

PCBS

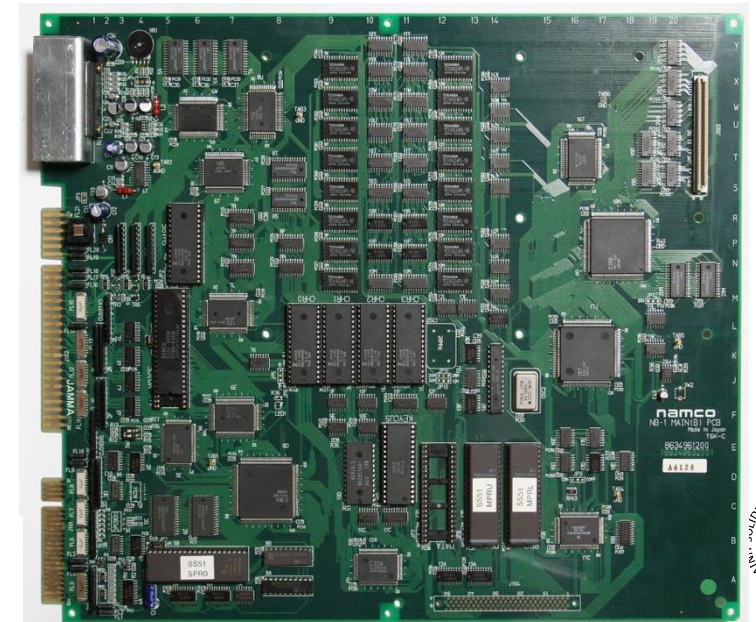
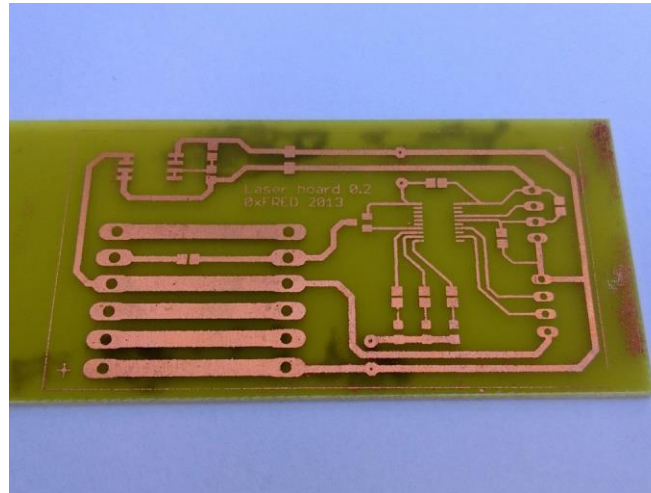
PRINTED CIRCUIT BOARDS

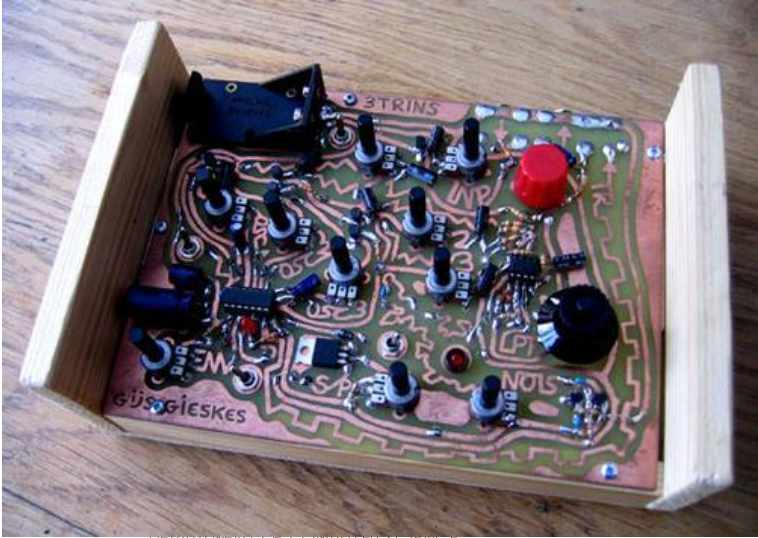
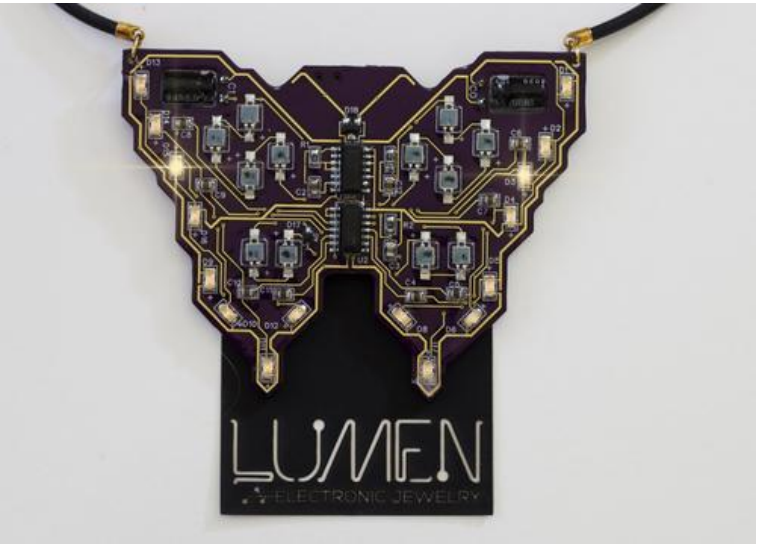
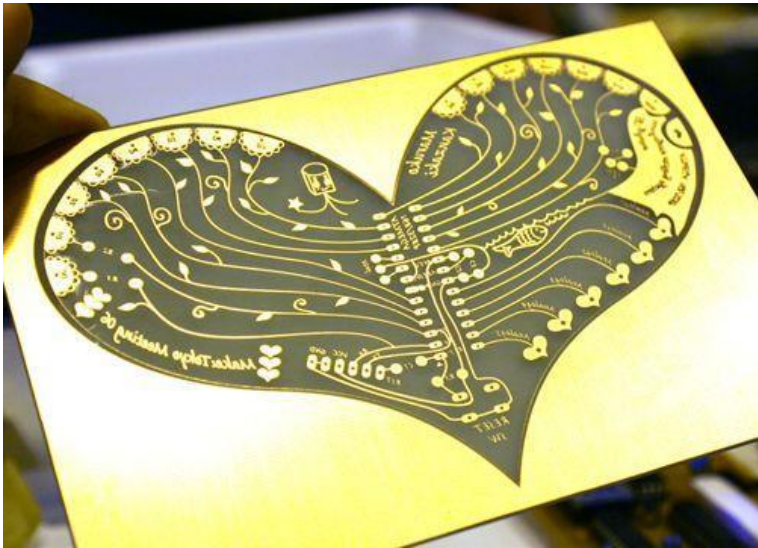
PCBS

