Construction of Electronic Systems

General tips for PCB design in Altium

Here we would just like to remind you of some of very useful tools and practices when designing the PCBs in Altium Designer.

Do not forget the "dual display" and "cross select mode"

For a good PCB design workflow, it is very highly recommended, if not crucial, that you are able to see both the circuit schematic and the PCB document at the same time. If possible, use two displays. If not, use "split vertical" option. And do not forget to enable the "cross select mode" ("**Tools -> Cross Select Mode**"), which performs components selection both in the PCB document and the schematics at the same time. This allows you to quickly recognize the component function from schematics during the PCB design.
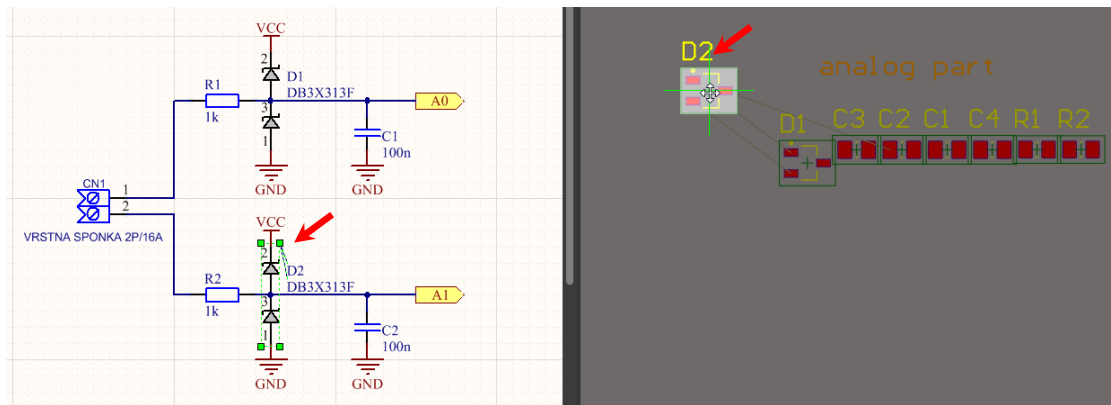

Figure 1 – Moving a component around in the PCB document. The cross select mode and being able to see both the PCB document and the schematic at the same time helps you decipher the role that the component plays in the circuit part.

Picking and placing the selected components in the desired order

One of very useful tools, especially *when you are making the first more detailed component placement* of the circuit part. Pick the components in a desired order in the schematic holding the SHIFT key, then go to the PCB document (left-click only once in the PCB document to put focus from the schematic on the PCB document) and the use the "**Tools -> Component Placement -> Reposition Selected Components**" (shortcut **TOC**).
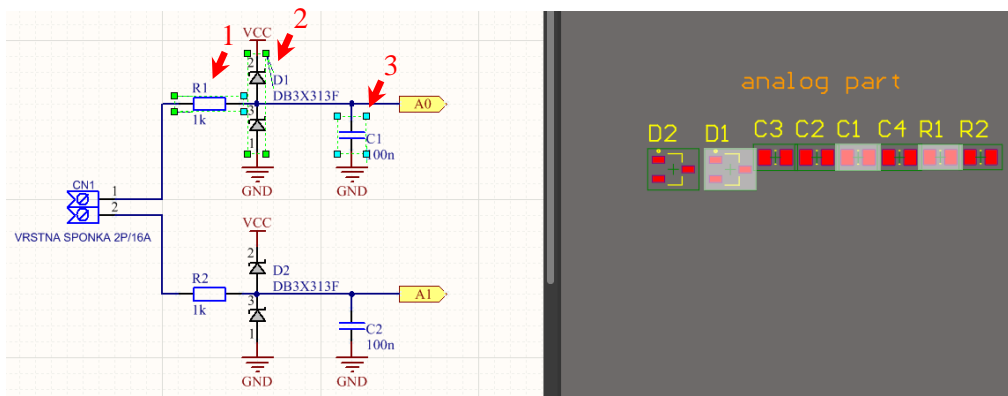

