Ecodesign of Electronic Devices

UNIT 11: Computer-aided design of electronic devices

Funded by the Erasmus+ Programme of the European Union

Ecoinnovation Skills for European Designers, Project number: 562573-EPP-1-2015-1-SI-EPPKA2-SSA.

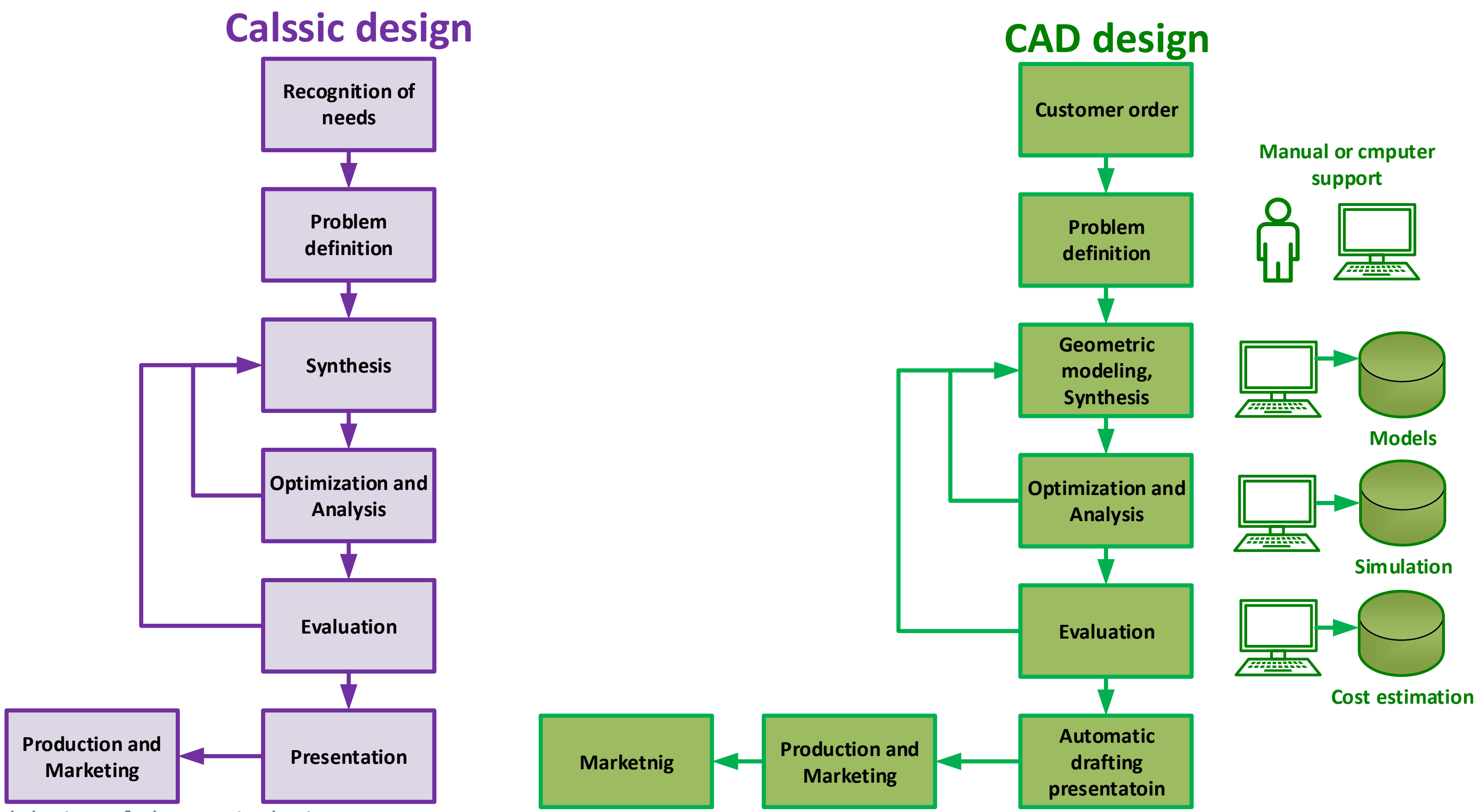
Computer-aided design of electronic devices

