

Hello and Welcome

As we are slowly moving back to instructed learning (which I prefer) there are still time complications that prevent the contact time necessary to make it fully instructed. This is my running document over the years.

These documents cover all the material and more for what you would need to achieve 100% in the MEC1003F course provided you are careful. Designing hardware is very detail specific and a willingness to deep dive is vital. Please start by looking at the first link in the following sentence if you are a 1st year.

With the current context of COVID19, I have decided to link useful material [here](#).

Only proceed further if you choose to do so...

The following notes are made for anyone wishing to take their PCB design further, if you are part of the 1st year class, we will go through this material as we progress through the class, but I will keep these notes up if any of my other students stumble across the material or a 1st year is especially keen.

Simplified KiCAD Module

It would be foolish to make notes/videos on KiCAD as there is the help functionality, a massive community and as new versions come out things might change slightly. V6 came out 2021 Dec 25th. So if you are stuck use F1 or Google.

However with that said there is always some underlying structure which helps with regard to good practises.

This starts out as a guide to making a simple PCB Project, but then as you need to delve deeper into the finer details - more information is linked in this document.

If you go through:

[Downloading KiCAD](#)

[KiCAD Cheatsheet](#)

[Project](#)

[Schematic](#)

[PCB](#)

[Gerbers](#)

[Assembly Files \(Optional\)](#)

[Designing for Manufacture](#)

You should have a completed PCB - If your board was simple enough.

Downloading KiCAD

KiCAD is a free service, it is cross platform and freely downloadable from [here](#). It isn't that small, so unless you have decent internet at home. I suggest you do it on campus.

The first step would be understanding the file structure

Project:

The project is effectively the container for all your documents, if you forget to add everything to a project, you cannot link your files to each other which is very NB... (When opening your work, open the project.)

1. File -> New -> Project.

In general I find if you click on the items from left to right you following the correct order.

Schematic:

1. Whenever you come back to a project or show a project to anyone else, the first point is to look at the information in the bottom right. Board Name, Who made it, which revision are you showing me, how old is it? **Board setup (2nd from left I think)**
2. Save (Ctrl+S)
3. Place your components (A)
4. Join the components together with wires(W)Wire. Use Supply symbols and Ports where possible
5. Save (Ctrl+S)
6. Think about test points, modularity, mounting holes then add them (Not necessary for 1st years)
7. Save (Ctrl+S)
8. Annotate your schematic
9. Run an ERC (Often need to set the rules for it first)
10. Assign Footprints

PCB:

This is where mechanical meets electrical, think connector placement, heating issues, ingress, good layout.

1. Setup -> Layer Setup
2. Design Rules
 - a. Width - There is no 1 size... Choose intelligently (applies to a,b,c)
 - b. Clearance Manufacturing Board Outline Clearance
 - c. Hole Size
 - d. Silkscreen

Some Intelligent numbers could be: (It really depends on PCB)

- a. 0.25mm | 0.6mm | 2mm
 - b. 0.4mm (easy board - noob soldering) | 0.25mm (difficult board - skilled soldering)
 - c. 0.5mm
 - d. 0.6mm min
 - e. Not important
3. Save
4. Import from schematic
5. Move components around find suitable modularization
6. Wire Sections
 - a. Manual Routing / Active Routing / Guided Routing
7. Create Outline (Sometimes moved to point 5)
8. Legend/Silkscreen

Gerbers:

In order to manufacture your boards, you need to submit Gerbers to the manufacturing plant. A Gerber file is associated for each process in the manufacturing step. This would typically be each layer in your stackup.

Top Silk
Top Stop/Mask
Top Cu
Edge Outline
Bottom Cu
Bottom Stop/Mask
Bottom Silk

Plated Through Hole (PTH)
No(n/t) Plated Through Hole (NPTH)
Optional:
Top Paste and Bottom Paste

Notes regarding my KiCAD [Lecture 3](#)

I made a file for historically manufacturing from a local supplier Trax.

[Assistance to Use the TraX PCB manufacturing discount](#) - Legacy

Regrettably, the cost benefits and timeframes with TraX are no longer deemed optimal and most of my manufacturing is done in China.

To do it (without plugin):

In PCB view, File → Plot → Gerber files.

In the layers tab select the layers you want.

OK (That will give you 7 files)

Then for the final file (NC Drill File)

In PCB view, File → Fabrication Outputs → Drill file.

OK (That will give you 2 files)

X is your Filename.

Convert from

Convert to

KiCAD	Altium	Trax (Legacy)
X-F.Cu.gbr	X.gtl (Gerber Top Layer)	Top Layer.gtl
X-F.SilkS.gbr	X.gto (Gerber Top Overlay)	Top Legend.gto
X-F.Mask.gbr	X.gts (Gerber Top Stop)	Top Stop.gts
X-Edge.Cuts.gbr	X.gm1 (Mechanical 1 Layer)	Outline.gm1
X-PTH.drl	X.TXT (NC Drill file)	Drill.TXT
X-NPTH.drl	(No Non Plated Holes)	NPTH.drl
X-B.Cu.gbr	X.gbl (Gerber Bottom Layer)	Bottom Layer.gbl
X-B.SilkS.gbr	X.gbo (Gerber Bottom Overlay)	Bottom Legend.gbo
X-B.Mask.gbr	X.gbs (Gerber Bottom Stop)	Bottom Stop.gbs

X-F.Paste.gbr		
X-B.Paste.gbr		

Assembly Files (Optional)

If you are taking files to an assembly plant. There are a few requirements you would need to submit.