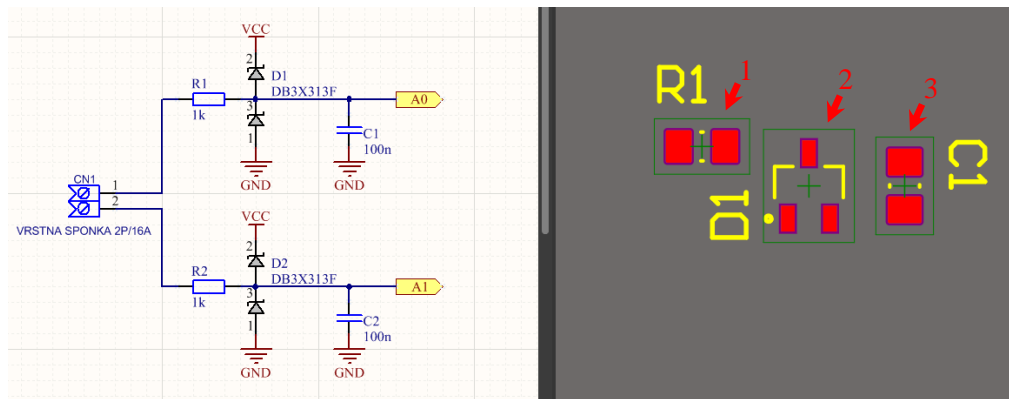*Figure 2 – First select the components in the schematics in the desired order…*

Figure 3 - ... and then place these components one after another in the PCB document. If the schematics is good, you can try and imitate the component placement from the schematics as well.

Finding components in the schematics from the PCB document

Sometimes in the PCB document happens that you are not sure where some component belongs and you cannot find it in the schematics. Then you can search for this component in the schematics using the "**Tools -> Cross Probe**" function and clicking on the component in the PCB document. This function will then open the schematic document that contains this component, select the component in the schematic and dim all other components out. See the example below. You can "undim" the schematic components using the **SHIFT+C** shortcut.
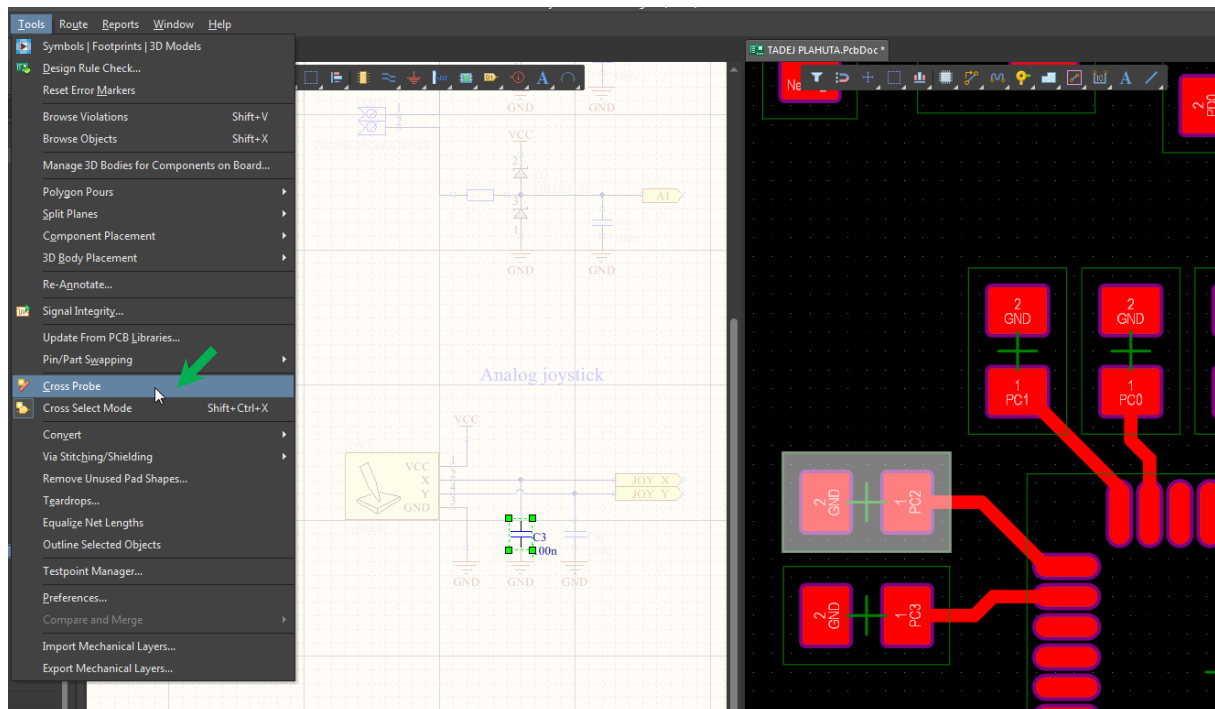

