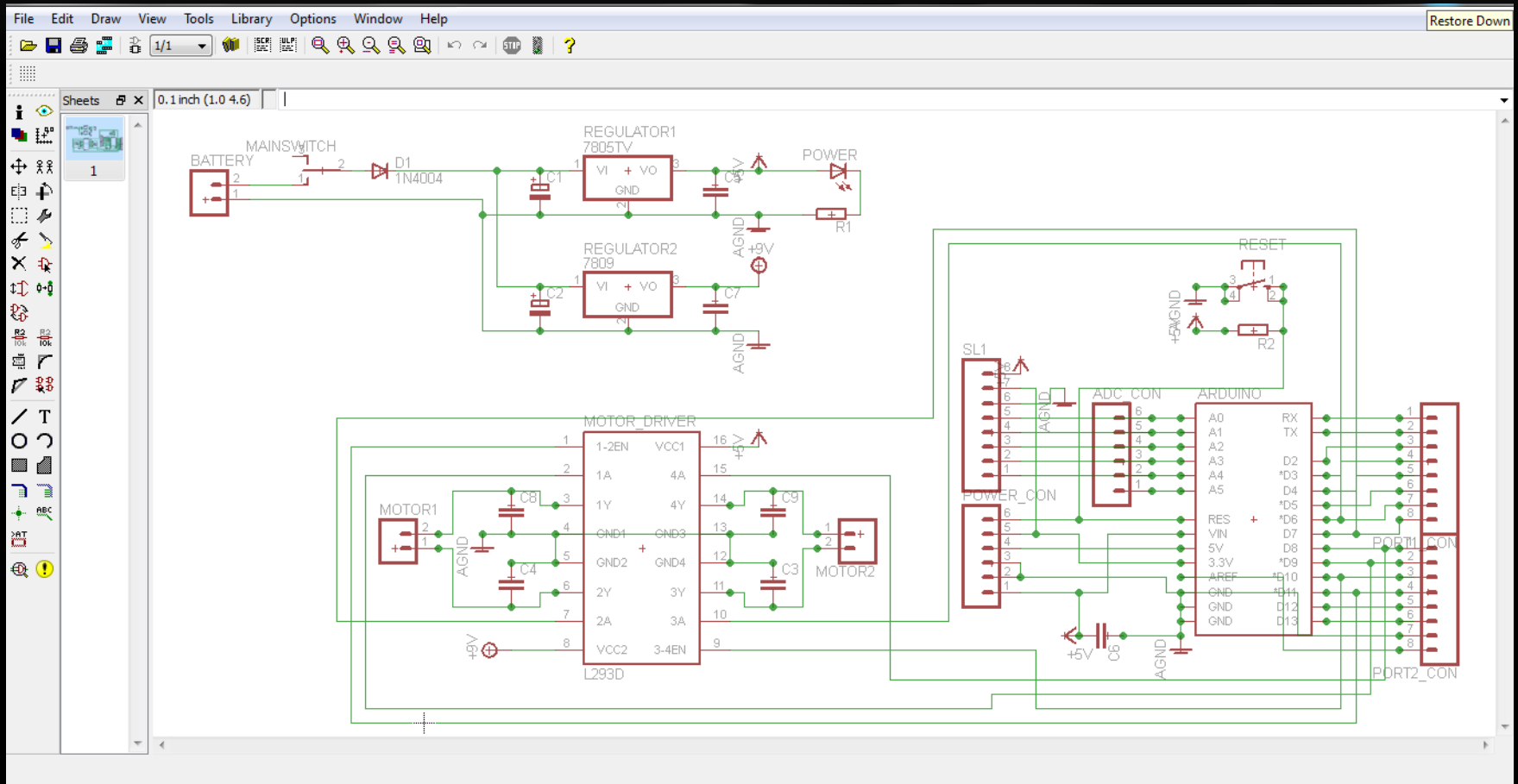
- The Layout Editor

This is where you design the layout of the schematic you made above. ie How your circuit will physically look like on a PCB. NOTE THAT YOU ALWAYS NEED A SCHEMATIC TO MAKE A LAYOUT.

