Ecodesign of electronic devices

UNIT 11: Computer-aided design of electronic devices

Author: Simon Pevec.

11.1. Computer-aided design of electronic devices 1
11.2. Designing printed circuit boards 4
11.3. Examples of using Altium Designer program................................ 13

Chapter summary:

– CAD tools
– Using CAD tools for printed circuit board
– CAD tools for 3D modeling

Funded by the Erasmus+ Programme of the European Union

Ecoinnovation Skills for European Designers, Project number: 562573-EPP-1-2015-1-SI-EPPKA2-SSA. Disclaimer: The European Commission support for the production of this publication does not constitute endorsement of the contents which reflects the views only of the authors, and the Commission cannot be held responsi-ble for any use which may be made of the information contained therein

11.1. 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. Many tools already contain tools that help analyze and evaluate if the design is ecological. Some tools also contain datasockets with elements and materials that are part of the design. 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.

The first CAD tools have been developed for space and automotive industry. Latter, CAD tools have expanded to other engineer areas, such as electronic and textile industry, packaging, etc.

CAD tools have originally been intended for automatization of design processes and system modeling. Modern CAD tools support most activities in the design phase. They contain information on product characteristics and used materials. These tools also serve as a common platform for data exchange between different design groups and teams. They can contain information on production and are often called CAD/CAM tools (computer aided manufacturing). Image 1 presents the process of ordinary and CAM/CAD design. CAD tool shortens design time, which consequently means that the product will be on the market sooner and product development will be cheaper. CAD tools also save design process, which enables faster and easier upgrade of the initial version. Image 2 presents average time spent on product design and its use. The shorter design time is, the longer the usefulness of the device on the market. This condition is only valid if the design is done correctly.

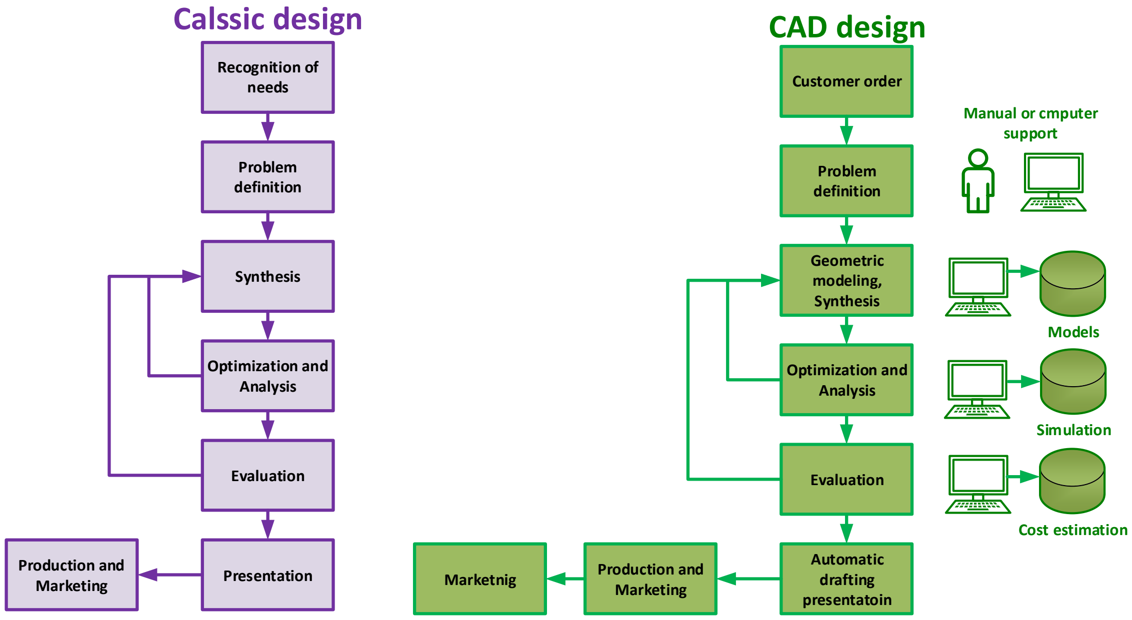


IMAGE 1: EXAMPLE OF CLASSIC AND CAD/CAM DESIGN.

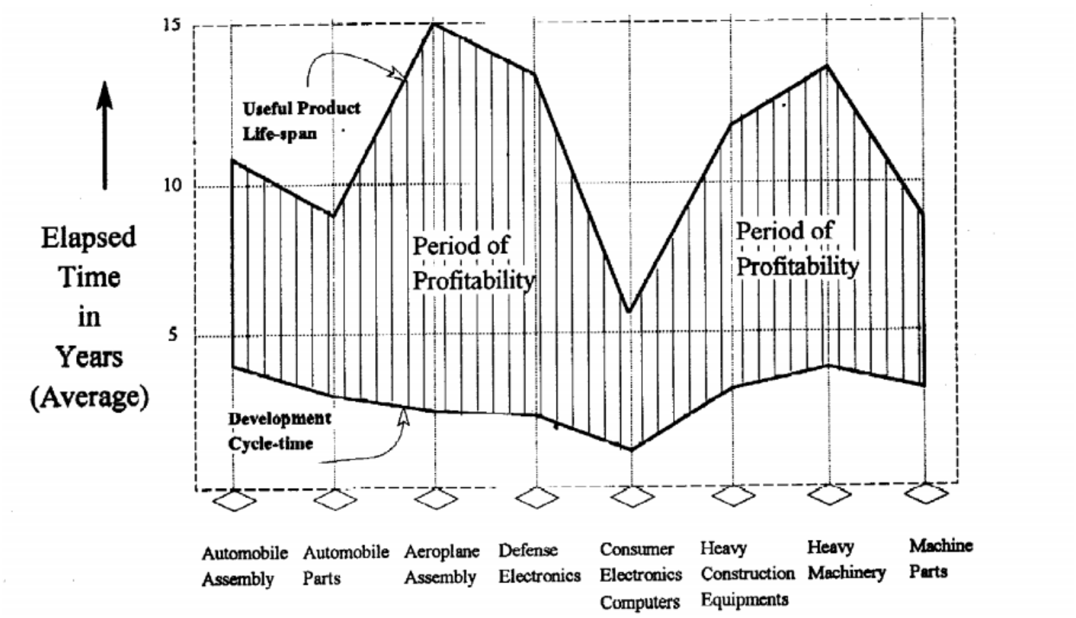


IMAGE 2: AVERAGE TIME OF DESIGN AND PRODUCT USE.

In the following chapter, we will present CAD tools that are intended for designing electronic devices.

11.2. Designing printed circuit boards

Designing printed circuit boards PCB is the key task of each development process. Well designed circuit and printed matter itself influence the quality, as well as device reliability. With design process, we determine all elements and the size of the circuit. When choosing components and printed matter size, we can meet many ecological guidelines. Usually, we use components that are smaller, less energy consuming and the design circuits that require less space. Today we know many advanced technologies that enable multi-layer printed matter. These enable smaller circuit surface and consequently lower material consumption. On the other hand, we need to know that production technology is much more expensive and uses more energy, which is less eco-friendly. When designing printed matter, we need to find a compromise between chosen elements and printed matter size that it will have a lower ecological impact in the production phase. Apart from these aspects, we need to consider certain guidelines and regulation on printed matter design with the intention of achieving high reliability and quality of printed matter and the final device.

The standards for printed matter design are monitored by IPC association. They manage printed matter production standardization and material use. The main document that covers design of printed circuit boards is document ICP-2221 - Generic Standard on Printed circuit board Design.

The standard steps in printed matter design are:

- Project specification.
- Electrical scheme design.
- Circuit design.
- Prototyping.
- Testing.
- Production.

Currently, there are many tools on the market that enable designing of printed circuits. Here are some of the most commonly used:

- **AutoTRAX** (Scheme, PCB design with built-in Spice simulator)
- **Advanced Design System** (Intended for RF electronics - mobile phones, WiFi network, satellite communication, radars, VF circuit - VF simulators)
- **Eagle** (Scheme, PCB design, available free version for smaller projects, academic environment, 3D view, etc.).
- **Altium designer** (Used to be Protel, schemes, PCB, support for FPGA – Field Programmable gate array, with the programming option, translation of program code, 3D view)
- **OrCAD tools** (Scheme, PCB)
- **CADSTAR** (Scheme, diagrams, PCB, available free version with basic functions CADSTAR Express)
- **KICAD** (open-code environment, scheme, PCB and 3D view)

11.2.1 Designing electronic schemes

When designing electronic schemes, it is important that the scheme is organized, relations are logical, and there is as little crossing as possible. A good scheme is also designed in a way that it is very similar to the final printed matter. For example, if we want to place a capacitor near a certain electronic component, then we also draw it near the symbol or component in the scheme. We stick to the unwritten rule that all inputs are on the left and all outputs on the right side of the scheme. If necessary, we also use comments and notes. In complex circuits, we define scheme areas for better visibility. All areas together make the complete device scheme. The areas are only virtual scheme sections that round up the parts of the circuits. For example, we draw the power supply part separately or on a certain area of the scheme. The controlling part is also designed on its own area, etc. Image 3 presents the classical scheme for printed circuit board design.

