

Construction of electronic systems

Exercise 3: USB DAQ project Power supply schematics

Hints and guidelines

Draw standard circuits the standard way!

Remember our first exercise where we talked about an important guideline for designing good schematics:

"Draw standard circuits the standard way."

In this exercise you will find a perfect example of a bad datasheet schematic, where the *standard* SEPIC and Ćuk converter topologies are drawn in a *non-standard* way. See the schematic from the datasheet below.

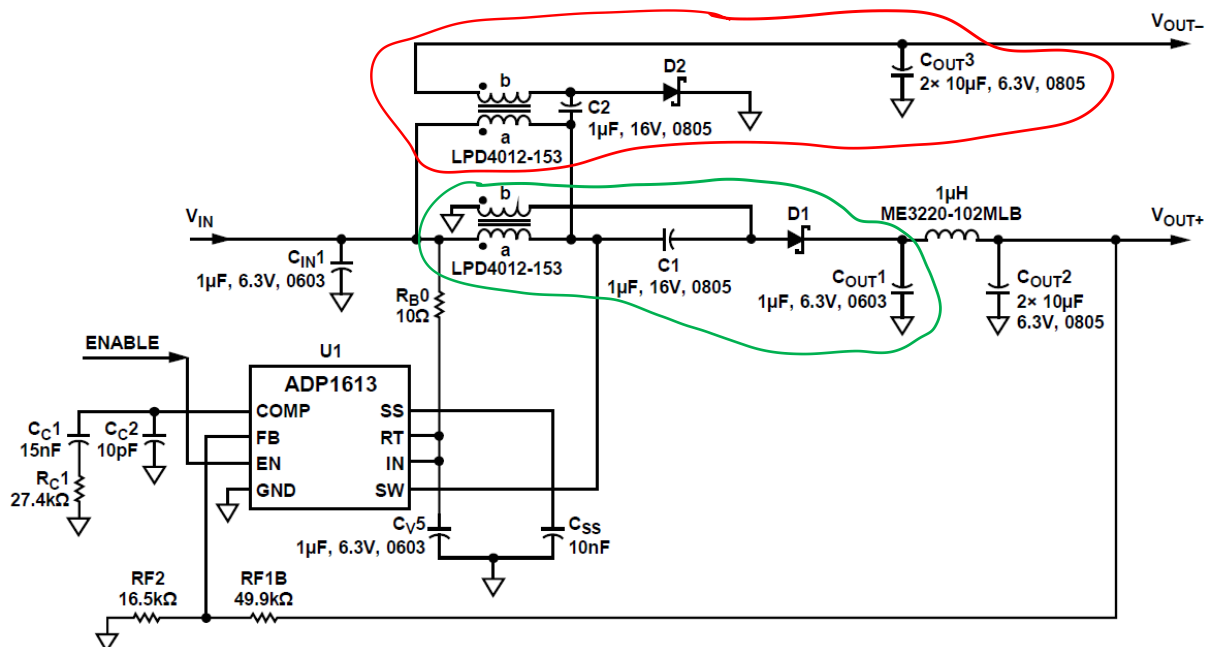


Figure 1 – the standard Ćuk and SEPIC converters are drawn in a non-standard way.
And the chip symbol is also really awkward.

The red part of the schematic is actually the Ćuk converter and the green part is the SEPIC converter. But this is far from evident from the given schematic. It could be made more evident, if we drew these two parts in the standard way. See the schematics below.

This is how the schematics for the Ćuk and SEPIC converters are usually drawn.

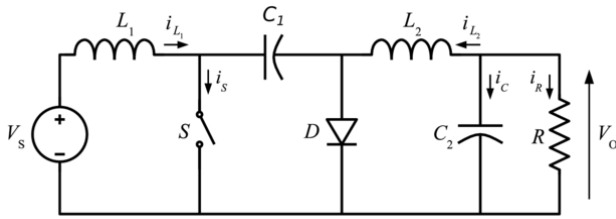


Figure 2 - [Ćuk converter](#) standard schematic

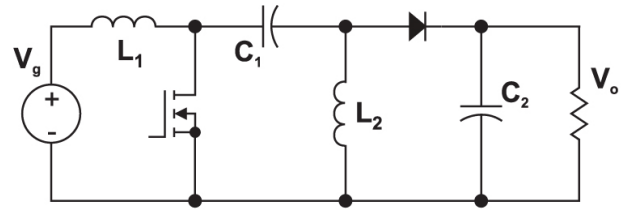


Figure 3 - [SEPIC converter](#) standard schematic

And this is how they actually drawn the both converters in the first figure of the application note (see below), but at the final schematic, they really "made a mess" of this.

