

# Electronic and electromechanical prototyping

## *Introduction - ECAD*

Corso LM 'Materiali Intelligenti e Biomimetici' – Prof. A. Ahluwalia

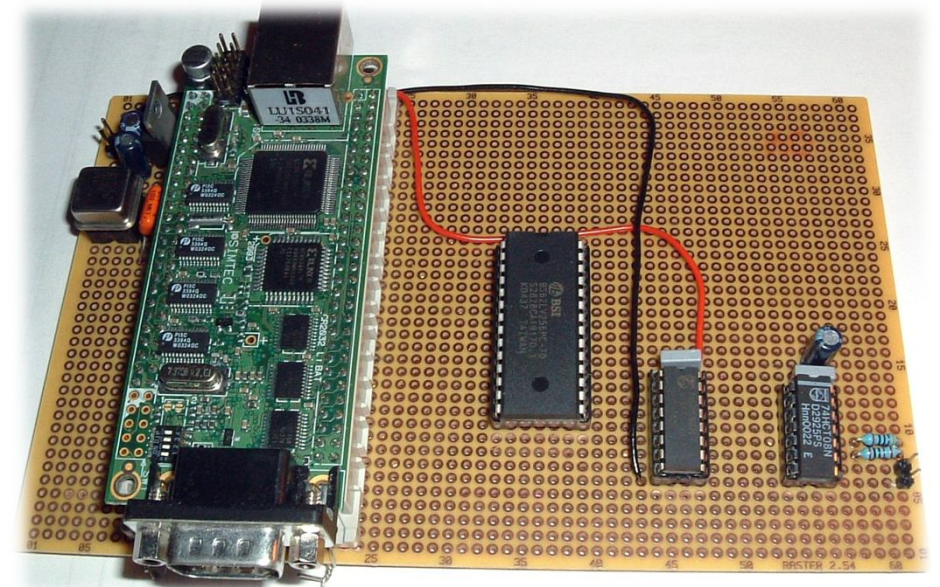
4/05/2017

*ludovica.cacopardo@ing.unipi.it*

# After Breadboards: Matrix Boards

We use breadboards for quick construction, **Matrix Boards** for laying out a project so it can be copied to make a Printed Circuit Board.

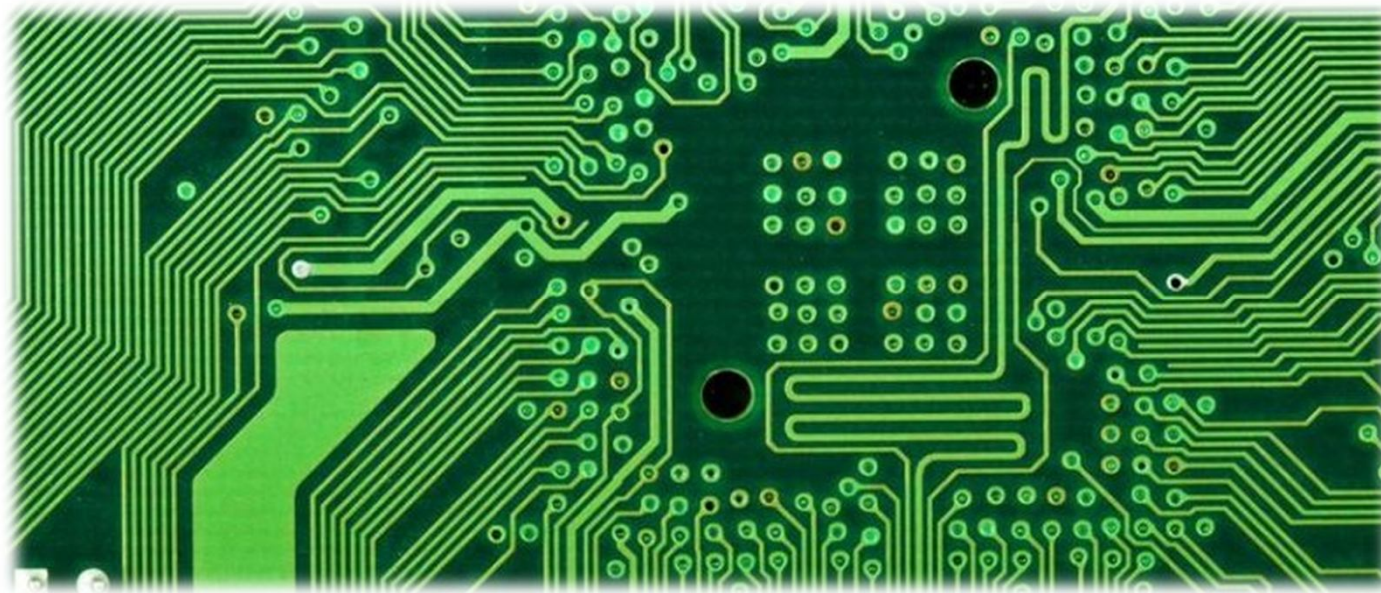
This is a prototyping board, with *copper pads in a matrix layout*. You solder the components in place, and then simply cut pieces of wire, and solder them to make the circuit



# Printed Circuit Board

The PCB is the physical board that holds and connects all of the electronic components.

