

# Construction of Electronic Systems

## Exercise 6: USB DAQ project

### Initial (preliminary) component placement

---

#### Tips for the initial component placement

##### Hiding layers that are not important for component placement

When you are making the initial component placement, there is no need to display all the layers that the PCB document contains. If you do, the view can become crowded and cluttered. Therefore, we *usually hide most of the layers* (e.g. the overlay, component outline etc.) and *leave just the signal layers, board outline and courtyards*. The information on the courtyard layer about the "private space of the component" becomes quite useful now. See the example below. You could also use so-called *layer sets* to predefine the groups of layers used for different purposes.

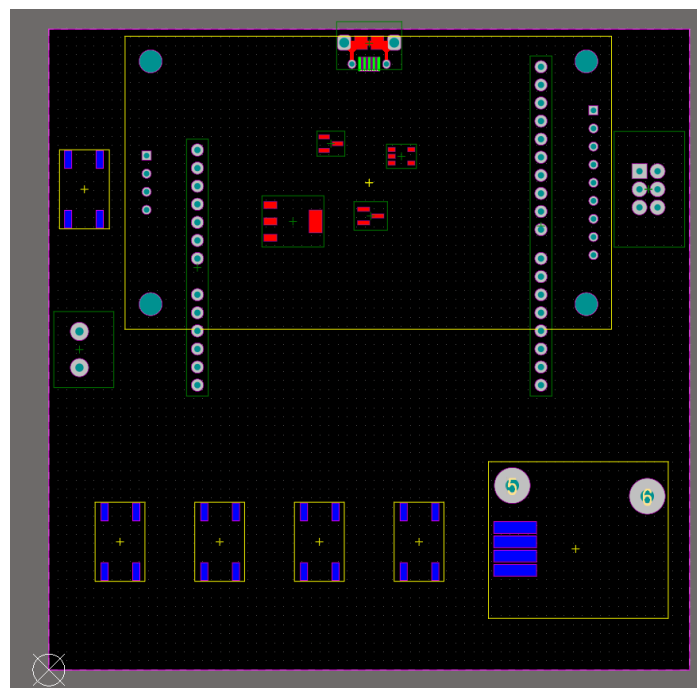


Figure 1 - when making the component placement we usually hide the layers that are of no use during the placement

## Separating components and organizing them into groups

Sometimes it is useful to organize the components into groups before we even start with the initial component placement. We usually form these groups based on the functionality of the circuit parts. Such an organization can give us a better feeling about how much space certain circuit parts require. See the example below. For example, you can immediately see that the USB part requires the most PCB space.

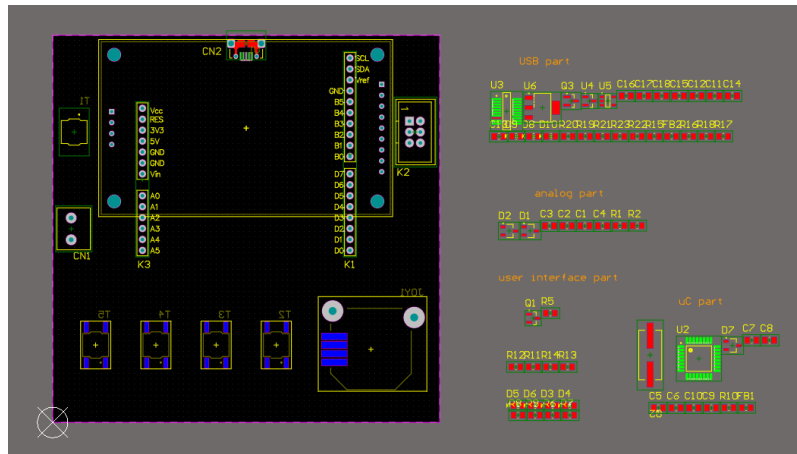


Figure 2 - organizing the components into groups before the initial component placement

### And how can we make such a component reorganization quickly?

Here we can use various features that Altium Designer offers. We will concentrate on *two very important concepts when it comes to component placement*.

The first concept is the so-called **cross select mode**. When this mode is enabled (see Tools → Cross Selection Mode), it allows you to select components in the schematics and the same components will be selected in the PCB document as well – and vice versa. So when you have a large number of components all thrown together in the PCB document, you can *easily pick out only specific the components by selecting them not in the PCB document but in the schematics!* It is usually prudent to use the selection filter that limits the selection in the schematics only to the components. See the example below.

