Printed Circuit Board (PCB) Design

Lecture 17

Microprocessor-based Systems (E155)

Prof. Josh Brake

μP

Survey Feedback

- Most important things you've learned
  - Datasheets!
  - Understanding the "plumbing"
- Things still unclear
  - Interrupts
  - Redundancy/error checking
  - CMSIS
- Things to change
  - Start earlier
  - Refreshers

TBD lectures no longer TBD

- Direct Memory Access (DMA)
  - Load from or store to memory without using CPU
- Real-time operating systems
  - Why?
    - Multi-tasking
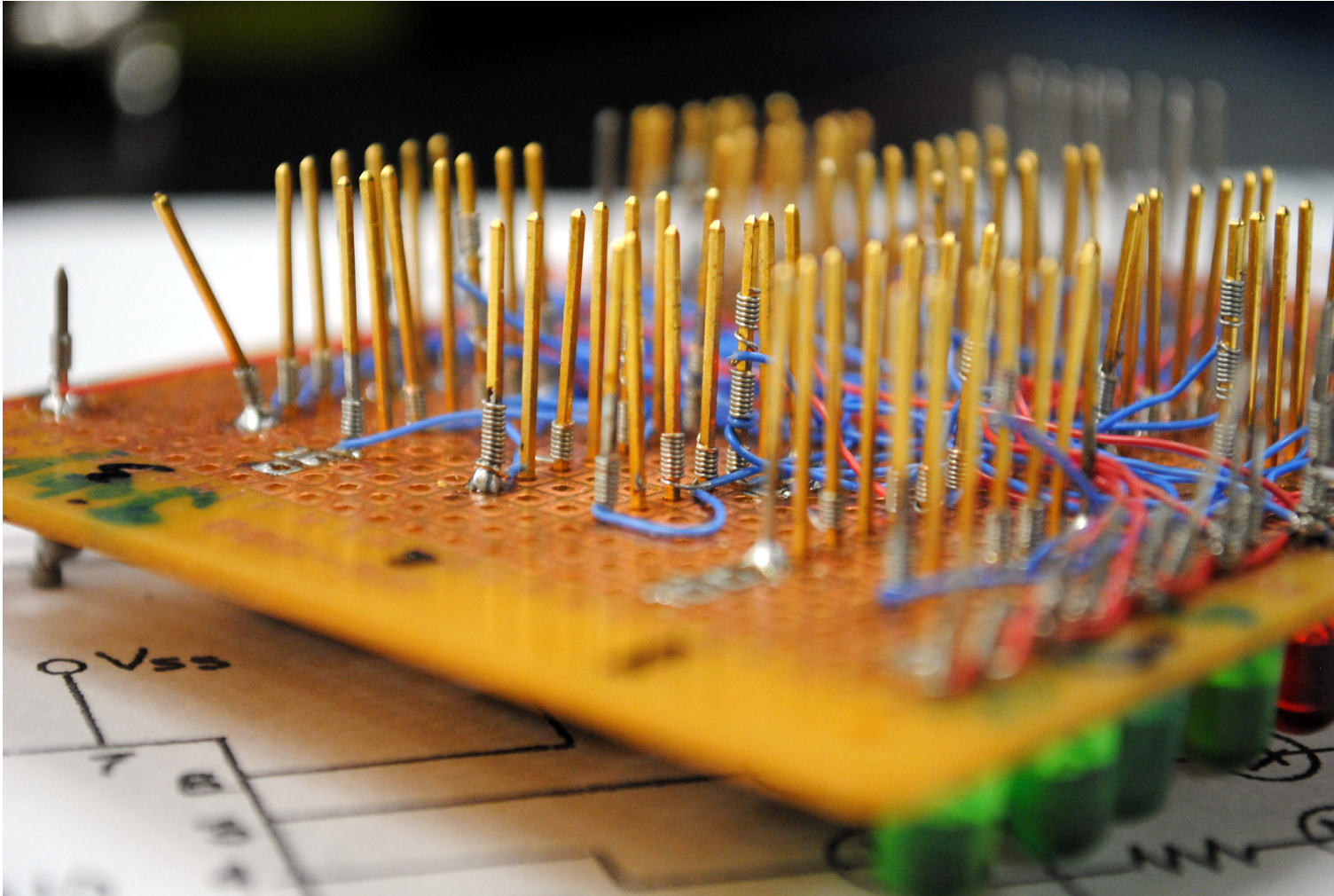   - Scheduling

