MITMECHE

Introduction to PCB

MITMECHE

Workshop TiM THE BEAVER

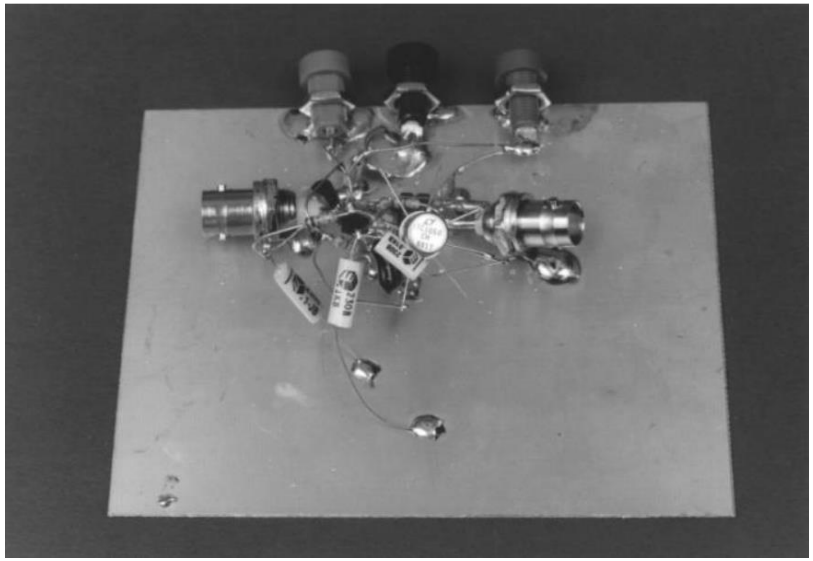
Outline

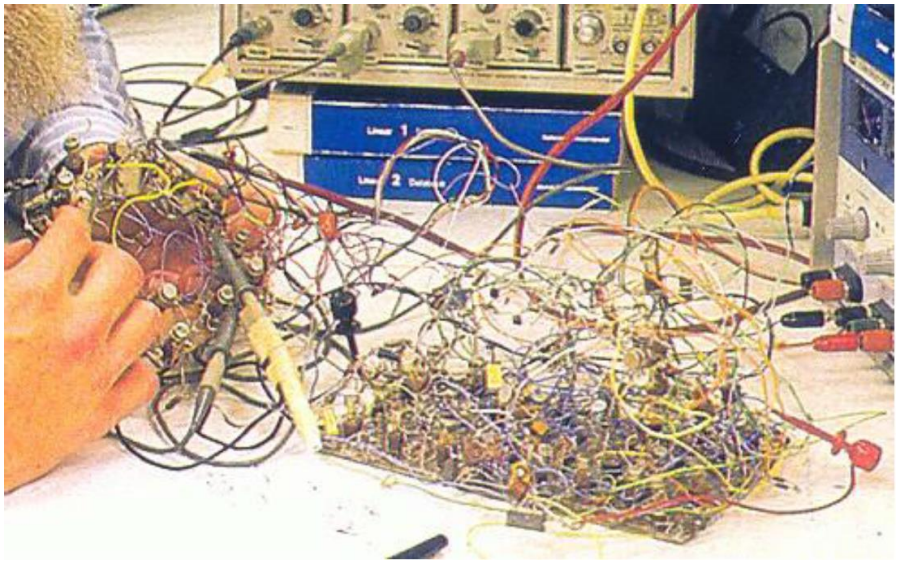
- Overview
- What is a PCB and how it is made
- PCB Design procedure
  - Design Intent
  - Schematic Capture
  - Board Layout
  - Design Transfer

MITMECHE

Workshop TiM THE BEAVER

Why use PCBs





Pros and Cons

- Pros
  - Repeatable with low incremental labor cost
  - Lower and more repeatable parasitic capacitance
  - Better signal integrity
  - Shorter route length
  - Ground planes
- Cons
  - High initial investment: $30 to $1k
  - Several days of waiting for fabrication
  - Need careful planning
  - Hard to change (less flexibility compared to bread board)