- Computer-aided design CAD is established in many industrial areas.
- CAD design means that we use tools of information technologies IT in the design process.
- CAD design consists of hardware, specific software and external devices, and interfaces. The core of CAD design is program package that uses graphics for display, different datasockets and drivers for peripheral devices.
- CAD design does not change design process but significantly facilitates and speeds it up. These tools are very efficient especially at designing devices that need to be ecologically efficient.
- The essence of CAD design is summed in the following points:
  - Precise graphical presentation of the product. It is also easier to analyze, modify or upgrade it.
  - Enables complex design in very short time.
  - Enables simulation of different events, such as electrical, chemical, thermal and mechanical.
  - With simulation tools, it is easier to provide optimal approach to design and the product itself.

Funded by the
Erasmus+ Programme
of the European Union

Computer-aided design of electronic devices

- Classic vs. CAD design:



Funded by the Erasmus+ Programme of the European Union

Printed circuit board-PCB design

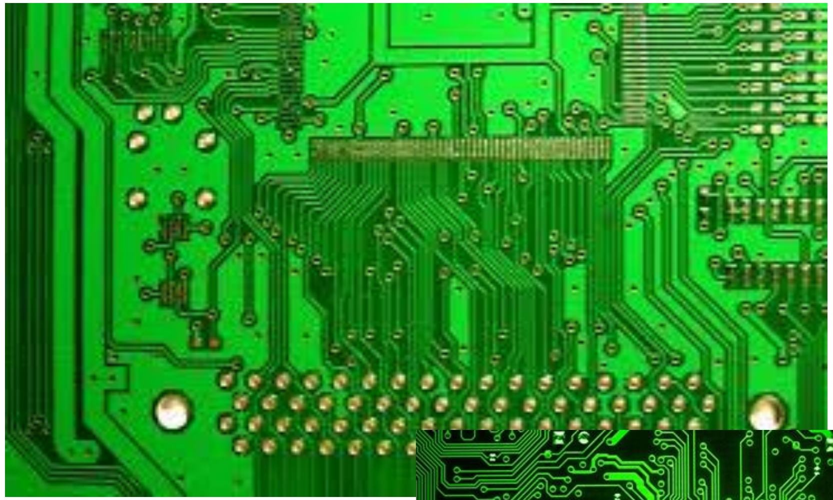
- The design of printed circuit boards - PCB ('Printed Circuit Board') in the design of electronic devices is a key task of any development.
- All the elements and the size of the circuit itself are determined from the printing design procedures. When breaking out the components and the size of the printout, we can meet many environmental guidelines.
- Nowadays, we know many advanced technologies that enable multiple layered printing. Several layered prints provide a smaller surface area and, consequently, a slightly lower print material consumption.
- In making prints, it is necessary to find a compromise between the selected elements and the size of the printing press, which have the lowest ecological impact in the production phase.
- The print design standards are controlled by IPC. IPC takes care of the standardization of the production of printed circuits and the use of materials. The main document covering the design of printed circuit boards is the ICP-2221-'Generic Standard on Printed Board Design 'document.

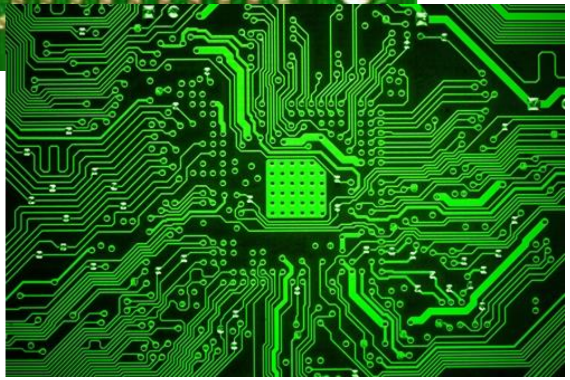
Funded by the Erasmus+ Programme of the European Union

