Construction of Electronic Systems

Exercise 6: USB DAQ project
Initial (preliminary) component placement

In this exercise you will start *placing components onto the PCB*. First you will place the components which have *their location (more or less) strictly defined*. Typically, these are the components that provide some kind of *interface to the electronic system*. In our case, these are the electromechanical components (e.g. connectors, switches etc.) and the user interface components (e.g. LEDs, displays etc.). When these "fixed components" are placed, you will start placing the rest of the components. This process is called *"the initial/preliminary component placement"* where the aim is to *approximately place* individual *segments* of your electronic system on the PCB. As a result, you will have all the components very approximately arranged on the PCB and waiting to start with the more precise placement and routing of the individual segments.

Exercise tasks

Setting the component clearance design rules:

1. ***Try to place the encoder E1 on the bottom side of the PCB.***

   What happens? Why is "everything green"?

2. ***Create a design rule that makes an exception for component clearance for encoder E1 and the OLED display U14.***

   In our design, these two components will be placed on the bottom PCB side and will *collide* with the 3D model of the bottom enclosure part. In real design, the bottom enclosure would have cutouts so that the encoder and the display would not cause any collisions. Therefore, we can allow collisions between the encoder / display and the bottom enclosure part in our design.

   Use the following procedure to make the rule exception:

   a) **use "Find Similar Objects" tool to create a "*search query expression*"** for the bottom enclosure 3D model only:

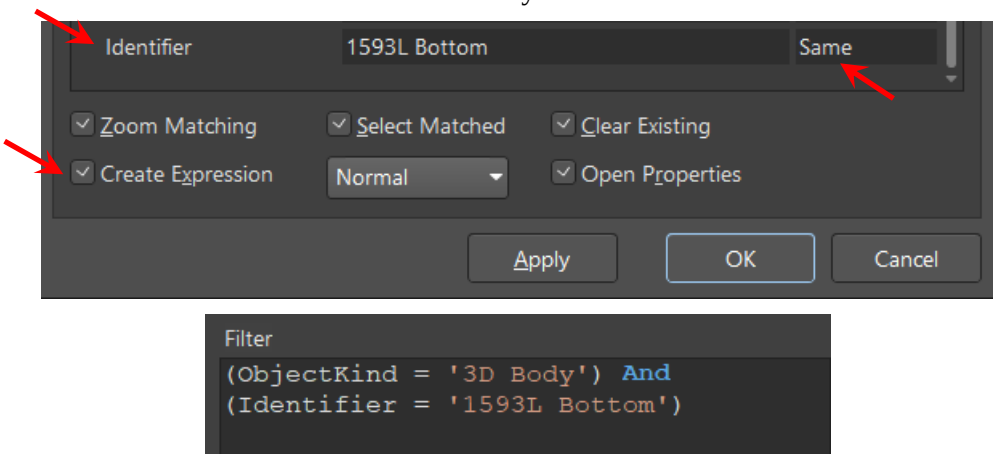

Figure 1 - using the "Find Similar Objects" (above) to generate the search query expression for the bottom enclosure 3D model (below)

b) Define the new component clearance design rule.

Create a new design rule in

"Design rules → Placement → Component clearance → New rule"

Specify the components that the new rule applies to by using the filter-generated search expression obtained in the previous step. Set the minimal allowed clearance to 0 mm, which means that the components can collide. See the example below.

