Construction of Electronic Systems

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

About the initial component placement

Here we will have a look at how to go about the initial placement and which tools can be used to speed up the process.

Deciding where to place the circuit parts

In order to *make the initial component placement as good as possible*, we have to bear in mind at least two important aspects:

1. the *good component placement strategies* for minimizing the EMI and
2. the *nature* of *interconnections* between the *circuit parts*

The good component placement strategies for minimizing the EMI are based on the understanding of the *EMI generating mechanisms* and are, to some degree, summarized in the bulleted list of the task six:

- ***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).

And what do we mean by "the nature of the interconnections between the circuit parts"? We mean that when you are deciding where to place the individual circuit *part* on the PCB, you have to take into account how this individual circuit part is connected to the other circuit *parts* on the PCB:

- are there *many* connections to some other part?
- are some of these connections to other parts *critical*?
  - are they noisy?
  - are they susceptible to interferences?
  - are they problematic due to large currents?
- etc. etc.

For instance, ***if one circuit part has many connections to some other circuit part, we should really consider if we can place these two circuit parts near to each other***, ensuring that we won't have to place a large number of long connections across the PCB just due to the unsuitable component placement.

Note that by "circuit *part*" we usually do not mean just one single component, but a *group of components providing a certain functionality*. Although, if the component is complex and has many connections (e.g. the microcontroller), this could be considered as a circuit part.

Visualizing the interconnection between circuit modules

Using the top level schematics

The discussion above made us realize that it is important to have the *insight* into how the circuit parts are interconnected. How can we see these interconnections? We have to *visualize the circuit part interconnections* somehow. How can Altium help us with that?

A good starting point to get the feeling how the circuit parts are interconnected is to *use the top-level schematic diagram* of your hierarchical project. This should give you a good abstract image of your *circuit organization* and *interconnection between certain parts*. This is an especially helpful starting point if you didn't make the schematic yourself, but you are only designing the PCB. See the figure below.

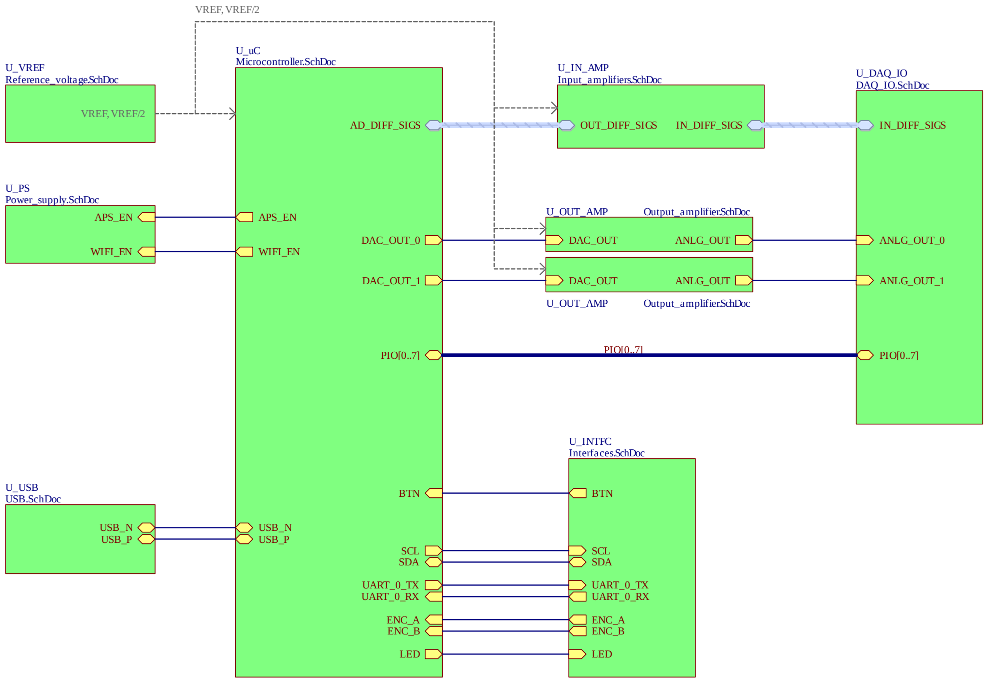

Figure 1 - a top-level schematics can give good overview of your main circuit parts and how they are interconnected. For example, you can immediately notice that the analog signal processing blocks come between the DAQ I/O and the microcontroller.