The circuits are formed by a **thin layer of conducting material deposited**, or "printed," **on the surface of an insulating board known as the substrate**. Individual electronic components are placed on the surface of the substrate and soldered to the interconnecting circuits.

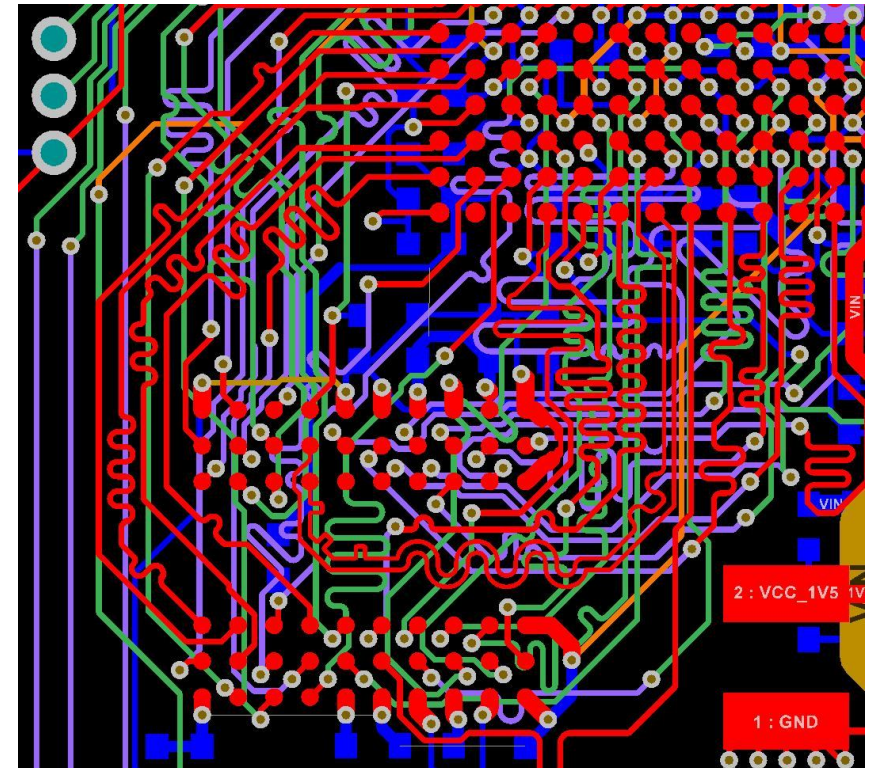


# ECAD

**Electronic computer-aided design (ECAD)** or Electronic design automation (EDA) is a category of **software tools for designing electronic systems** such as integrated circuits and printed circuit boards. The tools work together in a design flow that chip designers use to design and analyze entire semiconductor chips.

*Before EDA*, integrated circuits were designed by hand and manually laid out.

By the mid-1970s, developers started to automate the design along with the drafting. The first **placement and routing** tools were developed.