Outline

- What is a PCB?
  - History
  - Components
- PCB Design process
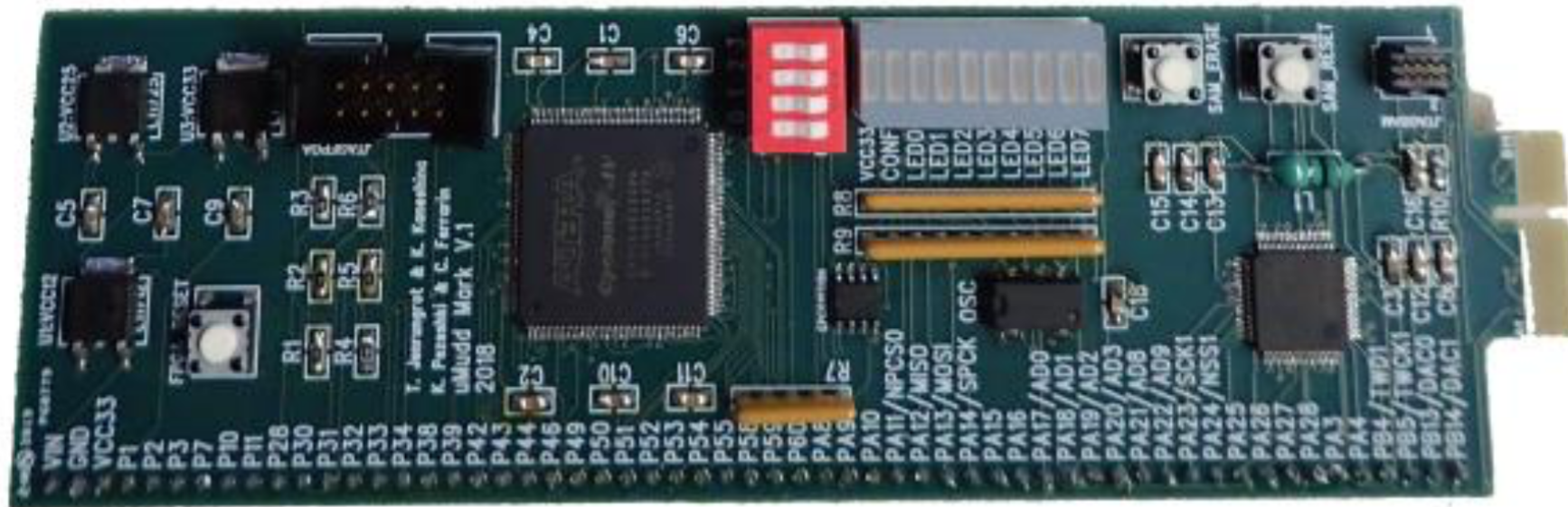- Layout considerations
- KiCad Demo
- Further resources
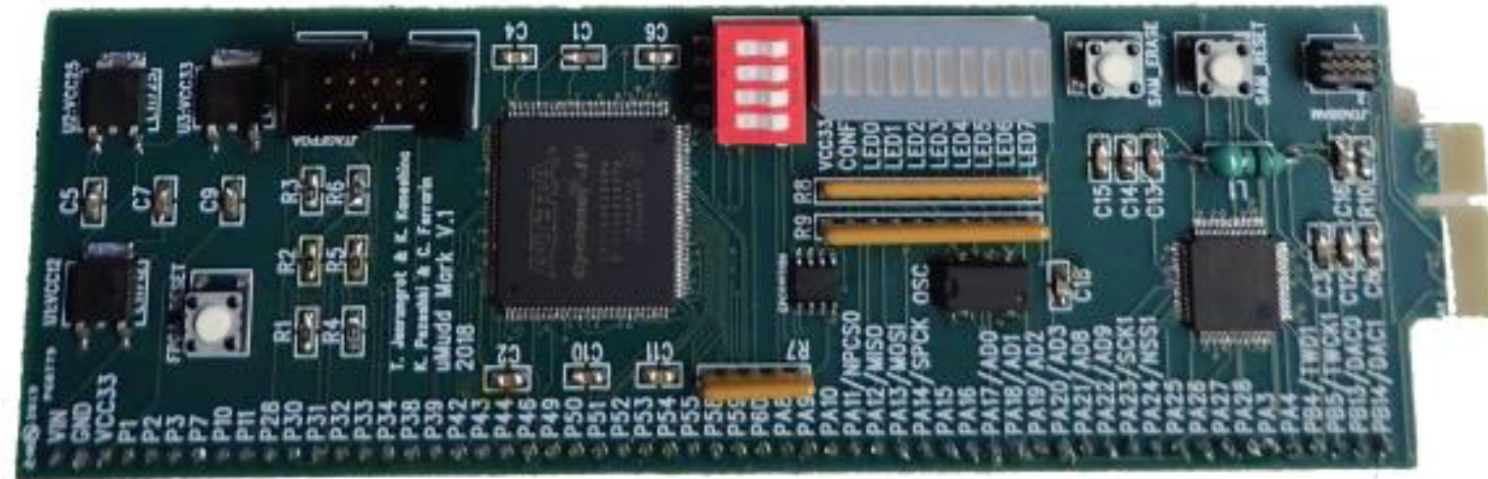
HMC2019: Cyclone IV + ATSAM4S4BA