Printed circuit board-PCB design

The standard steps of the PCB design are:

- Specification of the project.
- Planning an electrical scheme.
- Design of the printed circuit board.
- Prototyping
- Testing
- Production

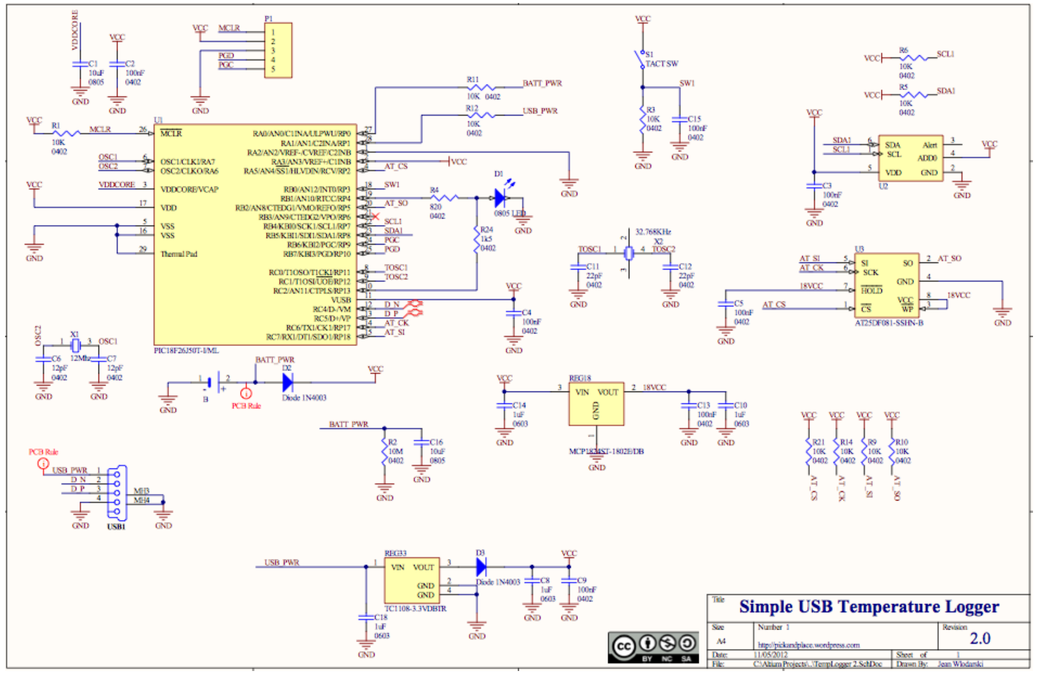




Funded by the Erasmus+ Programme of the European Union

Designing electronic schemes

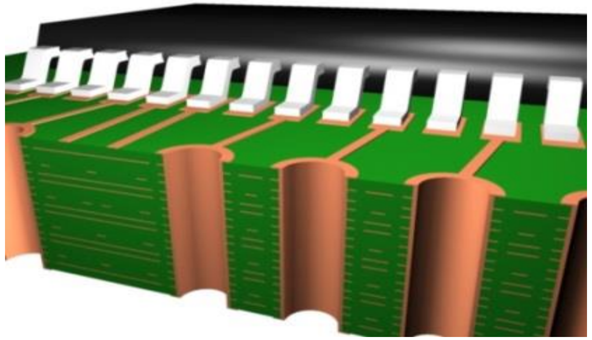
- When designing electrical schemes, it is important that the scheme is arranged, the connections being logical and minimizing intersection with each other.
- Also, a good scheme is designed so that it is very similar to the final prints.
- Let's stick to the unwritten rule all the inputs are on the left and all outputs on the right side of the schema. We make comments and a comment if necessary.
- In the case of complex circuits, the areas of the scheme are defined for better transparency.