Figure 4 – Using "Cross Probe" function you can quickly locate the component in the circuit schematic

Rotating components with a 45-degree step

If you decide to use the 45-degree orientation of the microprocessor, then you might find the following setting very useful, since it will help you rotate all the surrounding components such as bypass capacitors to be rotated in 45-degree steps when you will use the SPACE key during component placement. You can change the rotation step in the general PCB editor settings (click that cog icon in the top right corner and navigate to "**PCB Editor -> General: Rotation step**"). See the figure below.
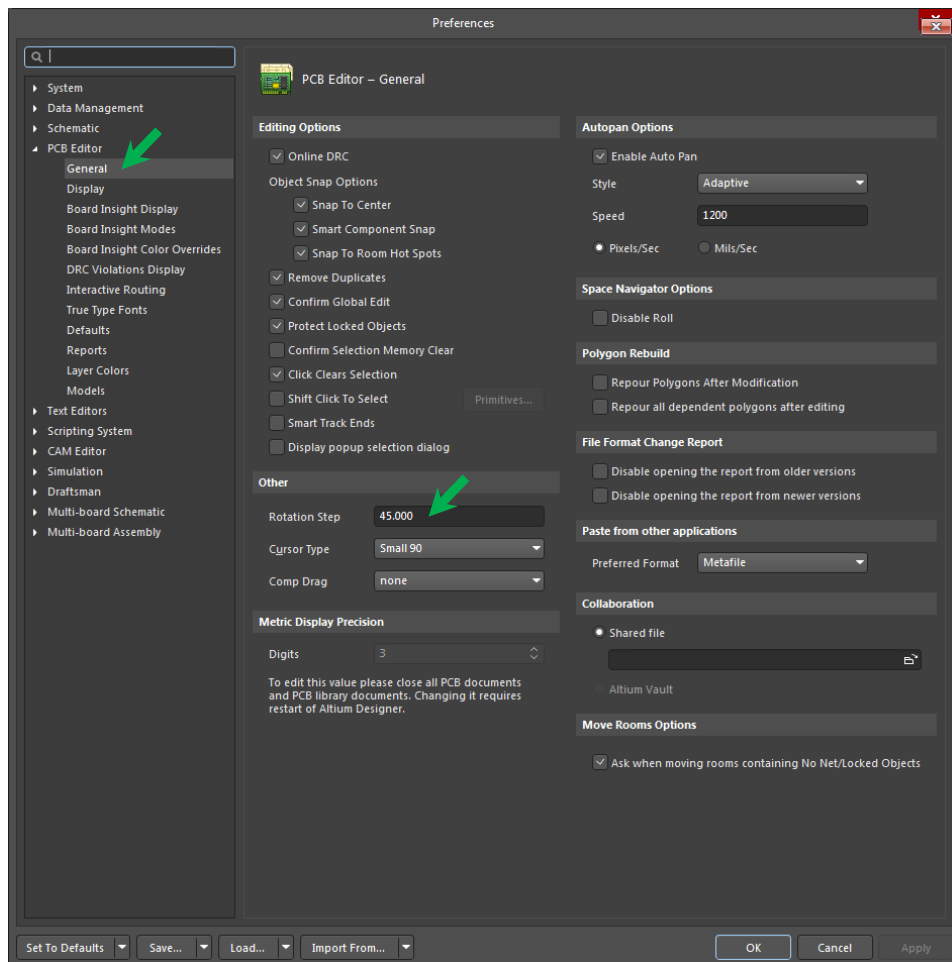