Why PCBs?

Components

- Two main types
  - Through-hole
  - Surface mount technology (SMT)


HMC2019: Cyclone IV + ATSAM4S4BA

Helpful Terms

- Trace – copper "wire" on board
- Via – hole through the board
- Pad – exposed copper area where part is soldered
- Package – physical size of part and pins
- Surface mount – part which is attached to one side of PCB
- Through-hole – part which is inserted through PCB

PCB vs. Breadboard

- Pros for PCB
  - Stability and reliability
  - More space efficient
  - Better grounding
  - Physical mounting is more stable
- Pros for breadboard
  - Flexible
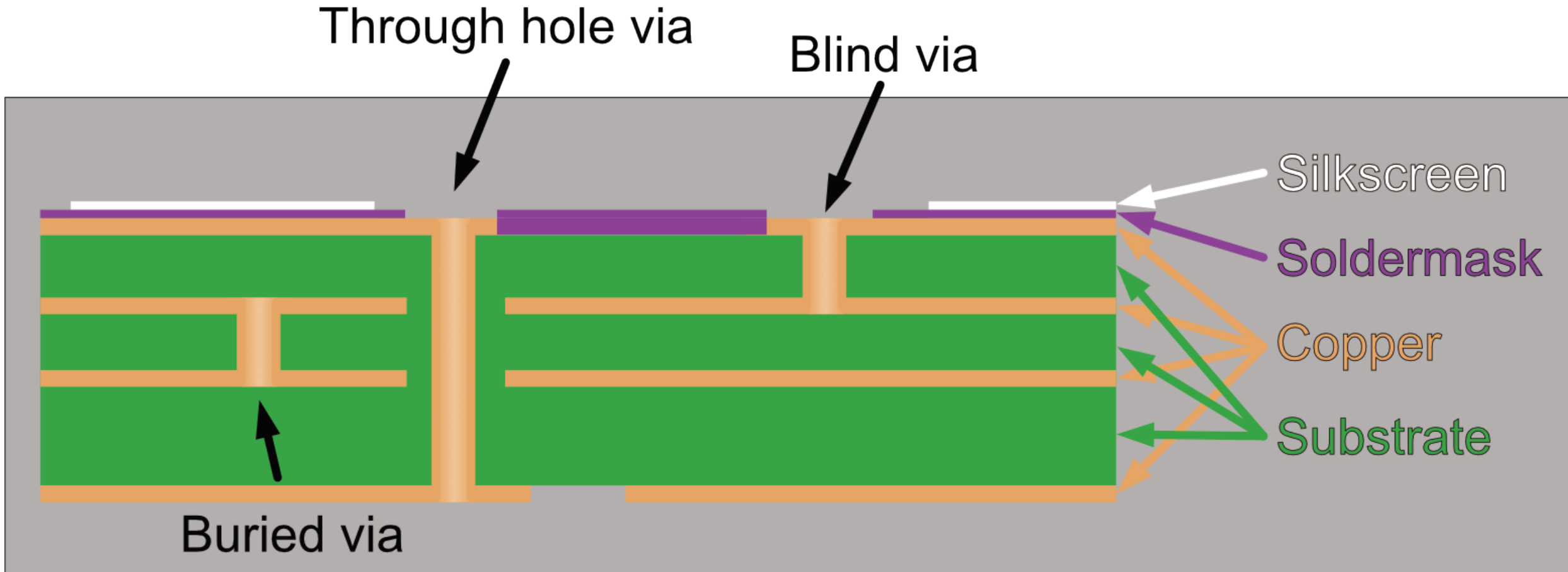 - Ease of debugging and isolating problems
  - Fast!