MITMECHE

Workshop TiM THE BEAVER

History

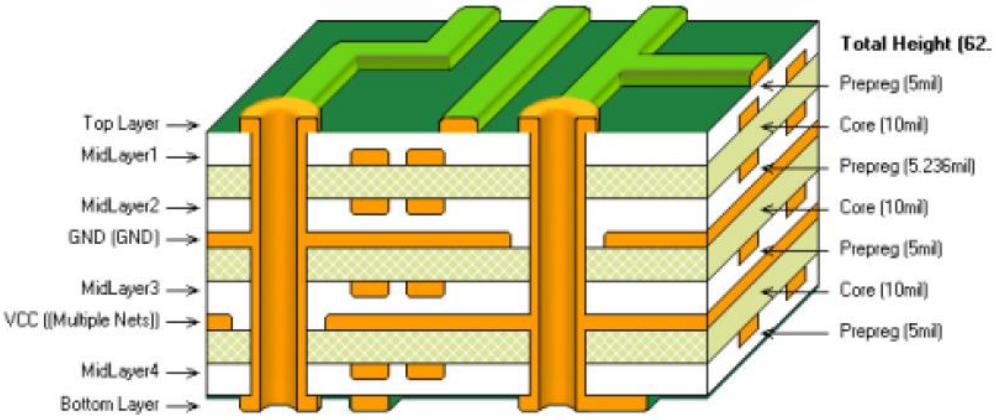
- Invented in 1903
- Common use by the 1920s
  - Radios, gramophones
  - Sheets of Bakelite, cardboard, wood, etc
  - Brass rivets for connections, all Through hole components
  - Single sided
- 1947 - First 2 layer PCBs
- 1960 – First 4 layer PCBs
- 1960s / 1980s

MITMECHE

Workshop TIM THE BEAVER

What is a PCB

- Printed Circuit Board
- Holds and connects components in a circuit
- Patterned sheets or copper foil
- Laminated sheets with multiple interconnected layers



Layers

- Conductor
  - Almost always copper
  - Alternatives: silver, gold
- Board material
  - Fiber-glass + resin (glass transition at 135 C, decomposition at 315 C
  - Polyimide for flexible PCBs
  - Metal for extreme power densities
  - Teflon for high frequency boards
- Solder mask for trace cover
  - Often green, makes soldering easier
  - Environment protection but not reliable electrical insulation
- Silk screen
  - Typical white color text or documentation on board

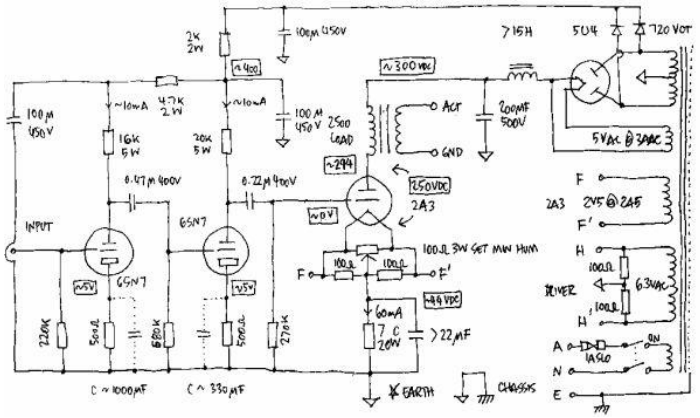
MITMECHE

Workshop TIM THE BEAVER

How is a PCB made

PCB Design Intent

- Draw out functional block diagram of circuit system
- Draw out roughly implementation of each functional block
- Choose key components (IC chips, operational amplifiers, etc.)
- Confirm that you actually need a PCB
- Double check bread board implementation
- Make sure that no significant modification is needed
- Identify constrains in bandwidth, operation range, power and etc.