Figure 5 - changing the rotation step to 45 degrees

Finding all the available tool shortcuts during tool operation

Many Altium Designer tools allow you to *use shortcuts while the tool is in operation*. For example, while you are placing a track using the "Place -> Track", you can change the track corner style with pressing the SHIFT+SPACE shortcut (i.e. right angle corner, 45-degree angle corner, direct connection, etc.). And for such tools Altium can show you a list of all available shortcuts! How to find this list? During the tool operation, press the key between the TAB and ESC (see below). Or press the **SHIFT+F1** combination.
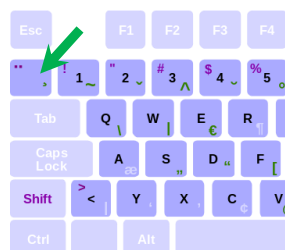

Figure 6 – an important key that displays a list of all available shortcuts during the tool operation

Hitting this key during the tool operation will display a list of all possible shortcuts that allow you to set the tool parameters/settings. See the figure below.
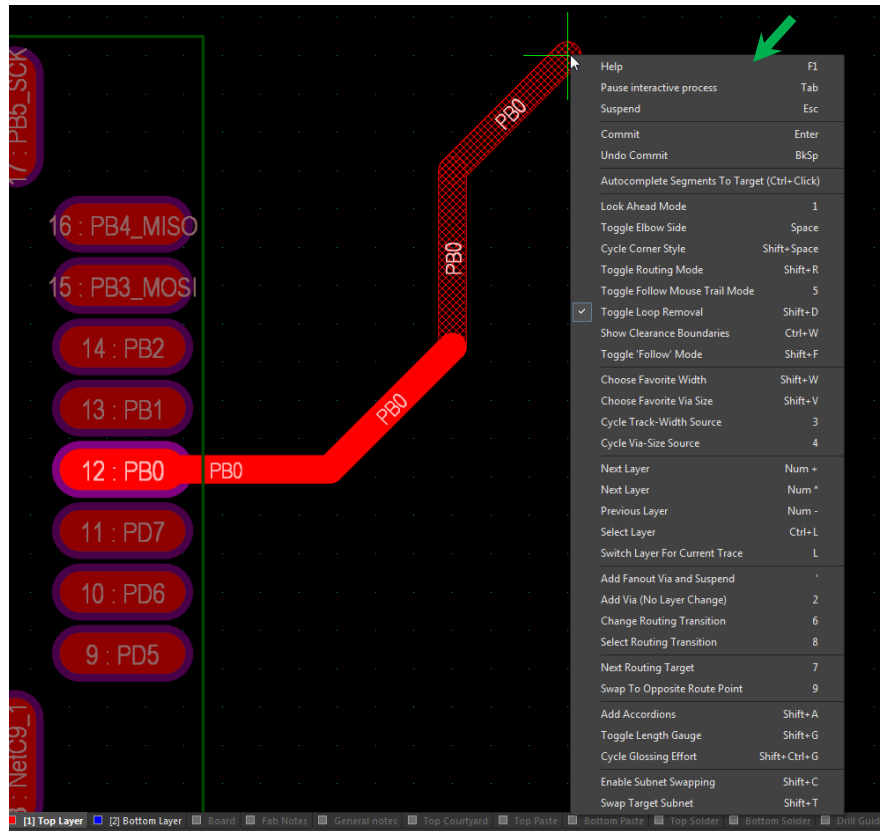

Figure 7 – a list of all available shortcuts and setting for the currently used tool! A very powerful way to learn how to use various Altium tools!