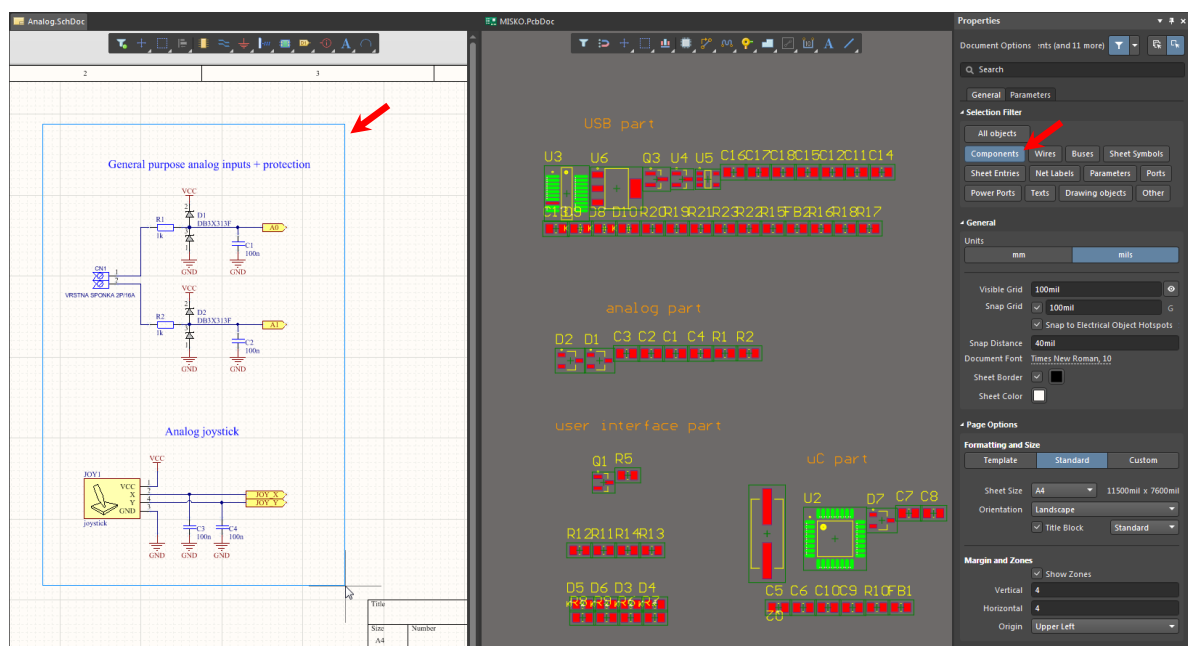


Figure 3 – using the selection filter and cross select mode we can quickly select only components that belong to a certain circuit part. When the left mouse click on the left side is released, all of the components selected in the schematics will also be selected in the PCB document. You can use CTRL+A to select all components in the schematics.

Now you know how to select only certain circuit part components using the schematics. Now you need to know how to group these selected components in the PCB document. Here we use the second important Altium functionality: *tools for component placement* (Tools → Component Placement). See the example below. If we want to group the components in one place, we use the "Arrange Within Rectangle" tool. It is useful to remember the shortcut: **TOL**. Try it yourself.

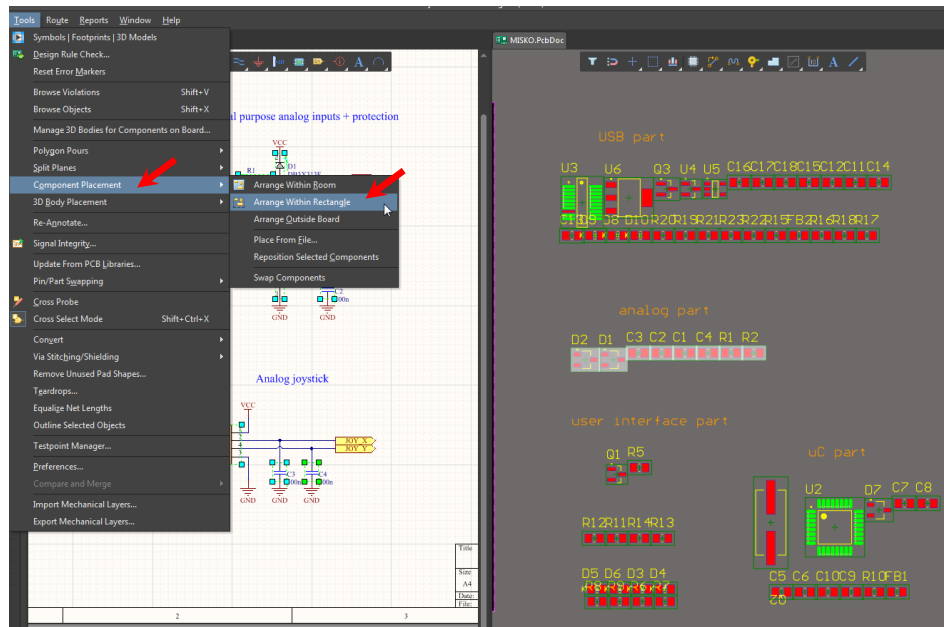
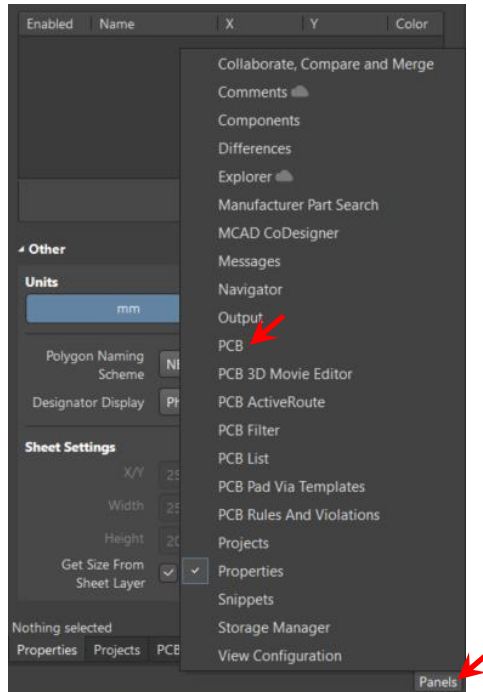