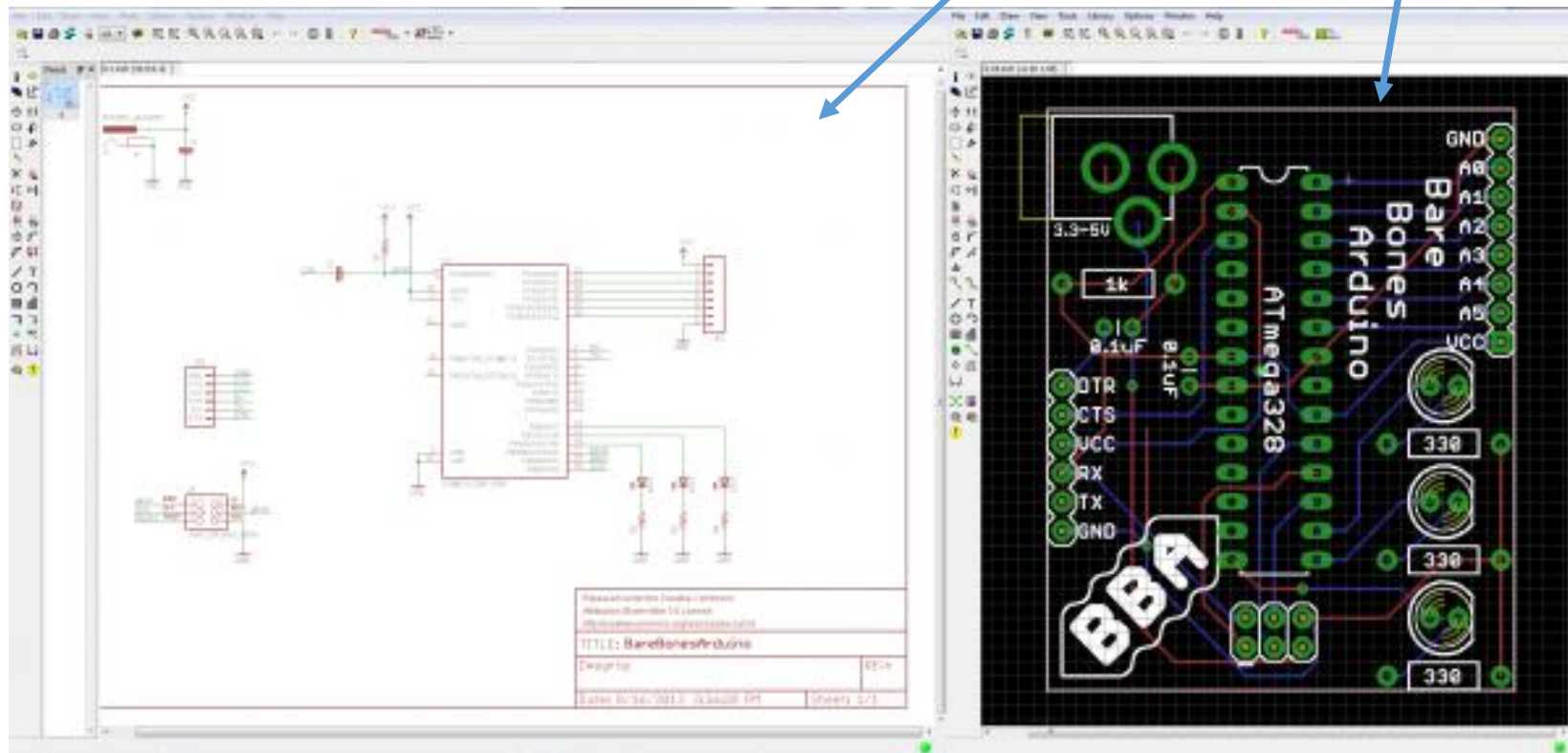




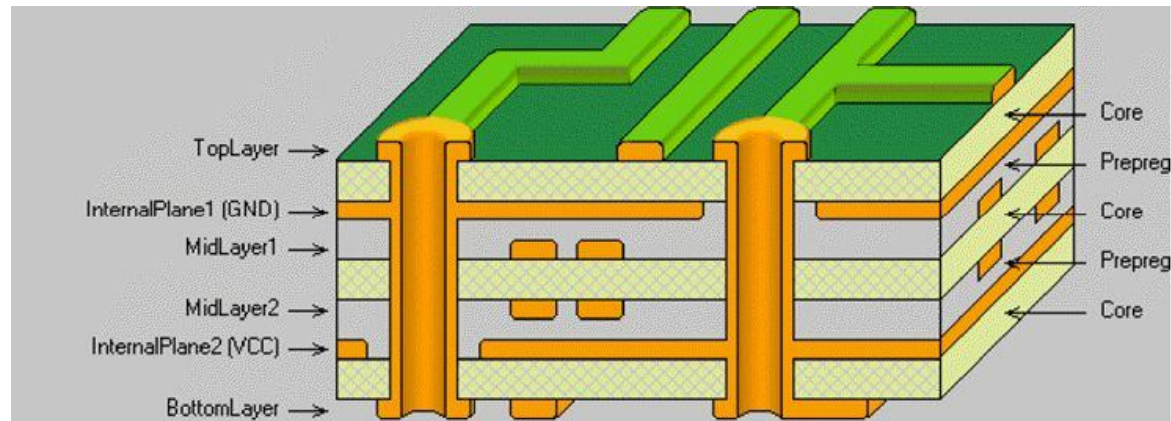
# Printed Circuit Board Design

## PCB ECAD Software (Ex. Eagle):

PCB design in EAGLE is a two-step process. First you design your **schematic**, then you **lay out** a PCB based on that schematic.



# PCB Design (2)



Your *circuit design software* will allow you to output the PCB layout in a format called **Gerber** with one file for each PCB layer (copper layers, solder mask, legend or silk) to allow manufacturing.

During board fabrication Gerber is the standard input format for photoplotters and all other fabrication equipment.

The final step is having all of the electronic **components soldered onto the board**.

The logo for Fritzing, featuring the word "fritzing" in a white, lowercase, sans-serif font with small circles at the ends of the letters, set against a red rectangular background.