Highlighting the module connections only

Of course, in the PCB document Altium Designer displays all of the connections using the so-called airlines (see the figure below). This is useful, although the large number of airlines can obscure the actual information about the interconnections. One way to work around this problem is to use the trick presented in Figure 3.

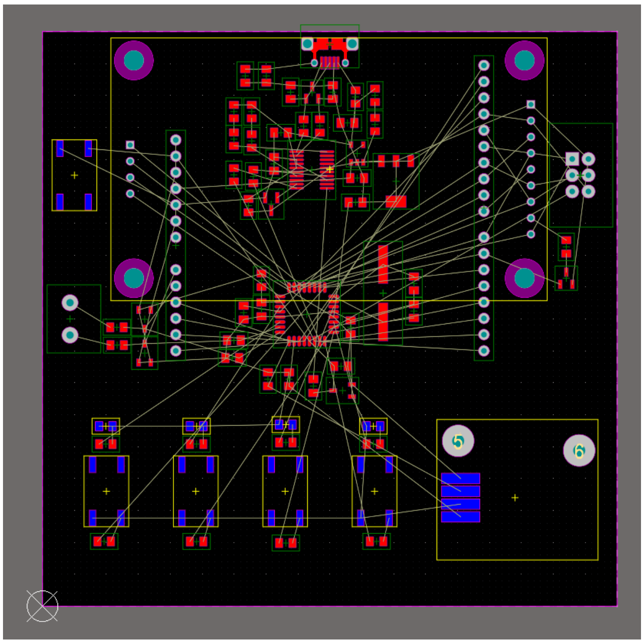

Figure 2 - airlines give us information how the circuit parts are interconnected. But this view can get pretty crowded soon.

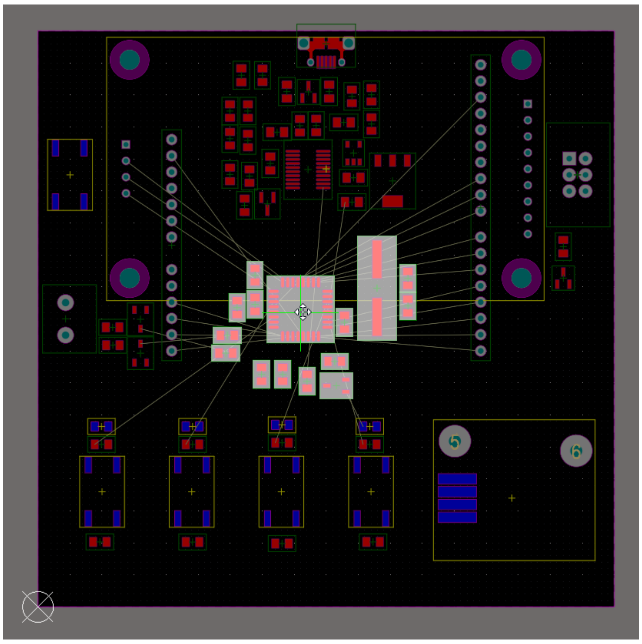

Figure 3 - if we select a circuit part and move it around a bit, the Altium Designer hides all other connections and shows only the connections to the circuit part that we are moving. This makes the connections to this part more evident.

Using "color-coding" for connection types

Another way to better visualize the interconnections between the circuit parts is to <u>*color the connections*</u> *according to the functionality of the connections* (or the nature of the connections). For example, see the figure below. In your project the microcontroller is, in a way, *the <u>central hub</u> for many connections*, so it is quite difficult to sort these connections out. But if we use additional coloring (i.e. color coded information), we can get a better idea what is the nature of signals connected to the microcontroller pins.