Printed Circuit Boards

You can create your own PCB by using different techniques to convert a copper-covered board into a custom designed one.

You can construct highly reliable, compact circuits.





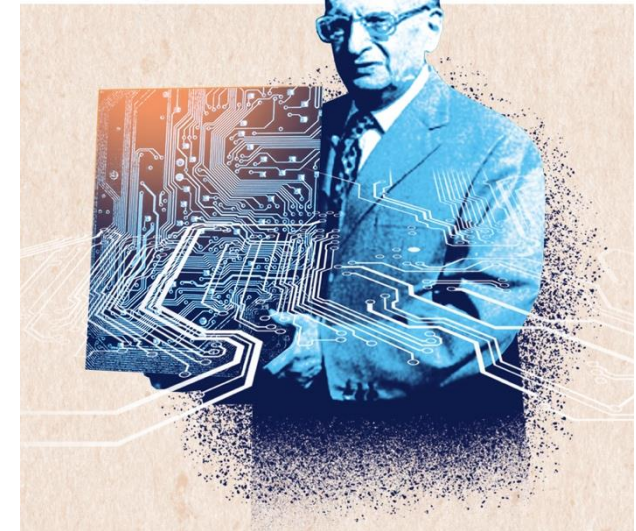
HISTORY

1936: Austrian engineer Paul Eisler invented the printed circuit as part of a radio set while working in the UK

c. 1943: USA began to use the technology on a large scale to make proximity fuses for use in World War II

1948: USA released the invention for commercial use.

1950s: PCBs became common in consumer electronics.



—

ADDITIVE AND SUBTRACTIVE METHODS

ADDITIVE AND SUBTRACTIVE METHODS

Subtractive methods: e.g. milling.

- A mill is used to remove unwanted copper from a prefabricated copper sheet laminated on top of a dielectric board.
- Acid is used to remove unwanted copper from a prefabricated laminate.

Additive methods: the pattern is electroplated onto a bare substrate.

- The advantage of the additive method is that less material is needed and less waste is produced.
- The future? Voltera etc. believes so.
- It is possible to mod a regular inkjet printer to print special silver particles onto special flexible dielectric sheets.

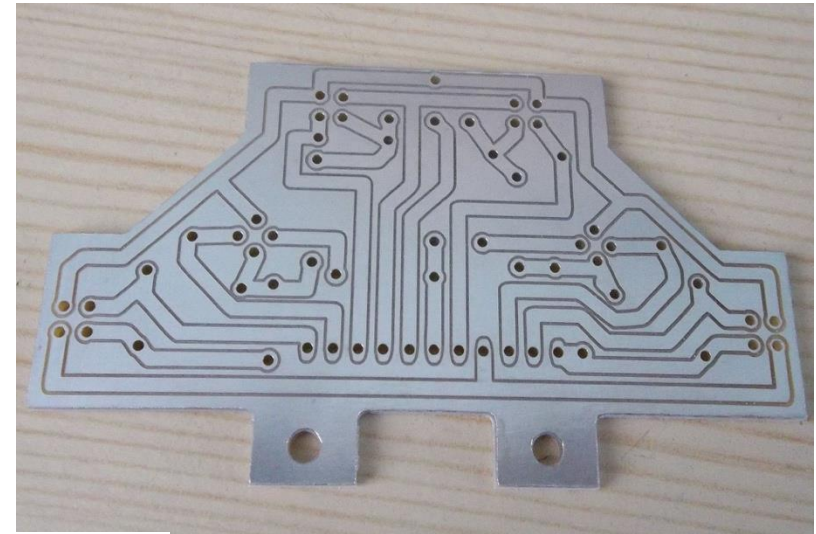


PCB MILLING (E.G. WITH THE 3018)

No chemicals!

CNC can be used for milling (and drilling, if you are brave)

Subtractive method



ETCH YOUR OWN PCB WITH AN IRON

—
... No excuses!