You can use this shortcut lists as a good source of information about Altium tools and learn about these tools as you go! Below you will find some more important shortcuts that you might find useful during the PCB design.

Changing the track elbow side

A very basic shortcut while placing a track. Pressing **SPACE** will change the track elbow side. See the example below.
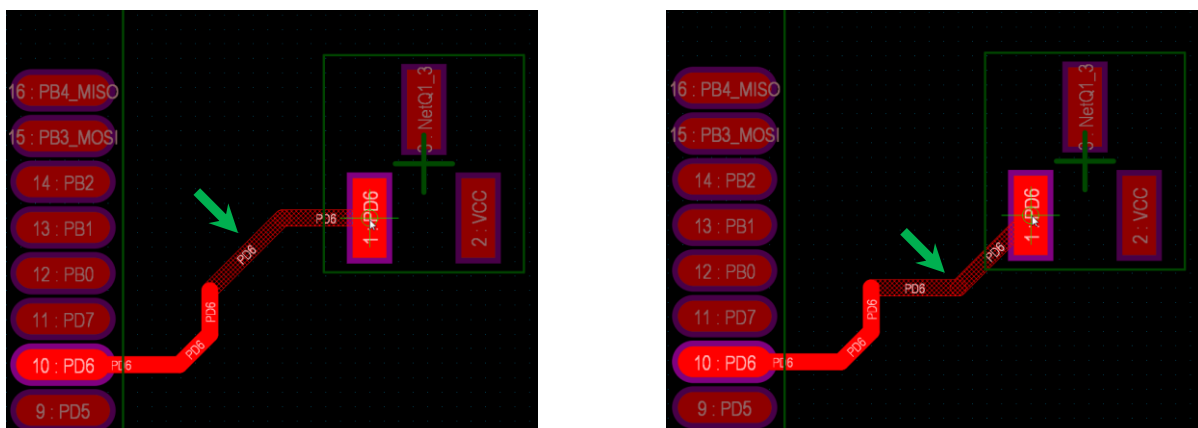

Figure 8 – pressing SPACE while placing a track will change the track elbow side

Selecting track width among the favorite values

Sometimes it is useful if you can change the track width while you are laying the track on the PCB. You can do this by hitting a TAB key and setting the desired width. Or even better, you can hit the **SHIFT+W** combination to open a list of favorite (preferred) track widths and simply select the desired width. See the example below.
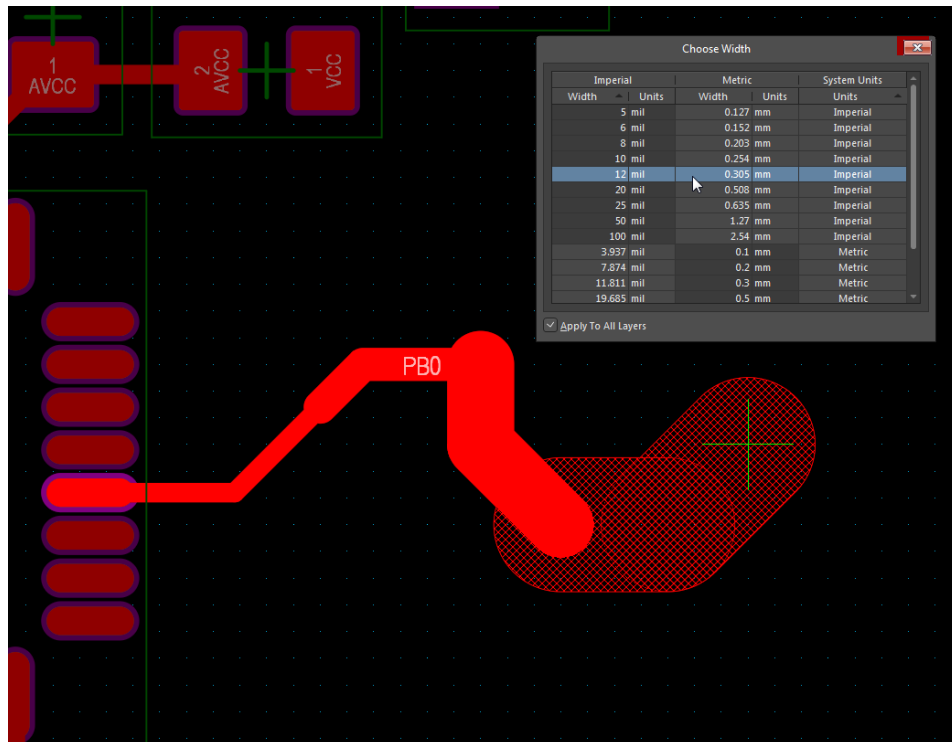