Funded by the Erasmus+ Programme of the European Union

PCB design

- Equally important, as the drawing up of the scheme, the next step is the production of a printout.
- Designing a printout involves the layout of the elements and the connection between them. The final appearance of a tight circuit is determined by printing.
- When making a printout PCB, it is very important to use poper element libraries.
- The following rules must be followed when making a print:
  - Choice of printing (material, thickness).
  - Number of print layers.
  - Layer layout.
  - Distribution of circuits on the printed matter.
  - Consideration of parasitic effects.
  - Placement of components on the printed matter.



Funded by the Erasmus+ Programme of the European Union

PCB-Copper lines

- Copper line thickness is chosen depending on electrical requirements and space on printed matter.
- Thicker lines give faster responses and better results. Wider and thicker lines have lower resistance and shorter length, their production is easier and cheaper, they are easier to be repaired and examined.
- When choosing printed matter manufacturer, we need to know what are some of the closest lines and the smallest spacings that the manufacturer can still provide.

Preffered Line Width (mil) 1mil=0.0254mm		
Current [A]	1oz Width (µm)	2oz Width (µm)
1	350	175
2	1050	525
3	1750	875
4	2800	1400
5	3850	1925
6	5250	2625
7	6300	3150
8	7700	3850
9	9100	4550
10	10500	5250

$$R = \frac{\rho_{Cu} l}{S}, \qquad \rho_{Cu} = 1.724 \times 10^{-6} \Omega cm \qquad L = \frac{1}{20}\lambda = \frac{1c}{20v},$$

Funded by the Erasmus+ Programme of the European Union

PCB-Copper lines

- Copper wire resistance is calculated with the formula:

$$R = \frac{\rho_{Cu} l}{S}, \quad \rho_{Cu} = 1.724 \times 10^{-6} \Omega cm,$$

- here R is resistance, l length, S cross-section and ρ_{Cu} specific copper resistivity. Line length is conditioned by anticipated signal frequency. Example of good practice gives an estimation on line length depending on signal frequency:

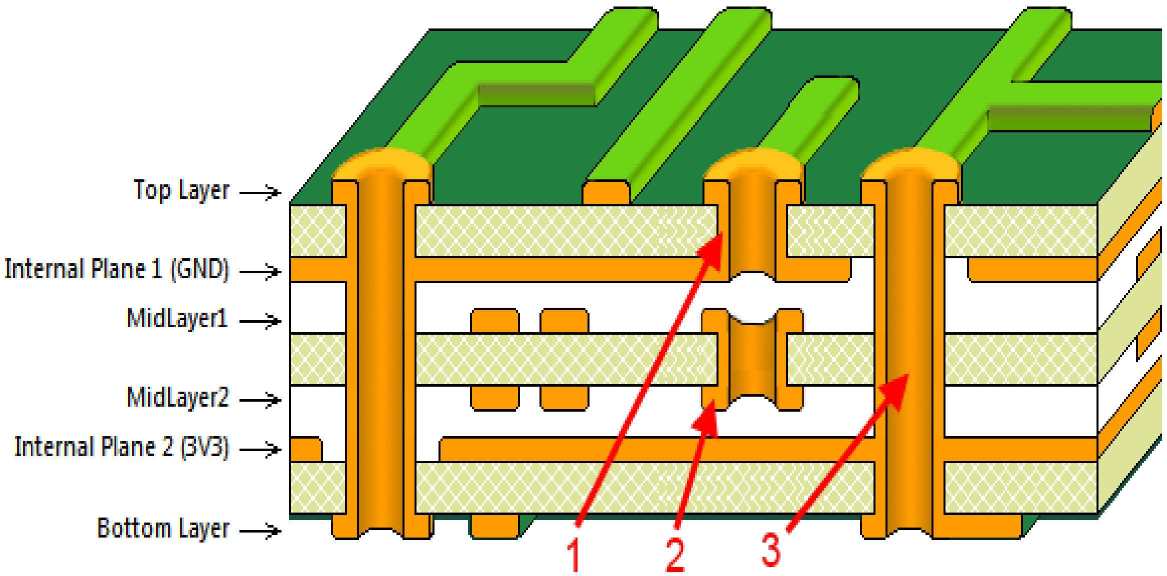
$$L = \frac{1}{20}\lambda = \frac{1c}{20\upsilon} \text{ ,}$$