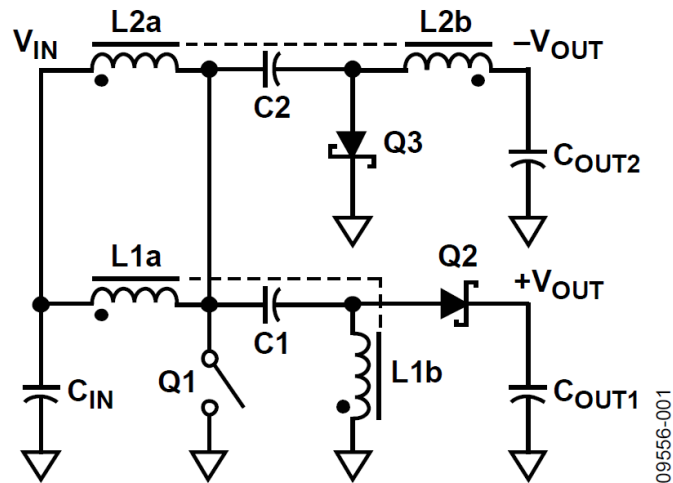


Figure 1. Schematic of the SEPIC-Ćuk Converter

How come that there is a "standard way" to draw these two schematics? Because Ćuk and SEPIC converters have been well studied and documented through several years. And in these studies, they were typically drawn in such a way, that then became "standard".

Therefore, a challenge: try to create a good schematic for the power supply by incorporating the standard layout for these two topologies.

Using the multipart components in Altium Designer

The Ćuk and SEPIC converters can be implemented using the so-called *coupled inductors*. In fact, that proves to actually be beneficial.

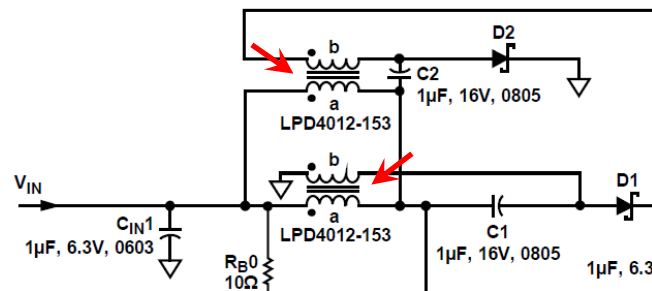


Figure 4 - using coupled inductors to implement Ćuk and SEPIC converters

As the name already implies, the coupled inductors in our case are actually two inductors that share the same ferrite core. Therefore, it makes sense that when we are creating a component symbol for such coupled inductors, that we make a so-called *multipart component* – each part represents one of the two coupled inductors. See below.

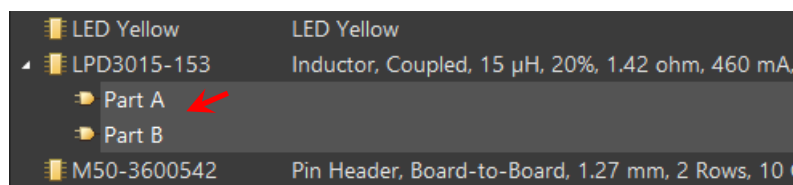
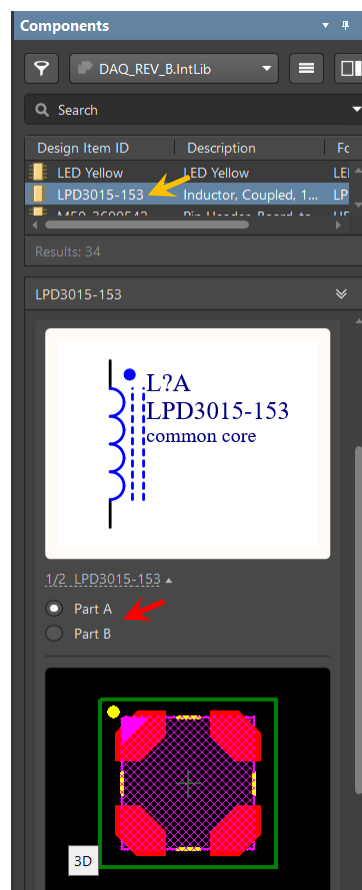


Figure 5 – in Altium Designer we represented the coupled inductors by a multipart component: each part is one inductor

Just a short hint how to properly use the multipart components in the Altium Designer while drawing the schematic.



While placing the multipart component, you can select which part of the multipart component you want to place. See the figure to the left (red arrow).

If you decide to place a multipart component by double-clicking the component ID in the list of the component (orange arrow), then you will be able to place all of the parts cyclically: first part A, then part B, then part A again etc.