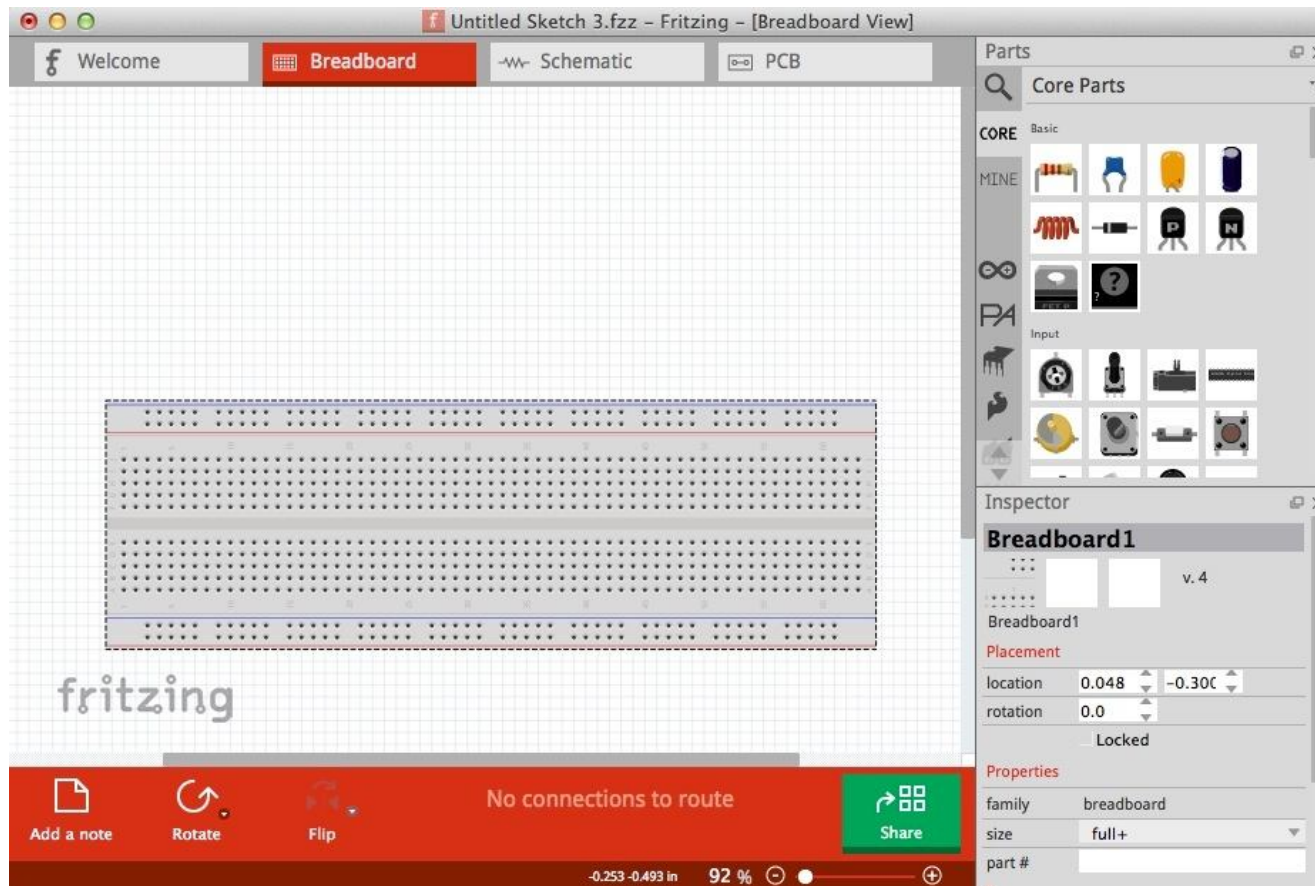
fritzing

Fritzing is an open source tool for anyone to prototype electronic projects. It allows you to **design a schematic** which can then be added to **wiring diagrams**. You can even **design your own PCBs** and have them fabricated from the files you design.

There are **three views / design modes** which are the Breadboard, the Schematic, and the PCB mode. You may choose what suits your particular design needs best and start designing. At any time, you can switch between the view modes to check how the changes are reflected.

<http://fritzing.org/home/>

# Step 1: Design on Breadboard



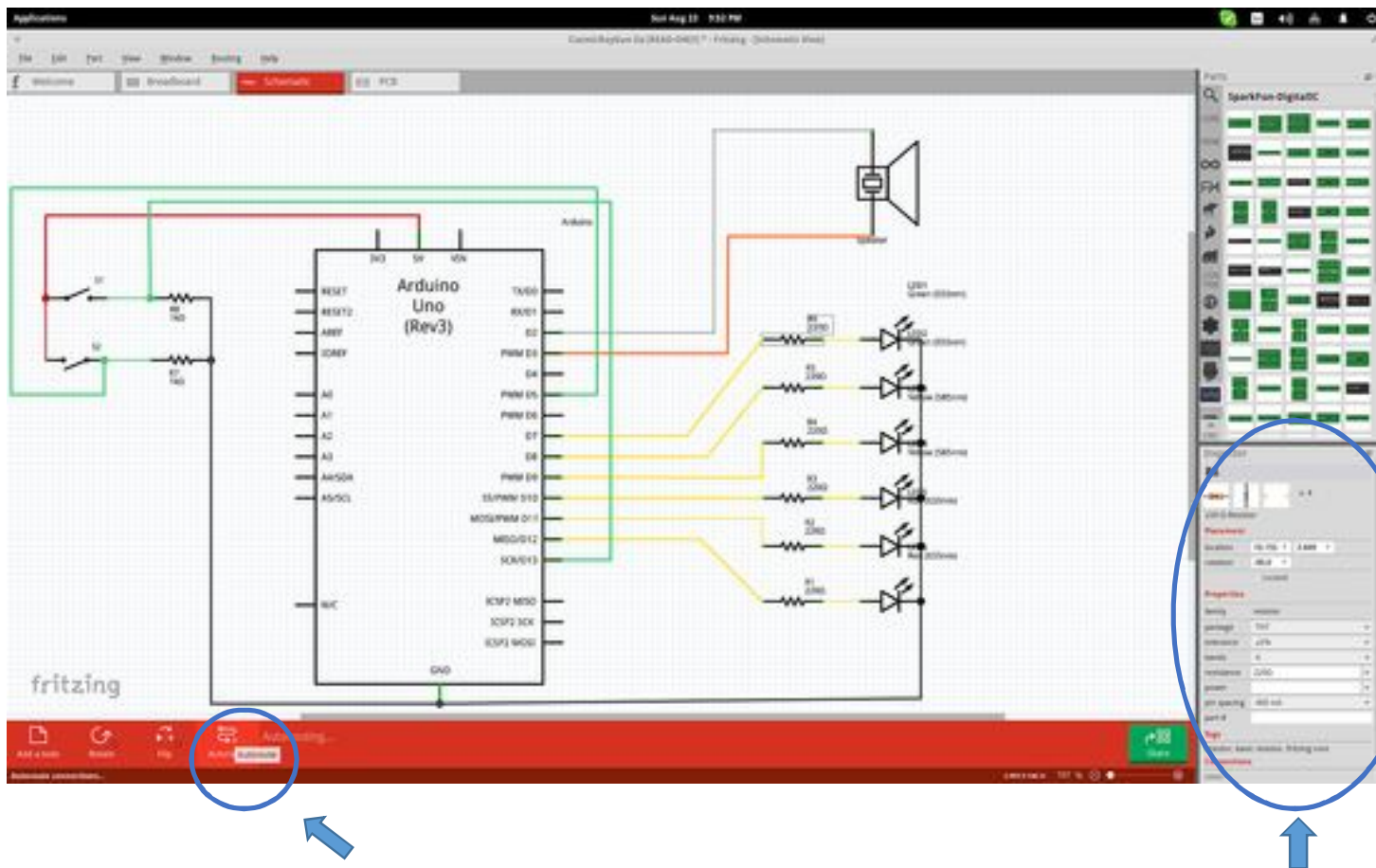
The **parts** are found on the right side and are *categorized based on their type*.