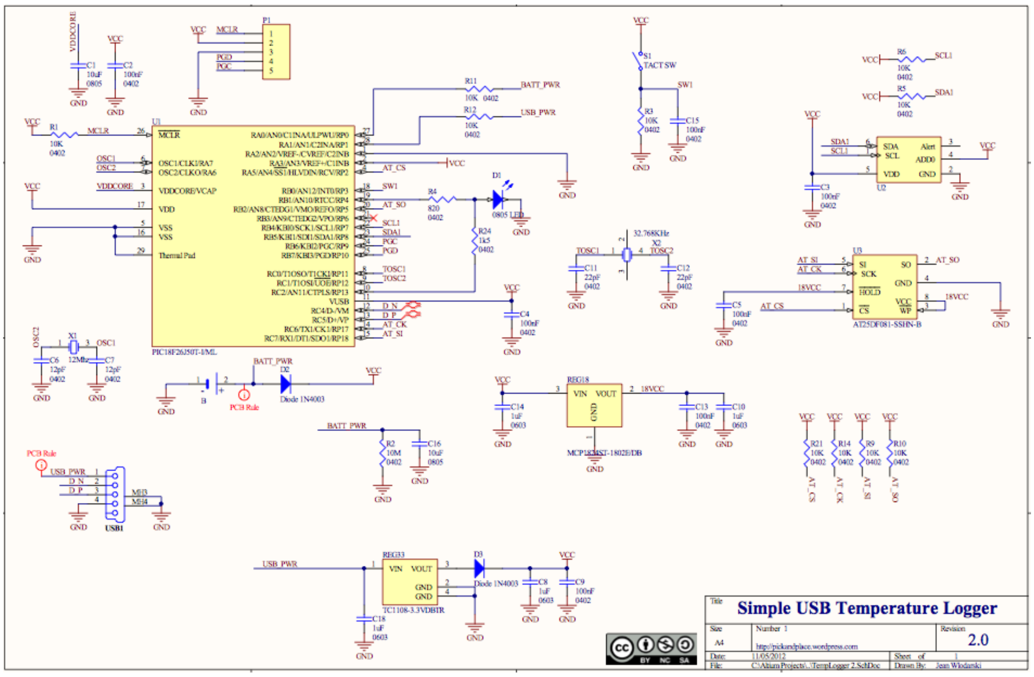


IMAGE 3: SCHEME FOR DESIGNING A PRINTED CIRCUIT BOARD.

11.2.2 Designing printed matter

Equally important is making the scheme for the next step in printed matter production. Designing printed matter consists of element arrangement and connections between them. With production of printed matter, we determine the final appearance of printed matter. In the production, it is very important that we use element libraries. In these, we can find characteristics and dimension of the element, as well as names of connected clamps. In printed matter production we need to consider the following rules:

- Printed matter choice (material, width).

- A number of printed matter layers.
- Layer arrangement.
- Circuit arrangement on printed matter.
- Consideration of parasitic impacts.
- The component layout on printed matter.

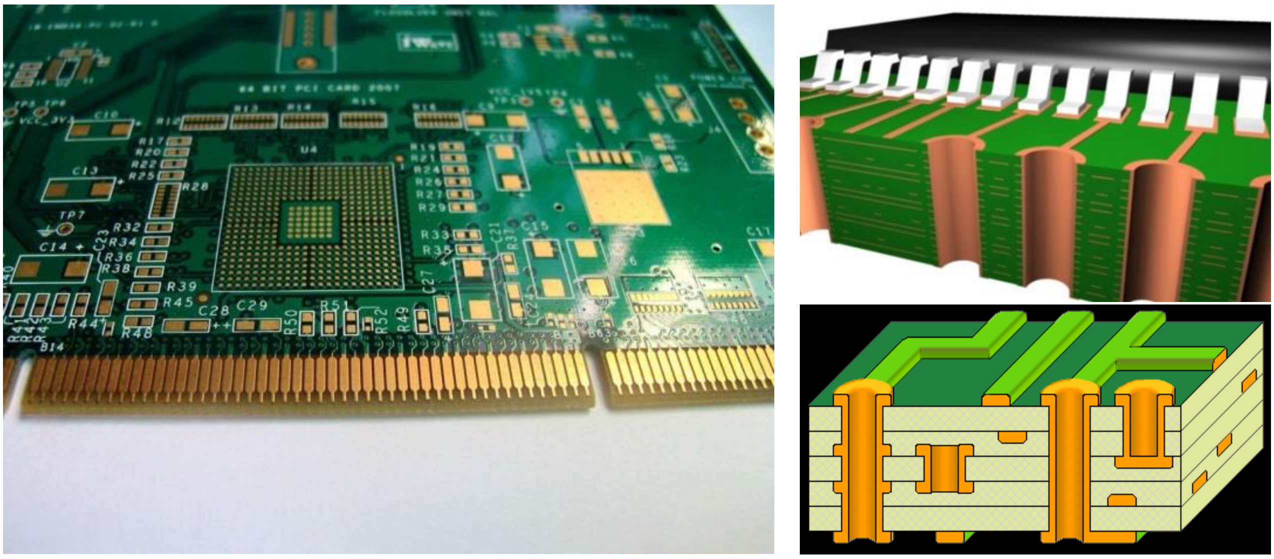


IMAGE 4: PRINTED MATTER - PCB.

In printed matter production, we often encounter different metric systems. Electronic components usually have dimensions and arrangement of connection legs presented in inch units. In Europe, metric units are used more often. Inch unit is more often used for dimensions of connections, copper connectors, and pads. Metric units are used for determining hole dimension, printed matter size and circuit housing dimensions.

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.

For example, offer 10/8 means that the lines cans can be at least 10 mil wide and the distance between them can be at least 8 mil. The typical offers are 10/10 or 8/8. Generally, printed matter with 12/12 can be produced by almost every manufacturer. IPC standard recommends the lowest limit up to 4/4. Smaller distances can mean the costs of producing printed matter are significantly higher. An example of good practice of using a wider line and its narrowing at places where it is necessary is presented in image 5. This way, we can retain lower total impedance.

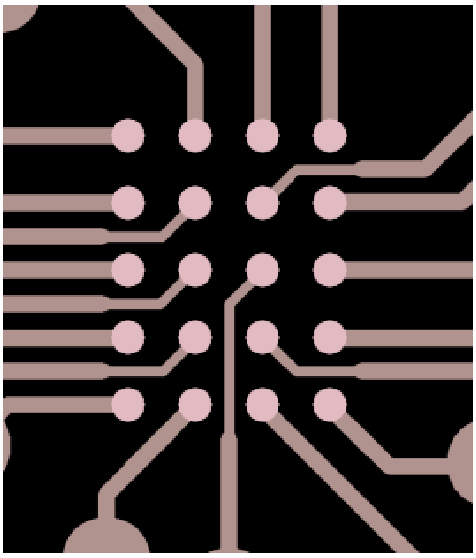


IMAGE 5: EXAMPLE OF NARROWING LINE .

Wire width and length is determined by electrical current and electrical signal frequency. For higher currents, it is recommended that we use as thick lines as possible. More narrow lines have higher resistance, which causes losses and unwanted conductor heating. Table 1 below presents recommended line width depending on the current at a temperature increase of 10˚C.

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

TABLE 1: COPPER LINE WIDTH IN PRINTED MATTER BY CURRENT.

Copper wire resistance is calculated with the formula:

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

Where 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:

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

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.

Vertical interconnect access

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.

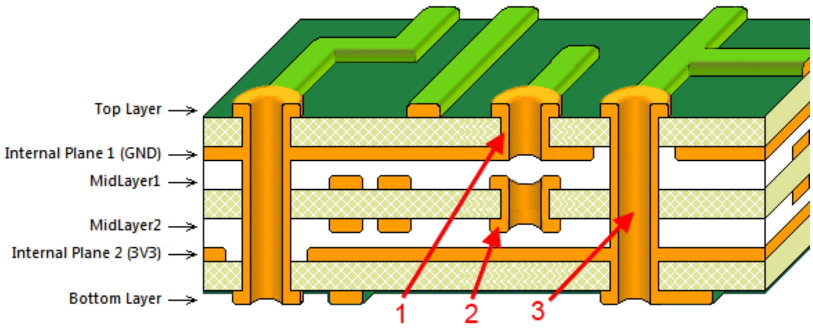
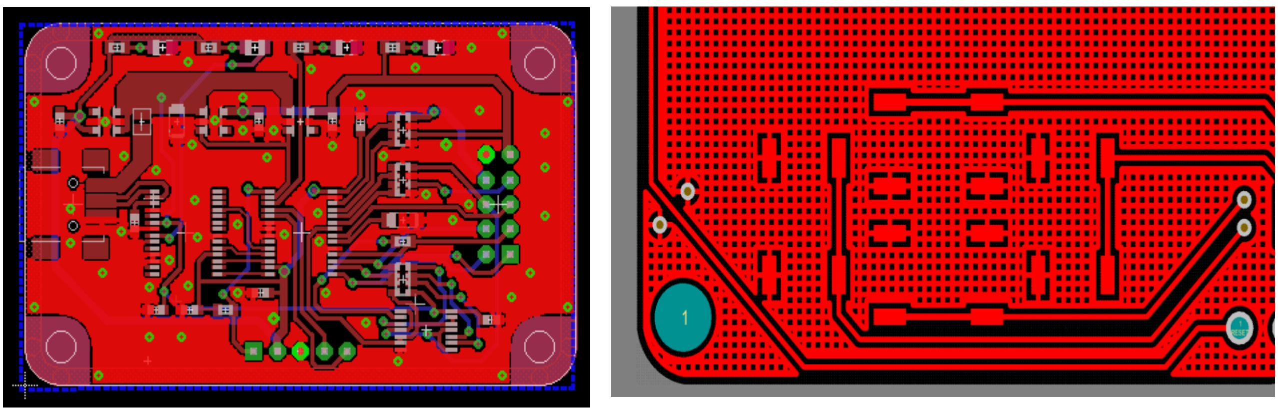


IMAGE 6: VIA IN MULTILAYER PRINTED MATTER.

Polygons

Polygons are used for filling larger areas with pure copper or copper texture, seen in image 7. 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.



a) *b)*

IMAGE 7:PRINTED MATTER POLYGON; A) TOP LAYER, B) COPPER TEXTURE.

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. It depends on the manufacturer and production technology. The common rule is 15 mil for wire components and 8 to 10 mil as the lower limit for elements that are montaged on the surface. For circuits that work at network voltage 230 V/110 V, we need to consider safety standards that are valid for a certain geographic area. The basic rule is that between phase and zero current has to be at least 3.2.mm of distance. The smallest distance between high-voltage parts and parts with which user can come in contact with is 8mm. A simple guideline is that the distances should be bigger rather than smaller when it is inside the granted dimensions!

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 2 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:

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

where E_p is electrical breakthrough strength, U_p is electric voltage and d is an isolant dimension. Different isolants have different electric breakthrough strengths, presented in table 3.