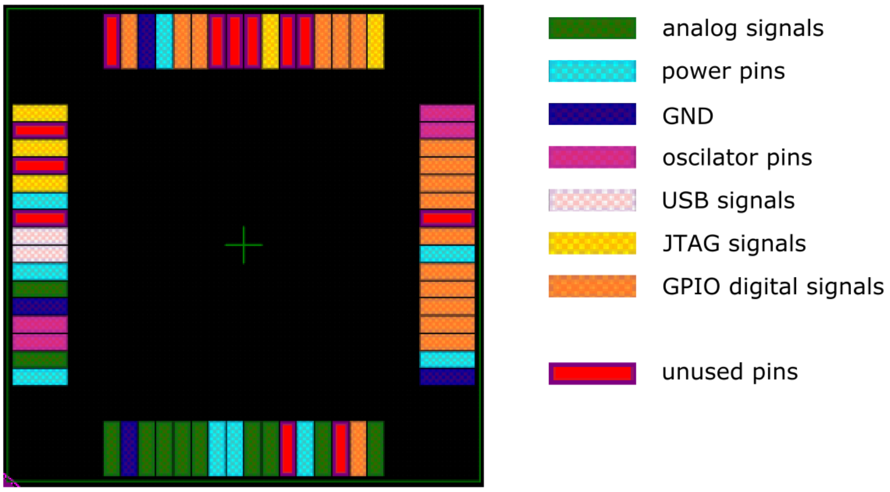

Figure 4 – connections to the microcontroller in your project. Using additional connection coloring can help you better visualize the nature of connections to the individual circuit part.

The color-coded visualization of the microcontroller pins can *give us quick additional information* about the nature of pin connections. If we concentrate on the group of connections for each edge of the microcontroller chip, we see the following:

- the bottom edge contains *mostly analog* signals,
- the right edge contains *mostly digital* signals,
- the top edge contains *mostly digital* signals and
- the left edge contains *mixed types of signals*: analog, digital and also the fast USB connection.
- The power-supply-related pins are, of course, distributed all around the microcontroller.

The importance of component orientation

This seems an appropriate place to **emphasize the following**: since the microcontroller is a complex part of your circuit, it contains *many connections of different types*. In such a case, during the initial component placement, it is not only important where to place such a component, but also *how the component is oriented*! The orientation of such a connection-rich part is going to crucially define how the circuit parts on the PCB will be connected with tracks.

In addition, it turns out that in some cases we *can benefit from the 45-degree rotation* of such a connection-rich component such as microcontroller. See the idea below. Such a component orientation can be useful since, for example, now the right side of the PCB "sees" not only ¼ of all the microcontroller pins but ½ of all the microcontroller pins. The same goes for the top, left and bottom sides of the PCB, relative to the location of the microcontroller.

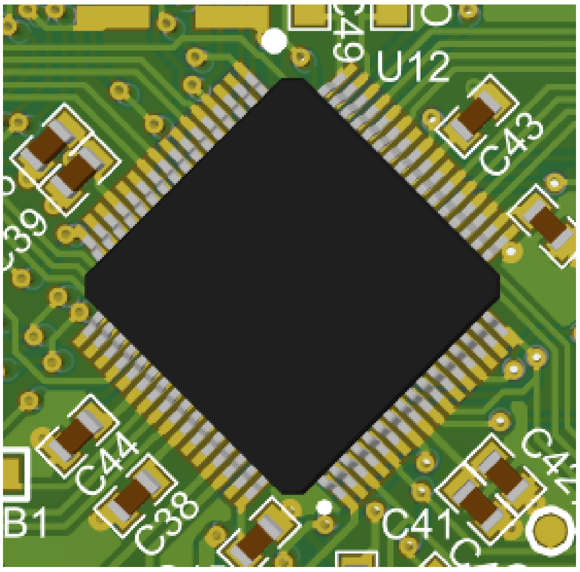

Figure 5 - rotating the microcontroller for 45 degrees can be beneficial sometimes

Another important circuit part that contains many connections it the DAQ I/O connector. Therefore, it makes sense to use the similar color coding for these connections. See the figure below. You can immediately see that the I/O connector contains mainly the analog signals, but one part of the connector is entirely dedicated to the digital GPIO connections.