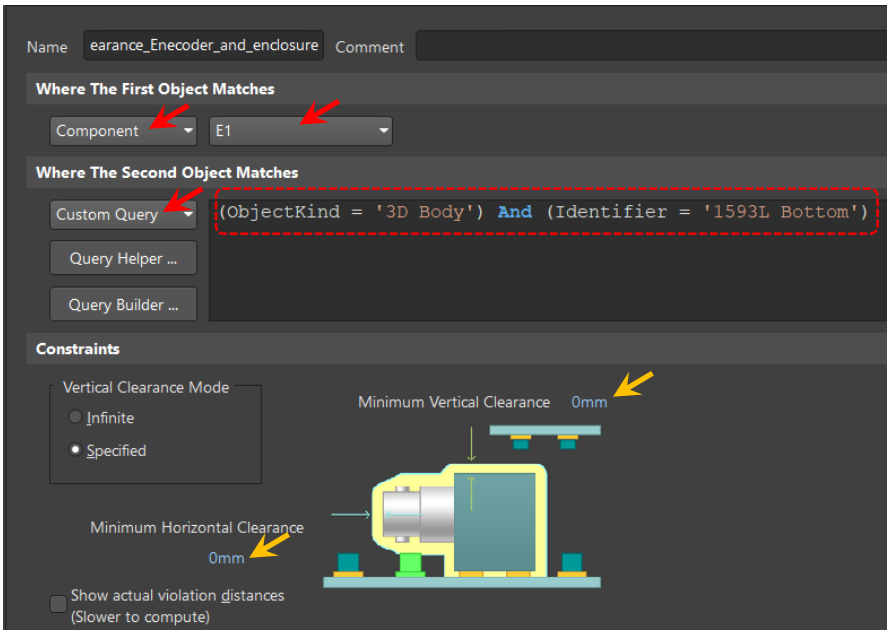

Figure 2 – an example of a component clearance design rule for the encoder E1 and the bottom enclosure 3D model. The encoder can be specified as a component with the designator E1, while the enclosure is specified using the custom query and the filter-generated expression.

When these two collision exceptions are added, the "green error marking" should disappear and you can proceed with the component placement below.

Fixed (locked) component placement

3. ***Place*** the I/O connector (J1), the USB connector (J2), the encoder (E1) and the OLED module (U14) according to the scheme presented in the document "PCB preparation scheme" (see the eFE "attachments" folder).

   You can double-check the "locked component" location by comparing your design with the 3D model of the PCB presented in the document "Prepared PCB - 3D model".

4. **After you have successfully placed these components, make sure that you *lock their location and prevent any further changes to their position* on the PCB.**

   Hint: you can "protect" the locked components and prevent accidental changes to their location. Go to general settings ( ) and find the PCB Editor → General → Protect Locked Objects.

Initial/preliminary component placement

5. **First, make sure that the *area surrounding the PCB mounting holes will stay clear of any components or copper* during the PCB design.**

   Refer to the *clearance specifications* in the "PCB preparation scheme.pdf".

   This step will help you *define the available PCB space before you start with the initial component placement*.

   Also ensure that there will be no solder mask in this space surrounding the holes: not on the top side nor on the bottom side. See the figure below.

   Hint: use the "keep-out layer" and the "solder mask layers" to achieve this. See the end of this tutorial.

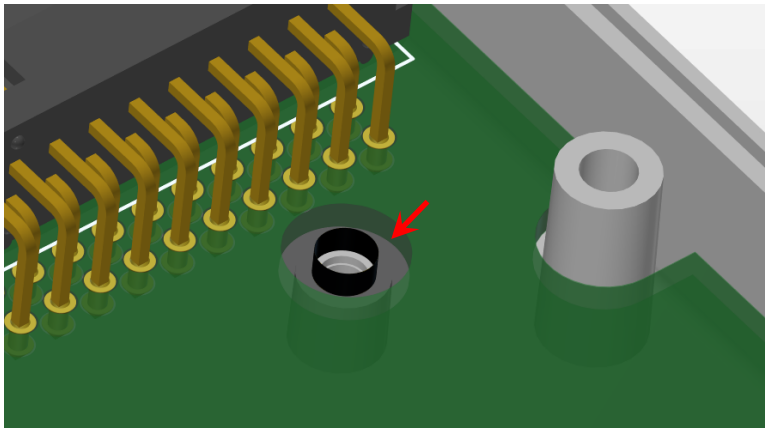

Figure 3 – ensuring a clear space surrounding the non-plated mounting holes. In addition, the green solder mask must be removed near the hole in order to prevent solder mask delamination due to mechanical stress.