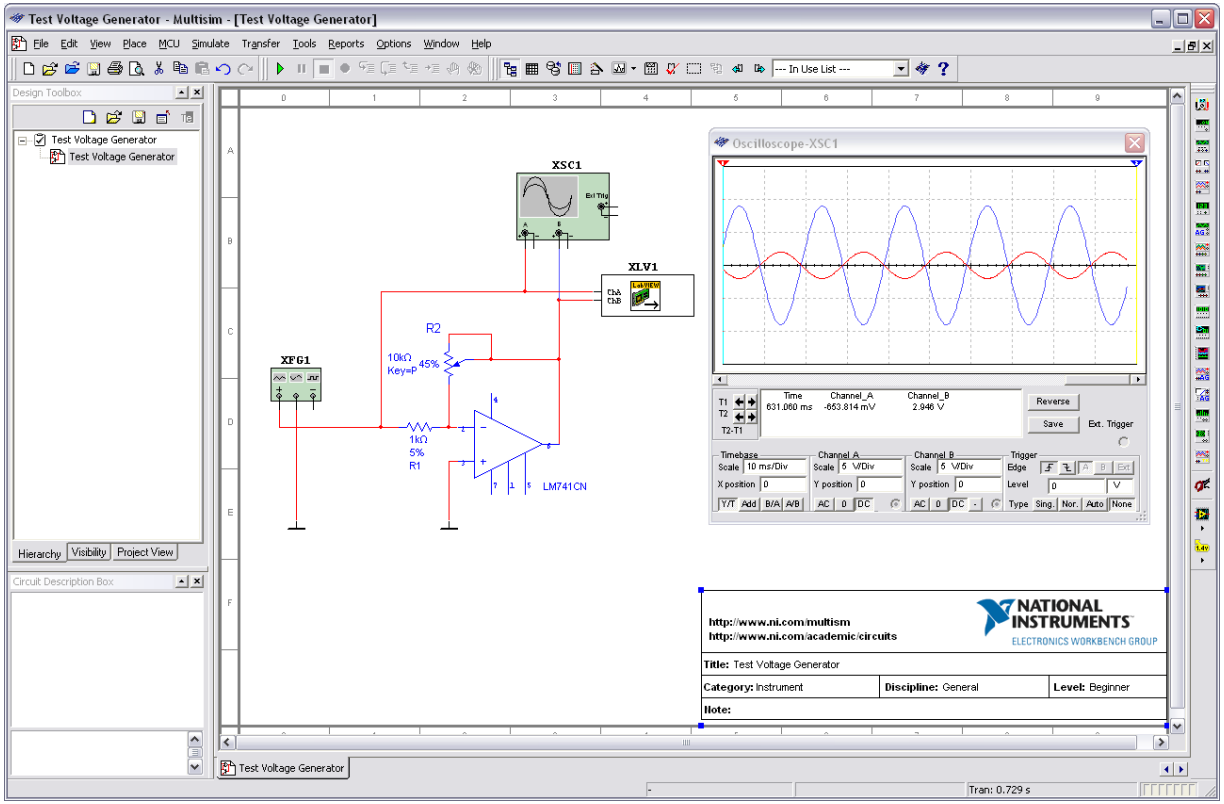
PCB Design Software

- Altium Designer
- NI Multisim/Ultiboard (We will use this)
- CircuitMaker
- Upverter
- KiCad
- DipTrace
- Eagle
- PADS/Dx Disaster