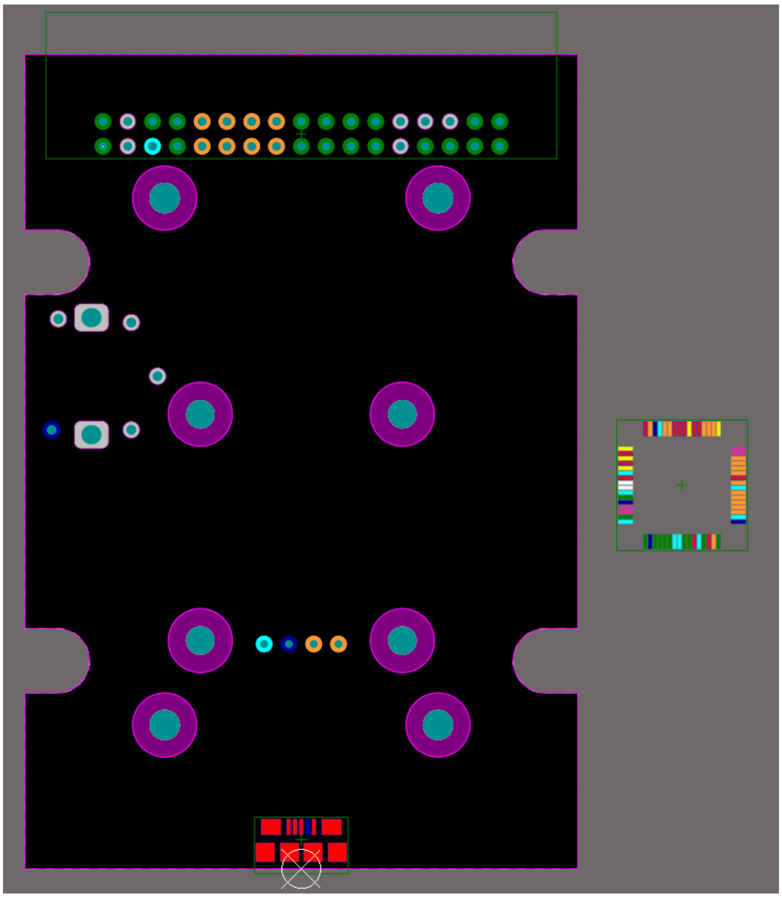

Figure 6 - the DAQ I/O connector also contains many connections, therefore we can get some additional insight in the nature of these connections using the same type of color-coding for these connections

Observations, conclusions and design decisions for initial component placement

With these ***well-visualized connections*** for the two key circuit components above, we can start planning our initial component placement. Let's use the figure above as our starting point to *draw out some observations* that you can use for the initial component placement, if you wish:

1. the *microcontroller is a digital component with fast voltage transients* inside its package. Therefore, we must ensure that it is placed far enough from the PCB edge. But where? The mounting holes and board "cut-aways" at the sides somehow *split the PCB in three sections*. We cannot place the microcontroller in the upper section, because there we need space for the analog signal processing parts (see Figure 1 with module inter-connections). We probably also cannot place it in the bottom section since there must be some space for the power supply and USB part (see Figure 1 again). This leaves us the middle section, which makes sense. The microcontroller is, of course, the *central* processing unit. ☺ See, in a way, there wasn't much choice where to put the microcontroller – we ruled out other possibilities.

2. Remember: we need to be careful about the *microcontroller orientation* since it contains so many connections. How should we orient it? Since one group of microcontroller pins is almost completely "analog", it makes sense to try to orient it so that they are faced towards the analog connection coming from the DAQ I/O connector. Also, be careful about other critical signals that connect to the microcontroller. How will the fast USB line connect to it? Try to optimize the microcontroller orientation so to make the important connections to the microcontroller as elegant as possible.
3. Another circuit part that also doesn't leave us with much choice where to place it is the analog part that process the analog signals. Obviously, it has to lie somewhere between the microcontroller and the DAQ I/O connector. This means that we must place it somewhere in the upper PCB section. This also complies with the good EMC strategy to place components near the connectors.
4. Similarly, we are limited with where to place the SMPS. The SMPS draws its input power from the USB power line. Therefore, it makes sense to place it near the USB connector. We only have to decide on which side of the connector: the left or the right? If we place it on the left side connector, that makes sense, because the USB power line is on the left side of the connector. But there is a possible trap here. Look at the power supply schematics below. There are quite a few additional power *components following* the SMPS: the analog voltage regulators, the CPU voltage regulator and the WiFi voltage regulator.