Ways to manufacture

- Etching
- Milling
- Printing
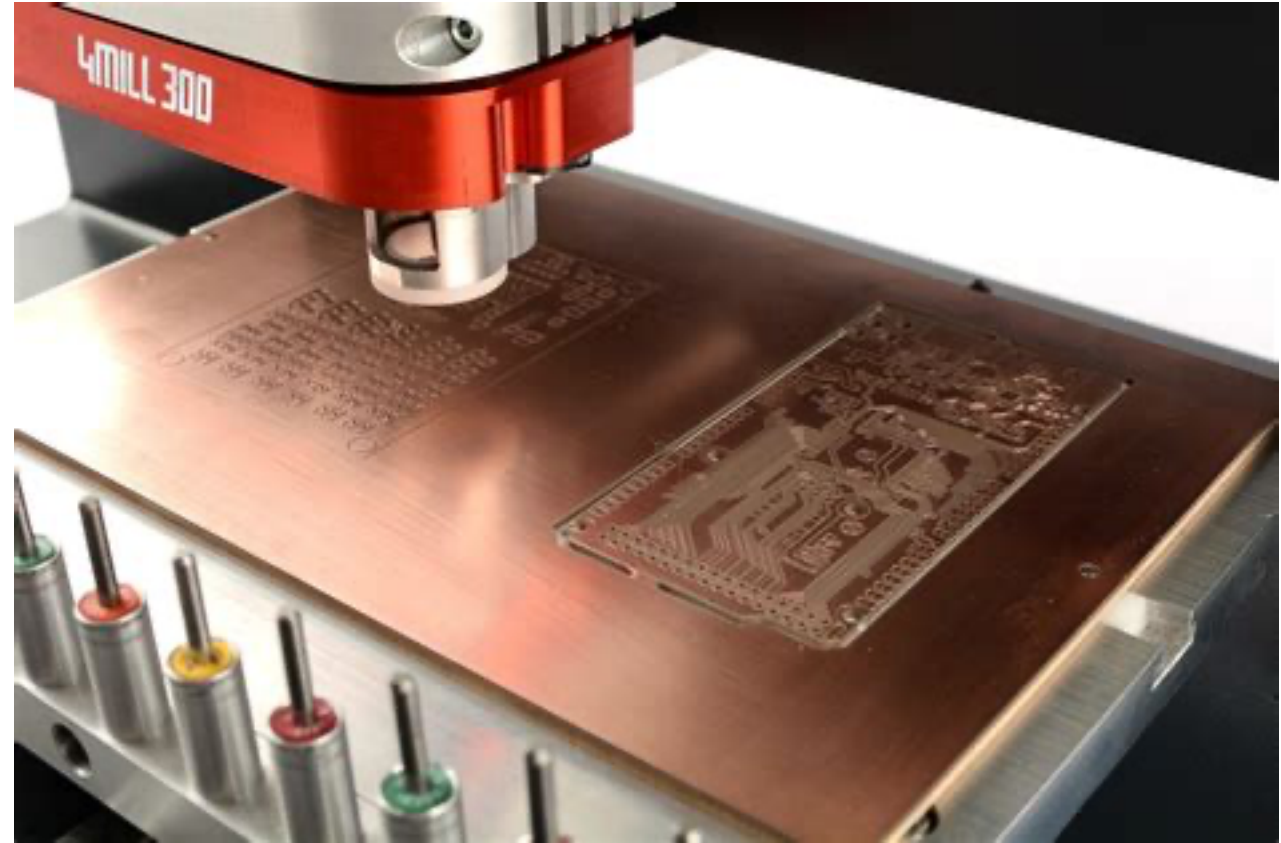
http://www.webshop.mipec.eu/fotky6870/fotos/_vyrp14_71DSC_7924.jpg

PCB Structure

Design Process

KiCad

Budget around 6-8 weeks per iteration if you are new at it.

| Step	KiCad Tool Name
Design circuit	
Create symbols	library editor
Schematic capture	eeschema
Create footprints	footprint editor
Generate netlist	cvpcb
Board layout	pcbnew
Production	GerbView
Generate BOM	Digi-Key