—

BOARD MATERIALS

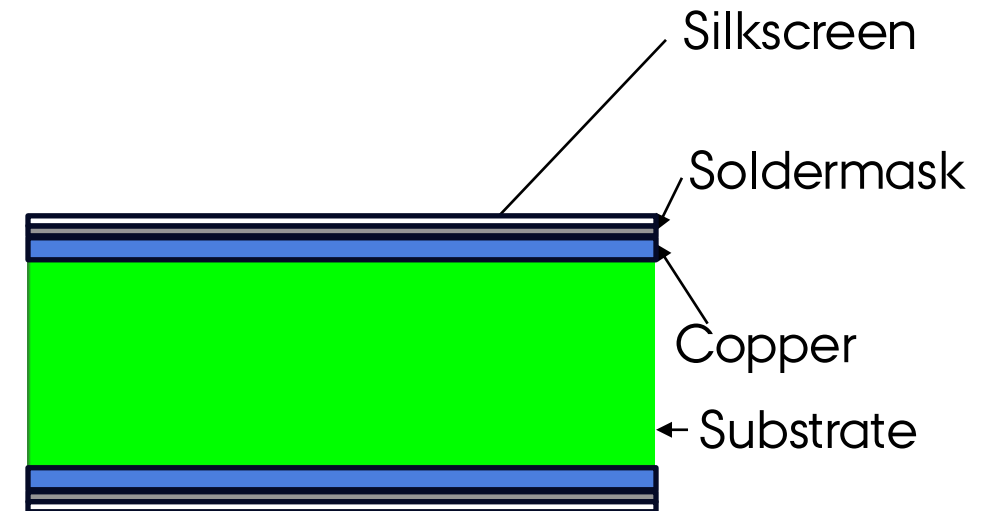
BOARD MATERIALS

Substrate:

- usually fiberglass.
- flexible PCBs have a base made of high-temperature plastic (f.x Kapton).

Copper Layer:

- laminated to the substrate with heat and adhesive



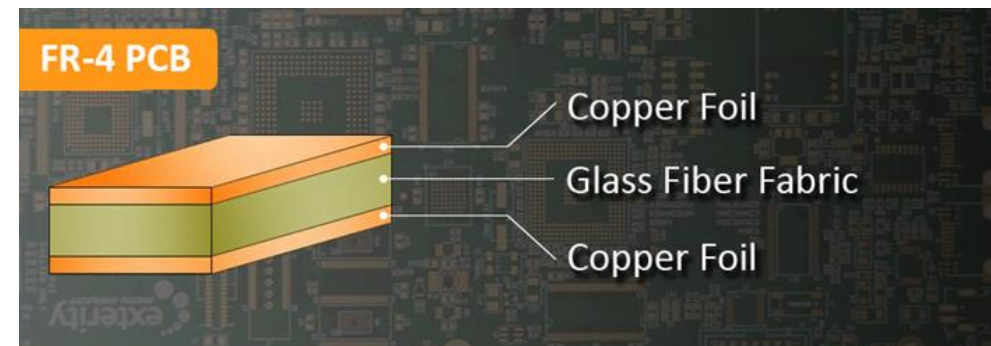
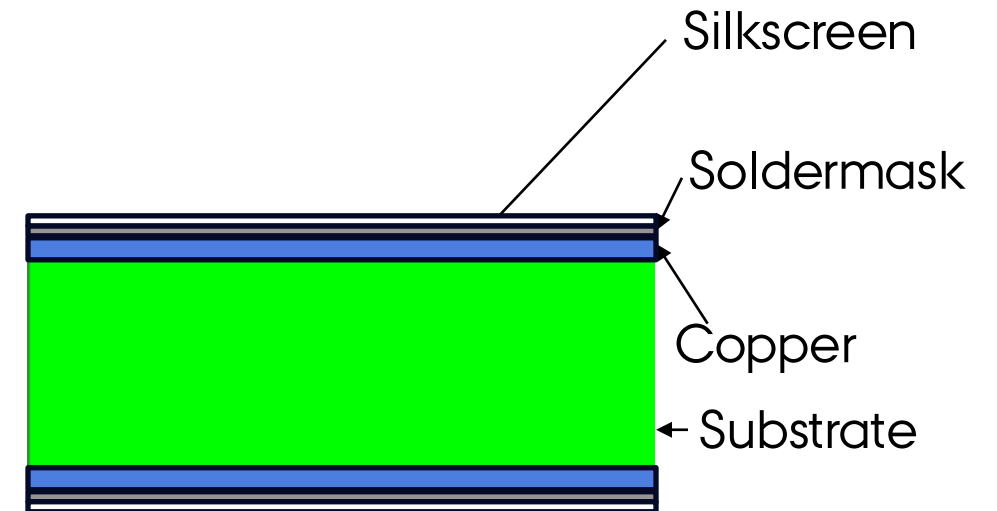
BOARD MATERIALS

Soldermask Layer:

- gives the PCB its green colour.
- insulates the copper traces from accidental contact with other metal, solder, or conductive bits.

Silkscreen Layer:

- adds letters, numbers, and symbols to the PCB
- usually white, but can be other colours