Figure 4 – quickly arranging the selected components into a rectangular-shaped group using the TOL shortcut

This "component-grouping tool" described above is perfectly useful when you are trying to make the first initial component placement on the PCB. Remember, the initial component placement is an approximate component placement where you trying to roughly arrange the circuit parts across the PCB. You use this tool and try to place all the functional groups on the PCB, following the strategy previously described.

## Using the component classes to quickly select the module components

If your project generates so-called "*component classes*", you can use them to quickly select all of the components from a certain block (i.e. module). In the PCB design, open the PCB panel.



In the PCB panel, select "Components" from a drop down menu. You will then be able to see the "Component classes". These classes are based on the schematic documents used in your PCB design. Make sure that "Select" and "Clear existing" fields are checked. See the idea in the figure below.

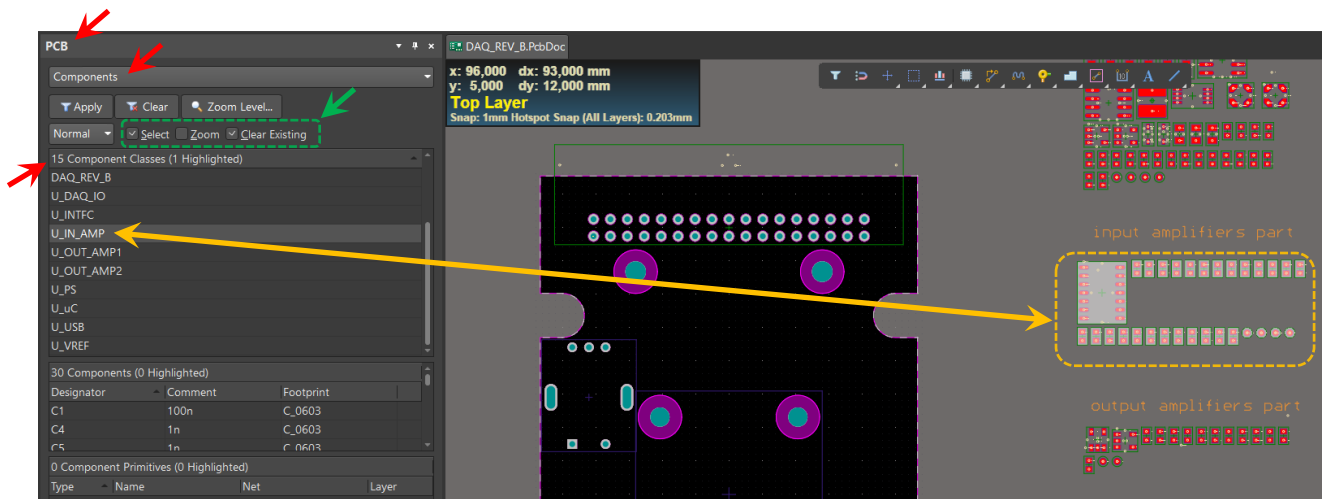


Figure 5 - you can use "Component classes" to quickly select all the components belonging to a certain module.