Create/Import Symbols

- Open eeschema
- Add/create libraries
- Select or import symbol
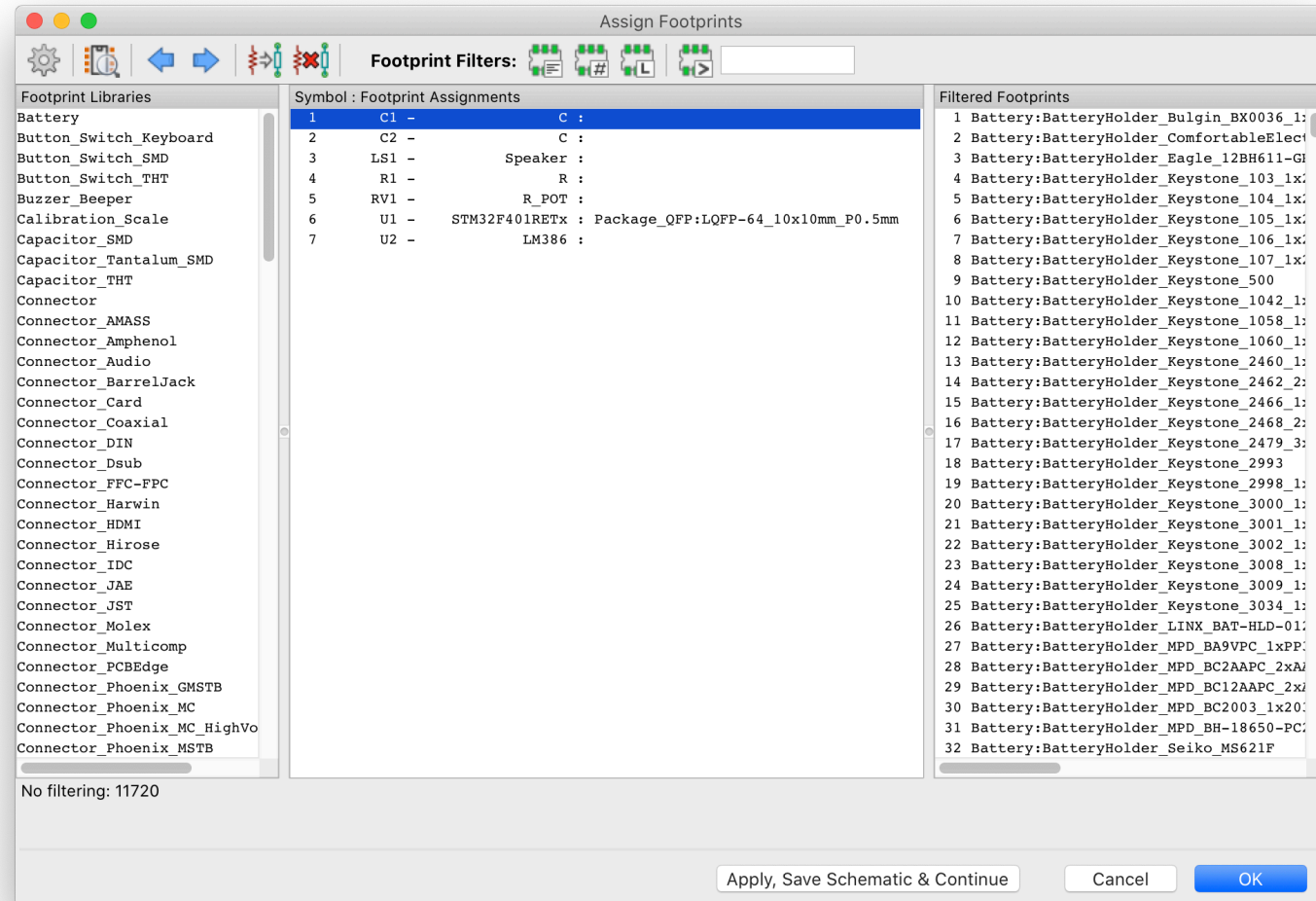- Symbols