SURFACE MOUNT VS THROUGH HOLE

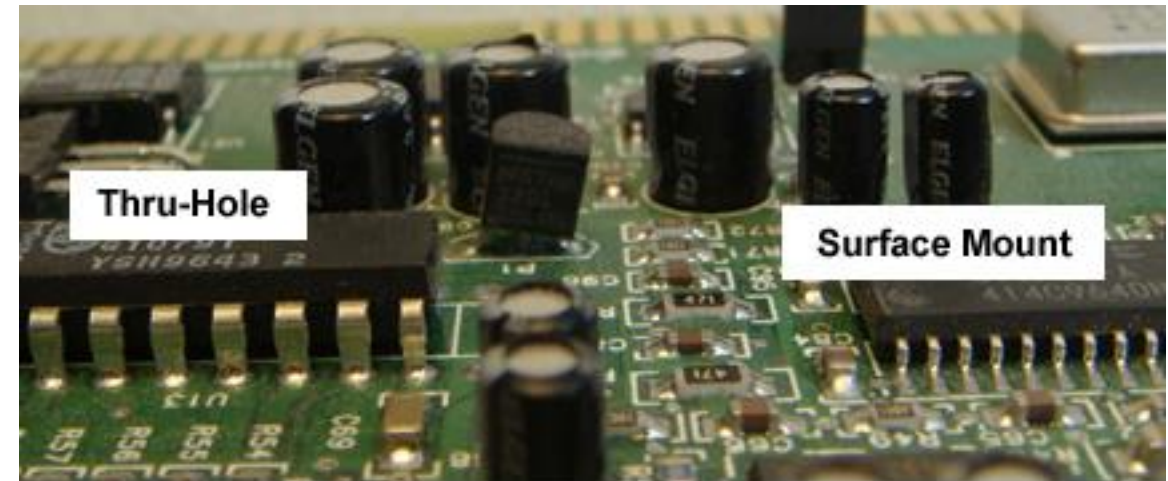
Different ways of attaching electronic components to a PCB

In SMT:

- solder joint forms the mechanical and electrical connection
- connection does not use holes
- denser layout
- cheaper pcbs
- (often) improved frequency response

Through-Hole:

- component pins inserted through holes and soldered underneath (sometimes DIP (dual in-line package) like ATmega328)
- easier to assemble by hand and to prototype



From Computer Desktop Encyclopedia



AARHUS
UNIVERSITY

— (GENTLE) INTRO TO ELECTRONIC DESIGN WITH FUSION360



SO... HOW DO YOU DESIGN PCBs?

Popular programs: Fusion360/Eagle, KiCad and Fritzing

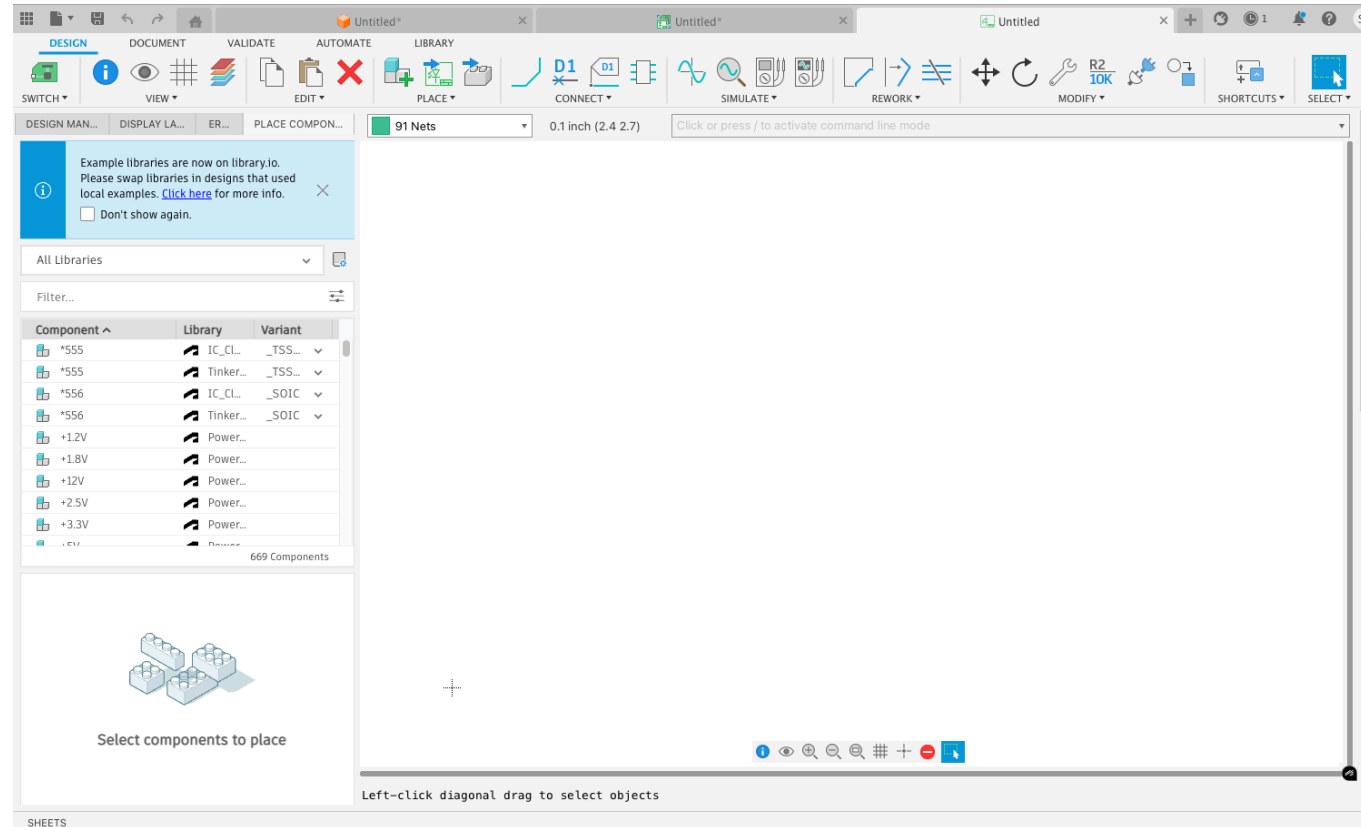
We focus on Fusion360 – choose another program at own risk (or for aesthetic reasons)