6. **Make the *initial component placement* by *approximately placing* the rest of the circuit components on the empty space of the already prepared PCB.**

   The aim of this initial component placement is to ***divide*** *the PCB space among the* ***segments*** *of the electronic system* and to ***check*** *whether there is* ***enough space*** for the component arrangement that you have in mind.

   Although you are making an *approximate* component placement, *you should already bear in mind the most fundamental strategies of good component placement in light of minimizing the EMI* (remember the lecture about the EMI generating mechanisms). Below you will find a few very general guidelines to help you with this:

   - ***external* signals that arrive on the PCB via the cables and connectors should be *"terminated" on the PCB as soon as possible***, where by "terminated" we mean here that they reach the components that process or require these signals. This ensures that external signals have very short traces on the PCB, reducing the risk of EMI coupling to the external cables.
   - **Electrical components that produce very *fast voltage transients* (i.e. large dV/dt) should be *placed far enough from the PCB edge***, ensuring that the fast changing electric field stays coupled within the PCB space and is not coupled to nearby electronics or radiated away from the PCB.
   - **For the *critical parts* of the electric circuit think whether you can use component placement to help mitigate the sensitive nature of this circuit part** (e.g. placing the sensitive analog section in the "quiet" part of the PCB away from the noisy section).

Hint: use the "Tools → Component Placement → Arrange Within Rectangle" tool for initial component placement (shortcut: TOL). First select all the components of a single module in the schematic and then place all these components on the PCB using the TOL shortcut. In this way you will create segments on the PCB that belong to different system modules (see the idea below).