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

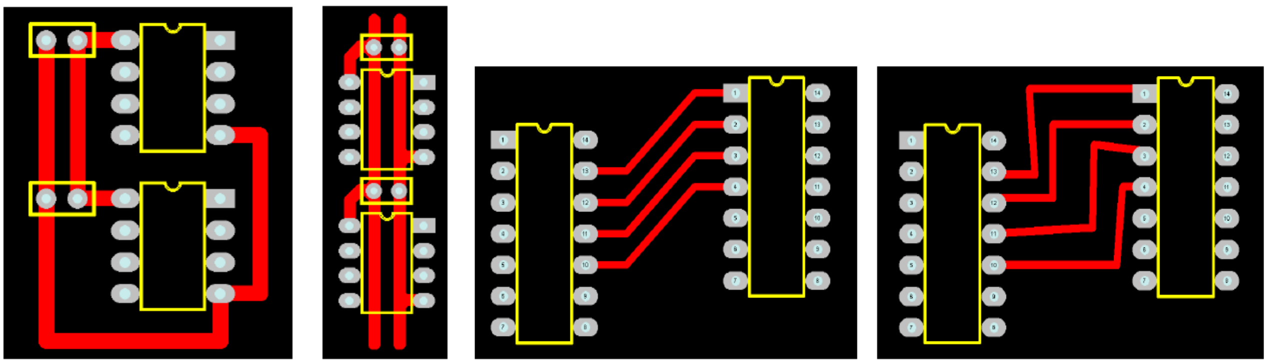
TABLE 2: PRESCRIBED DISTANCES FOR GALVANIC ISOLATION OF PRINTED MATTER LINES.

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

TABLE 3: ELECTRIC BREAKTHROUGH STRENGTH OF SOME MATERIALS.

Basic rules for connecting printed matter components

The connection of components means installation of copper lines on printed matter between component connectors. Electric connections between two or more connection pads are called electric signal network. We strive for shorter electric signal network because longer lines cause more serious parasitic effects. The lines should break at 45° angle, or we can use rounded lines. Copper lines are connected at the middle of connection pads, for which we use working network or function “snap to object”, which depends on program package. The connection between two points can consist of only one line. For higher currents is necessary to use larger VIAs, which decrease impedance and increase reliability. Between connection pads on distance 100mil, we decrease line width. Power supply and grounding lines should be wider for higher current loads. Power supply and grounding lines have to be set as closely together as possible, which can be efficiently blocked by capacitance elements. There should be no copper “islands” without connections. These islands have to be grounded or deleted. Image 8 presents different ways of connecting elements.



a) *b)* *c)*

IMAGE 8: PRINTED MATTER CONNECTIONS: A) APPROPRIATE, B) APPROPRIATE, C) NOT APPROPRIATE.

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. Table 4 presents an overview of materials for production of printed matter.

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 for cryogenic temperatures to over 260 °C.*

TABLE 4: TYPES OF MATERIALS FOR PRINTED CIRCUITS.

Label FR stands for flame retardant material. The thickness of the copper coating on printed matter is by standards 0.5oz(18 μm), 1oz(35 μm) or 2oz(70 μm). There are also other standards that are less often used (12 μm) and (105 μm). The printed matters with aluminum or metal core have a copper coating of even 70μm to 400μm. The most often used material in industrial environments is FR-4. This material is the optimal choice regarding price and causality.

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.
- **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.

- **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. It also better blocks EMI/RFI disturbances. Production of multi-layer printed matters is significantly more expensive in comparison to those circuits with fewer layers.

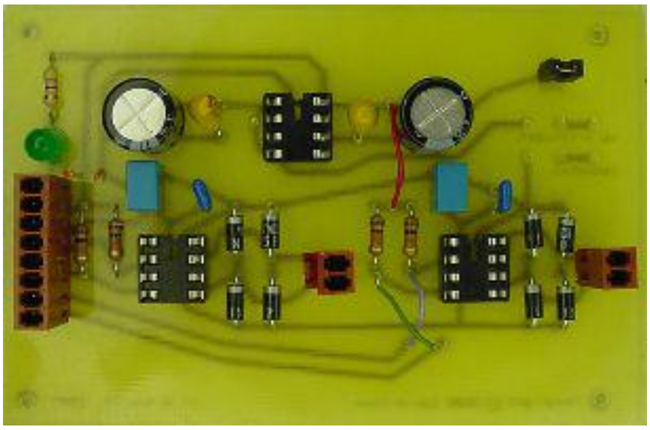


IMAGE 9: ONE-SIDED PRINTED MATTER.

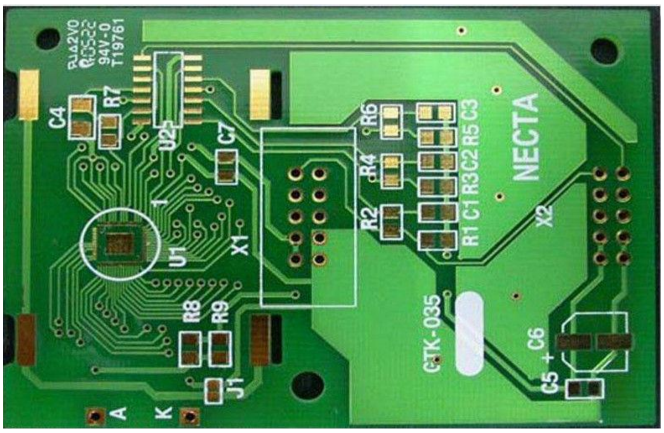


IMAGE 10: TWO-SIDED PRINTED MATTER.

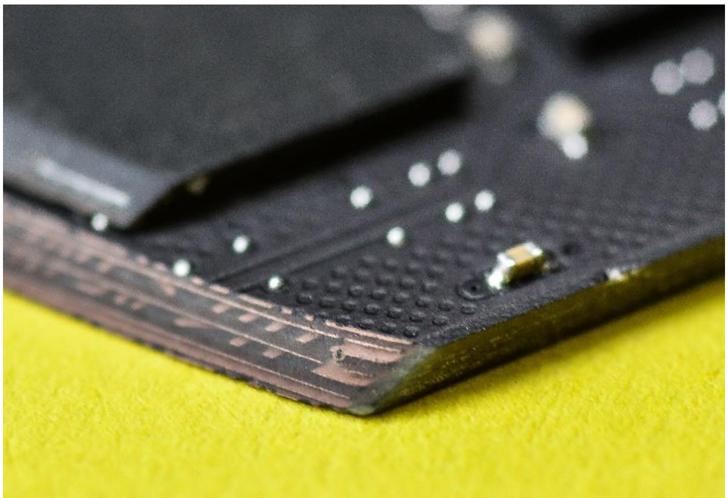


IMAGE 11: MULTI-LAYER PRINTED MATTER.

11.3. Examples of using Altium Designer program

Program Altium-Designer is a professional environment for designing printed circuits in all phases, which are:

- Designing of flowcharts and block diagrams.
- Designing of printed circuit boards-PCB.
- Designing software for FPGA (Field Programmable Gate Array).
- System solutions for FPGA and debugging (when working with suitable development boards, such as Altium NanoBoard).
- Designing built-in systems.
- Simulation tools for digital and analog circuits.
- Signal quality analyses.
- Management of PCB production processes.

Altium includes editors and program interfaces for all steps of electronic device design. Writing and editing of program code together with translating is done inside Altium Designer environment. In our case, we will pay more attention to drawing and designing of the printed circuit board for an electronic device. Content is designed in a way that offers an overview of different PCB design phases. The first design phase includes an outline of the electrical scheme. The second phase includes translation of the scheme into printed circuit. In this phase, we determine printed matter dimensions, element arrangement, and connections. In the third phase, we will show how to present the printed matter in a 3D environment and the possibilities for using it in other CAD programs. For this case, we will take a look at current generator circuit, seen in image 12.

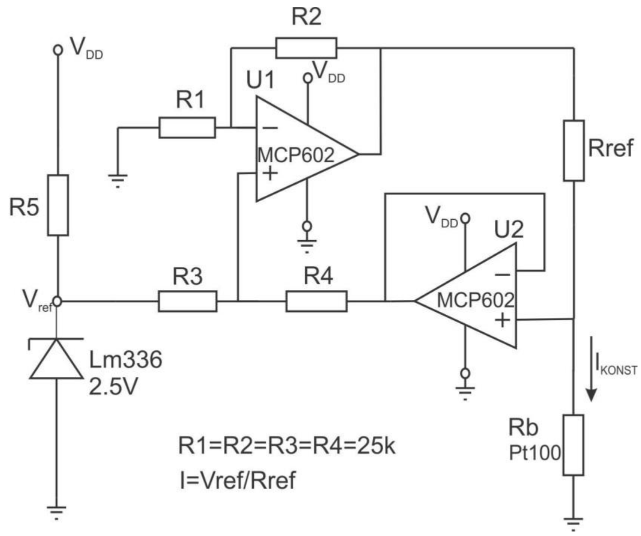


IMAGE 12: CURRENT GENERATOR SCHEME.

11.3.1 Creating a new project

For each new printed matter, it is recommended to create a new project because in a project we have saved all document and settings that are related to the design process. Project file xxx.PrjPCB is ASCII file in which are written all output documents and settings, such as printing settings and CAM. Documents that are not related to the project are called free documents. We also add links to electrical schemes, PCB, FPGA, built-in VHDL (Verilog hardware description language) and libraries. When the project is completed, all designs are synchronized inside the documents in the project. The project is created with the following set of commands:

- Choose **File → New → Project → PCB Project,** image 13.