Fusion360: <https://www.autodesk.com/education/edu-software/overview?sorting=featured&filters=individual>

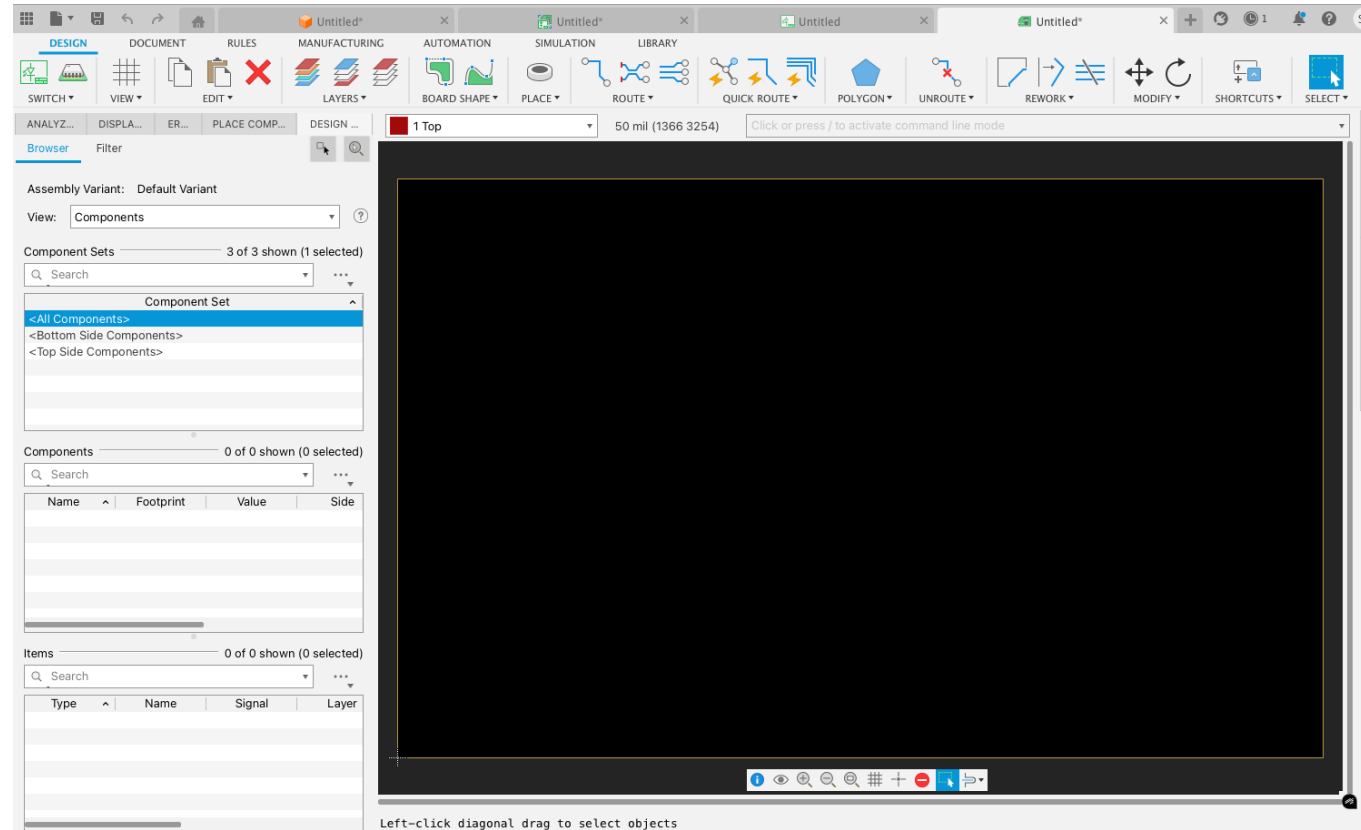
Tutorials:

- Labtools tutorial on PCB manufacturing with Fusion360 and 3018 Pro CNC: <https://gilded-sidewalk-baf.notion.site/Designing-PCB-Programming-CNC-3018-PRO-4e08230d29914d42bed1495d756c2f66?pvs=143>