BOM	(Schematic View)
PDF of Docs	(Anywhere)
Pick and Place Files	(PCB View)
Paste file(s) for Stencil	(PCB View)

Useful Information I have learnt through my mistakes

- Whenever you make a mistake - it doesn't matter who's fault it actually was - you look like the idiot. You are responsible for the projects you make and you need to check everything before having the opinion that everything will work.

Designing for Manufacture

Simply connecting tracks to pins might produce a PCB that works, but there is vastly more that can be done to optimise the process. I am compiling a document to highlight some of the lessons I have learnt over the years with regard to designing for manufacture.

[Help in designing for manufacture](#)

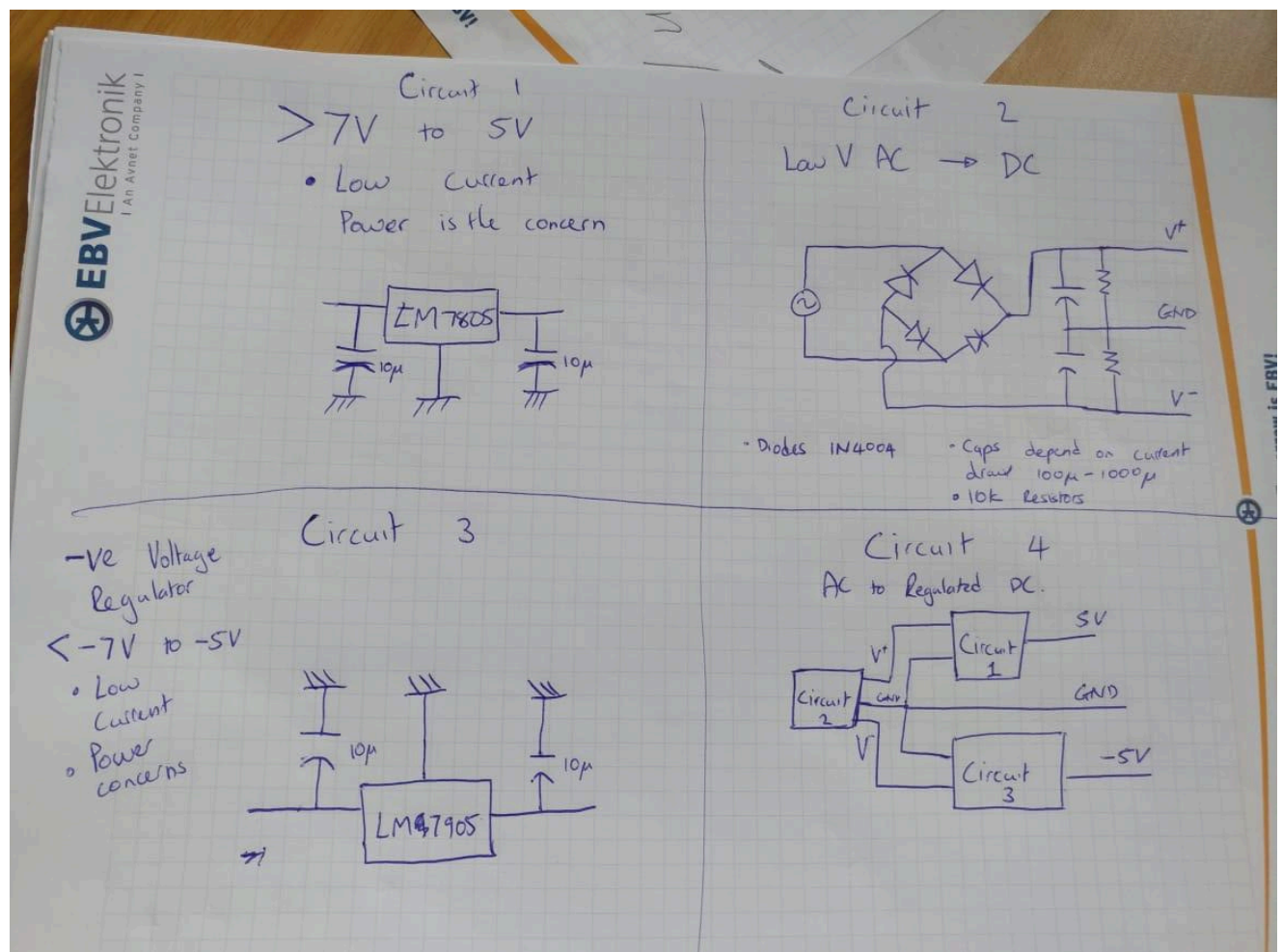
<http://www.ti.com/lit/ds/symlink/lm556.pdf>

