

Altium Designer Tutorial

Getting started setting up project files

Putting all files within a project allows them to be linked together.

File->New->Project

Choose <PCB Project>, filling in the location and Name. Click <OK>

File->New->Schematic

File->New->PCB

File->New->Library->Schematic Library

File->New->Library->PCB Library

Hover over each of the 4 items you have created and <Right Click> choosing <Save As>.

Schematic Library Editor

This is where we can define our own schematic symbols.

Place->Pin

Place->Rectangle

Select the drawn rectangle by clicking on the yellow area.

Edit->Move->Send To Back

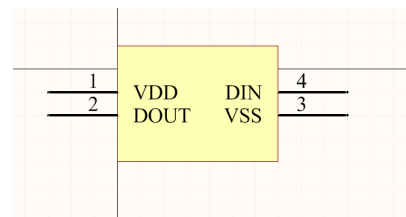
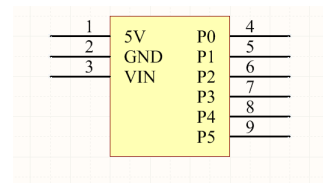
<Space bar> to rotate by 90 degrees, x to mirror pin in x plane

Come back later and add Footprint

Tools->Rename Component

Tools->New Component

Tools->Update Schematics



PCB Library Editor

This is where we can define our own PCB footprints.

<Right Click> on main page and choose Snap Grid to suite your design.

Place->Pad

To change properties of the pad press <Tab>

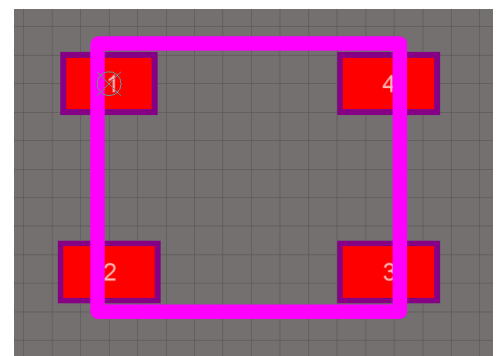
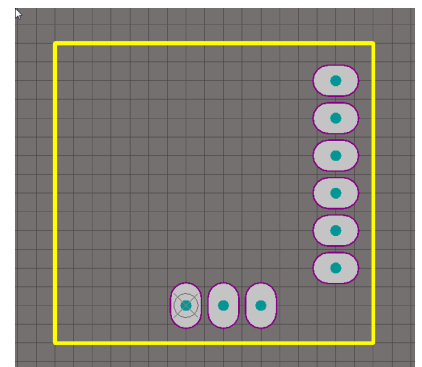
From here hole size, designator, layer, pad size and shape can be set.

Place->Line

Edit->Set Reference->Pin 1

Tools->Component Properties to set component name

Tools->Update PCB With Current Footprint



Schematic Design

Place the following components using **Place->Part**

Parts....