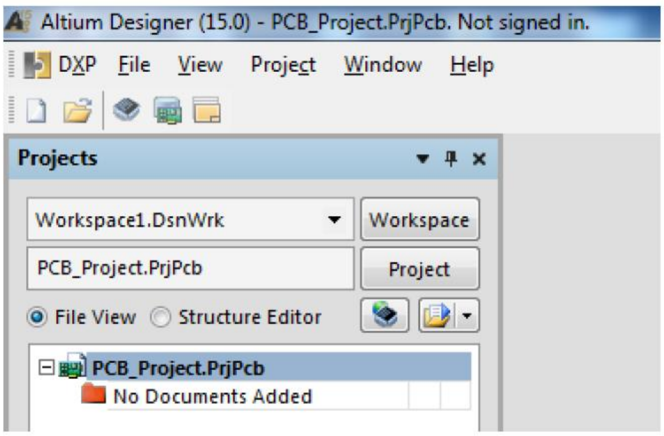


IMAGE 13: CREATING A NEW PROJECT.

- We can see window **Projects**, where project **PCB_Projects.PrjPcb** is listed without attached documents.

- Rename the project, in our case to **Current_ generator.PrjPcb**, so it can be saved on disc location of our choice with the next command. **File → Save Project As → Current_generator.PrjPcb.**

Then create a new file for designing electrical scheme 'schematic'. Create new scheme.

- Choose **File → New → Schematic**. We can see file **Sheet1.SchDoc** that is added to the previously created project automatically, seen in image 14.
- Rename scheme to **Current_generator.SchDoc**. Save the scheme, **File → Save As → Current_generator.SchDoc**.

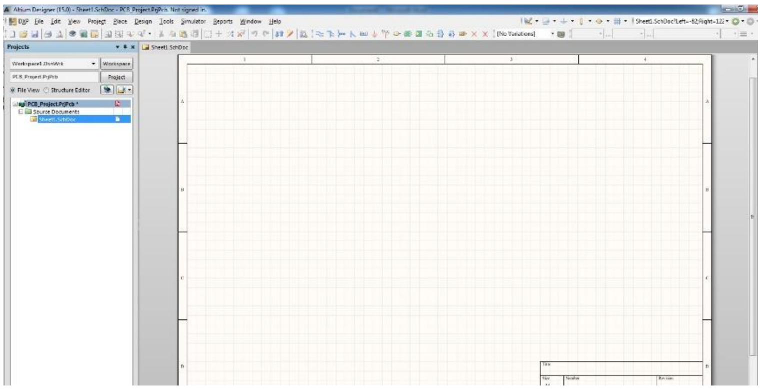


IMAGE 14: CREATE CIRCUIT SCHEME.

When an empty scheme is opened, we can see a completely changed view with different keys, menus, etc. Now we are in the editor for electrical schemes. If we want to add already outlined scheme of another project to the created scheme, then choose **Add Existing to Project**.

Before we start designing electrical scheme, we need to set the following:

- Setting format for worksheet: **Design → Document**, Option – choose format A4.
- The worksheet can be enlarged with a combination of CTRL key + scroll wheel; view can also be fit to full screen with command **View → Fit Document** [shortcut: V, D].
- There are many settings that can be accessed from the menu **Tools → Schematic Preferences** [shortcut: T, P], where the settings will influence all schemes in the working project.
- Click on the scheme and select **Default Primitives** and enable option **Permanent**, which offers basic preset values when we select an electrical element from the library and not the settings that we set in the previous step.

Scheme design begins with choosing an element, seen in image 12. In the library of elements, we search for all elements, starting with operational amplifier MCP602. Altium has very strong support for libraries of electrical components, such as schemes, sockets and 3D models. These libraries are not yet installed, but we can download them free of charge from Altium website (https://designcontent.live.altium.com/.) if we have a valid license. We can also download freely accessible libraries from 2004 and extend the folder where we have already installed basic library by Altium.

To search for correct amplifier MCP602, click **Libraries** and then **Search** (or click **Tools → Find Components**) and then opens dialogue window **Libraries Search**.

We need to be careful that we have all components selected and that we search in mode **Libraries on path** where we have to have written the correct path for installed libraries. For the higher possibility of results when searching do not enter all signs, but only the main ones because different manufacturers have different prefixes and suffixes. To do this, enter the search string between two asterisks (in our case: *602*). If we choose a component from the list that is not yet installed, then we get the option to confirm that we can install it immediately which can later be executed. For MCP602 we need to previously install library by manufacturer Microchip (Microchip Linear Devices.IntLib), and then MCP602 is available, so we can select it as seen in image 15.

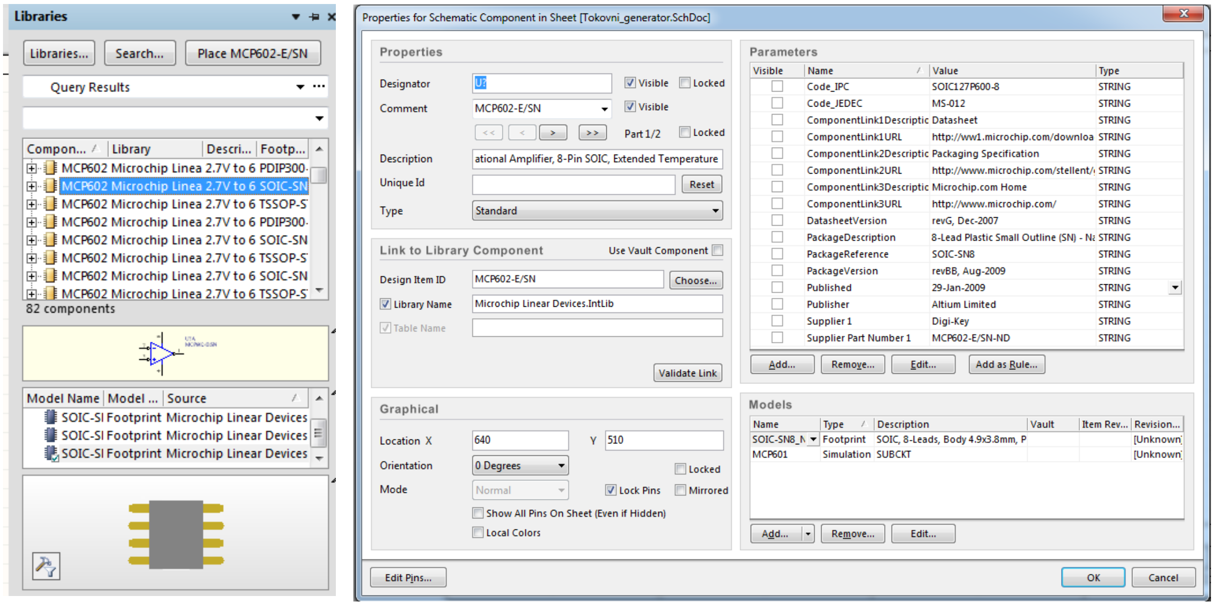


IMAGE 15: CHOOSING MCP602 FROM THE LIBRARY.

To insert, double-click the component which enables moving of the element into the working area with the cursor. Before we move it to a certain place, we can edit labels and properties which can be enabled by pressing the tab key. Then in the field Designator enter a component label that we will see in the scheme (in our case: U1A). At the model check, if the component has a correct socket (in our case: SMD SOIC – 8 connectors). When we have set everything needed, we can start with inserting components into the scheme. The component can be set to the wanted position by clicking or pressing ENTER. Then move the cursor and see that with the next click we can insert another OPA from which the label U1A is automatically increased to U1B, and other preset properties remain unchanged. Altium environment enables multiple insertions of components of the same type which is revoked by key ESC.

In the next step, we will insert 7 resistors. For inserting, we will use a basic library that is installed in Altium environment under name **Miscellaneous Devices.IntLib**. Type ***Res*** in the search engine and then select resistor with socket SMD type 2010 that is labeled as **Res3**. Select value 25k that is the most common value and can be later changed at certain resistors. Reset Designator to the lowest value R1 and start with entering values, as seen in the previous example. Then rename two resistors to Rref and Rb by clicking on each one separately and change the name in field Designator, as seen in image 16.

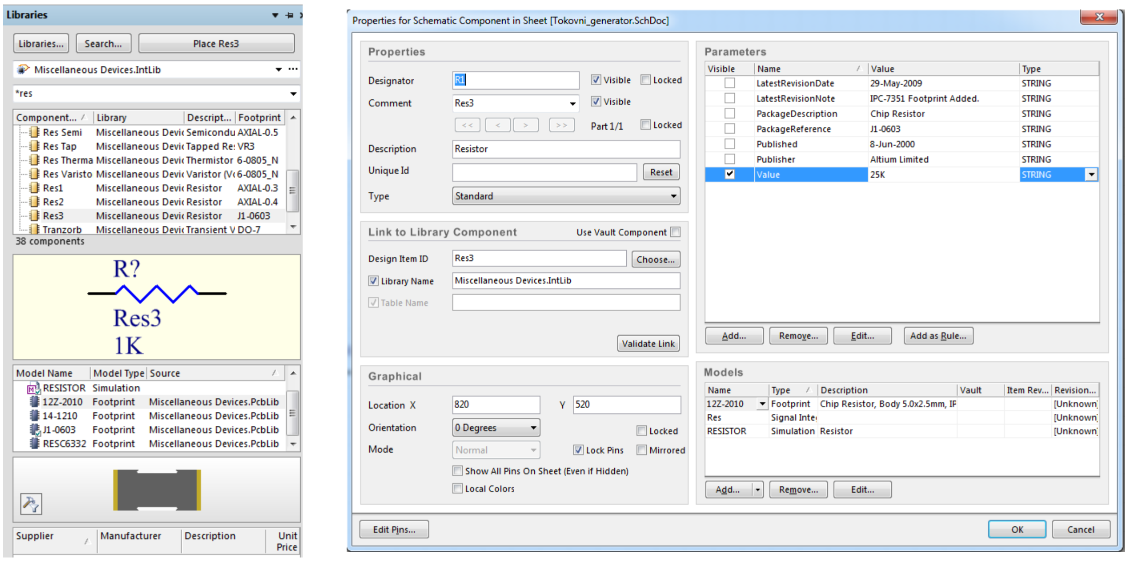


IMAGE 16: CHOOSING RESISTOR FROM THE LIBRARY.

Do the same with elements voltage stabilizer LM336 (2.5V, housing TO92) from library ST-Electronics '**Power Mgt Voltage Reference.IntLib**', capacitors (search string ***Cap***) and connector terminals (search string ***Header***) that can be found in library **Miscellaneous Devices.IntLib**, seen in image 17.