for many common parts already exist
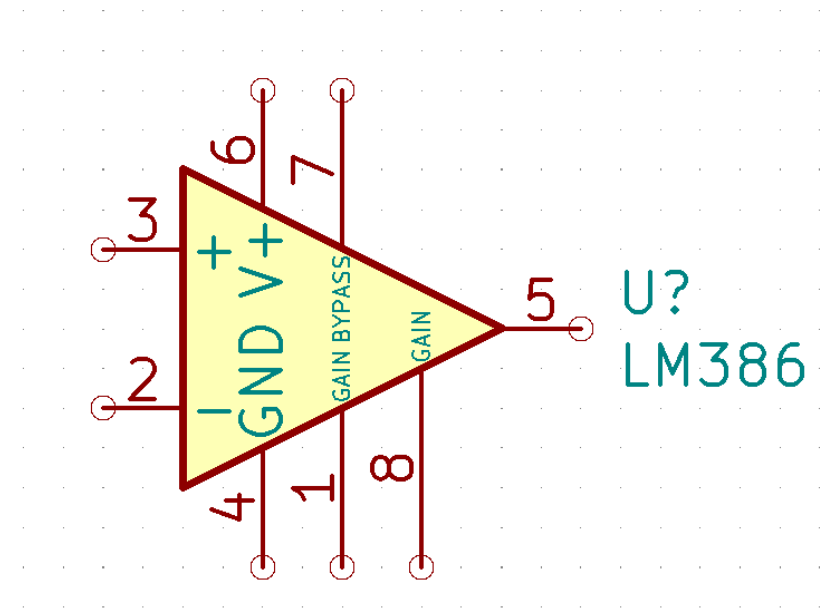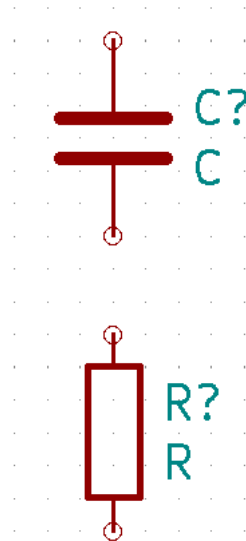
Schematic Capture