As the part lists are very long, Fritzing also offers a *handy part search engine* that users can invoke by pressing the “magnifier” icon on the right.

Once you find the part you want to use, you may simply drag and drop it on the drawing space in the main window.



# Step 2: Schematic view

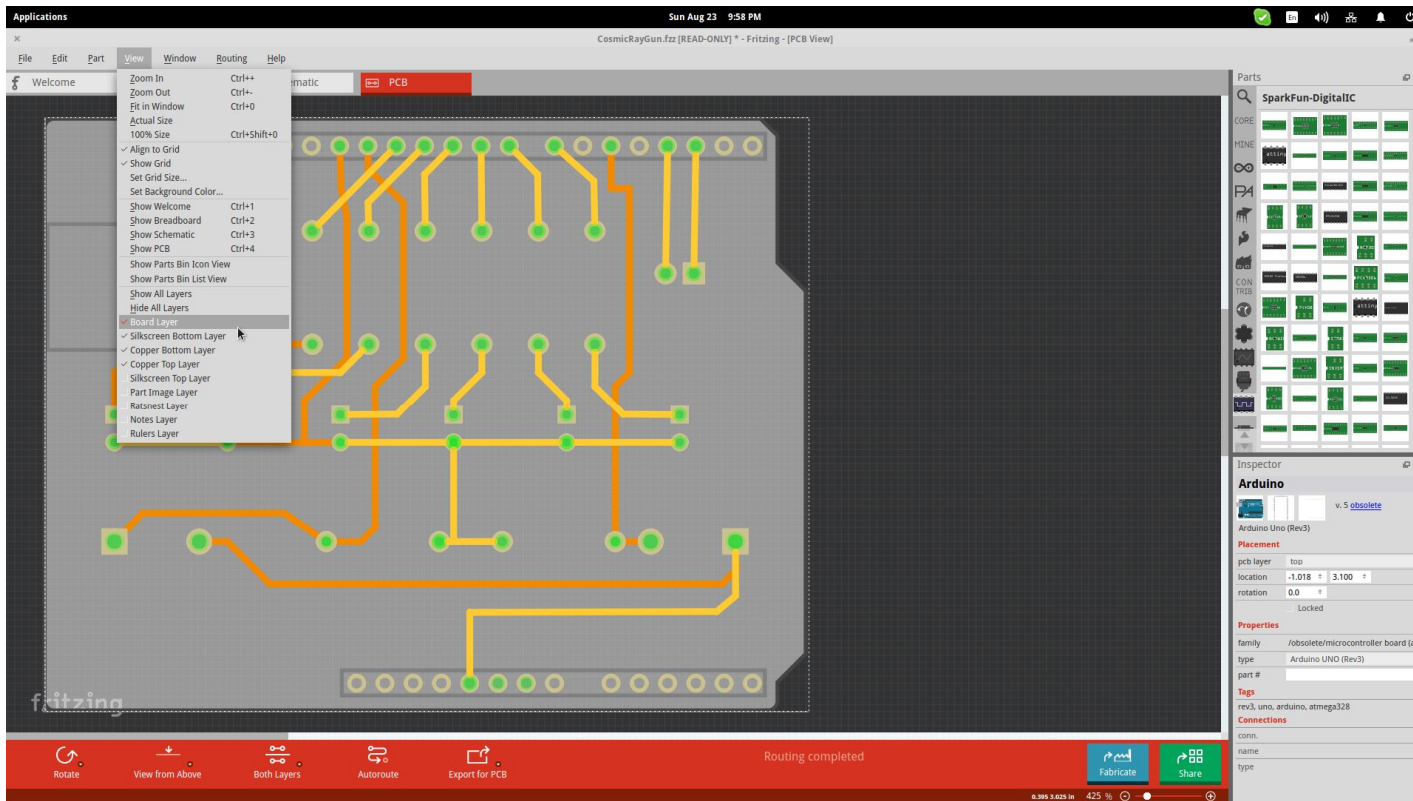


The parts feature **ports** are best shown in the “Schematic” view mode so that the user can comfortably connect what is needed.