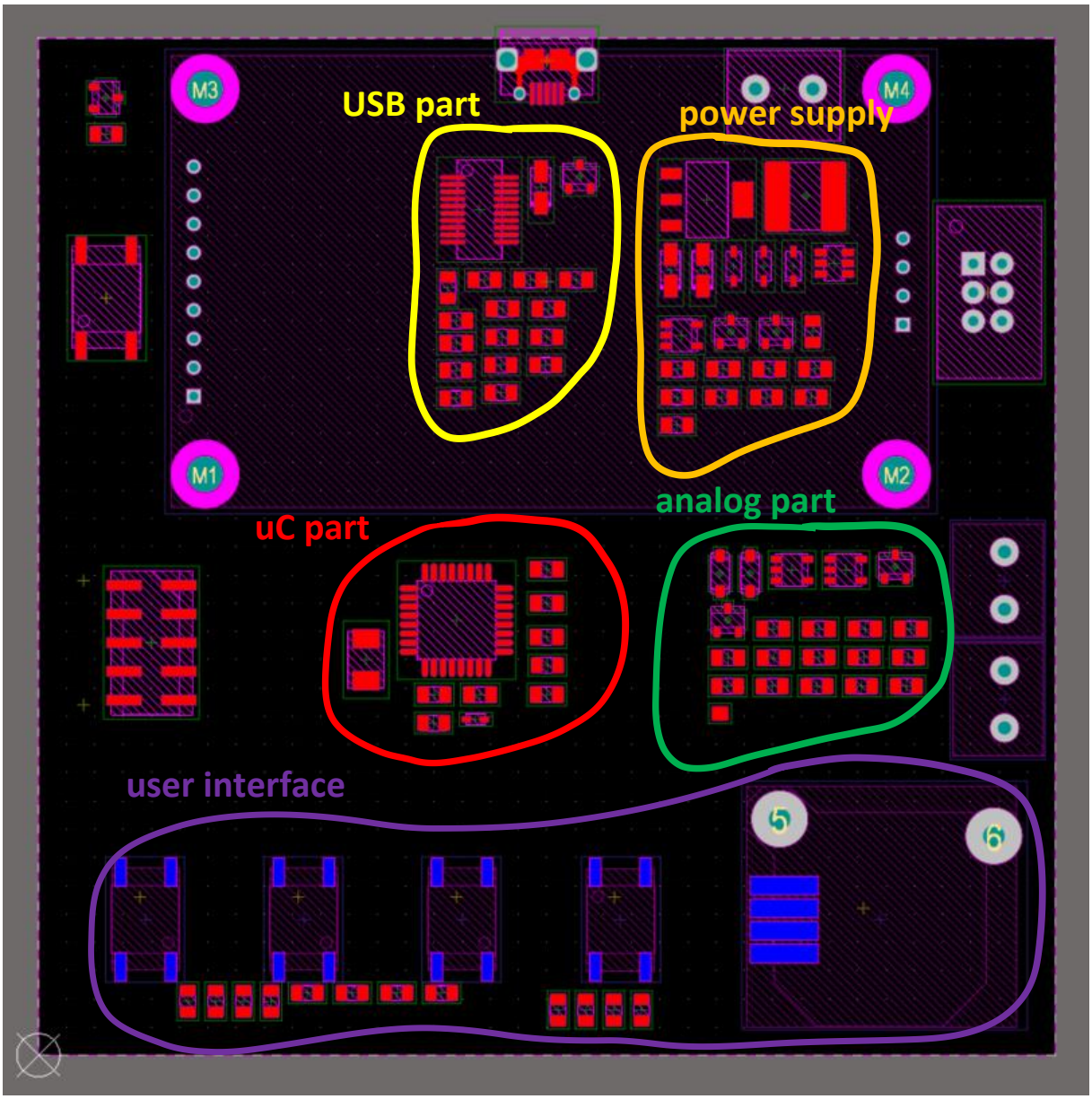

Figure 4 - an example of the initial component placement for a simpler project. Using the "TOL shortcut" you can easily group and arrange module components on the PCB and therefore divide the PCB into segments.

Explanation of the exercise

A part of the *"PCB preparation process"* is also the placement of those key components that have their position on the PCB more or less strictly defined, typically by the enclosure. These components are usually the electromechanical components (e.g. connectors, switches etc.) and the user interface components (e.g. LEDs, displays etc.). And because their location is strictly defined, it makes sense that, once placed, we *lock* the location of these components. What we also usually do at the beginning is that we *define which space on the PCB cannot be used for the PCB design due to various reasons* (e.g. clearances from the edges, clearances from some parts due to mechanical reasons such as mounting etc.).

The *initial component placement* (a.k.a. *preliminary* component placement, *an initial layout*) is an important step in the PCB design because *all of the subsequent PCB design depends on it*! With this *initial layout* you will *set a general course in your PCB design* since it will *roughly dictate* how the main circuit parts will be interconnected on the actual printed circuit board. In other words: now you are starting to give your *abstract* electric circuit from the schematic the *actual physical* shape.

The main purpose of such an initial component placement is to *decide* how you will *divide the PCB space* among the electrical circuit parts. This division, in turn, has several purposes:

1. you *allocate enough space* for you circuit parts and check whether your *component arrangement* is even *feasible;*
2. by smartly placing the circuit parts on the PCB you are *already working toward the good EMC* of your PCB.

When you finish with this initial component placement, you should have all the components on the PCB and there should be enough room around them for the tracks that will be routed later. The *orientation* of the most components is still not important at this point and also the *relative position* of the components within their electrical circuit part. *These are the details that will be fine-tuned in the next step*.

You can probably already feel that **the PCB design is an *iterative process*, where you are *gradually improving the current design situation step-by-step***. You can understand this initial component placement as the first approximation of the solution for your PCB design, similar to the situation where you are trying to iteratively solve a complex equation. In the next steps, you try to improve this current solution towards the better solution. By the way, you can also feel that this process of initial component placement requires some *foresight*, since you are initially trying to *predict* how the PCB design will continue in the future, making some kind of *design plan*.