MITMECHE

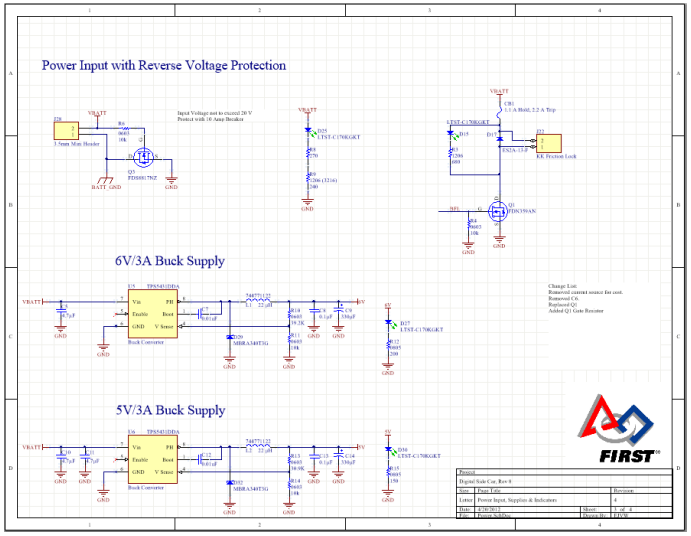
Workshop TIM THE BEAVER

Schematic Capture



Schematic Capture

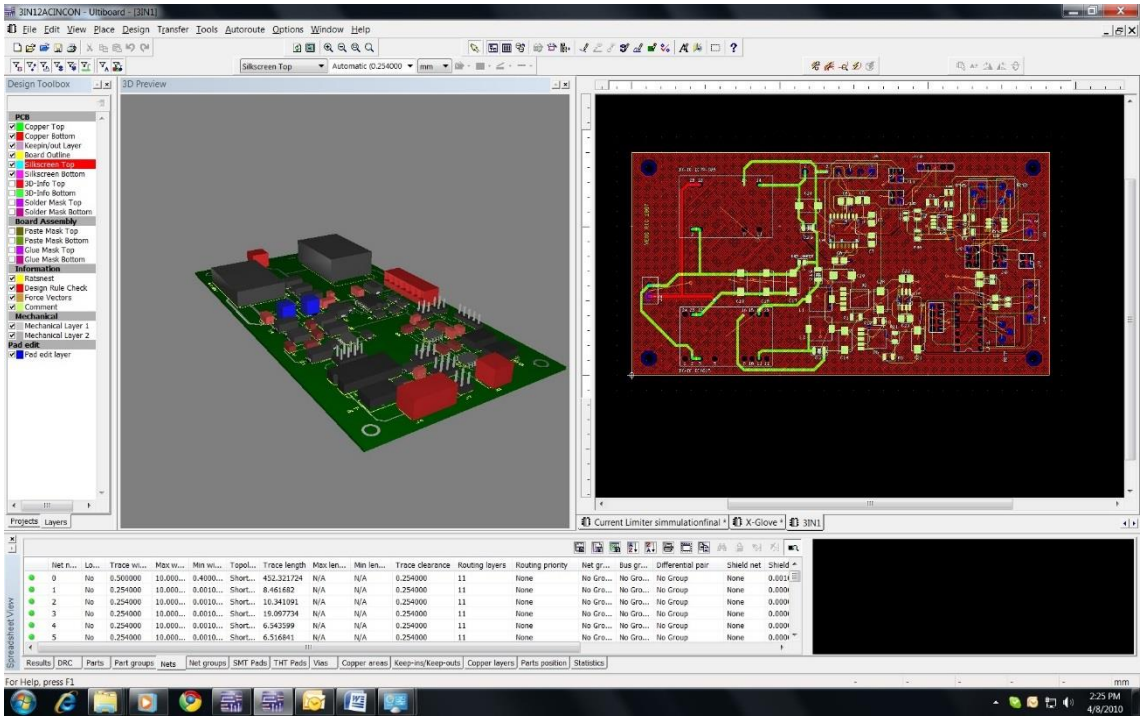
- Captures and formalizes design intent
- Select components
  - Add to library if necessary
  - Download SPICE model from vendor
- Group into sub-circuits
- Add connectivity
- Conduct simulation if needed
- Export to PDF for discussion
- Primary communication method
  - Include relevant calculations
  - Leave brief comment and pin function
  - Use neat connection