There's also an “**auto rooting**” option (press Ctrl+Shift+A) that helps users who are unsure about how to wire their boards in the best way.

Finally, you can change the **technical specifications** of a part by selecting it (after placement) and setting the values from the boxes on the lower right.

# Step 3: PCB View



Especially for the PCB, it would be useful to **check it both from above and from below** before finalizing the positions of each part.

Another helpful viewing feature is the **layer selection** that is done from the "View" menu found on the top panel.

# Starting a new project

Let's start by opening Fritzing, **naming and saving** our project.

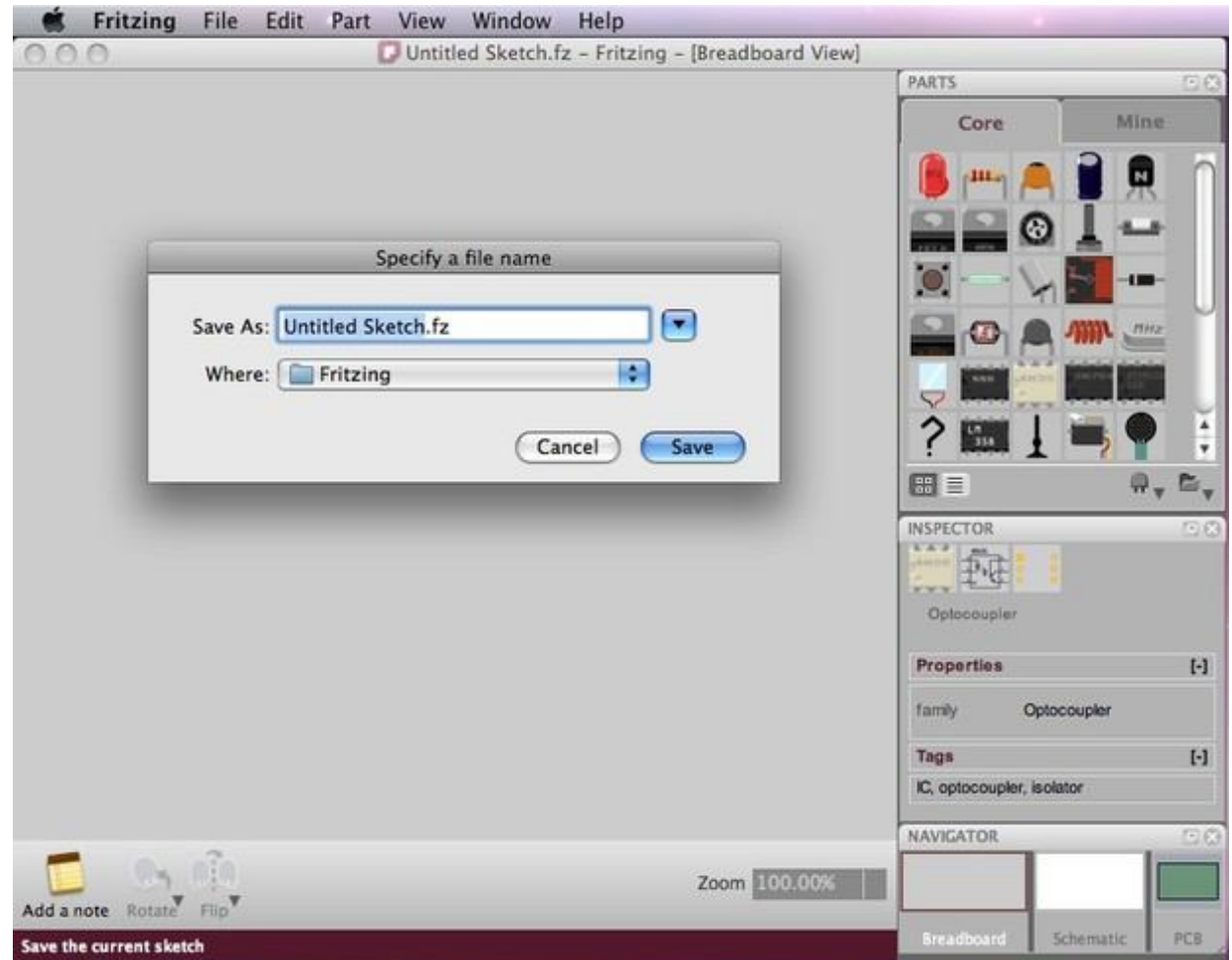
Saving a project is highly recommended at start and every now and then while working, since

Fritzing is still Alpha and unfortunately might sometimes crash...

From the Fritzing menu bar, select

**File > Save As...**

Specify a name and location for the project and click Save.