Figure 9 – SHIFT+W: select the preferred track width and change it while you are routing

Note: you can only select the track widths that agree with the track width design rule!

Change layer while routing a track

If you want to change the layer while placing a track, you have several options to do so:

1. pressing "**2**" will place via but you will stay on the same layer,
2. pressing "*" on the numeric pad will place a via and change the layer at the same time,
3. the **SHIFT + CTRL + [mouse wheel scroll]** will place a via and change layers,
4. **CTRL+L** will offer you a list of layers where the via should connect.

Choosing among minimum, maximum, preferred and custom geometry

While you are placing a track or via, you can use shortcuts to cycle between the four different geometries:

1. minimum – minimal geometry (i.e. track width or via size)

2. preferred – preferred geometry (i.e. track width or via size)

3. maximum – maximum geometry (i.e. track width or via size)

4. custom – custom geometry, typically the last used custom geometry

Of course, you must define the minimum, maximum and preferred geometry in the design rules first. The shortcut keys for track width and via size are "**3**" and "**4**" (see below).
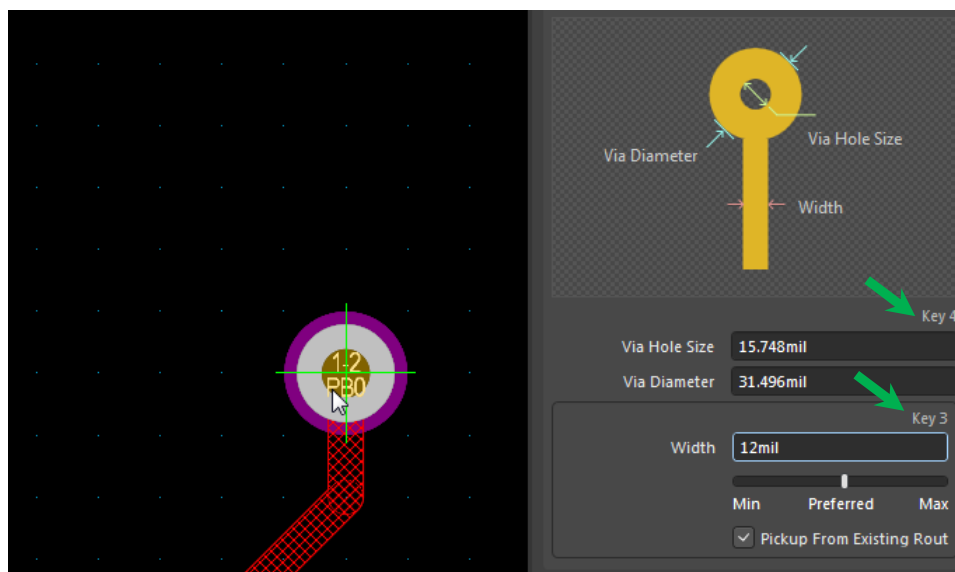

Figure 10 – setting the geometry while placing the track or via. The shortcuts can be seen also in the "Properties" menu (green arrows).

