Altium Tutorial

Introduction

This is not a tutorial to teach you all about PCB design, this is just a short introductory lesson to introduce you to Altium Designer 6. Before you start making your own PCB, you should go through Altium's 'TU0117 - Getting Started with PCB Design.pdf'.

When Altium is first opened the Home page is displayed. If you accidently close this page and have to access it again, close on this icon in the top right hand side of the screen. You can access help documents and such from the home page.

Creating a Project

Before we start with our design, we need to create a Project which will house all of our relevant files (Schematics, PCB files, libraries etc).

- Create a project by clicking **File >> New >> Project >> PCB Project**

Before we do anything else, we should save our project.

- Save your project by right clicking on it in the Projects Panel and selecting **Save Project As.**

Creating a Circuit Schematic

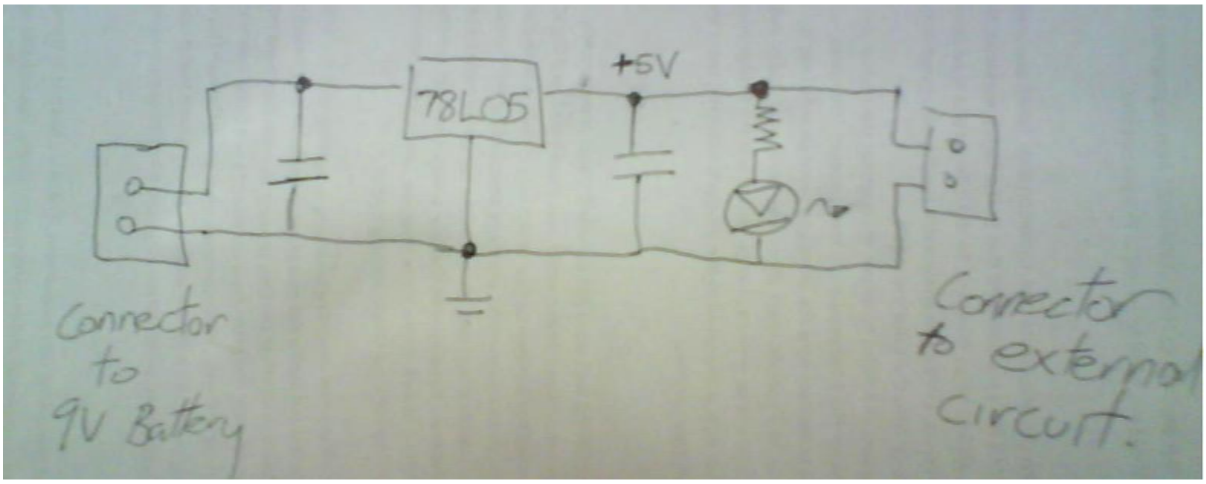
Now that we have a project, let's add our circuit schematic file to it:

- Create a schematic by clicking **File >> New >> Schematic.** Check the projects panel, if the schematic file isn't included under your project, then click and drag it into your projects source files.
- Save your schematic file by right clicking on it in the Projects Panel and selecting **Save As.**

Let's start populating our circuit schematic, we're going to build a very simple 5V power supply using a LM78L05 3-Terminal Positive Regulator.

- Using your favourite web browser, open the following link: http://www.national.com/mpf/LM/LM78L05.html

We're going to wire up the following circuit:



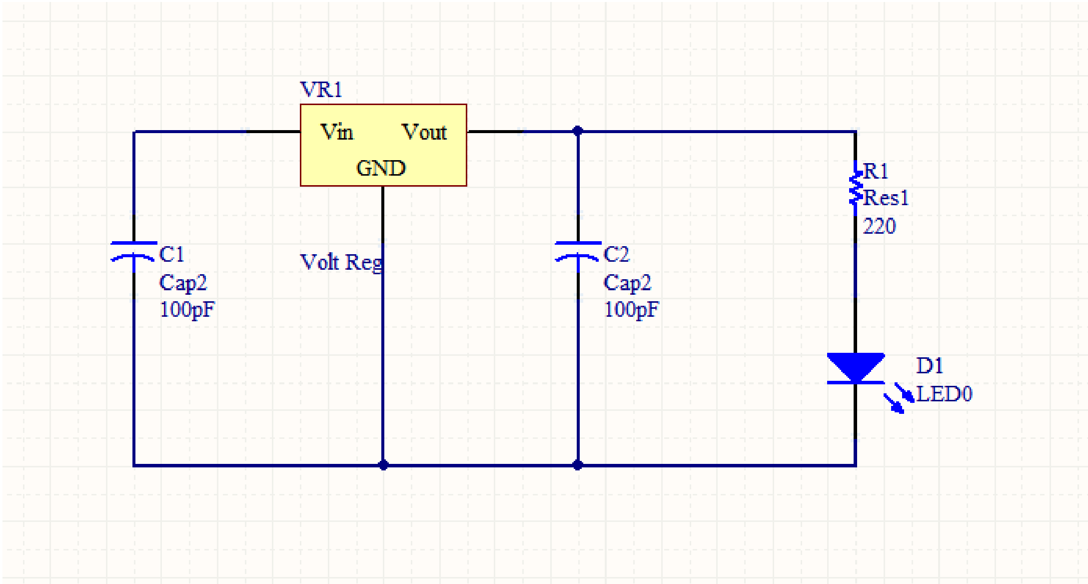
Let's ignore the connectors for now, and wire up the rest of the circuit:

- Click on the **Libraries** tab on the right hand side of your screen to access Altium's Schematic libraries. This is were we will our parts.
- Change the active library to **Miscellaneous Devices.**
- Let's place the resistor first, Search for **res1.** Select res 1 and click **Place Part.**
- Before we place the resistor, hit the tab key to edit its properties. Give your resistor a sensible ***Designator*** and ***Value,*** and take note of the ***Footprint***, but we'll come back to that soon***.***
- Now let's place our part, hit the space bar if you want to rotate it before placement.

Fantastic! Now let's repeat these steps for the rest of our components (except the connectors). You'll find the capacitors under the name **Cap** or **Cap2** (use cap2)**,** the voltage regulator under **Volt reg,** and the LED under **LED0.**

Now for a quick note about wiring schematics. Placement, Nets, Values, Manual Junctions.

At this stage, hopefully your schematic will look something like this:



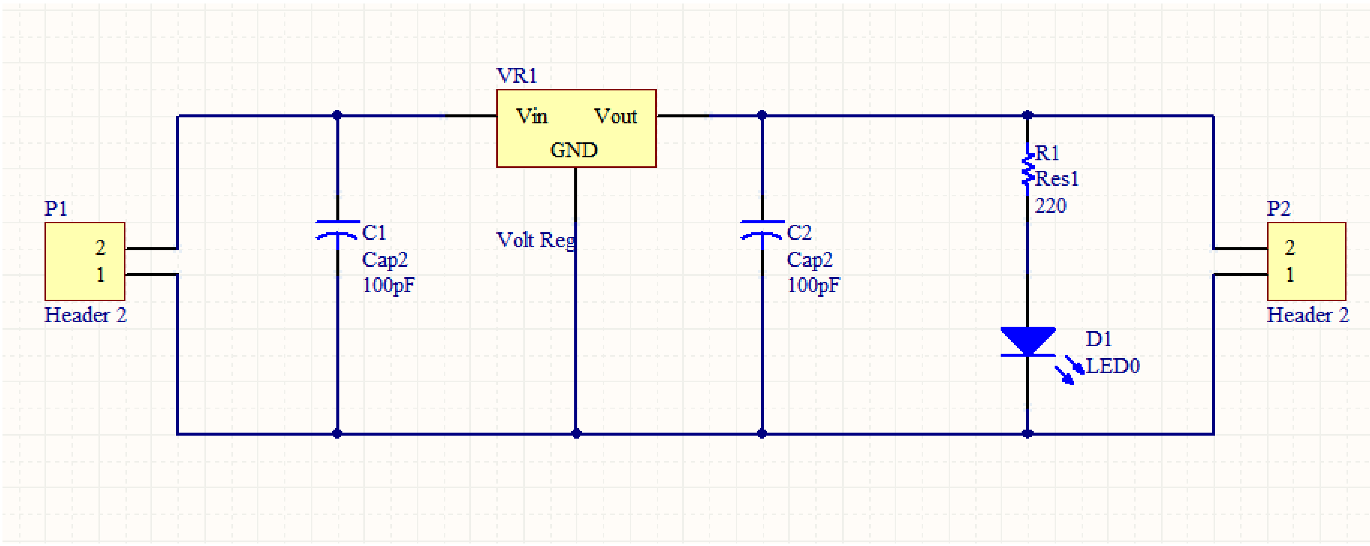
Now let's add our connectors:

- Change the active library to **Miscellaneous Connectors**
- Find the part **Header 2**, place two of these on your schematic. *Try to keep GND pins as the same pin number.*

It is a smashingly good idea to use headers in your PCB designs, especially when connecting to things not mounted to your PCB (push buttons, leds, etc). We'll be telling you more about making connectors and cables during the construction lecture!

In addition, headers can be used for debugging! Eg, toggling a microcontroller pin to check if you've fried your poor AVR or not! (Just as well you budgeted for a spare, right?).