# Build your circuit

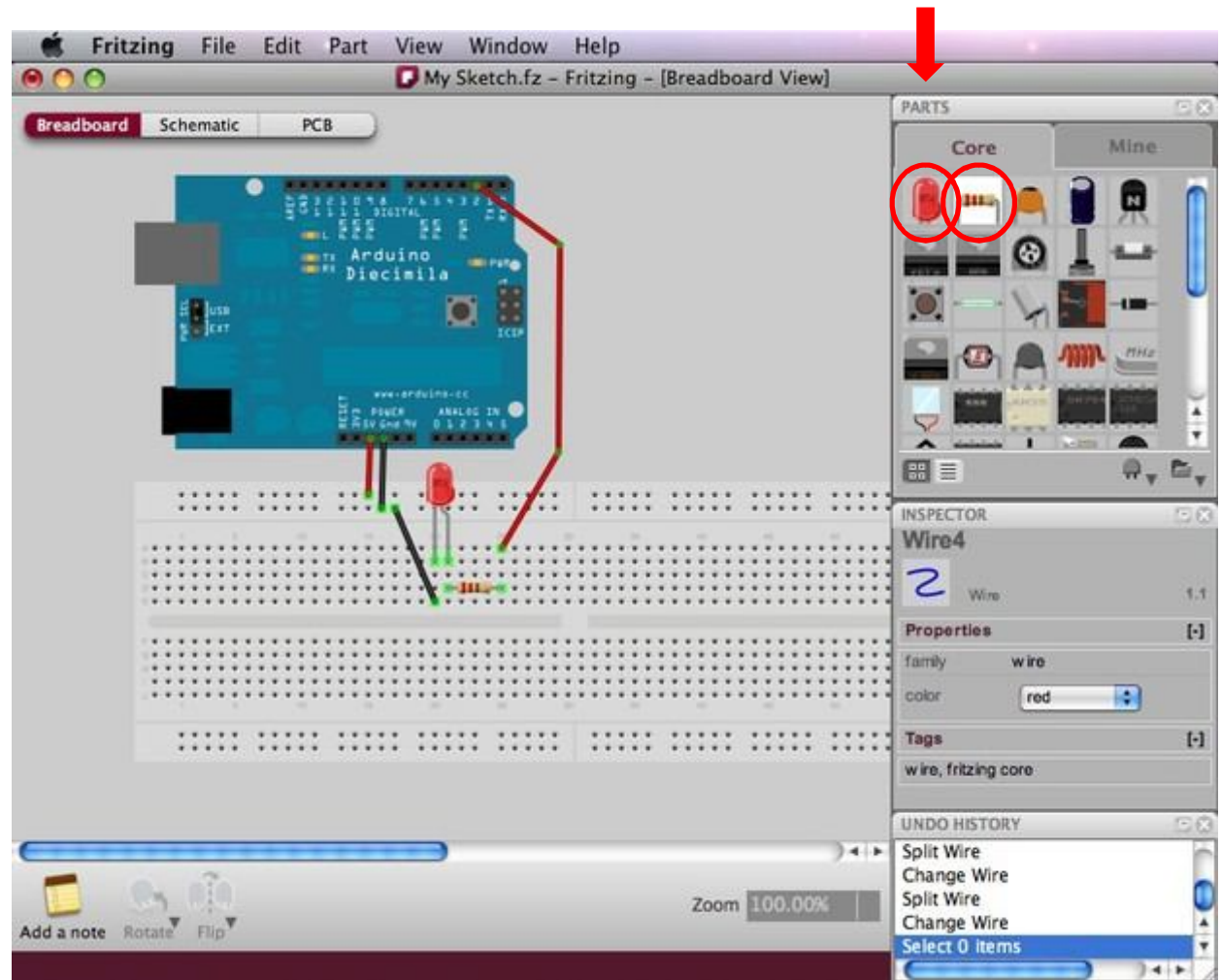
**Drag & drop** the parts you need (Arduino, LED, R) from the Parts palette window to the Project View.

If you cannot find a part in the library, use the Mystery Part (icon looks like a question mark - ?). The Mystery Part will let you quickly define a new part and its connectors (through the Inspector).

You can **arrange parts** by selecting, dragging and dropping, or by using the functions in the menu bar, located under Part.

To delete a part, simply select and press **BACKSPACE**.

Click & drag the Arduino **+5V connector**. *This should create a wire.* Drop the wire on one of the breadboard's connectors. The connection is confirmed by a small green circle or square.





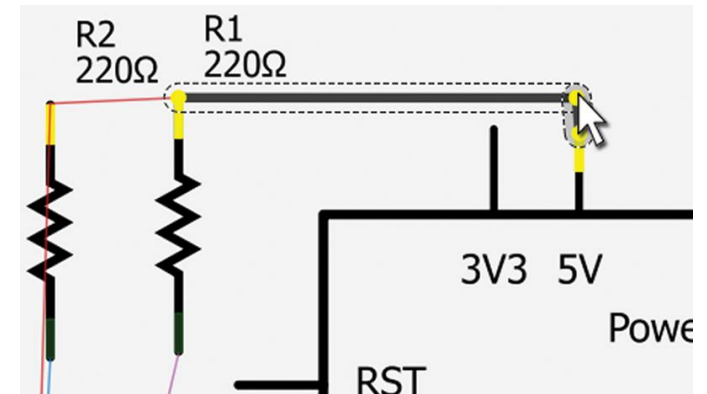
# Build your circuit (2)

**Connect all parts** until the circuit looks exactly like your circuit in the real world. Notice that *connectors that are not properly connected are painted red*.