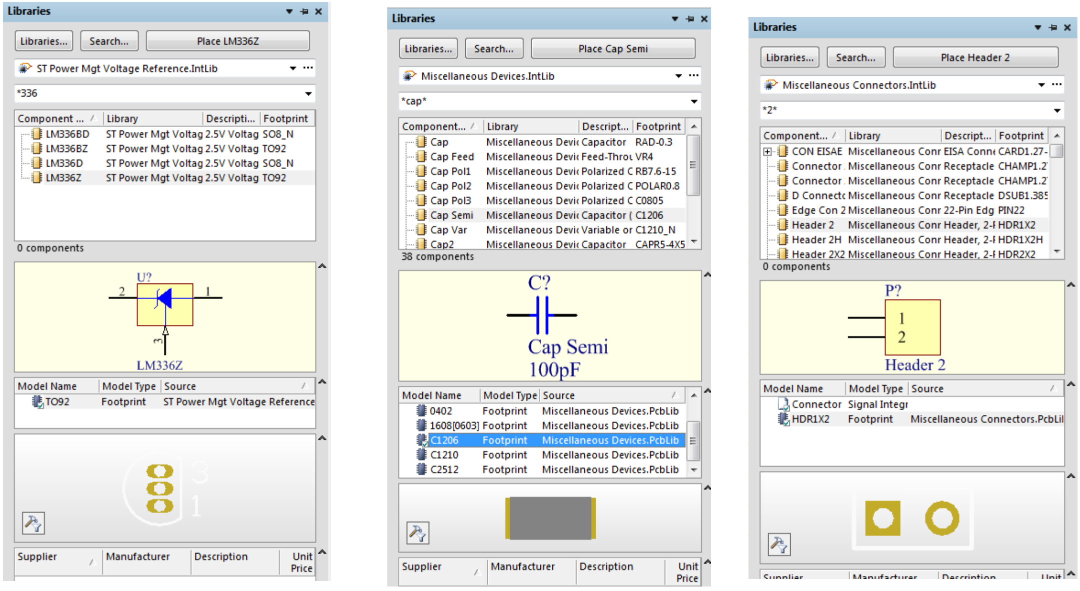


IMAGE 17: CHOOSING LM336, CAPACITOR AND CONNECTION TERMINALS.

Next, arrange the components correctly in the scheme while considering the rule inputs on the left and outputs on the right site. We need to be careful that there is enough space between the components for connections because if we connect electrical connection over connection pins, then the program will automatically connect it with the crossed connector, image 18.

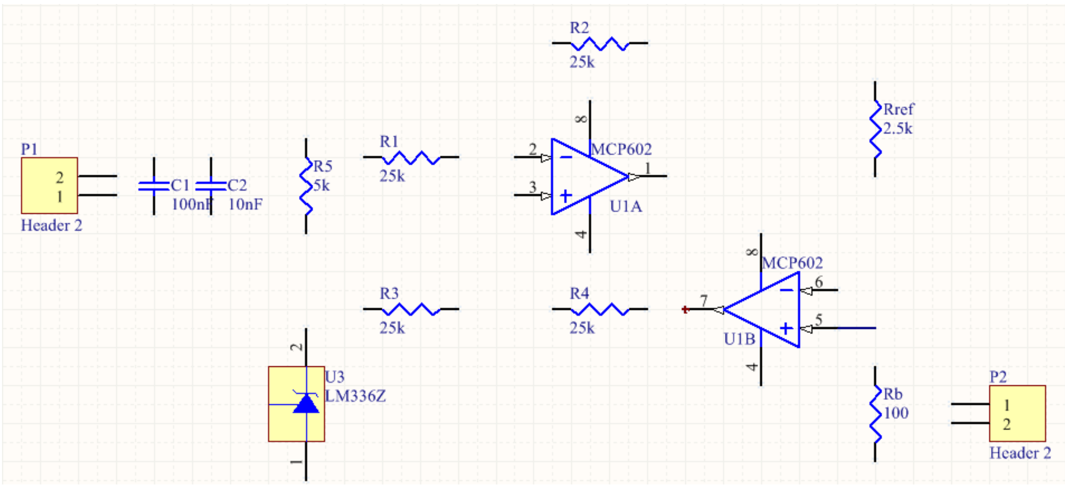


IMAGE 18: ELEMENT ARRANGEMENT.

To the existing components, we also need to add voltage potentials, such as zero-grounding and power supply . For test purposes, we will insert GND and power supply Vdd (insert them at several places to retain better scheme visibility). When connecting components, we need to consider the following rules:

- We need to have a good overview of the complete scheme.

- Draw connections with the tool **Place Wire** [shortcut: P,W]. Pay attention to interconnections to prevent short circuits.
- If we want to move components after we have connected them, we need to delete current connections, move the components and then reconnect them again. We can also use combination CTRL + moves with the mouse, which enables moving of components together with previously made connections.

The finished wiring diagram can be seen in image 19.

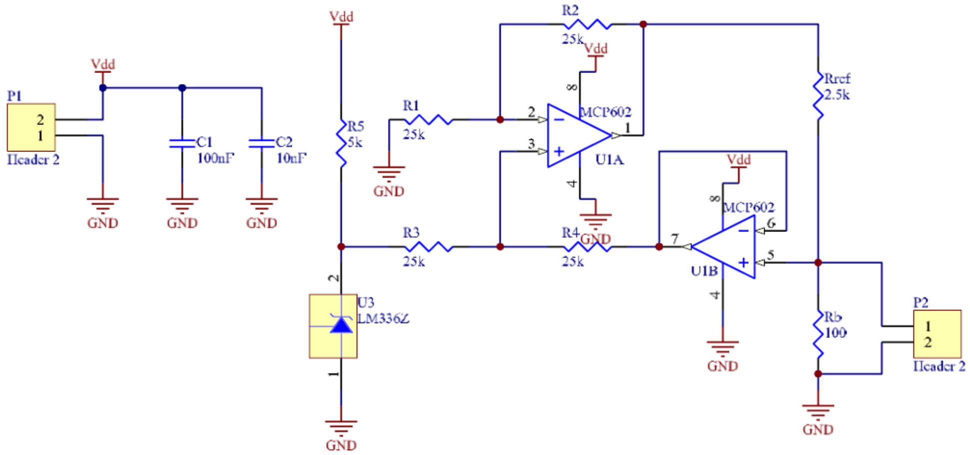


IMAGE 19: CURRENT GENERATOR SCHEME.

We also need to add to the scheme names of individual connections (nets and net labels) that can be later used in printed matter. Connection names are not essential for circuit functioning but serve as notes or references when reviewing finalized circuit or for later repairs. Component connectors are connected with nets. Some connections with higher importance and potentials in the electrical scheme can be named with the name of our choice. By using symbols for GND and Vdd we have automatically set two names. When choosing Menu → Place → Net label [shortcut: P, N] we get the option to set name to any connection. By pressing TAB, we can change settings, such as color, arrangement and font type for net label before we define to which connection we will set the new name, as seen in image 20.

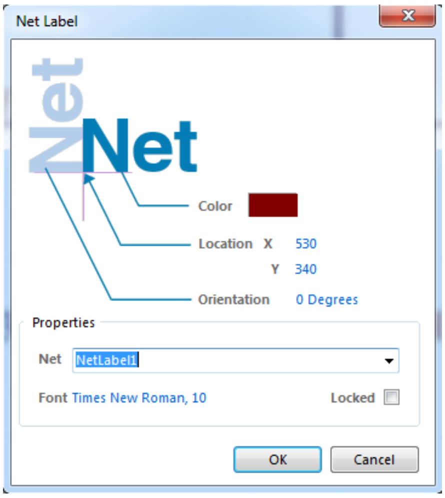


IMAGE 20: WINDOW FOR SETTING CONNECTION NAME, FONT TYPE, COLOR, SIZE, ETC.

In our circuit, we will use connection name for reference voltage Vref shown in image 21.

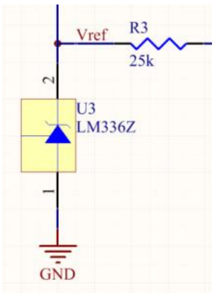


IMAGE 21: DEFINED NAME FOR CONNECTION V_{REF}.

With this, we have finished drawing and designing electrical scheme, which is the basis for moving onto designing the printed circuit board. Before we start translating the scheme into printed matter, it is recommended to configure project settings. This includes settings for detecting errors, connection matrix, class generator, settings for the comparator, generating warning messages, settings for printing, etc. Altium uses all these settings when translating a project. When the project is translated, the complete design and electrical rules that were set are checked. After all errors are solved, the scheme is again translated and uploaded into the target project with for example PCB document. Project comparator enables searching for differences between the source and target files that we see and can be confirmed or rejected (synchronized) from both directions. All settings are available under menu **Project → Project options**.

11.3.2 Determining printed matter

After wiring diagram is finished and the settings are correct, we can start translating wiring scheme into printed matter. In the transaction phase, all electronic scheme design rules are checked with possible error tracking and the repair option. To translate given project **Current_generator.PrjPcb** we need to choose **Project → Compile PCB project**. When translation is finished, the translator message is visible in the message window. To access this window, we need to go to **View → Workspace Panels → System → Messages**. Translated documents will be recorded in window **View → Workspace Panels → Design Compiler → Navigator**, where we can monitor document structure, list of components and connection types (when clicking the connection we can also see it with marked lines in the scheme). In case of errors and unsuccessful translation, these events are recorded in window **Error Messages**. If the translation is successful, we can move to a new document in which we can design printed matter with the given electrical scheme. First, we create empty PCB. The easiest way is by using interface **PCB Board Wizard** with which we can choose between standardized industrial formats or own dimensions. Later these settings can be changed. **PCB Board Wizard** can be found under menu **Files**, which needs to be enabled in order to be visible, as seen in image 22. If file choosing is not enabled, then we have a row of shortcuts to certain function at the right bottom of the scheme, and under tab **System,** we can tick **Files**.