- Layout components
- Then go and connect with wires or labels
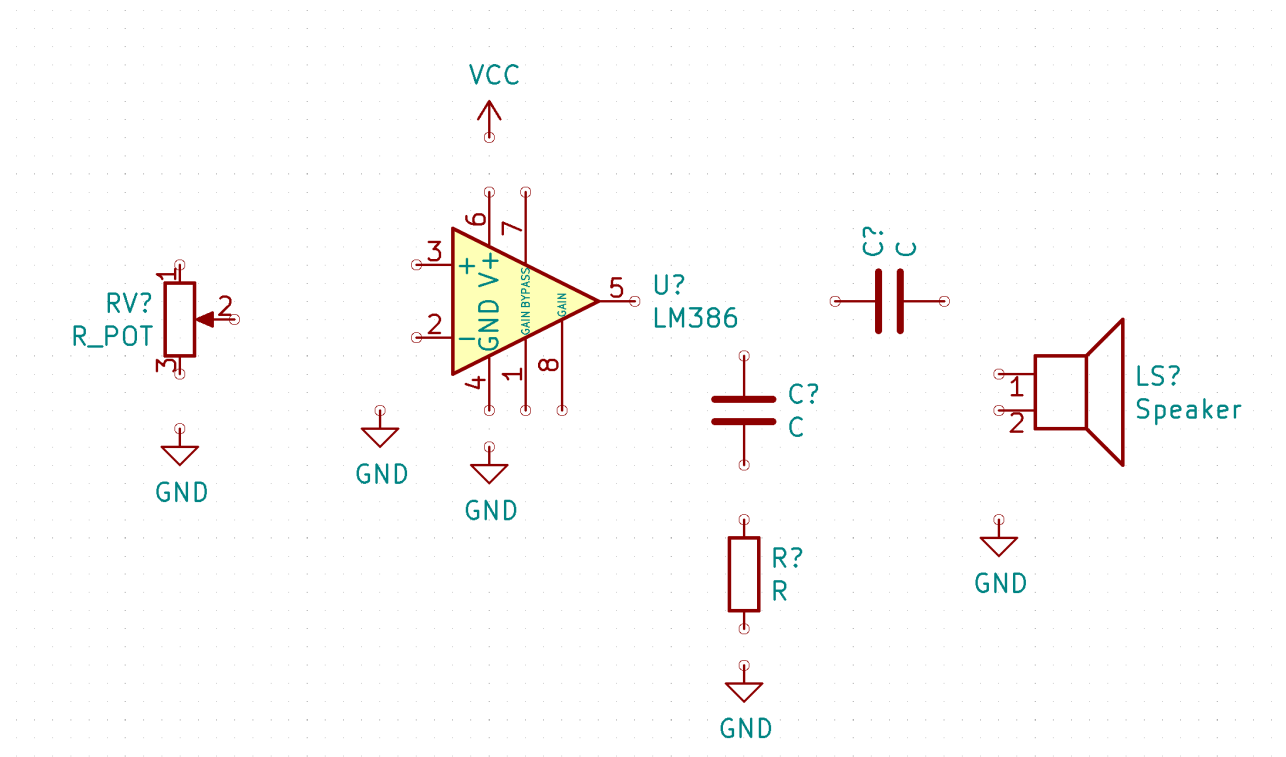
Create/Import Footprints
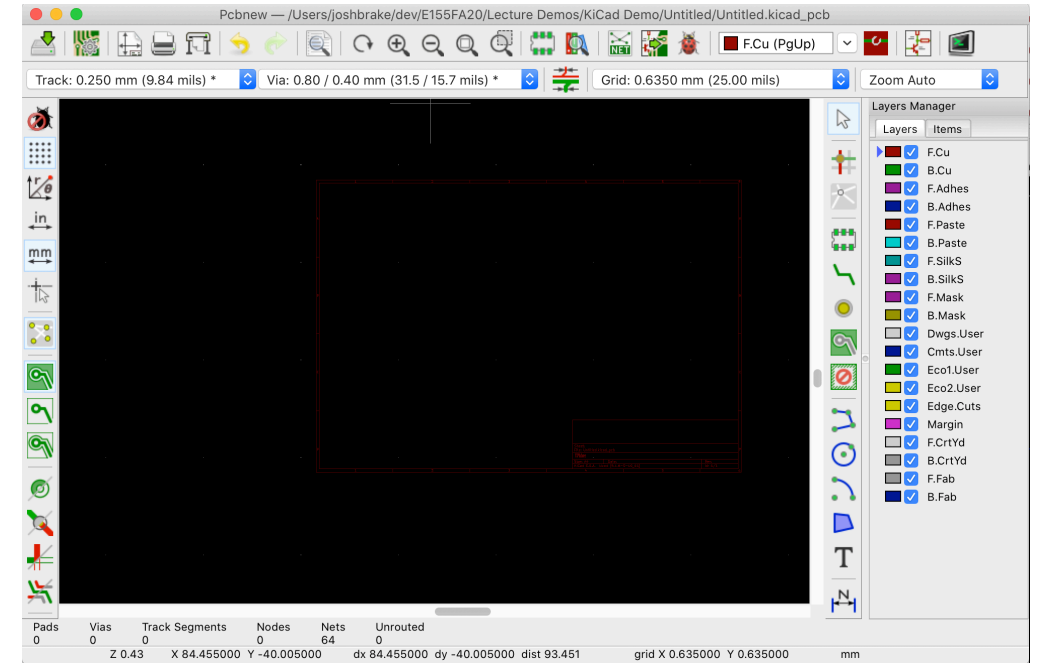
- Connect schematic symbols to their physical shape