If you click and hold on a connector, Fritzing will highlight all *equipotential connectors*. This can really be useful if you want to see the whole set of connections attached to this particular connection.

You can **bend wires** by adding bend-points. Just drag them out of a wire.

Select the schematic and pcb tabs to watch or edit your circuit in these **views**.



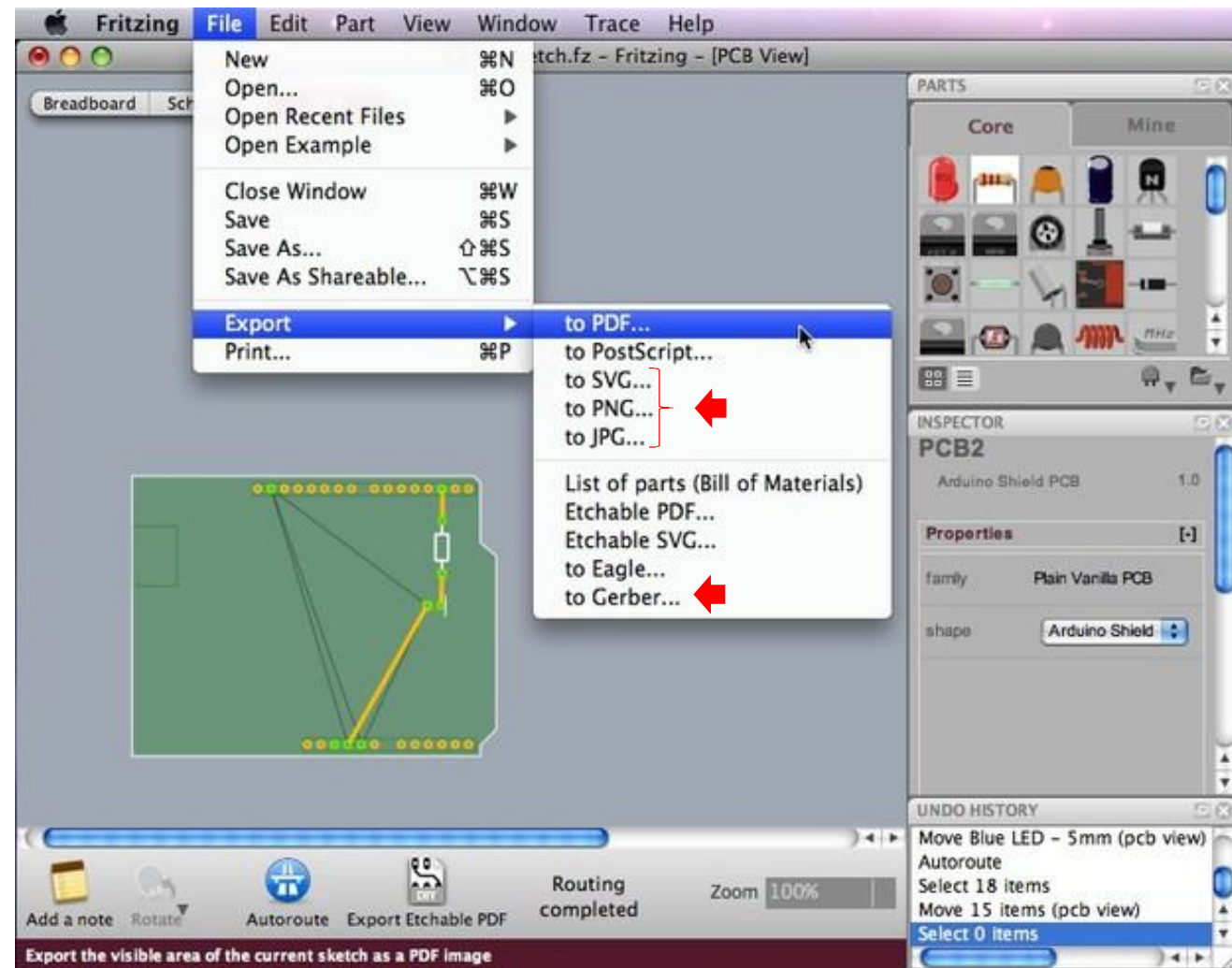
# Exporting a circuit

After finishing building the circuit, save your project. You can export your circuit in different format.

**Select the desired Project View to be exported** (breadboard, schematic or pcb).

From the Fritzing menu bar, select **File > Export >** and the desired format.

**SVG (Scalable Vector Graphics), PNG and JPEG** formats are supported (except for the part pcb graphic which must be an SVG for the gerber files export to work well).



# References

<https://learn.sparkfun.com/tutorials/make-your-own-fritzing-parts>

<http://fritzing.org/building-circuit/>

<http://www.pcb.electrosoft-engineering.com/04-articles-custom-system-design-and-pcb/01-printed-circuit-board-concepts/printed-circuit-board-pcb-concepts.html>