Using the polarized capacitors

Note that the schematic in the application note is also problematic because it uses a not-that-typical symbol for the [polarized capacitor](#). See below.

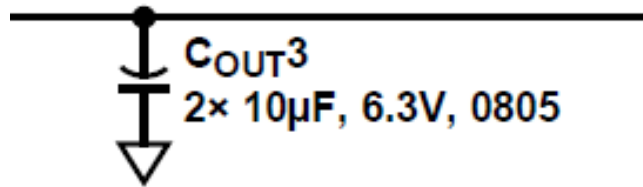


Figure 6 – you might recognize a symbol for a polarized electrolytic capacitor, but which side of the capacitor is positive and which side is negative?

The problem occurs because the schematic uses an older American symbol. See an excerpt from the book titled "Beginner's Guide to Reading Schematics" by Stan Gibilisco below.

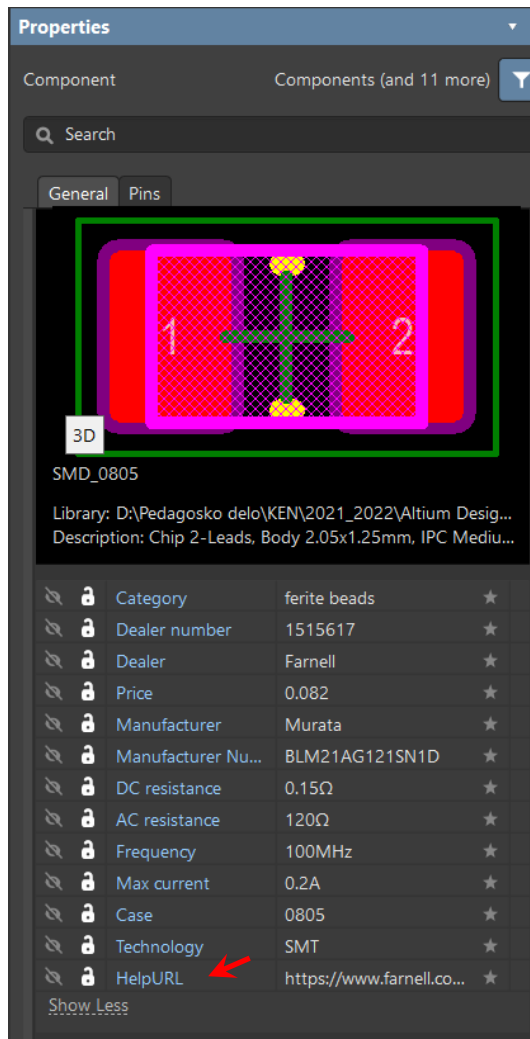


FIG. 3-9. *Standard symbol for a fixed capacitor. The curved line represents the plate (or set of plates) electrically closer to ground.*

So, the "curved side" of the capacitor is the negative side. Details.

You can see, how the standardization of the electronic symbols plays an important part in understanding the schematics.

Getting a quick help for a component in the schematic

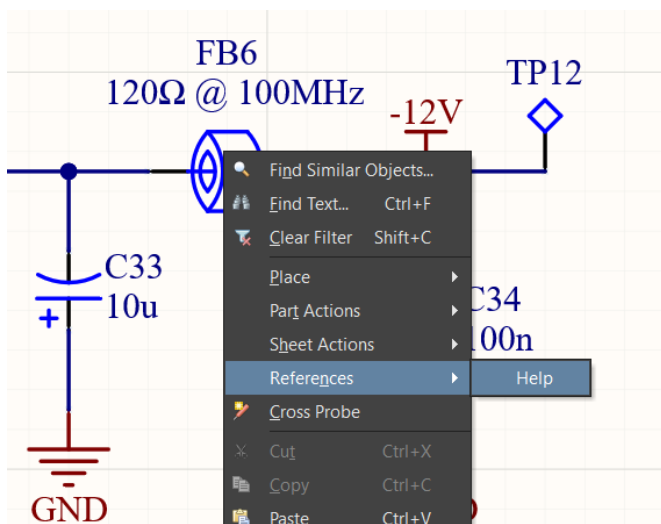


If you create your Altium Designer components in such a way, that they have a parameter named "HelpURL" (see figure to the left), then you can use this parameter to specify a link to the document or a web page that provides some kind of additional helping information about the component.

Typically, you would enter here the link to the official web site of the component or link to the datasheet of the component.

And how can you use this help? In the schematics, right click on the components and chose "References -> Help".

See below.



You can use this kind of help for most of the specific components that you will find in the project library, so you can quickly get additional information about the USB DAQ components.