Workshop 3 - DSO Top Level Design and Project Introduction

Introduction

In this workshop we will cover:

- Project introduction
- Altium sheet symbols and hierarchical design
- DSO project research and development

Learning objectives:

- Create a hierarchical block diagram within Altium using sheet symbols
- To become familiar with the Altium Designer schematic capture environment
- To design using a specification as a guideline
- To understand the methodology of datasheet based research
- To generate experimental analog designs ready for simulation

This workshop naturally follows on from Workshop 1 and Workshop 2. You will become aware of the difficulty involved in developing a working manufactured product through the R&D process, so if you have not completed the previous workshops or were not paying full attention, it's highly recommended you begin there.

Note: In 2018 this project will be much closer to a real representation of a typical product development relationship between an engineer and a client (i.e. the class leaders). To reiterate, demonstrators and the coordinator will **not** answer any questions you have - they will guide you to the answer. You are provided with all the material necessary to build a working product from the ground up, and will be expected to do so with minimal input from the client other than specifications.

As such, ELEN90053 should no longer be considered an easy subject. You are required to spend a lot of time and effort in research, design, development, and troubleshooting when things do not work as expected - just as in the real world.

Workshop Tasks

1. Download the DSO Altium template from the LMS Project page.
2. Read and understand the DSO Specifications and Introduction documents.
3. Read the restricted BoM (unless you are purchasing your own components as per the advanced track) and the included component datasheets
4. Open the Altium project file DSO.PrjPCB.
5. Examine all schematics.

   You will find details missing for,

   a. DSO_Top – Requires completion of the sheet symbol connectivity diagram.
   b. DSO_Front – Requires the design of the scopes analog front-end circuitry.
   c. DSO_ADC – Requires the design of level shifting, ADC voltage reference, and antialias filtering circuitry.
   d. DSO_Trig – Requires the design of simple triggering circuit including digitally controlled analog voltage reference generation, DSO input level comparison circuitry and a logic level shifter.
   e. DSO_Clock – Requires the design of a clock generation circuit for ADC and RAM synchronisation.
   f. DSO_Control – Requires wire connection to appropriate IO pins of the microcontroller.
   g. DSO_Power – Requires the design of a ±5V supply for your DSO.
   h. DSO_IO – Requires data isolation and USB connector.
   i. DSO_CPLD – Does **not** require any changes. This block control the clock logic for the DSO. **See the specifications document**. Must be understood.
   j. DSO_Counters – This circuitry has been upgraded with a CPLD replacing the logic gates and flip-flops.
   k. DSO_RAM – This circuitry is complete but must be understood.
6. Create a top level sheet symbol schematic as follows,
   a. Double-click on the file DSO_Top.SchDoc within Altiums project explorer (left pane).
   b. Right click on the open schematic DSO_Top in the schematic page area.
   c. “Select Sheet Actions” -> “Create Sheet Symbol from Sheet or HDL”.
   d. Select a schematic sheet and click OK.
   e. Do this for all remaining project schematics, except for the CPLD sheet, which is already in the Counters sheet.
   f. The corresponding ports for each sheet will be automatically generated from their respective schematic files.
   g. You must connect up these sheets for correct operation of your DSO.
   h. You will need both wire and bus connections to various sheet symbol ports.

7. Explore the projects schematic library and start to understand what each component does and how it functions. Examine the IO and power requirements of each IC. If you think you know which schematic sheet to place the components on start filling in the blanks including the analog sections which we have already covered in class.

   NOTE: You will need to make sure your library is up to date - as in the real world.
8. Over the coming weeks we will address any gaps you have in your knowledge so that it is reasonable to expect that **you should be able to complete all schematics unassisted.**