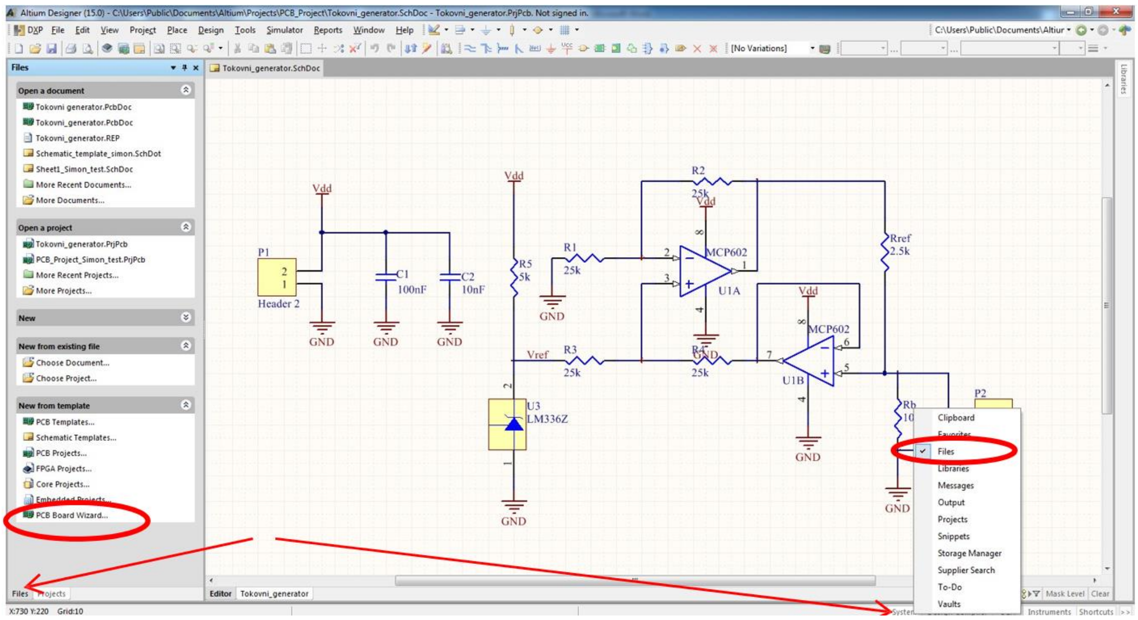


IMAGE 22: CHOOSING SHORTCUT TO FILES.

Document preparation follows these steps:

1. First click **PCB Board Wizard** and in the second windows select metric units **Metric**, image 23.

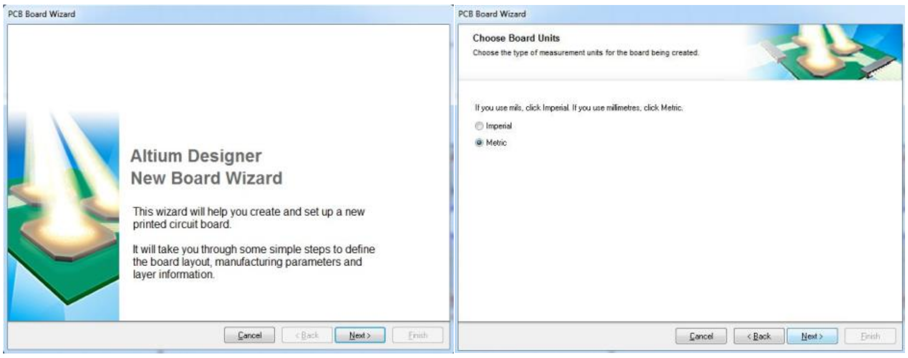
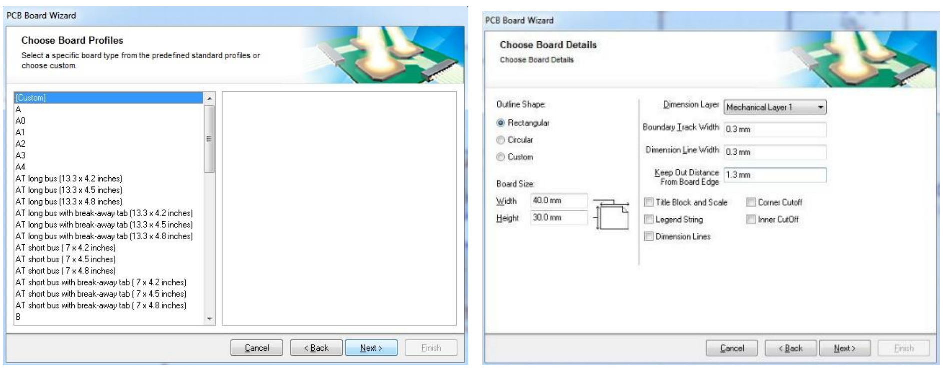


IMAGE 23: WIZARD FOR CREATING PCB.

2. In the third window choose between PCB formats. We can choose **Custom** and then enter dimensions W=40 mm, H=30 mm. For designation of PCB edge, select line type **Mechanical Layer1.**
3. In the next two windows choose a number of layers and type of VIAs that go through the whole PCB thickness.
4. Then select which component type we will mostly install on PCB (SMD, PDIP, etc., depending on the socket) and on how many sides. Most components are surface-mount and are installed only on one side.
5. In the following window select recommended settings for connections and VIAs:
    a) The smallest connection thickness = 0,25 mm
    b) The smallest VIA width = 1 mm
    c) The smallest VIA diameter = 0,6 mm
    d) The smallest distance between connections = 0,25 mm

Image 24 presents two wizard windows for printed matter settings.



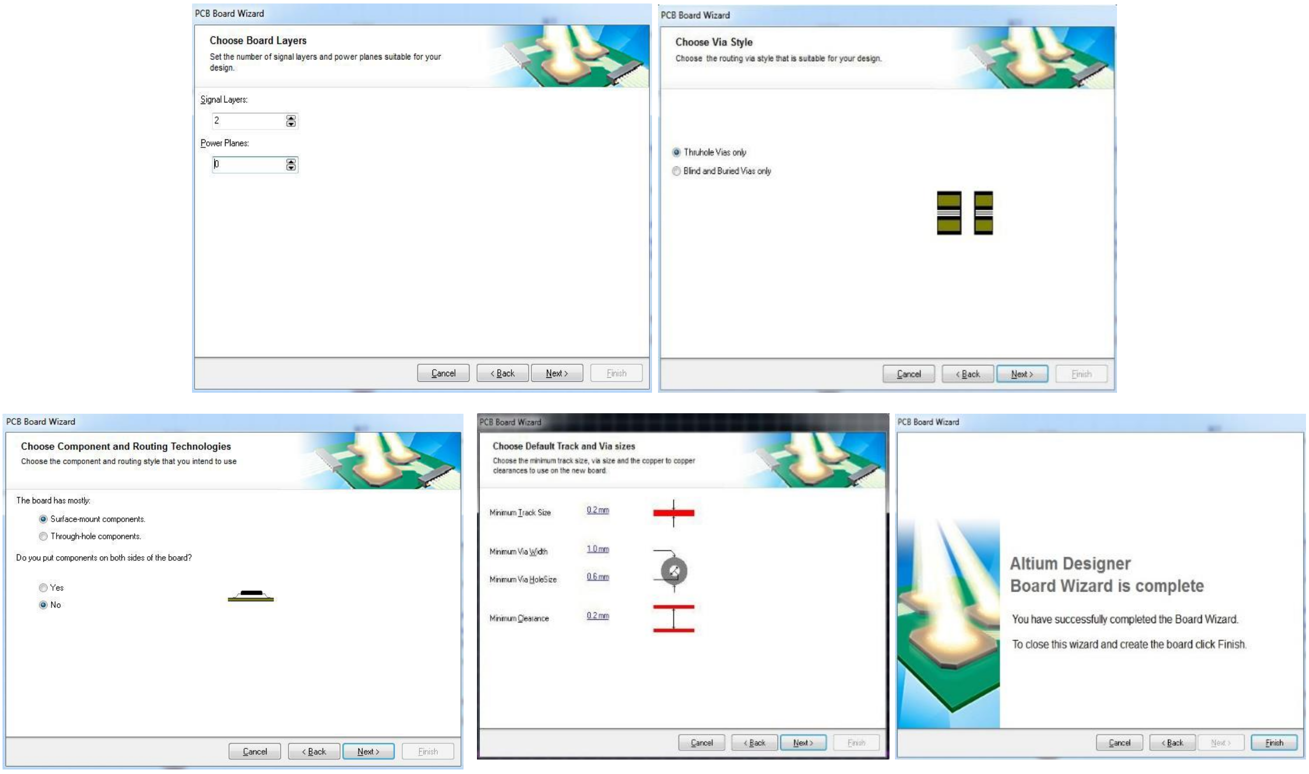
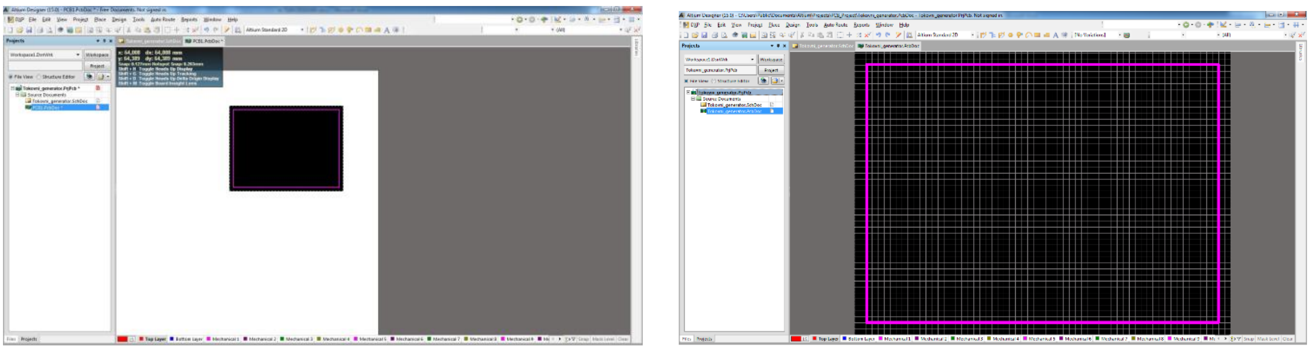


IMAGE 24: STEPS IN PRINTED MATTER SETTINGS FROM 2 TO 5.