- where L is permissible line length, λ is wavelength, υ is wave frequency, c is speed of the light. At high frequency signals the, condition cannot be fulfilled, so it is necessary to consider signal propagation time on the given line, which causes time delay. In parallel conductors, it is important that they are shorter and equal in length. Most program packages correct lengths of critical conductors. At high frequency lines, we also avoid RF connectors, which cause disturbances and signal loss.

Funded by the
Erasmus+ Programme
of the European Union

PCB - Vertical interconnect access VIAs

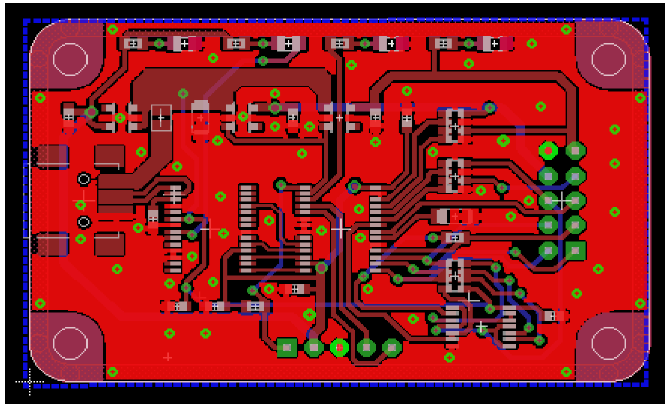
- VIAs are metalized copper lines with different layers.
- They are very similar to connecting pads that we must not confuse them with.
- The connection pad is part of the component socket, and VIA only bridges connections between different printed matter layers. Image 6 presents VIA in multilayer printed matter.

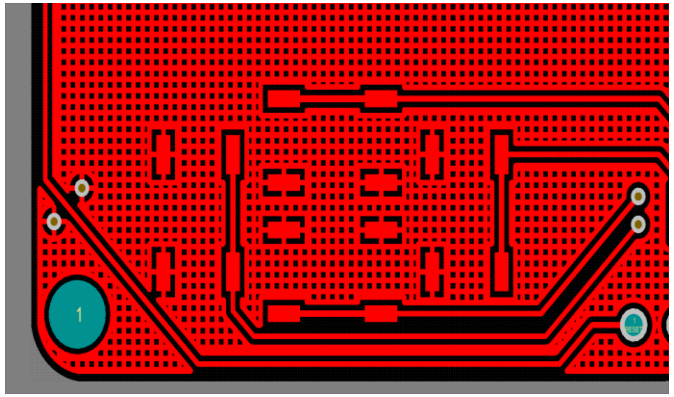


Funded by the
Erasmus+ Programme
of the European Union

PCB -Polygons

- Polygons are used for filling larger areas with pure copper or copper texture.
- Polygons are interconnected by connection pads and VIAs. Usually, they are used for substituting grounding and power supply surfaces. We install them at the end after all other copper lines are already plotted.
- When plotting polygon and lines, we need to consider the empty space between lines or polygon. Too small distances are not preferable because they can cause hairline short circuits that can occur in the production phase. We also need to consider the lower limit for producing printed matter.