MITMECHE

Workshop TIM THE BEAVER

PCB Layout

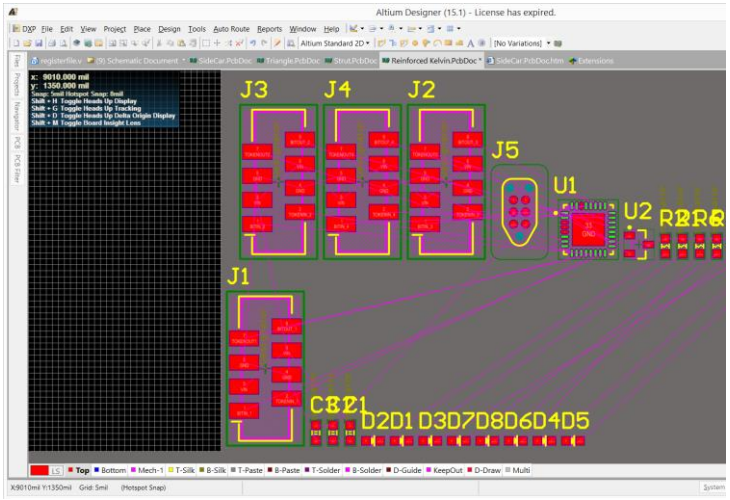


MITMECHE

Workshop TIM THE BEAVER

Layout

- Pull parts and connections from schematic
- Arrange by sub-scircuit into rough groups
- Initial placement
  - Connectors, power, switches
  - Fast digital, analog
  - Slow digital, analog
  - High current or voltage
  - Radio frequency communication
- Route the wires
  - Consider wire thickness
  - Use ground plate and/or vias
- Refine routing and placement



MITMECHE

Workshop TIM THE BEAVER

Trace Space

- Determines maximum voltage
- •~1.5V to 25V per mil (0.001 in.)
- Wider trace and space is cheaper

Table 4 – CLEARANCES and CREEPAGE DISTANCES for MAINS CIRCUITS

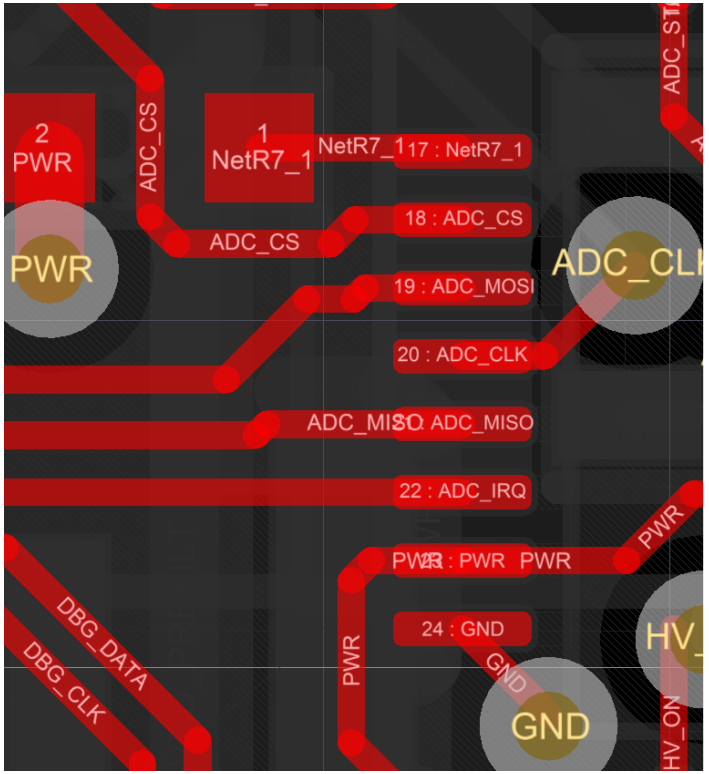
Voltage line-to-neutral a.c. r.m.s. or d.c.	Values for CLEARANCE (see note 1)	Values for CREEPAGE DISTANCES								
		POLLUTION DEGREE 1		POLLUTION DEGREE 2				POLLUTION DEGREE 3		
		Printed wiring boards	All material groups	Printed wiring boards	Material group I	Material group II	Material group III	Material group I	Material group II	Material group III
		CTI ≥ 100	CTI ≥ 100	CTI ≥ 100	CTI ≥ 600	CTI ≥ 400	CTI ≥ 100	CTI ≥ 600	CTI ≥ 400	CTI ≥ 100
V	Mm	mm	mm	mm	mm	mm	mm	mm	mm	mm
>50 ≤100	0.1	0.1	0.25	0.16	0.71	1.0	1.4	1.8	2.0	2.2
>100 ≤150	0.5	0.5	0.5	0.5	0.8	1.1	1.6	2.0	2.2	2.5
>150 ≤300	1.5	1.5	1.5	1.5	1.5	2.1	3.0	3.8	4.1	4.7
>300 ≤600	3.0	3.0	3.0	3.0	3.0	4.3	6.0	7.5	8.3	9.4