SCHEMATIC SIDE



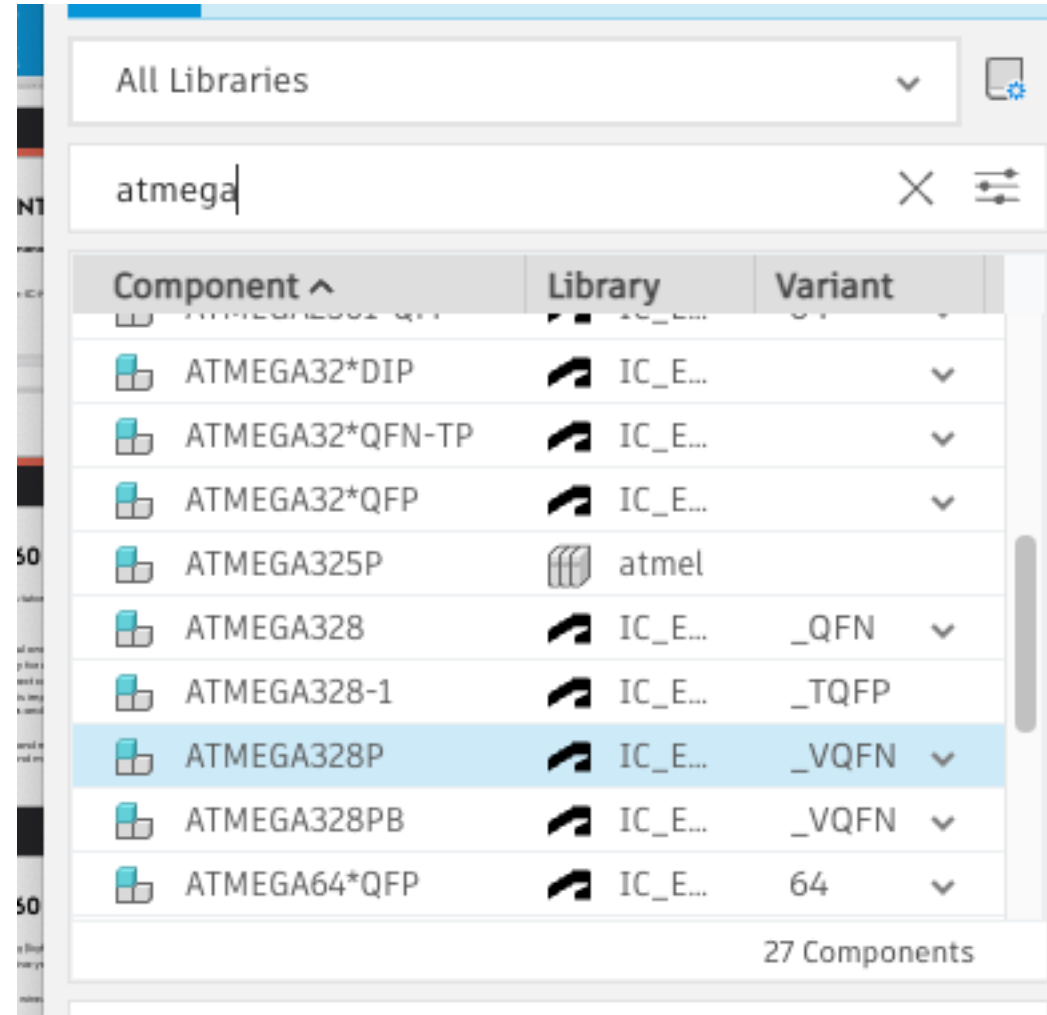
BOARD SIDE



IMPORTANT LIBRARIES

Many components are default!

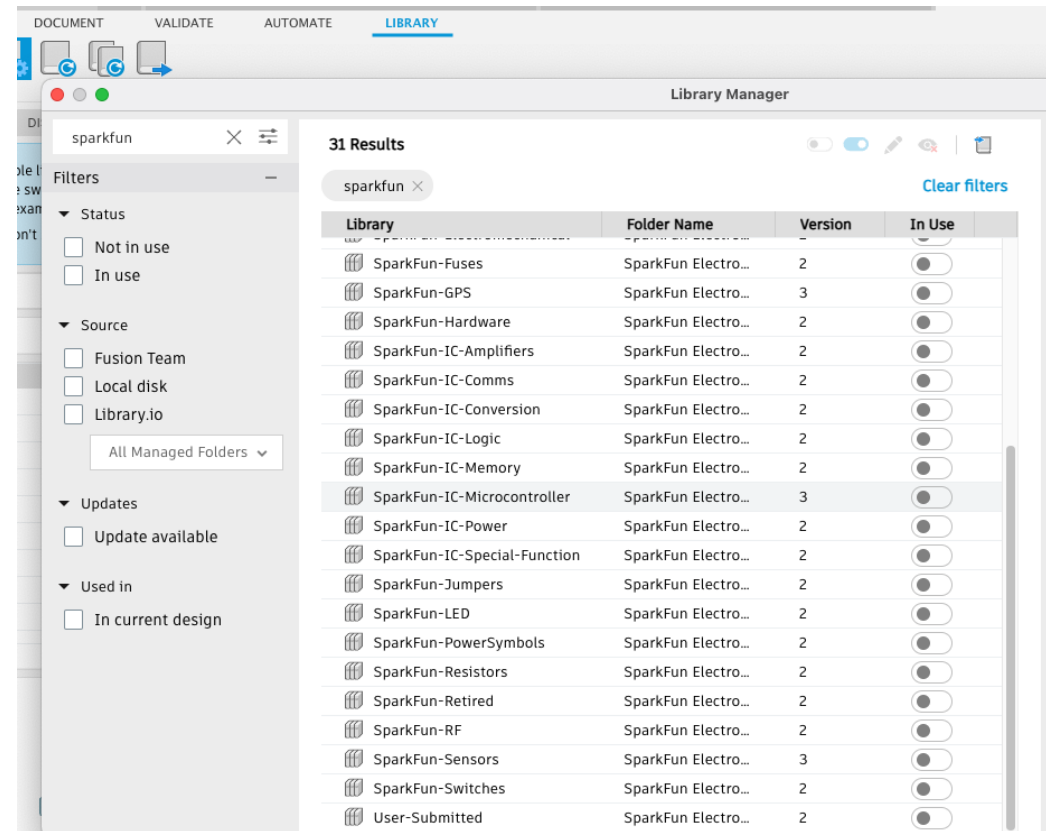
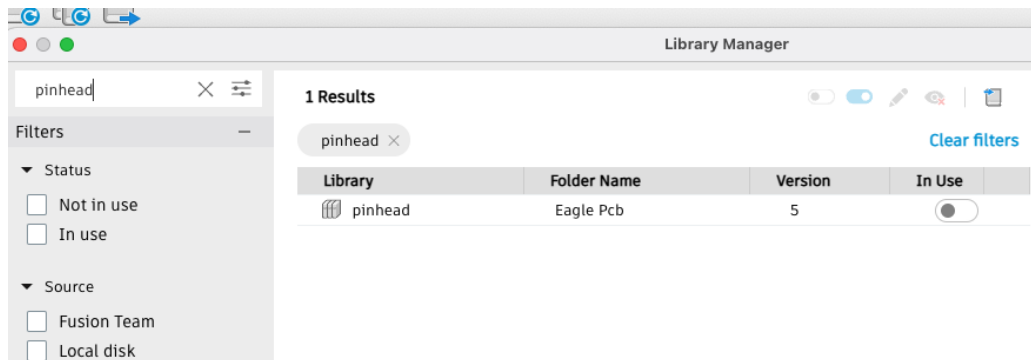
Some libraries include the 3D model as well