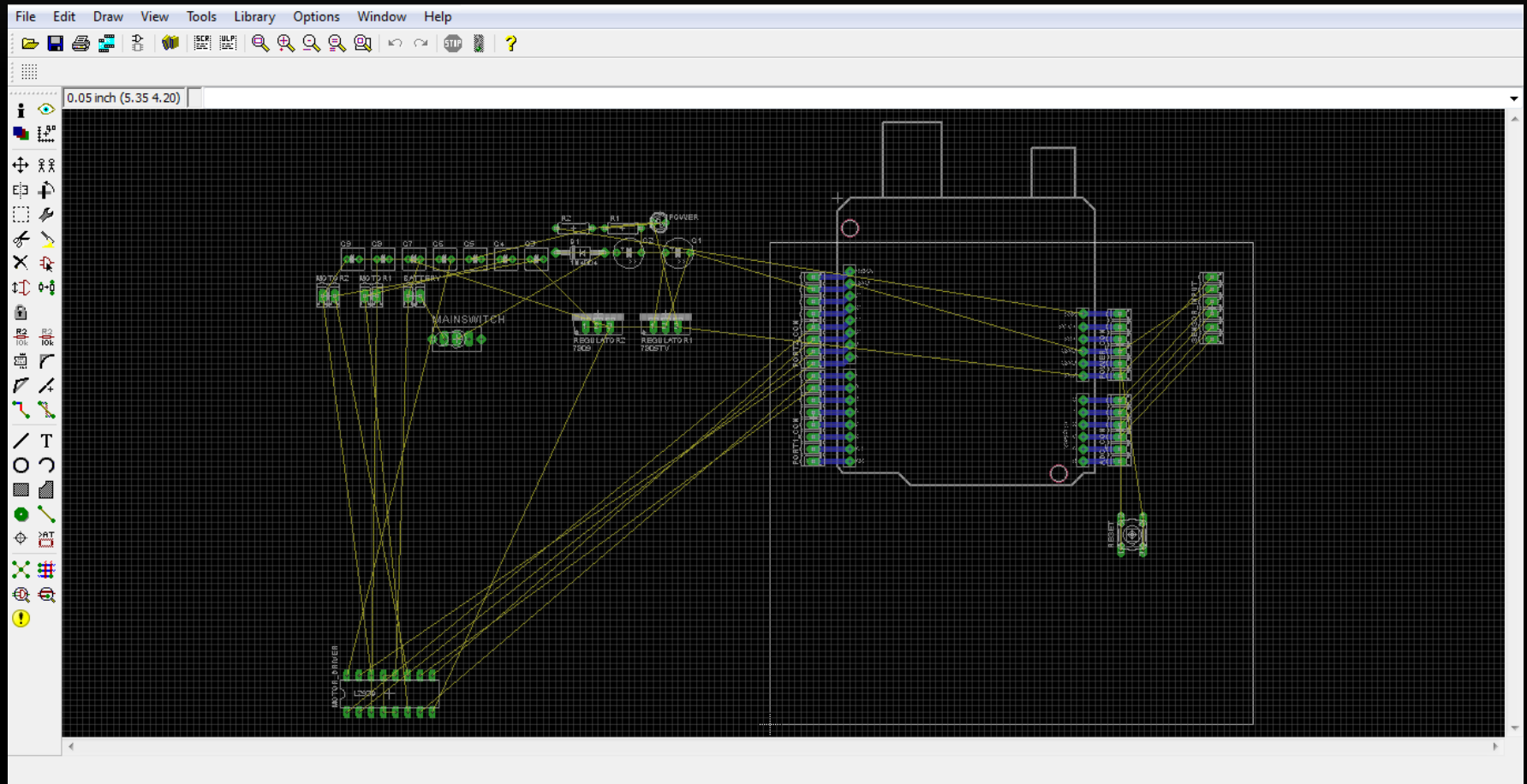
- The Component Editor

Eagle has many libraries of parts which you directly use. But it of course does not have all the parts used in the world. Sometimes you have to make you own part. That is where the Component Editor is used.

THE SCHEMATIC EDITOR



THE LAYOUT EDITOR



THE GENERAL METHODOLOGY

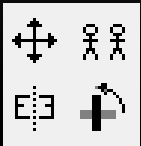
- Know what your circuit should do (very important)! And approximately what parts will be necessary to make it.
- Make a rough circuit diagram starting with the most basic circuit. Add on to it as you proceed (It happens that you realize mid-way that “Oh! A capacitor would be required over here”).
- Collect all the parts required (This is necessary early on because since you have most of the parts at hand, you know what they look like & how much space they will need on the actual PCB/ which part to use from the EAGLE libraries.)
- Complete all the connections in the circuit.
- Design the physical layout which you will actually solder / get printed. Make sure to start this after finalizing the schematic.
- Print/Solder the circuit & test for short circuits & correct it appropriately (I once had a micrometre thin line causing a short circuit which frustrated me for an entire day).