NOTE 1 Minimum values for clearance for different POLLUTION DEGREES are

POLLUTION DEGREE 2: 0.2 mm;

POLLUTION DEGREE 3: 0.8 mm.

NOTE 2 Values specified are for BASIC INSULATION or SUPPLEMENTARY INSULATION. Values for REINFORCED INSULATION are twice the values for basic insulation.



MITMECHE

Workshop TIM THE BEAVER

Trace Width

- Main consideration for thermal dissipation
- Trace dissipation determines current capacity
  - Outer layers dissipate more
Layer	0.5 oz	1 oz	2oz
Inner	100 mil/A	50 mil/A	25 mil/A
Outer	20 mil/A	10 mil/A	5 mil/A

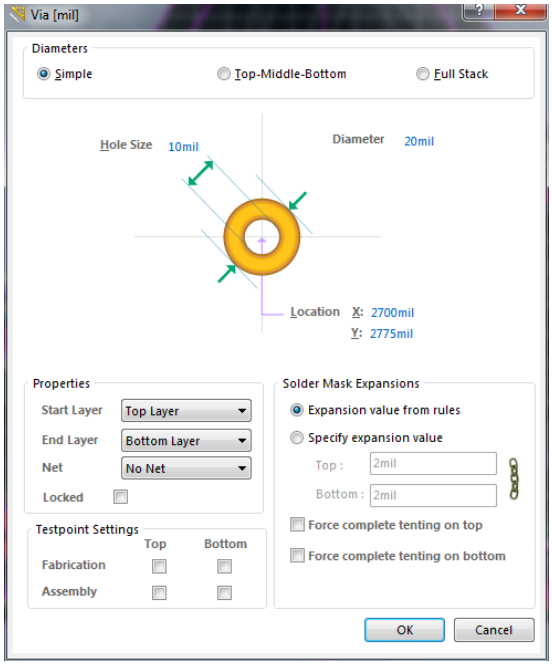
- Most online calculators use very old data
  - Modern glue delaminates at higher temperatures
  - Assume infinite trace with no extra heat sinks

MITMECHE

Workshop TIM THE BEAVER

PCB Drilling

- Denoted as Via in software
- Minimize number of drilling
- Maximize minimum drill bit size
- Minimize number of drill sizes
- Blind and buried drilling are expensive
  - Holes that start/stop in the middle of board



Symbol	Hit Count	Tool Size	Plated	Hole Type
□	101	20mil (0.508mm)	PTH	Round
○	4	40mil (1.016mm)	PTH	Round
✡	4	52mil (1.321mm)	PTH	Round
▽	8	66.929mil (1.7mm)	PTH	Round
⌑	2	125.984mil (3.2mm)	PTH	Round
⌬	4	190mil (4.826mm)	PTH	Round
	123 Total			