Funded by the Erasmus+ Programme of the European Union

PCB -Polygons

- By dimensions, we also need to consider galvanic isolation. These distances are defined by IPC standard. Distances differ by whether they are inside or outside of printed matter and the area where the electronic component will be used (humid environment, altitude, etc.). We often protect printed matter with lacquer coating that increases galvanic corrosion resistance and protects the circuit from external effects. Table presents standard distances and layers for different components of a printed circuit by voltage. Galvanic isolation is determined by breakdown voltage. Breakdown strength is a material characteristic that is given by the following formula:

$$E_p = \frac{U_p}{d} \quad \left[\frac{V}{m}\right],$$

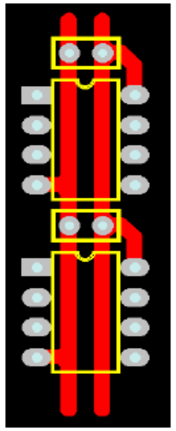
Voltage	Internal layers	External conductors uncoated	External conductors coated
[V]	[mm]	[mm]	[mm]
15	0.05	0.1	0.05
20	0.05	0.1	0.05
50	0.1	0.6	0.13
100	0.1	0.6	0.13
150	0.2	0.6	0.4
170	0.2	1.25	0.4
250	0.2	1.25	0.4
300	0.2	1.25	0.4
500	0.25	2.5	0.8
1000	1.5	5	2.33
4000	9	20	11.48
5000	11.5	25	11.53

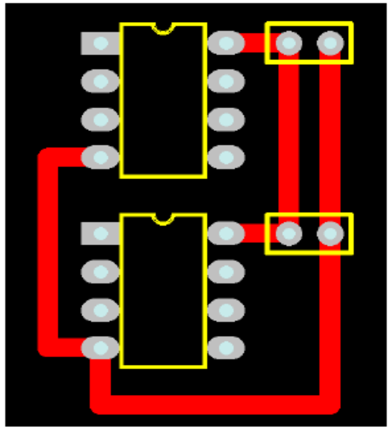
Isolant (20˚C)	$E_p \left[\frac{V}{m}\right] \times 10^6$
Air	3
Paper	10
Pubber	10
Transformer oil	15
Porcelain	20
Polyvinyl Chloride-PVC	50
Polystyrol	80

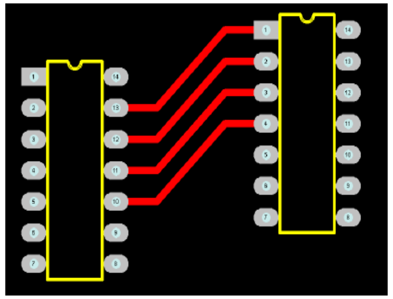
Funded by the Erasmus+ Programme of the European Union

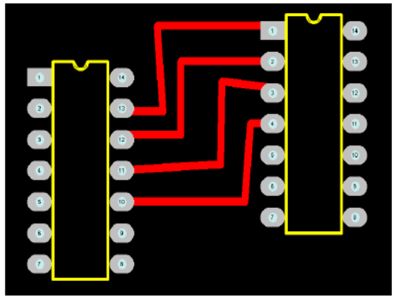
Basic rules for connecting printed matter components

- Connecting components is the installation of copper connections to print between component connectors.
- The electrical connections between two or more connecting pins are called an electrical signaling network.
- We focus on these shorter electrical signaling networks, longer connections result in greater parasitic effects. The links should be broken at angles of 45 ° or we use rounded links.
- Power and ground connections should be as wide as possible due to higher current load. We run the power and ground connection as closely as possible, which can be effectively blocked by the capacitive elements.
- There must be no unconnected copper islands on the print.