Generate Netlist

- Gives list of network connections between components

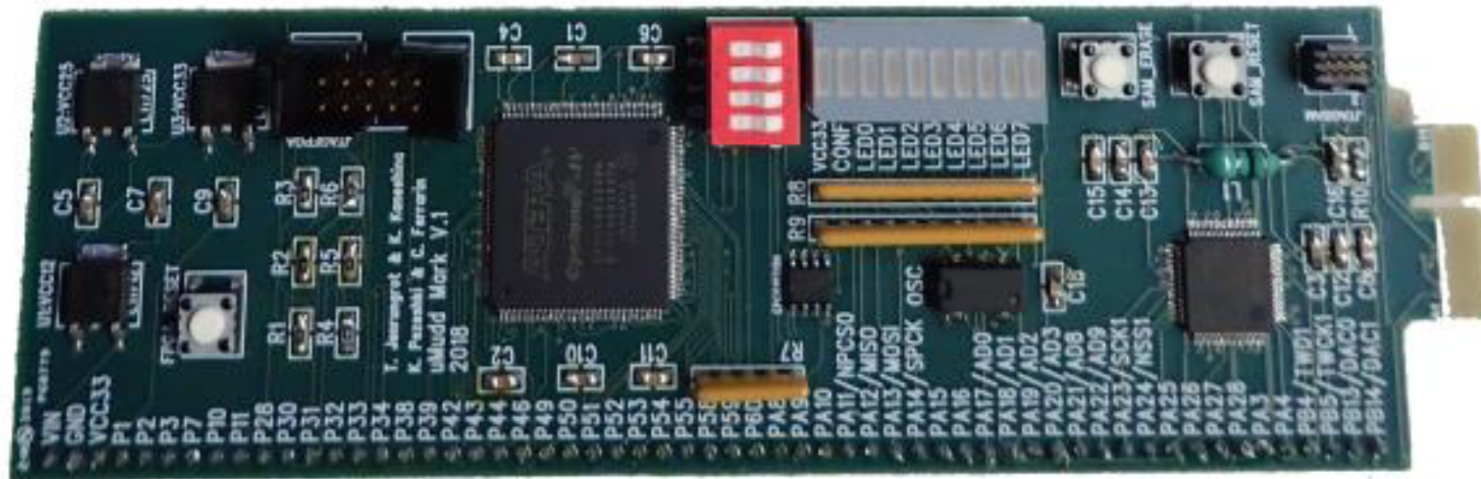# Board Layout
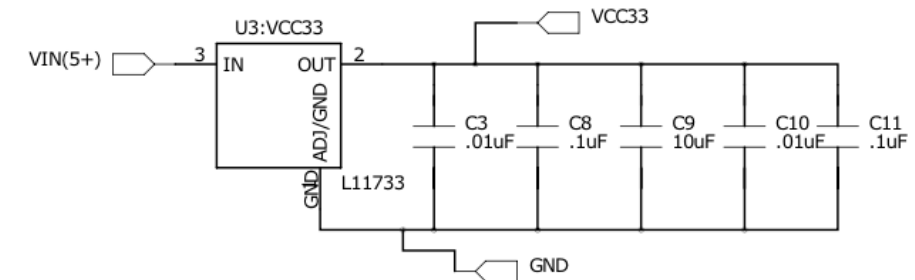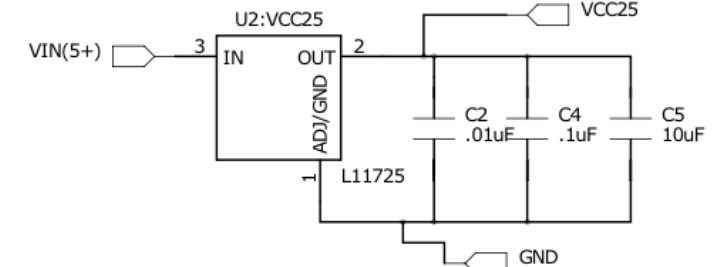
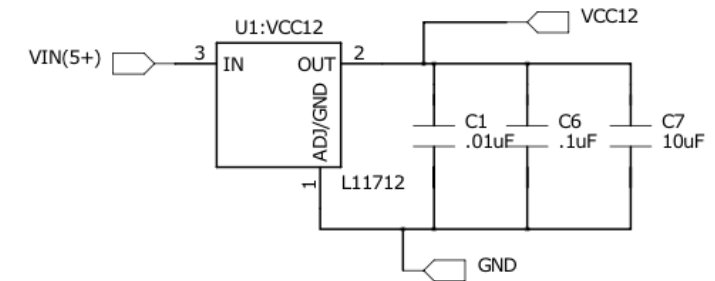- Open PCB new and import schematic

Production

- Check out the Gerber with an online viewer
- Gerber file is industry standard file for machines with manufacture PCBs
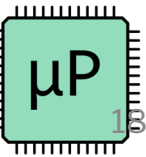
Layout Considerations

- Trace Widths

- Decoupling capacitors: Rules of thumb
  - ~10 µF for next to power inputs
  - 10-100 nF next to ICs

HMC2019: Cyclone IV + ATSAM4S4BA

KiCad Demo

References

- https://cei-lab.github.io/ece3400-2018/lectures/Lecture15-PCB_Design.pdf

- https://ocw.mit.edu/courses/mechanical-engineering/2-996-biomedical-devices-design-laboratory-fall-2007/lecture-notes/lecture02.pdf

- https://www.protoexpress.com/blog/7-pcb-design-tips-solve-emi-emc-issues/

Further Resources

- Shawn Hymel YouTube series - [KiCad Digi-Key Tutorial](#)
- Chris Gammel Video series - [Getting to Blinky 4.0](#)
- Prof. Spencer PDF - [How to make a PCB for me](#)
- Clinic Tutorial

Up Next

- Wednesday: Motors and Speakers
- Lab 7: The Internet of Things
  - Final project proposal due today