After successful creation of printed matter, we see a window with selected dimensions. In order not to have white page background, remove it in settings by clicking **Design → Board options → Display sheet**. PCB view can be adjusted with command **View → Fit** board or shortcut [V,F], seen in image 25.



a) b)

IMAGE 25: ADJUSTED VIEW OF PRINTED MATTER A) WITH BACKGROUND, B) WITHOUT BACKGROUND.

PCB dimensions can be later changed. The easiest way is to draw desired frame shape which can be used for determining PCB borders. This can be done by clicking all lines that define PCB edges with mouse and then select the function from menu **Design → Board Shape → Define from selected objects**, to trim PCB to the desired shape, image 26.

IMAGE 26: CUSTOM PCB SHAPE.

11.3.3 Project translation - Compiling

First go into electrical scheme environment and translate **Design → Update PCB**. Click **Validate Changes**, where error probability is checked. If everything is correct, there will be confirmation checkmarks under status. If the changes are not correct, we need to return back, check error messages and solve the errors. When the error check is successful, click **Execute Changes**, which transfers components and connections to PCB in the project, image 27.

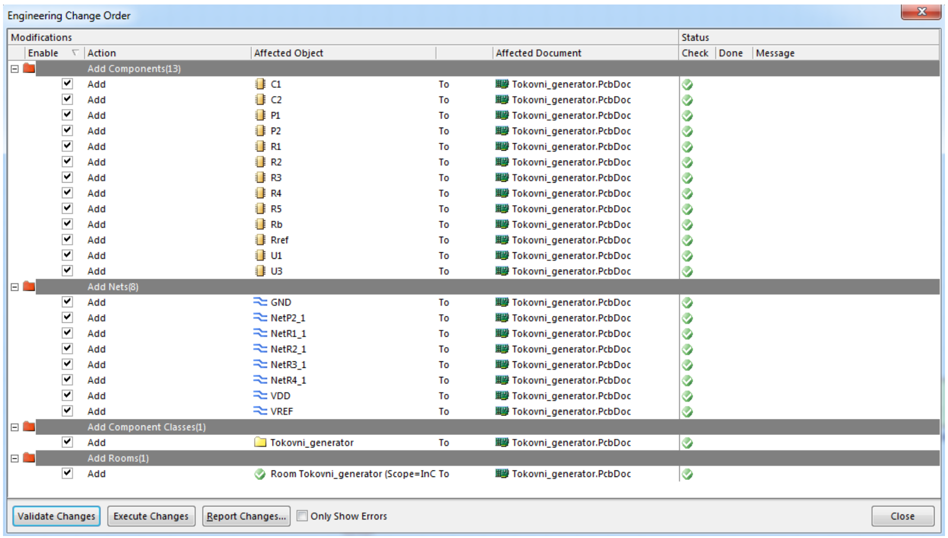


IMAGE 27: CHECKING SCHEME DURING COMPILING.

Transfer of components to PCB printed matter, image 28.

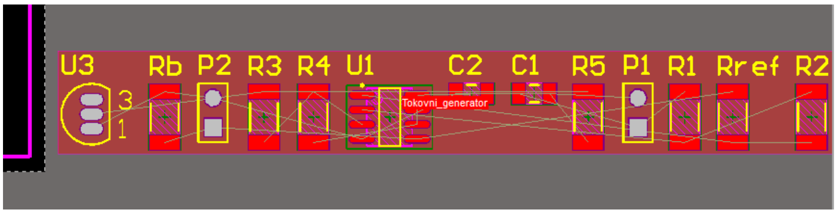


IMAGE 28: SHAPE OF ELEMENTS FOR PRINTED MATTER.

From image 28 we can see that arrangement of elements to printed matter is up to the designer. In this case, it is sensible to consider guidelines and instructions of good printed matter design. In the next step follows the arrangement of components to printed matter. Components are arranged by clicking on the component and moving it to the correct position (click with the left button and press it while moving the component). Stay within PCB limits. With spacebar we can rotate the component while transferring it. Text that is connected to the components can also be moved the same way as components.

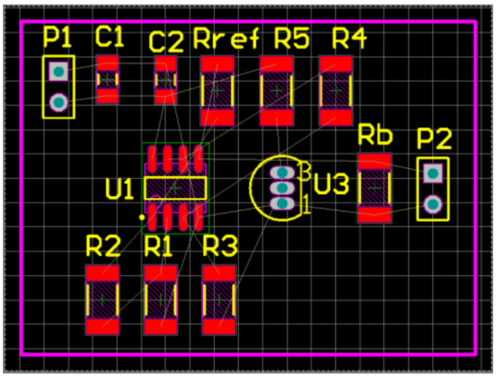


IMAGE 29: ARRANGED COMPONENTS ON PRINTED MATTER.

Altium environment has function support for the easier component arrangement. Several components can be aligned horizontally, vertically, by different edges or center line. They can also be aligned with uniform spacing. These functions can be accessed if we select component group, where we use key SHIFT or round around the components we want to select. Then click with right mouse button and choose **Align → Align** [shortcut: A, A] and mark wanted functions on X and Y axle. Also, when we select one or multiple components, we can move them with mouse or use key combination: CTRL + SHIFT and arrow keys. This means a combination of an arrow key and CTRL causes smaller move (snap grid x 1) in wanted direction than when we press both keys CTRL + SHIFT, which causes the components to be moved for larger distance (snap grid x 10).

The connection of arranged elements can be made manually or automatically. In manual connection, we have a better overview of connections and the work takes longer to be done. Manual connecting is most often used in larger printed matters. Automatical connecting is generally more useful in less complex circuits.

Manual connecting of PCB printed matter

Manual PCB connecting means establishment of connections and VIAs on printed matter in order to connect all components as presented by the electric scheme. In manual mode, we usually design more complex printed matters, where many design aspects are important, such as EMI influence, RF circuit design, combined analog and digital circuits, combined power electronics, and digital/analog circuits, etc. Many times the designer has to decide and take compromises between certain design parts.

In our case, we will do manual connecting, although the circuit is simple. We will try to draw signal connection on the top layer, for lower layer we will use GND mass, which we will access through VIAs and in certain cases with connection pads (connectors P1 and P2 and reference U3). The connections consist of sequential segments. At each change starts a new connection segment. Generally, Altium enables arrangement of connections in vertical, horizontal arrangement and at 45° angle (at advanced settings we can also use curves) to get a final professional result.

We will use standard settings. First, we will choose view **Top layer**. Use shortcut L to get to the menu **View configuration**, where we remove the checkmark at Bottom layer and confirm the change, seen in image 30.

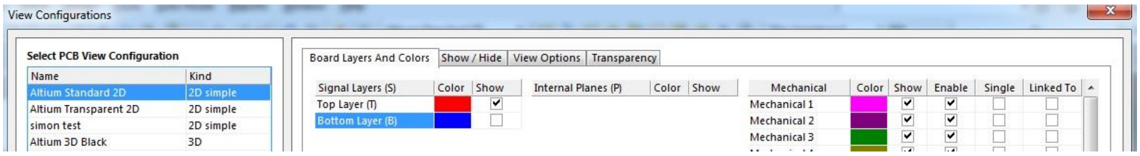


IMAGE 30: CHOOSING PRINTED MATTER TOP LAYER .

Connecting start with . We can only connect those components or connectors that are interconnected electrically in the scheme, which is in PCB environment presented by thin straight lines between connection pads. By pressing key CTRL and clicking certain connection pad, we can easily check which pads are interconnected. When we create a connection between two connectors, we can finish it with automatic function **Auto-complete**. This can be done by clicking the first connector and then the second one and during that press CTRL and arrow key >. The connection will be created in the shortest way between both connectors. This function is successful if the path is fairly simple without major obstacles between both points. The second way is by using key ENTER when connections are finished gradually, with which we decrease the number of segments in one connection. Altium enables other ways of manual connecting where we can press SHIFT + R during moving and select between different methods:

- **Push –** In this mode, the connection will try to break through to the other point by moving already established connections and VIAs within the rules.
- **Walkaround –** Path to the second point is trying to find by drawing the connection around obstacles without moving existing connections and VIAs.

- **Hug & Push** – This way is a combination of previous ones.
- **Ignore** – This method allows establishment of connections anywhere without breaking the rules.

We need to be also aware of the components that we set on the top surface, where are also signal connections and those that have connection pads routed though PCB to the other side. Such components are soldered below, so we need to connect them with signal connections on the top side with VIAs. VIAs can be found with key Place via . Because we will use the bottom layer for mass, VIAs will be installed at connectors P1, P2, and the reference.

Apart from components and VIAs, it is also good to predict how PCB will be attached to the housing because we usually need screw holes. Those can be created by selecting **Place Pad** . Do not connect it to any electrical connection. Mark option Plated, so there will be no error messages later on when checking PCB.

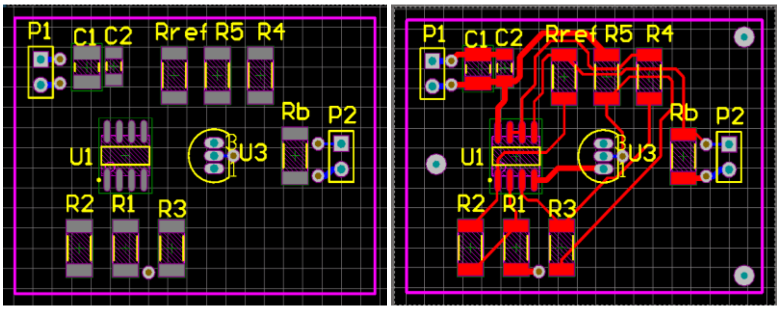


IMAGE 31: CONNECTED PRINTED MATTER.