Funded by the Erasmus+ Programme of the European Union

Choosing printed matter

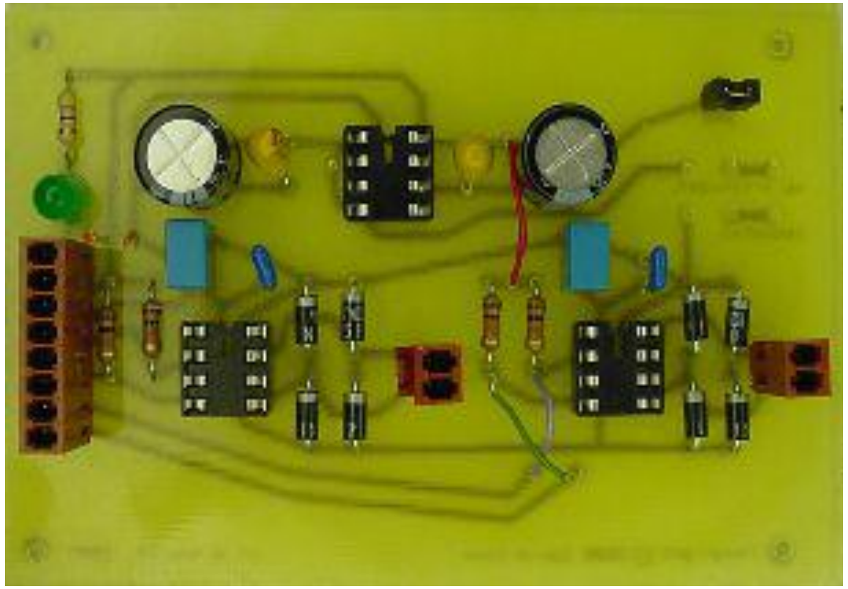
- When choosing materials for the printed matter, we have different possibilities that differ in several characteristics, such as: fire safety, temperature stability, moisture absorption. These characteristics are defined by international association NEMA - National Electrical Manufacturers Association.

Material	Comment
FR-1	*Bakelite: at room temperature poor moisture resistance.*
FR-2	*Bakelite: suitable for single-layered PCBs, good moisture resistance.*
FR-3	*Epoxy resins: a balanced material with good mechanical and electrical properties.*
FR-4	*Glass fibers: excellent mechanical and electrical properties.*
FR-5	*Glass fibers: high strength at high temperatures, self-extinguishing.*
G10	*Woven glass and epoxy: high insulation resistance, maximum mechanical strength, high moisture resistance.*
G11	*Woven glass and epoxy: resistant to bending at high temperatures, extreme solvent resistance.*
CEM-1	*Cotton paper and epoxy.*
CEM-2	*Cotton paper and epoxy.*
CEM-3	*Non-woven and epoxy.*
CEM-4	*Woven glass and epoxy.*
CEM-5	*Woven glass and polyester.*
PTFE	*Pure - expensive, low dielectric loss, for high frequency applications, very low moisture absorption (0.01%), mechanically soft. Difficult to laminate, rarely used in multilayer applications.*
RF-35	*Fiberglass-reinforced ceramics-filled PTFE. Relatively less expensive, good mechanical properties, good high-frequency properties.*
Alumina	*Ceramic: Hard, brittle, very expensive, very high performance, good thermal conductivity.*
Polyimide	*A high-temperature polymer: Expensive, high-performance, higher water absorption (0.4%). Can be used from cryogenic temperatures to over 260 °C.*

Funded by the Erasmus+ Programme of the European Union

Choosing printed matter

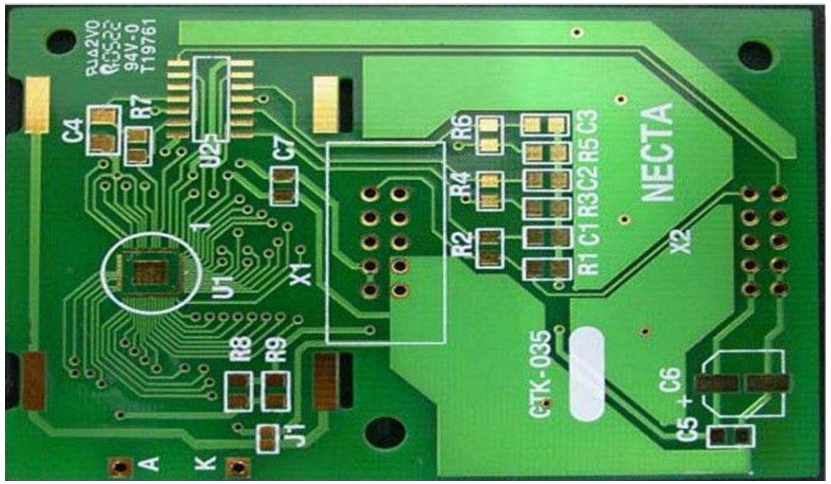
- When designing printed matter, we also choose a number of layers. The high number of layers generally increases production costs but also enables production of smaller and more resistant printed matters.
- **One-sided printed board:** It is suitable for simple low-frequency circuits, as seen in image 9. The circuit can be produced with many bridges. Such circuits have worse resistance to electromagnetic disturbances. Designing of complex circuits on one layer PCB requires much more effort and innovativeness. Usually, they are used for pilot devices and early component testing.