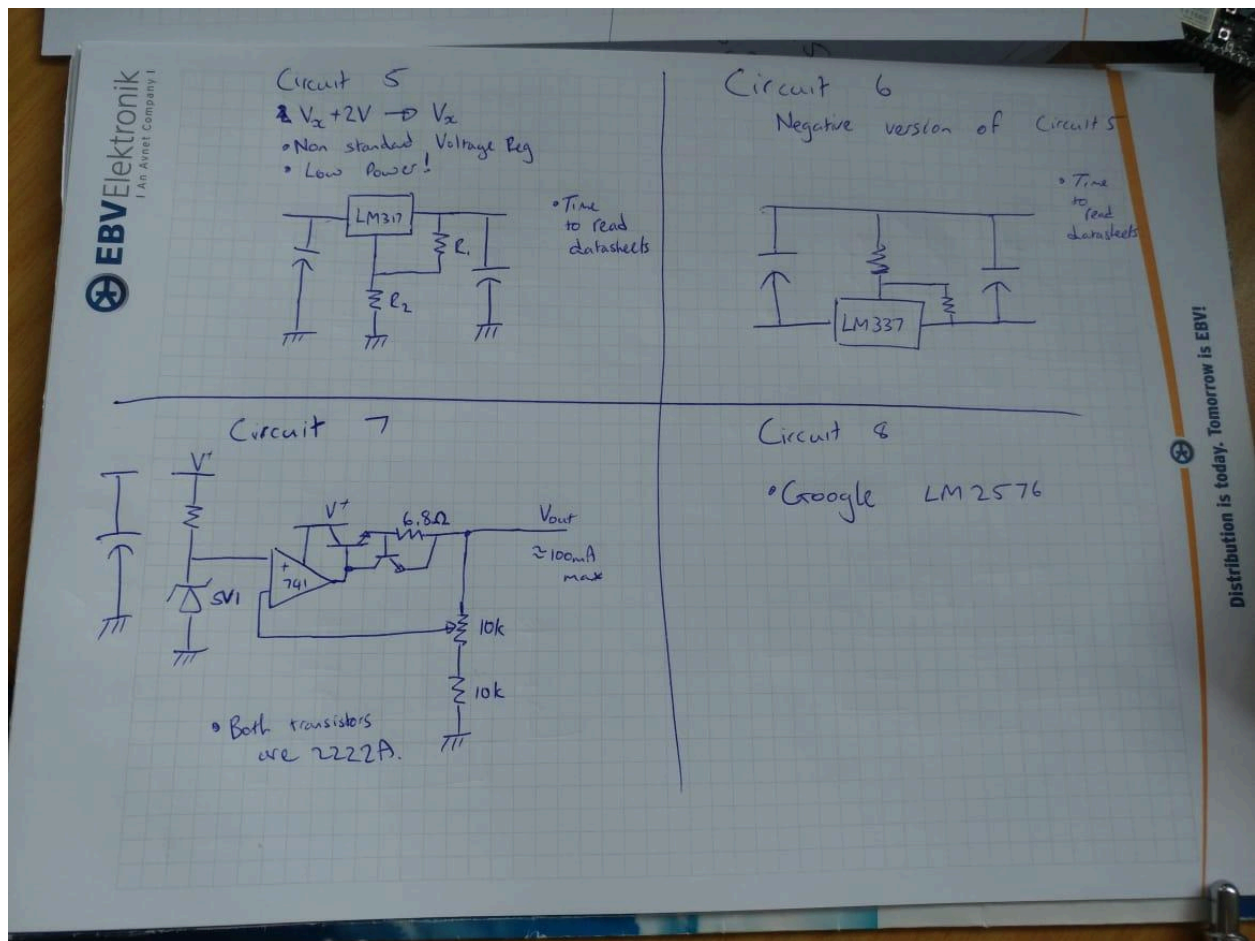
<http://www.seoulsemicon.com/upload2/STW8T16C.pdf>

2019 Session 4 - Kept here for reference

Lecture 4 - Application

This year, I thought starting the process of building a power supply would be useful. The first 6 circuits are solutions to the problem of how to make it work, however none of these circuits have the ability to survive when a mistake happens.





Then you could use the following components by still providing the necessary from Circuit 2.

OR

you could take apart a Computer PSU if you have one lying around. I have a how to on my Useful linker and then use the below links for further control.

[Purchasable Buck 1](#)

[Purchasable Buck 2](#)

Okay again these are the basics... I would STRONGLY recommend:

- fuses to ensure if/when something goes wrong the fuse blows and you don't start a fire.
- Power LEDs to confirm your voltages are working

[Voltage Monitoring](#) - Local

[Voltage Monitoring](#) - China

Could add the following device to get a readout for your voltage. Then we could also measure the current using an INA169 and measure the output voltage with the above voltage monitoring which would then be displaying the current.

Okay, I think that is enough to cover all the students in the class. Goodluck.

<https://www.youtube.com/watch?v=5Be7XOMmPQE>