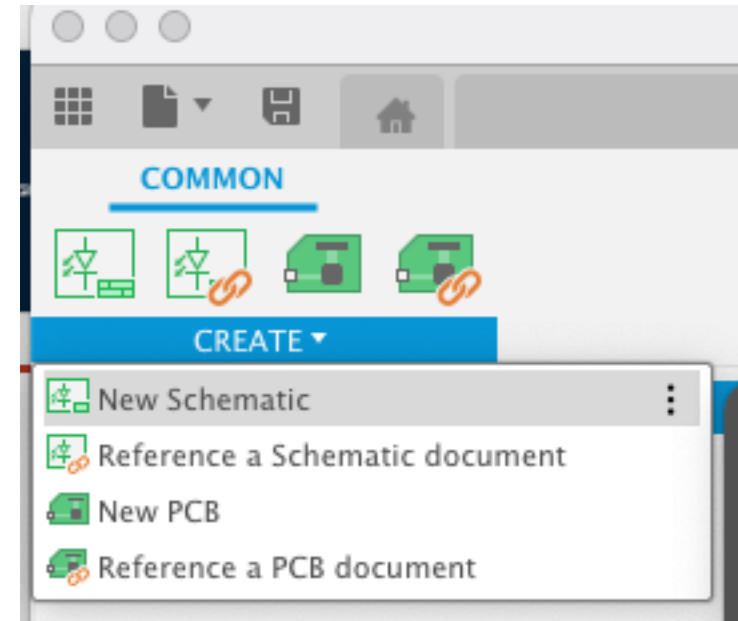
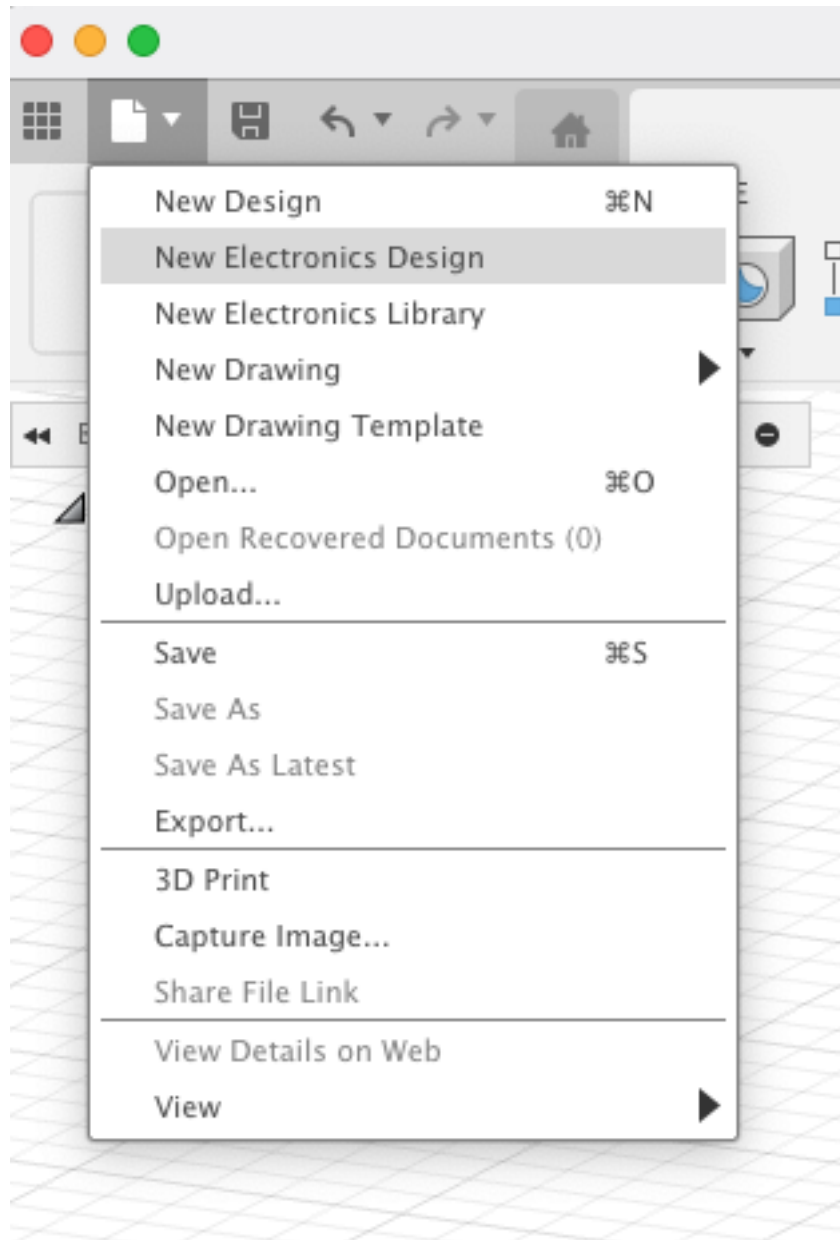


IMPORTANT LIBRARIES

Use the library manager to search and include additional libraries.