Quick selection of the entire track

Sometimes it is useful if you select the entire track with one click. You can do this with "**Select -> Physical Connection**" (shortcut **SC**). See the example below.
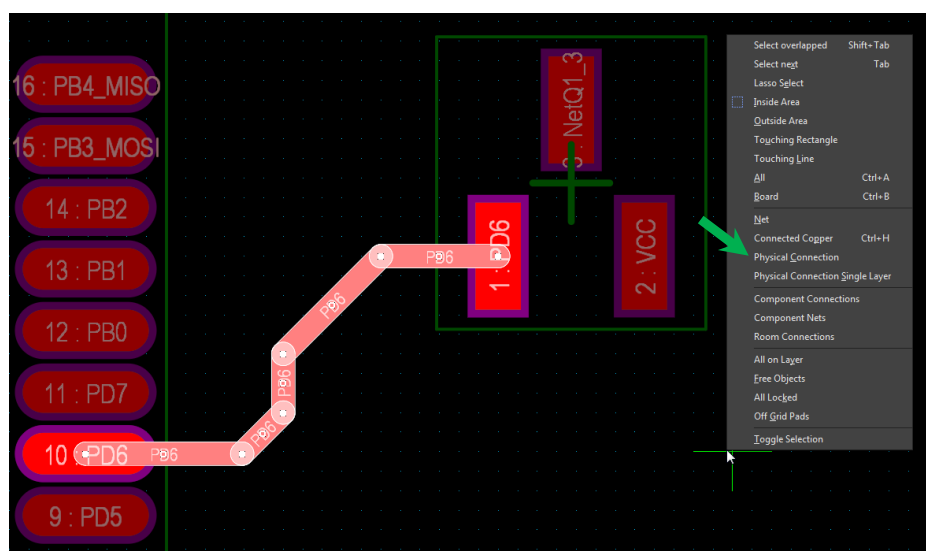

Figure 11 – a quick selection of the entire connection

A similar function: the "**Select -> Connected Copper**", which selects not only the tracks but also other connected copper constructs such as pads, fills etc.

Placing the GND vias in advance

We usually use the entire bottom PCB side for the ground copper pour, which we add at the end of the PCB design. You can add it sooner, but these large copper polygons are a bit annoying during the PCB design. With this GND copper pour in mind, it makes sense to prepare the GND vias that will connect to it at the end. This is a prudent design practice, since it immediately reserves the required room for GND vias. Also, it assures that we do not forget to place these vias at the end of our design. See the example below.
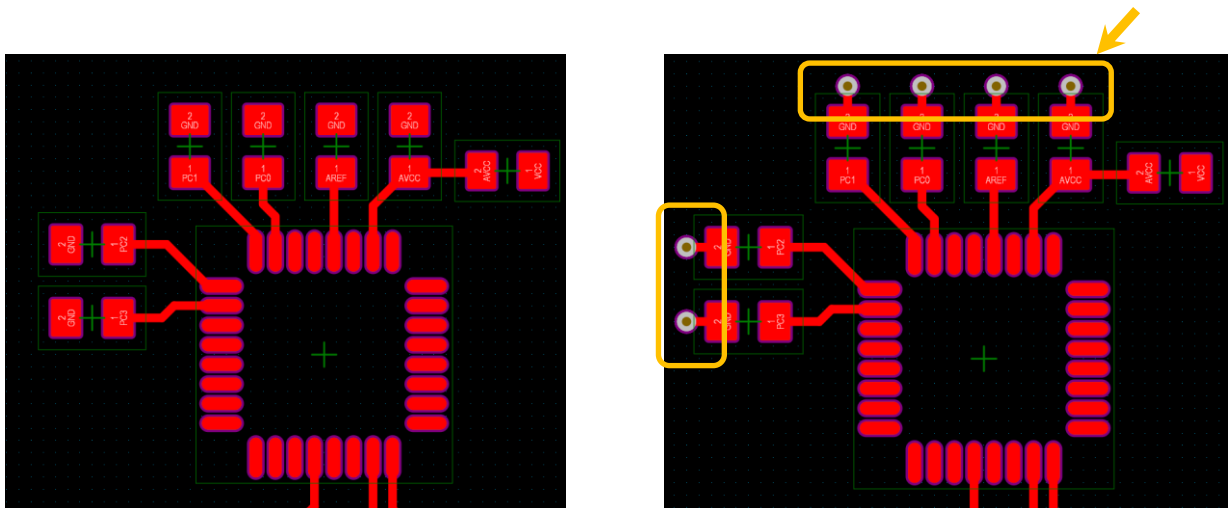

