

# CC2511 Assignment 1

## Task Description

You are to draw up a supplied circuit in Altium Designer. The objective of this task is to learn how to use Altium Designer to reproduce an existing design. The circuit you need to draw is provided in PDF format on LearnJCU. You will need to produce both the schematic and PCB layout. We will manufacture the board as per your drawings, and you will then need to solder the components to produce a working product.

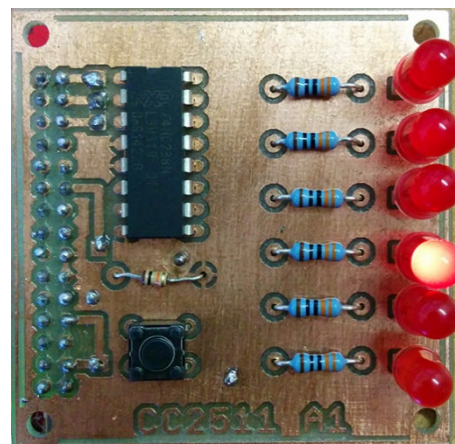
The circuit design is an electronic dice game that lights up a random LED when a button is pressed. It plugs into a FRDM board, allowing your software to detect when the button is pressed and then select a random LED to light up. The LEDs will be driven through a 3-to-8 line decoder so that three digital outputs can control up to eight LEDs (although we only use six).

Your hardware is to be a 50mm x 50mm square circuit board. It will connect to your development board via the headers that provide a power pin, which are:

- J9 and J10 on the FRDM K20D50M, or
- J24 and J25 on the FRDM K22F.

It will hang over the side of the development board so that you don't need to worry about your components hitting any jumpers on the FRDM board.

The photograph shows the finished product, as fabricated using a milling machine. Note that you are required to use a mix of through-hole and surface mount resistors, whereas the photograph shows only through-hole components.



## Due Dates

Start drawing in Altium	Week 2 laboratory
Electronic copy of your Altium project submitted via the LearnJCU drop box.	Wednesday 14/8/19 (week 3)
Approximate date from which manufactured boards will be available for collection from the workshop.	Monday 26/8/19 (week 5)
Soldering of components.	Any time before Friday 30/8/19 (week 5).
Development of software to drive the LEDs.	Any time before your week 6 lab.
Demonstration of your finished product.	During your weekly lab in week 6.

Note: you should aim to complete this task before you start working on Assignment 2, which begins in week 5.

## A note about PCB fabrication

**You must design your PCB for fabrication on a milling machine.** The capabilities of the JCU milling machines are documented on LearnJCU.

Note that milled boards do not have through hole plating. It is essential that you consider on which side of the board you will be soldering each component. For example, you will solder the 2x15 header on the top side of the board because the header must protrude from the base in order to plug into the FRDM board.

You must also check that there are no ground plane islands, and that the ground planes on both sides of the board are properly connected (with the use of vias).

Milled boards lack silkscreen so ensure that your name is written on the top layer (not the top overlay).

## Learning Outcomes

This is a formative assignment designed to teach you how to use the Altium software to prepare printed circuit boards, and to give you experience with circuit board manufacturing and prototyping. The criteria that are listed below will be applied strictly in order to teach “best practices” for future circuit board designs that you produce for the rest of your degree.

## Reference material

The following reference material is provided on LearnJCU:

- The reference design in PDF format, and
- The capabilities of the PCB manufacturer.

## Assessment Criteria

The assessment is on the basis of a Satisfactory or Unsatisfactory design. All of the categories below must be marked as Satisfactory to receive an overall Satisfactory grade.

Component	Satisfactory	Unsatisfactory
File names	The file names for the schematic and board file are clearly labelled with the student's name, subject code, and assignment number. Example filename: "BronsonPhilippa_CC2511_A1". Use initials if necessary. (This is to assist the electrical workshop who would otherwise receive 50 files called "Board".)	The file names for the schematic and board file are not clearly labelled with student's name, subject code, and assignment number.
Schematic	The submitted schematic accurately reproduces the reference design. The components and electrical connections are correct. The drawing is neat, legible, and of an overall professional standard.	The components and/or electrical connections are incorrect. The drawing fails to meet reasonable professional standards.
Circuit board layout	The submitted circuit board is suitable for immediate manufacturing on a milling machine. All electrical connections are present. Clearances, drill sizes, and track widths all match the capabilities of the PCB manufacturer. There are ground planes on both side of the board that are correctly connected without islands. Your name or initials are visible on the board on the top side copper.	The submitted circuit board cannot be manufactured until revisions are made, or electrical connections are absent or incorrect.
Product (including final soldered circuit board and software)	The finished product works correctly and satisfies the requirements of the task.	The finished product does not function correctly.

If an Unsatisfactory grade is received, students will be provided with feedback and required to revise their design accordingly. The revised design must be resubmitted as soon as possible. There is no limit on the number of re-attempts which may be made. All students must receive a Satisfactory grade before the end of semester in order to pass the subject.