Diode->Miscellaneous Devices.IntLib

LED2

Res2

Header 2->Miscellaneous Connectors.IntLib

Header 5

The following component is from an added library

MC78M05ACDT->ON Semi Power Mgt Voltage Regulator.IntLib

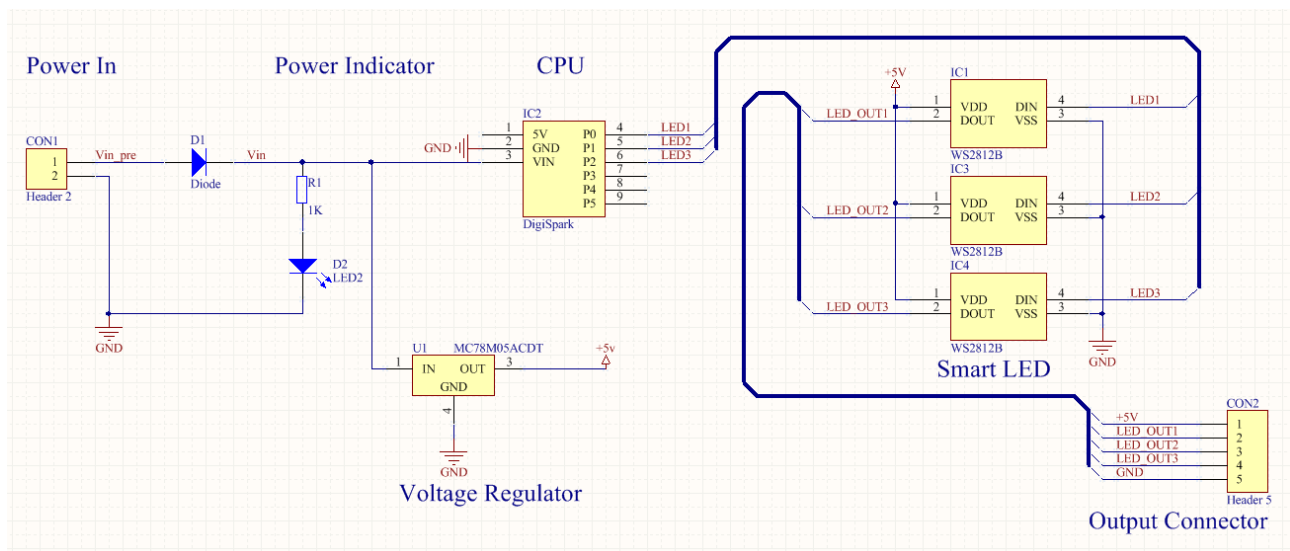
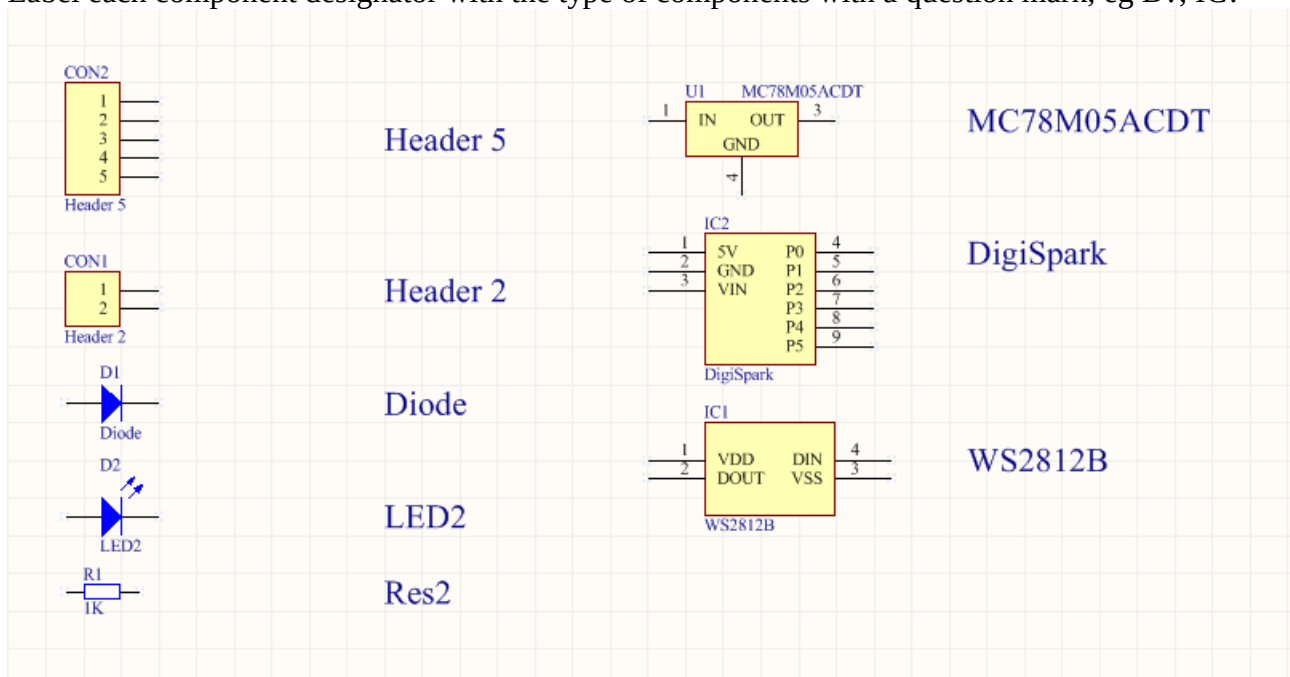
The following two components we will make ourselves.