Funded by the Erasmus+ Programme of the European Union

Choosing printed matter

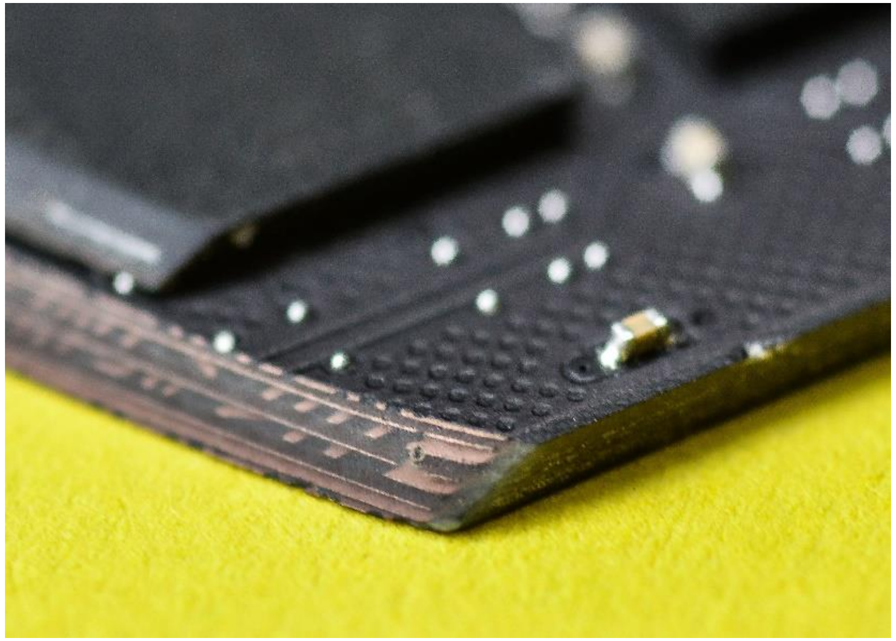
- **Two-sided printed board:** Often are made of material FR-4, as seen in image 10. The circuit is easier to be connected with. If possible, the bottom surface is intended for grounding, and other connections are left on the top surface. Advantages of grounding surface are increased mechanical stability of printed matter and lowered impedance of all grounding connections (reduces noise). It adds distributed capacitance to each connection on the top layer, which helps prevent electromagnetic disturbances. It acts as a shield from electromagnetic noise, the source of which can be printed matter environment.



Funded by the Erasmus+ Programme of the European Union

Choosing printed matter

- **Multi-sided printed matter:** There can be 4, 6, 8, 10 or up to 38 layers used. These are more suitable for sensitive high-frequency devices. The usual thickness of 2-sided printed matter is 1.5mm, which is too much. At a smaller distance between the upper and bottom layer, we achieve better distributed capacitance. It is also easier to connect the power supply and grounding connections (power supply and grounding layers). Connecting is easily done through VIAs. Other signal lines have a lot of space on all other layers, which significantly simplifies connecting. Higher capacitance distribution between the power supply and grounding layers which decreases high-frequency noise.



Funded by the Erasmus+ Programme of the European Union