MITMECHE

Workshop TIM THE BEAVER

Common Mistakes

- Poor thermal management
- Sub-circuit coupling
  - Switch mode power amplifier can generate ripple signals
  - Small spacing between high frequency lines
- Poor grounding
  - Loop management in general
- Poor bypassing
  - Bypass capacitors too far away
  - Coupled with signal line crossing
- Using the auto-router
  - Usually do a not so good job
- Almost impossible to solder

MITMECHE

Workshop TIM THE BEAVER

Thermal Relief

- Used for via connecting to ground copper plane
- Added for ease of soldering to save time and cost
- Gives better chance for desolder
- Reducing thermal conductivity significantly
- Reducing conductance marginally
- Not used for high frequency or high current channels

MITMECHE

Workshop TIM THE BEAVER

Design Transfer

- Gerbers
  - Image of one layer of PCB board
  - Copper, solder mask, silk screen, etc.
  - No real standard for which image is which layer
- NC Drill
  - Location and size of drill hits
- Bill of Materials
  - Components to purchase and solder
  - Can include vendor information if added
  - Biggest source of error in transfer

MITMECHE

Workshop TIM THE BEAVER

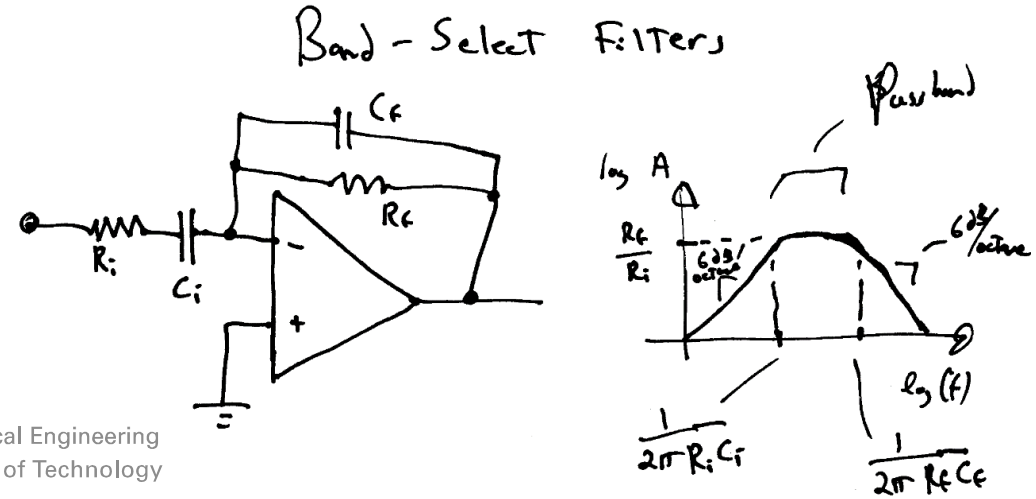
PCB Fabrication

- 3rd Party Vendors
  - Sunstone is quickest to respond via live chat
  - 4PCB offers student price package
  - OSH Park has nice service
  - Sencore is good for volume production but is slow to start
  - AMI has local stores
  - Use Taobao if you are in China, much cheaper even including shipping, good for most design

- In House Fabrication
  - 1 layer usually done a CNC router with tiny mill bits
  - Use FR1 substrate instead of FR4 material (harmful glass dust)
  - Various tooling available: Othermill, LPKF Machines

Homework Exercise

- Go through some tutorials for Multisim and Ultiboard online
- Create schematic capture in Multisim for a bandpass filter
- Computer values to allow band pass at 1 kHz to 100 kHz
- Utilize AD8021 or other OpAmps of your choice to simulate response
- Create board layout and remember to add bypass capacitors



Thank You!