MAKING THE SCHEMATIC

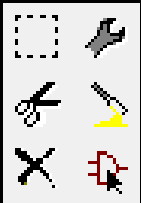
- All parts are present in libraries in the main control panel. Select the appropriate part (look at both the symbol & more importantly the package to be used). Then you can add it to the schematic. All parts are to be added this way.
- It doesn't matter how bad the schematic looks as long as the connections are properly made. But then neat Schematics are easier to debug.
- This is fairly Straight Forward. 1st bring the required parts to the schematic. Connect them. & that's it!
- The worst part of EAGLE is its library composition. Unless you know what exactly the part is called & who manufactured it, it's impossible to find. Make a note of where to find all the normally used parts. You can make your own library for this too.

*DEMO: make a schematic for a 555 timer with $R1=1k$, $R2=2k$, $C=0.1\mu F$

SCHEMATIC TOOLS



Move, Duplicate, Mirror, Rotate. Basic movement tools. Do what they say.



Group: Used to group a bunch of components so that they can be moved together.

Change: Change setting of the Schematic Display (Not usually necessary)

Cut, Paste, Delete: Standard usage

Add: Add parts to the Schematic. Avoid this method. Selecting from the control panel is easier.

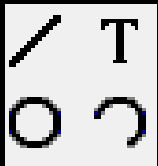


Change Name: Every part has a name which must be unique. If a IC has multiple parts (LM324 has 4 opamps), they are named as IC_NAME+ A/B/C/D

Change Value: Change the value of resistances etc. In case of some parts, you can change the IC number etc.

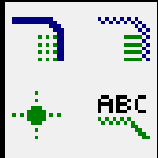
SCHEMATIC TOOLS

The most confusing part of the schematic editor. A WIRE does not connect components Properly, A NET does.



Wire & Wire type of tools.

BUS: just a thick wire



NET: This is what you should use for connections.

Junction: Used to make intersections.

Label: Every connection is also named uniquely. The label shows the name of the connection.

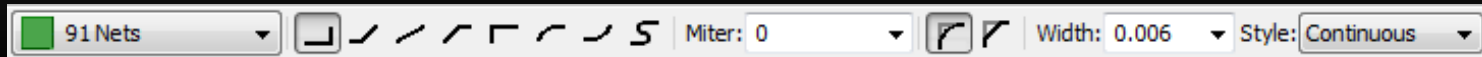


Attribute: Change some properties of the part.



There is also an invoke command which must be used with multi-part ICs. They will set the power supply for the IC etc.

THE WIRING TOOLBAR



The toolbar appears whenever the WIRE/NET is selected.

The 1st dropdown selects which Layer to write to.

Others select the way the wires are turned, their size, Roundness of the curves, etc.

Toolbar common to both Schematic/ Layout editor.



Switch to board when you are done with the Schematic.

LAYOUT EDITOR

All the parts mentioned in the schematic are to be placed in the box of appropriate dimensions.

The objective is to make the least messy board of the smallest size (since printing PCBs costs areawise).

Make sure to not stretch the limit of the PCB printer. They will most probably just spoil the board.

DESIGN Appropriately!

In the Layout editor, The WIRE actually makes the connection.

There are layers of connections, TOP / BOTTOM. Usually we make 1 sided boards where connections are made in the Bottom Layer & Components soldered in the TOP.

LAYOUT EDITOR

The unconnected signals are shown as yellow lines, disappear after all connections are made.

- Most tools are same as the Schematic Editor



RATSNEST: Used to optimize the signals ie the yellow lines look less of a mess once the components are placed in position. Make it easier to make connections.



VIA: Creates Holes on the PCB with Solder around them.



Polygon Tools: Create a filled area of Connections. Can be used as a Heat Sink.

To create a Polygon, 1st create the boundary & then rename it to the name of the Signal it wants to represent. The apply Ratsnest to actually show the polygon.

The AUTO connect tool makes automatic connections once all the components are placed inside the board based on some rules which you can specify. But this process is not UNDOable.

MAKING YOUR OWN COMPONENTS

Create a new library.



Device: Put the Part & Symbols together as a Device.

Package: Design the Part. The Pads are where the Connections are made in the Layout.

Symbol: Design the Part Symbol. The Pins are where connections are made in the Schematic.

The Device is then put together in the device menu where each pin is matched with the pad.

DEMO: Make a resistor.

NOTES

- Most Designs are made on the bottom layer. But printing is done the same way as you give the image. So remember to give a mirror-image of the layout for printing.

HAPPY DESIGNING!!!

