1. Getting used to Altium interface

Please download and extract the files that will be used:

https://robot-competition.epfl.ch/download/exercises/altium1.zip

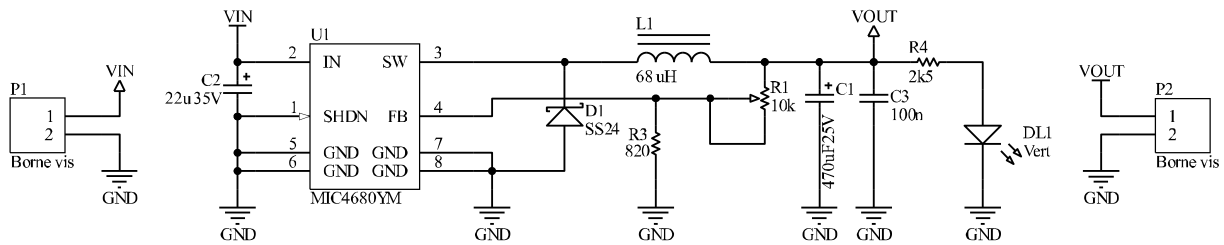
1.1. Document parameters and template

Load the schematics file ex101.SchDoc. Replace the document template (menu: *Design → Templates → General templates*) with the A4EPFL provided in the ZIP file. When asked, choose *not* to replace any parameters, as this would remove all the document information (title, etc.). If you want to use the EPFL template by default for new schematics, you can copy it to the templates directory (its location can be seen in the preferences under *Data management → Templates*), and select it as default in the schematic editor general preferences.

Once the new template is applied, edit the document parameters to set a custom title, author and company name. To access the parameters you can press the `K` key to display the *Panels* pop-up menu, and open the *Properties* panel. In the panel, you will find a *Parameters* tab, where you can edit the document parameters.

1.2. Moving and components, adding wires and power nets

Load the schematics file ex102.SchDoc; it contains all the components necessary for the following schematics of a step-down (DC/DC) converter:



Move and rotate the components as needed, and add the required wires and power net symbols to obtain the same circuit. It does not need to *look* the same, but it need to be *electrically equivalent*.

To draw the schematics, consider the following points:

- Don't care about the designators having undefined numbers (*i.e.*, `R?` rather than `R1`). EDA programs have a function named *automatic annotation* doing this automatically. Once your circuit is complete, look for *Annotation* in the *Tools* menu, and run the *Annotate schematics quietly* tool to automatically number component designators.
- The symbols of power nets (*e.g.*, ⍋ and ⊤) are not important, just be coherent about what you do. They can all be the same, or they can have a specific meaning (here ⍋ is used to tell that the signal *comes out* from there, whereas ⊤ is used in all the other cases). The only widely used convention is that ground (GND) symbols should point downwards and look different from the other ones; Altium clearly marks those as *ground* and *earth* symbols.

1.3. Schematic library

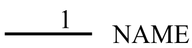
Open the provided file ex103.SchLib. This schematic library contains two components: the MIC4680 DC/DC converter chip visible in the previous exercise, and a bidirectional clamping diode. When opening the file, the *SCH Library* panel should automatically open to list the components present in the library; if not, you can open it by using the `K` shortcut key to display the *Panels* pop-up menu.

Click on the MIC4680 component to display it. Using the *Place pin* (use the `P` shortcut to open the *Place* pop-up menu), add the three missing ground pins (see the schematics of the previous exercise). Double click on pin 1 (SHDN) and change its type to *Input* so that the pin is displayed as an input pin. Double click on the MIC4680 in the *SCH Library* panel to open the properties of the component. Change its default designator to `U?` (U being a widely used prefix for integrated circuits). Also change the comment to the name of the chip itself.

Now click on the DC2 component. It is the body of a *bidirectional clamping diode*, a component used to protect power and signal lines from overvoltage. As you will notice, the component has no pins: you will need to add them. The symbol of the component should look like this:

The *designators* of the pins (which make the link to the footprint pads) must be 1 and 2, respectively. As the diode is bidirectional, it does not matter which one is which. As it typically happens with this kind of components, pins have no visible names or designators, therefore you will need to hide them (*hint*: double-click on the pins to edit their properties).

Important: as already mentioned during the tutorials, pins only have a single possible connection point. They have therefore to be correctly oriented. When adding a pin, it will have a visible designator (along the pin) and a name (near it):



The side where the connection is possible (which must be directed towards the outside of the component symbol, so that wires could be connected to it) is the one *opposite* to the pin's name (see for example on the MIC4680 symbol, names are *inside the rectangle* representing the component, therefore their connectable sides are outside). If you look close, the connectable side has a tiny white marking inside.

1.4. Associating footprints with components

Download the PCB libraries from Altium's website: http://valhalla.s3.amazonaws.com/AD10-Libraries/PCB.zip. Extract the ZIP file to a folder you will use for Altium libraries, then open the *Components* panel in Altium, click on the ≡ menu, and open the *File-based library preferences* window. From there, you could install the library named SOIC_127P_M.PcbLib[1]. Make sure you select *All installable libraries* as file type in the dialog box, so you can see the PCB footprint file.

Once you installed the library, you can return to the MIC4680 symbol, and add a new footprint link to it: SOIC127P600-8AM.

[1] The _M suffix in the file name refers to a *low density* (easier to solder) library. There are also equivalent libraries with _N (*medium density*), and _L (*high density*, mainly for machine mounting) suffixes.