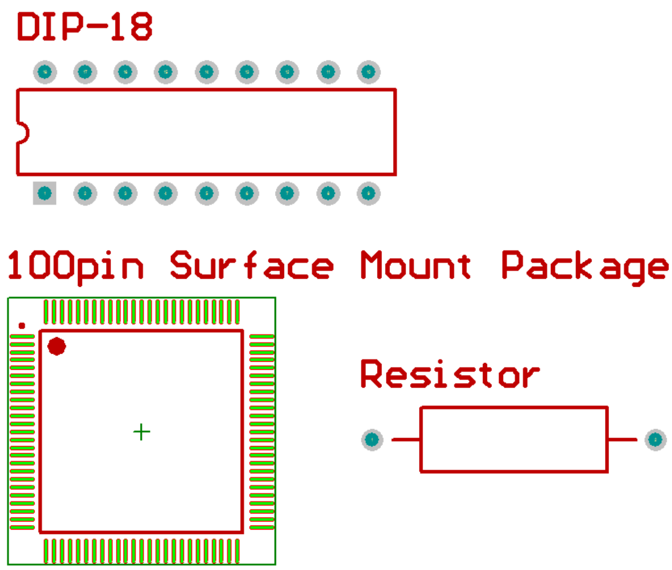
Right now your circuit schematic should look like this



Preparing the Schematic for the PCB Design

Now that we've wired up our schematic, it's time to prepare it for transferral to a PCB.

Each part that we have used in our schematic gets assigned a ***footprint.*** The footprint shows how a component will be mounted on to the PCB. Here's some example pictures of footprints.



Note the drill holes, pads, overlays.

We need to make sure that the footprints we assign to our parts are correct, Altium won't know the difference!

EG. Make sure you don't assign a surface mount footprint for a through hole component. Make sure your drill holes are large enough for your component.

Scared that you're going to make a mistake on your PCB by assigning the wrong footprint? Not a problem, ***print out*** *your PCB and lay down your components on top of the print out.*

Let's now assign some footprints to our components.

- Double click on your resistor, select the ***Footprint*** box and click ***EDIT.*** Change the PCB Library to **Any.** Now hit the **Browse** button and change your resistor's footprint to AXIAL-0.5.

- Let's repeat the same step for the capacitors, but the footprint we want to use doesn't come with Altium (oh no!).
    - Download the Altium libraries from the Electronics Workshop webpage.
    - For this example we only need the KBpcb02 library, save this file to a suitable directory.
    - Now, under the ***libraries tab***, hit the ***Libraries*** button.
    - Now we're in the available libraries section, click the ***Add Library*** button.
    - Find the library you've just downloaded, and add it to your project!
- Now edit your capacitor footprint, and change it to the **RB5.0** type footprint from the library we've downloaded.
- Change the Voltage Regulator footprint to the **TO-92** footprint (under Misc Devices). *What's the difference between the TO-92 and the TO-92A footprint?*
- Leave the other footprints as default.

Creating our PCB

Now it's time to create our PCB document.

On the left hand side of your screen, change to the files tab. Under 'New from Template' hit the **PCB Board Wizard,** if you can't see this button, shrink some of the other sections.

- Select the ***Imperial*** option, we can change this later to metric if we want to (for drillholes we're going to use metric). Hit next.
- Select custom, hit next.
- Change your board size to whatever you like, perhaps 2000mil by 2000mil. (NOTE: 1/1000th of an inch = 1 mil = 1 thou ≠ 1 mm)
- Keep the signal layers at 2, change the power planes to 0.
- Through-hole vias only.
- Through-hole components mostly, one track between adjacent pads.
- Leave these values.
- Finish!
- Save your PCB document. Make sure it's included in your project documents.

Now it's time to transfer our allocated footprints on to our PCB.

- Open your circuit schematic file, then under the **Design** menu, hit **Update PCB Document blah.PCB**.
- Mash the **Execute Changes** button.

All of our parts should now be on our PCB. You can delete the overlay covering your parts.

- Position your parts on your board.

*Double Check some of your footprints by using **Reports >> Measure Distance***

Let's route our PCB. Routing is the process of laying the tracks which connect up all our footprints.

- Hit the **Interactively Route Connections** button (shortcut P then T).
- Start routing your board!
    - ***Space bar*** changes the orientation of the track.
    - ***L*** will change the layer it is on (Top or Bottom in our case).

Congratulations, you've wired your first PCB. But wait, we're not done yet. We need to make sure that this board is suitable for manufacturing, otherwise it won't be sent to the manufacturer.

Preparing our PCB for manufacturing

ENGG2800 has strict rules that need to be followed in order for your PCB to be manufactured. If you don't follow these rules, your board will not be sent to the manufacturer.

- Navigate to the ENGG2800 website and find the PCB Page.
- Go through the ***Checklist for Submission***, make all these changes to your PCB.

Now let's have a look at our drill holes. But first we'll have a chat about how these holes are drilled by the manufacturer.

- On the left hand panel, open up the PCB tab. At the top of this tab, select **Hole Size Editor** from the drop-down menu.
- Hit ***Q*** to switch Altium to METRIC.
- Consolidate the drill-hole sizes (use as few as possible).

One final note

In this tutorial we have only touched the tip of the iceberg; there are a lot of topics which have not been covered in this very short tute. You still have a LOT to learn about PCB design.

You should set aside some time to go through Altium's 'TU0117 - Getting Started with PCB Design.pdf'.

The Altium lecture will cover some very good design practices and tips, you should attend this too.