Figure 12 – while placing the tracks on the top side, it is prudent to also place the GND vias that will at the end of the PCB design connect to the bottom GND copper pour

Highlighting the specific connections

Sometimes it is useful to make a specific connection more evident, more clear to observe – so we can better visualize the signal flow. You can do this by simply using the **CTRL + [left mouse click]** on the desired connection. This will highlight the clicked connection and dim all other connections. See the example below.
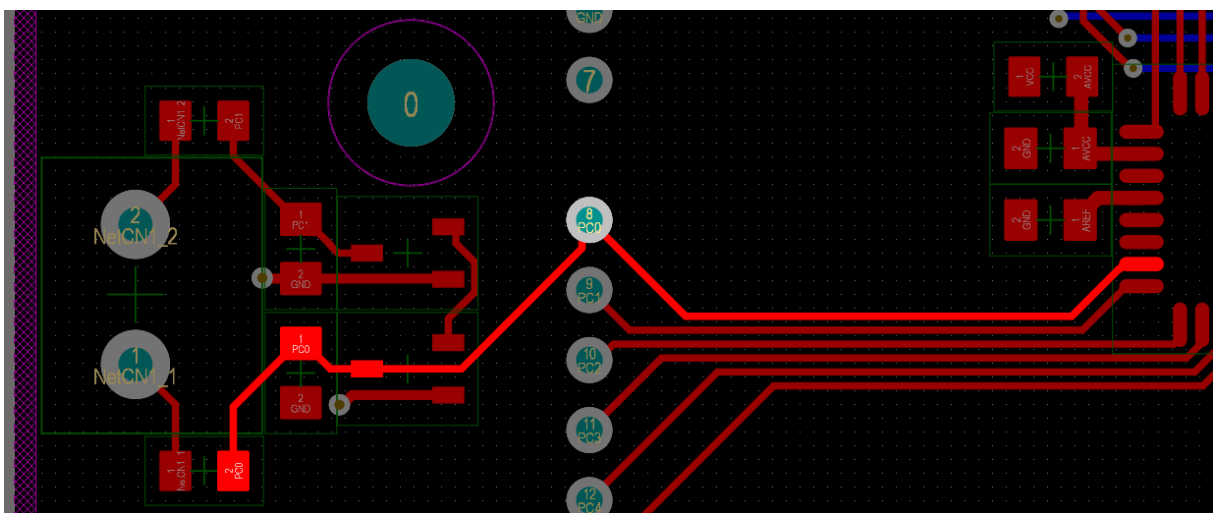

Figure 13 – temporarily highlighting one of the connections

The dimming level of all the other connections can be set by pressing the "**Š**" and "**Đ**". You can also highlight *several connections* by using **SHIFT + CTRL + [left mouse click]** .

To clear the highlighting, use the **SHIFT+C** shortcut.