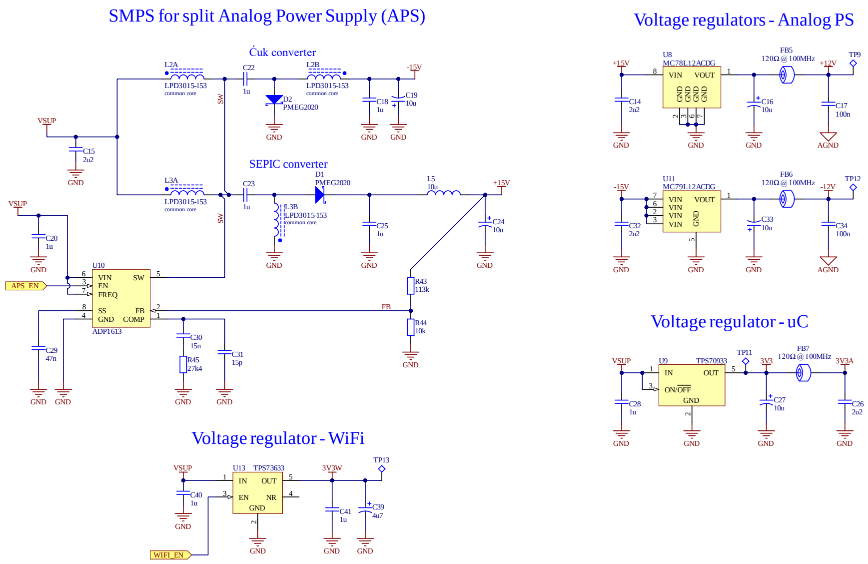

Figure 7 - the power supply circuit part has many subparts. Think how you will arrange these subparts and how the "signals flow from these subparts onwards".

This means that we need additional space somewhere next to the SMPS part where these voltage regulators will be place. Moreover, the analog voltage regulators have a quite large footprint (check it out). OK, the voltage regulators can be *distributed across the PCB* near to the parts which they supply. But be careful about the *circuit part interconnections* and the *"signal flow"*. The ±12 V analog power supply must be connected to the output amplifiers and the output amplifiers must be connected to the DAQ I/O connector. Have this in mind when deciding on which side of the PCB (left or right) will you place the analog power supply part. Also, look at the prepared PCB on Figure 6 once again. There is an obstacle on the left side of the PCB! Namely, the encoder with its many pins and holes. This could be present a problem and can complicate the "signal flow" towards the analog outputs. Think about it and then decide.

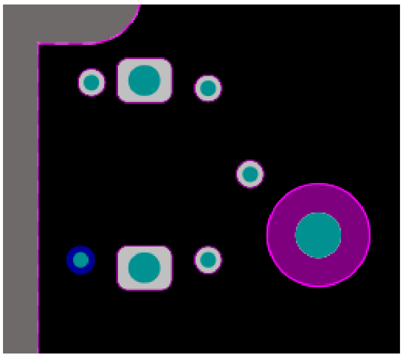

Figure 8 - encoder with its many pins and holes can become an obstacle for either signal flow or component placement

5. Now, depending on the selected orientation of the microcontroller and the placement of the power supply, you are left with empty PCB space for other circuit parts. ***Try to separate the analog parts of the circuit from the possibly noisy digital parts of the circuit*.**

Discussed above are some of the initial issues that have to be *considered before and during* the initial component placement. Also, **do not forget that the PCB design – even the initial component placement – is an iterative process!** This means that it is nothing wrong if you decide to completely redo the initial placement and try to find another, better one.

By now you should be equipped with the basic ***strategy*** for how to go about the initial component placement.