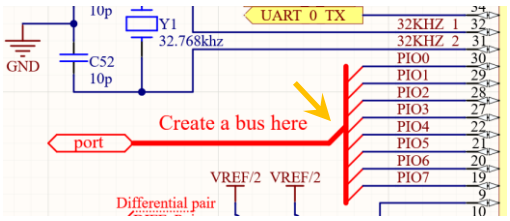
Construction of electronic systems

Exercise 4: USB DAQ project
Buses, harnesses and footprints

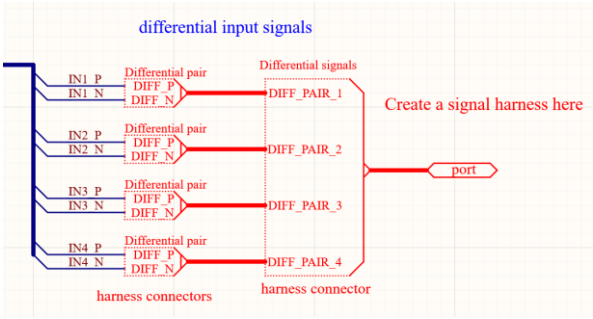
In this exercise you will continue to finalize the USB DAQ schematic. First you will add *buses* and *signal harnesses* to the schematic in order to *complete the connections* between system modules. Then you will create a PCB document based on the finalized circuit schematic. At this point you will notice that some of the components are missing their footprints. You will design the missing footprint yourself, add it to the project library and update the design. At the end of the exercise, everything will be ready to start working on the actual PCB design.

Exercise tasks:

1. ***Use a bus to connect*** **the programmable input/output signals** `PIO1...PIO7` from the microcontroller to the input/output connector of the USB DAQ board. The bus connection is indicated using red drawing lines (see below).



2. ***Use a <u>nested</u> signal harness to connect*** **the four analog differential pairs** `INx_P` and `INx_N` from the I/O connector to the input amplifiers and then further to the microcontroller analog-to-digital converter pins. The signal harness connection is indicated using red drawing lines (see below).



Note that the signal harness connections are *nested*: the "`Differential signals`" harness contains four "`Differential pair`" signal harnesses.

Hints: you need to define a signal harness connector *only once*! Then you can copy this harness connector and use it elsewhere. Alternatively, you can use this same harness connector from the "Place → Harness → Predefined Harness Connector". You can flip the connector using the "x" key while moving the connector. Do not forget to define the name of the harness connector!

Hint: use ALT+[click on a wire] to *highlight the connection*. This is how you can follow the flow of the signal through the schematics and double-check the connectivity. Use SHIFT+C to return to the normal mode.

3. **Add a new PCB document to the project and *update the PCB design using the schematic*.**

   After adding the missing connections from the tasks above, the schematic of the USB DAQ is finished. When updating the PCB design, take special care to *notice the errors* after executing the design update. Try to figure out what caused these errors.

4. **D*esign the missing footprint* for the linear voltage regulators** MC78L12ACDG and MC79L12ACDG.

   To be more specific, design the footprint for the SOIC-8 component package. Use the *"Altium Designer IPC Compliant Footprint Wizard"* and the SOIC-8 package dimensions described in the component datasheet (see the eFE attachments folder). Create and store the footprint in the ***footprint library*** "DAQ_REV_B.PcbLib".

5. ***Update the existing project component library*** "DAQ_REV_B.SchLib" by adding the missing footprint to the problematic voltage regulator components.

6. ***Update the problematic components in the schematic*** with the updated version of the component from the library.

   Hint: Use the "Tools → Update Selected From Libraries" function.

7. ***Finally, update the PCB design*** so that the missing footprints appear in the PcbDoc file.

   This time there should be no errors after executing the changes. Now everything is ready to actually start designing the PCB.

Explanation of the exercise

In this exercise you will learn how to *group wires* into *buses* and *signal harnesses*. Buses and signal harnesses provide and elegant way to connect *a large number of signals* from one part of the schematic to the other part of the schematic, while *still maintaining the graphical representation* of the *"signal flow"*, but without the cluttering of the schematic with many wires.

The exercise will also show you a quite common situation where the circuit schematic is finished, but when moving further on to the PCB design you realize that some of the component footprints are missing. You will learn how to use the Altium's *IPC compliant footprint wizard* to quickly create a footprint for a *standard* integrated circuit package.

Preparation for the lab. exercise

In order to prepare for this exercise you should see the part of the Altium Designer video tutorials that cover the more advanced schematic design. The key tutorials for this exercise are:

- Altium Schematics #01: Hierarchical schematics design
- Altium Schematics #05: Example of multilevel hierarchical design
- **Altium Schematics #04: Using signal harness in hierarchical design**
- Altium Schematics #07: Horizontal and vertical connectivity
- Altium Schematics #08: What gets on my nerves

- **Altium Designer Tutorial 6: How to use the Bus, Port and Bus Entry in the schematics**
- **How to use Harnesses in a Hierarchical Design**