- SparkFun-IC-
- Pinhead
- Atmel





FUSION360 ELECTRONICS DESIGN

Your job: Read our guides, watch tutorials, and familiarize yourself with the interface and machinery.

Tips:

- Create useful and well-named labels
 - Especially for ground and power connections!
- Use the correct components. If you cannot find them, mimic them with pinheaders.
- Readability is important, both in terms of text and symbols but also placement of components and wire crossing.

Take your time and make traces and connections in schematic AND board layout, that are easy to follow and maintain!

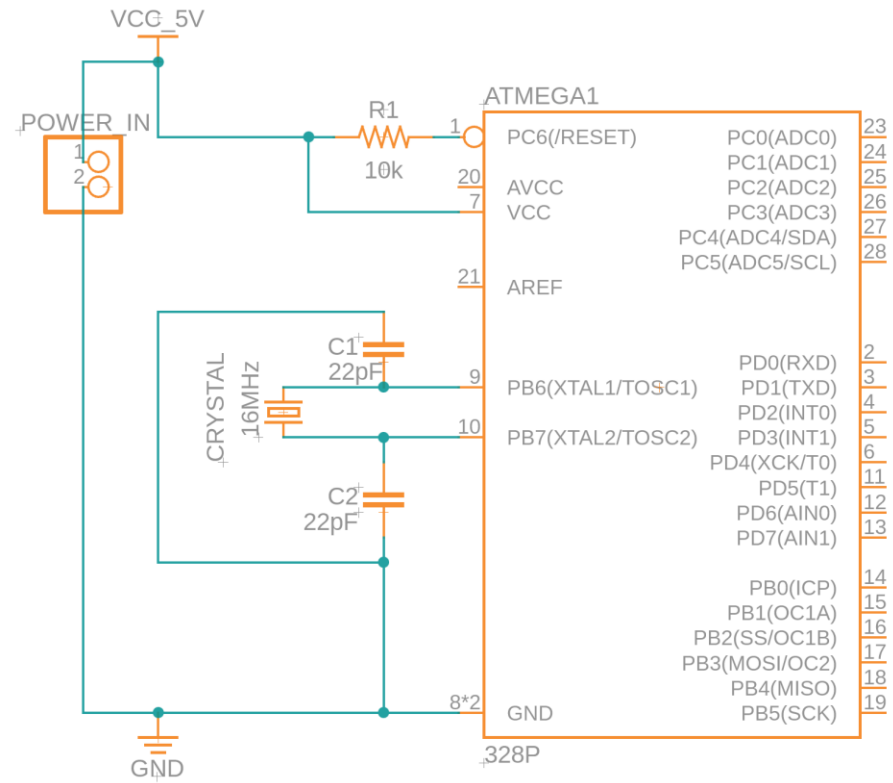


FUSION360 ELECTRONICS DESIGN (2)

Use auto-routing (lightning bolt)!
Even if it can't give you 100%...

Set appropriate wire/trace and net/pad dimension – and remember it in the autorouter as well!

- Appropriate = as big as possible
- DRC controls this – "Design Rule Check"
 - Use the Labtools DRC file to begin from
- ERC is an overview of errors and notes – always check this before doing a PCB.
 - Warnings can sometimes be ignored – check the guide



SCHEMATIC LAYOUT TIPS

Overall schematic layout

The lowest voltage should be shown at the bottom, and the highest voltage at the top.

The information should flow from left to right.

- Sensors and other input devices on the left, and outputs such as motors, LEDs, on the right.



BOARD LAYOUT TIPS

Try to make all conductive pathways short and direct, without crossings.
Avoid above 90 degree angles when “bending” pathways.

Make important nodes accessible (for debugging/attachment/reiterations etc).

Leave approx. 5mm border around the circuit board

Bring power supply leads or other input/output leads to the edge of the board.

Add polarity markings on the board next to devices such as diodes and electrolytic capacitors.

Use writing on your board to fix orientation and double check mirroring (e.g “SuperPCB V1337”).

PCB TIPS

Use the connector sockets for the ATmega instead of soldering it directly.

Use (screw)terminals etc. to attach power and externals.

Remember to include the oscillator circuit in your ATmega 328 projects.

Some even include connectors to directly upload to PCB (not required, but could be useful)

Do not waste material – make your PCBs as small as possible, and doublecheck routing before PCB development.





AARHUS
UNIVERSITY

PRACTICAL EXERCISE

PCB Design

DESIGN AND PRODUCE A PCB

Create a functional PCB based on the circuit you created in Practical Exercise 4 – Actuation

- Voltage divider circuit that takes input from an analog sensor and uses it to control an actuator.

Draw the schematic in Fusion360. Convert to board layout and make a PCB design.

Go to the CNC and mill the PCB.

Attach and solder components onto the PCB.

DESIGN AND PRODUCE A PCB

Remember to follow guidelines from the lecture

- label all components, add values
- Information should flow from left to right etc.

This is crucial, you will not pass without following the guidelines.

Group Exercise, make sure everyone participates

Use the labtools guide - <https://gilded-sidewalk-baf.notion.site/Designing-PCB-Programming-CNC-3018-PRO-4e08230d29914d42bed1495d756c2f66?pvs=143>



TO PASS -

Show your schematic and functioning PCB to Eve, Simon, Mai, or Mie.

Deadline: 04/11/2025 at 09:00



AARHUS
UNIVERSITY



AARHUS
UNIVERSITY