Now we need to fill the bottom side of printed matter with the mass GND. This can be achieved by clicking button **Place Polygon Plane** from the toolbox, set its characteristics and then define polygon limits by drawing them in PCB, as seen in image 32 and 33.

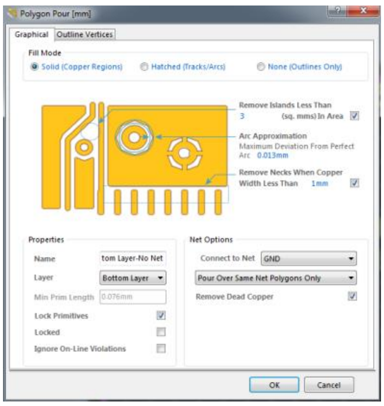


IMAGE 32: DETERMINING POLYGON FOR MASS ON THE BOTTOM PRINTED MATTER SIDE.

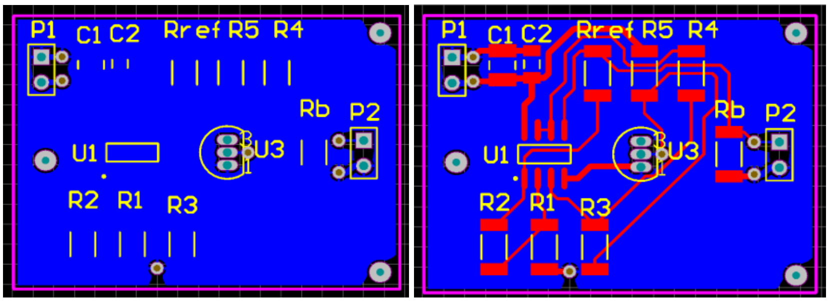


IMAGE 33: PRINTED MATTER WIT ADDED MASS GND ON THE BOTTOM SIDE.

Automatic routing

Simpler and less complex PCBs can be connected with an automated regime where all rules are strictly met. In our case, we want a signal line on the bottom side and the bottom side of PCB to be reserved for GND polygon. For this, we need to adjust the settings, so connections will be only on the top side. This can be done in rules. **Design → Rules** and under **Routing Layers** we enable only **Top Layer**, image 34. Then select from the menu **Auto Route → All** and click **Route All**.

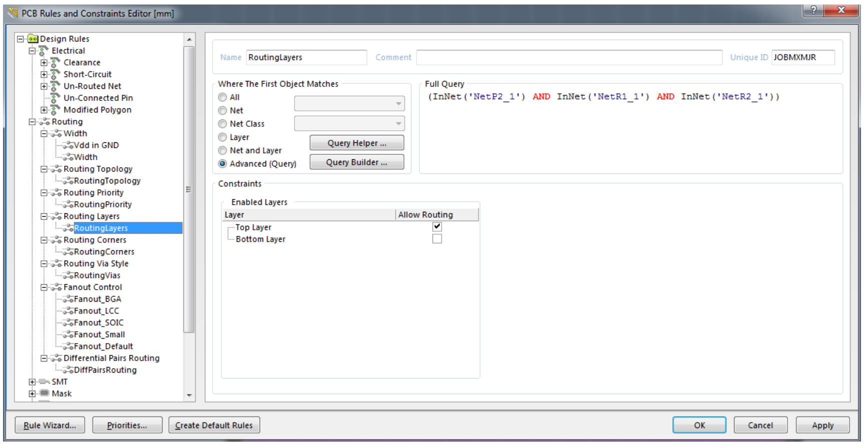


IMAGE 34: CHOOSING UPPER LAYER FOR AUTOMATIC CONNECTION BETWEEN COMPONENTS.

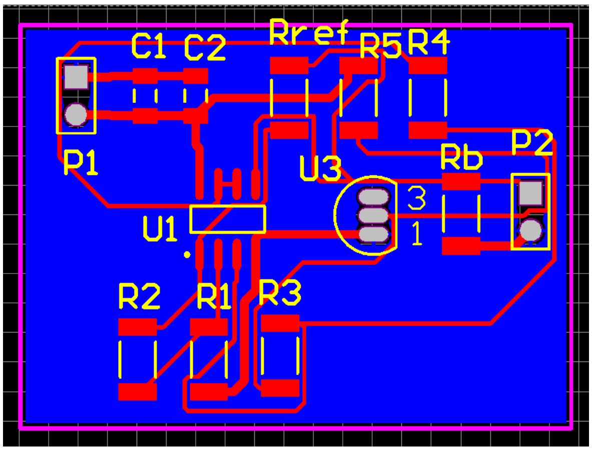


IMAGE 35: RESULT OF AUTOMATIC CONNECTING.

At first glance, the result is similar, but the detail that we manually entered makes a difference. There are quite a few mistakes that we can overlook in printed matter. It is also not considered that the components that are soldered on the bottom side the electrical contact only on the bottom side, so they need to be connected with connections with VIAs on the top side. VIAs are welcome at GND connections because we can make them shorter. If we connect automatically, it is sensible to use connection groups **Net classes**. These can be set through user interface **Design → Classes**. Here we can add groups by going to the tab Net classes and by right click choose **Add class**. We can see a window where all available electrical connections are written in the column **Non-members** and on the right is space where we can add wanted/marked connection with a click on arrow.

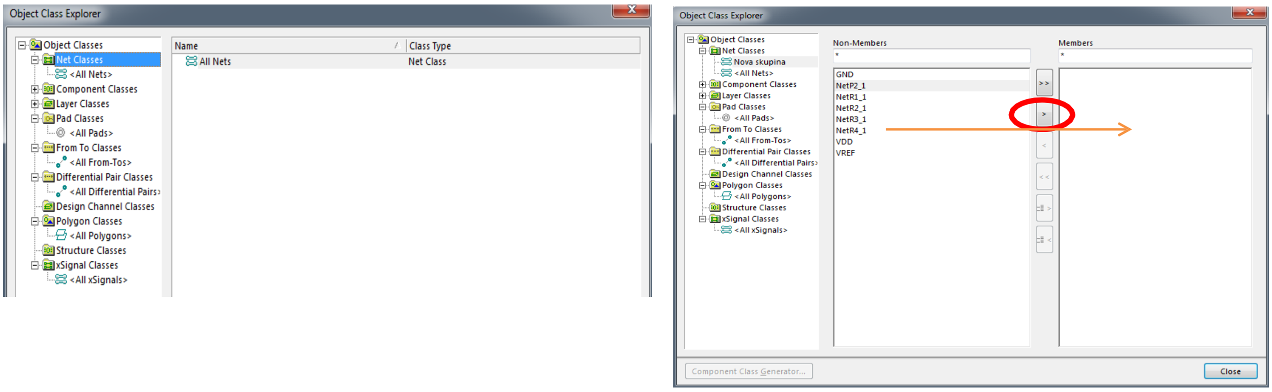


IMAGE 36: ADDING CONNECTION GROUPS.

3D view

3D view enables that we can see PCB in space from all perspectives. We can switch between views 1, 2 and 3. Number 3 is a shortcut for the 3D view, seen in image 37.

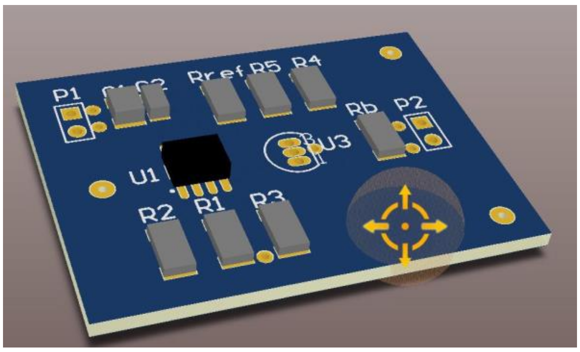


IMAGE 37: 3D VIEW OF PRINTED MATTER.

In the 3D view we can use the following shortcuts:

- Zooming in: CTRL + mouse scroll
- Moving up/down: mouse wheel, moving left/right: SHIFT + mouse scroll

- Rotation: SHIFT + right key on mouse and drag in the wanted direction that can be selected on sphere with arrows.

Altium enables import of 3D objects from different CAD tools in *.step* format. It is also possible to export in formats, such as: step, dwg/dxf. Exported objects can be used in other CAD programs that support mentioned formats.

Final documentation

Final or export documentation can be split into 5 groups:

1. **Parts list**
   a. Plans for automatic robotic component arrangement.
   b. Plans for placing components on both PCB sides..
2. **Output documentation**
   a. Component plans including copper connections.
   b. 3D views of PCB.
   c. Schematic display of electronic circuits.
3. **Documentation for printed matter production**
   a. Composite Drill Drawings – plans for holes, their position and size, all in one plan.
   b. Drill Drawing/Guides – plans for holes and instruction for making, their position and size in separate drawings.
   c. Final Artwork Prints – in one plan there is a lot of different output information.
   d. Gerber Files - Gerber files, made for each PCB layer separately.
   e. NC Drill Files - for numerical controlling (CNC) of drilling machines.
   f. ODB++ - Creating datasockets for manufacturer in ODB++ format, socketd on C++.
   g. Power-Plane Prints – plans for internal and separate layers.
   h. Solder/Paste Mask Prints – plans for protected layers.
   i. Test Point Report – report on testing points.
4. **Netlist Outputs – list of connections**
   a. The lists present logic connecting between components (in different text formats, CSV (comma separated values)
5. **Output reports**
   a. Bill of Materials – list of materials that we need for the making of the electrical circuit.
   b. Component Cross Reference Report – list of components based on the electrical scheme.
   c. Report Project Hierarchy – list of all documents.
   d. Report Single Pin Nets – list of connections that have only one

connector.

Documentation for production needs

In the last phase, we need to deliver all needed documents to the PCB manufacturer that are needed before making of PCB. For this, we need to send Gerber and NC drill files and material list.

Saving of Gerber files – Each Gerber file corresponds to one physical printed matter layer with all possible line profiles (Component overlay, signal layers, solder masking layers, etc.). Before **sending** and saving of Gerber files, it is recommended to consult the manufacturer to be unified with their requirements.