WS2812B->ENMT301.SchLib

DigiSpark

Place components as shown in schematic.

Label each component designator with the type of components with a question mark, eg D?, IC?,



Annotate Schematic

This is a process where each component is given a unique name.

Tools->Annotate Schematic

Click Button Update Changes List

Click Button Accept Changes

Click Button Execute Changes

Click Button Close

Click Button Close

Linking design to PCB

This is a process where changes made in schematic are transferred to the PCB design

Design->Update PCB

Click Button Execute Changes

Click Button Close

PCB Design

+ Key cycles through layers

Pressing ~ Will show a list of current available commands

Place->Interactive Routing

Place->Via

Place->Line

Place->String

Home to centre screen on cursor, Page Up to Zoom in, Page Down to Zoom out

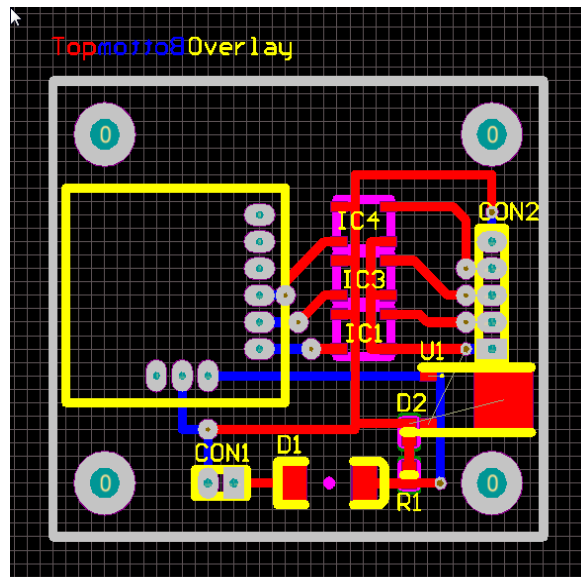
The top layer is RED. The bottom layer is BLUE.

Design Rules

Design->Rules

Most useful are Width, Clearance, and HoleSize.

Tools->Design Rule Check->Run Design Rule Check



Short-cuts Panel activator found on bottom right hand side.

Usefull Key Combinations

Shift + C	Clear current filter / mask
S, A	Select All (press S then A)
X, A	Deselect All
Z, A	Zoom All
Page Up/Page Down	Zoom in/out
Ctrl + Mouse wheel	Zoom in/out
Right Mouse Button	Pan
+ / -	next /prev layer
*	next signal layer.
Q	toggle units (mil / mm)
Backspace	Undo last wire/track.
Space	change wire/track mode

General Design Notes for Mech PCB Fabrication

Draw a board outline using Multilayer.

Place text on each layer(top, bottom, overlay), the bottom layer text is reversed in the **x** direction.

All through holes are 0.5mm(This allows the drill bit to auto centre).

Make pads as big as possible, pad should be twice the size of the hole.

Make tracks as fat as possible.

Thick about how you are going to mount the board and place these holes in the design.

Holes are NOT plated through. So through hole components, especially connectors should have tracks joining from the bottom layer.

Further Getting Started Tutorials

<http://techdocs.altium.com/display/ADOH/Tutorial+-+Getting+Started+with+PCB+Design>