This seminar is dedicated to students taking senior design project or attempting to take it very soon.

The focus is on practical aspects of engineering necessary to complete senior design project successfully and professionally.
Goals

✓ To learn practical technical aspects of designing and building your PCB
✓ To get advice before your full scale hands-on experience
✓ To avoid costly mistakes in implementing senior design projects
✓ To use senior design lab resources properly and efficiently
Topics

- ECE lab fabrication capabilities
- PCB technology
- PCB design guidance
- PCB fabrication guidance
- PCB cleaning and inspection
- Testing and Debugging tips

Note: Please listen carefully and keep notes. There will be a lot of information provided that is not on these slides.
ECE492/3 design and fabrication guidance (!!!)

- See ‘Resources’ on course web site. They include:
  - PCB Design Guidance
  - 3D Printing Guidance
  - Laser Cutting Guidance

GMU Library: electronic resources

- “Printed circuit board design techniques for EMC compliance: a handbook for designers,” by M.I. Montrose

Books:

- “Complete PCB Design Using OrCAD Capture and PCB Editor,” by K. Mitzer
- “Printed Circuit Boards,” by R. Khandpur

On-Line articles:

- PCB Design Tutorial:

Search youtube.com for soldering tips
Why PCB?

- SMT proliferation and gradual extinction of THT
  - Today, THT mainly exists due to the educational needs
- Big step towards being a professional engineer covering practical aspects of engineering
- Employers value PCB design skills – an art or science?
- PCB is also a circuit (!)
  - Ground loops
  - Cross-talk
  - RF aspects (!!!)
  - High speed aspects (>80MHz)
  - Manufacturability
- There is simply no other better technology
PCB in Senior Design Projects

- Ability to proceed through the entire engineering process
  ProblemDefinition ➔ Approach ➔ Design ➔ . . . ➔ Product
- Availability of electronic components as SMDs only
  - SMT has effectively replaced THT
- Gives a professional finish to your project (!)
- Industry and your resume factor
- Supported by the state-of-the-art professional software
  - OrCAD (Full version available in the lab); KiCAD open source
- Free ECE in-house fabrication capability !!!
- And it looks absolutely awesome
ECE Fabrication Lab (1)

- PCB fabrication
- Through hole plating
- Soldering stations
- Rework stations
- IR soldering
- PCB inspection
- 3D printing
- Laser cutting, etc.
You need to understand the fabrication process in order to design your PCB properly.

- PCB fabrication through precision milling
  - Copper traces and pads
  - Through holes
  - Mechanical holes/cutouts
  - Board cutoff
- Two-layer PCB as default
- Four-layer PCB, if truly needed
PCB Design Software

- You can use any PCB design software which gives you Gerber and Excellon files (but always check restrictions)

- Cadence OrCAD software
  - Full professional version in the lab
  - Free student edition available
    - Good for learning only; due to significant restrictions (netlist size)

- Eagle software
  - No longer an open-source (restrictions in place)
  - Not recommended: due to students having too many problems with proper design and file preparation for in-house fabrication
KiCAD PCB Design Software

- Open-source; very close to a professional software suite
  - Currently sponsored by CERN
  - Download from: http://iut-tice.ujf-grenoble.fr/kicad/

- No limitations on sheet, board, component size and complexity

- Problems:
  - Component and footprint database is rather small
  - Database search is rather inconvenient

- Recommendations:
  - Learn it by following a tutorial
  - Create a new component/footprint by modifying a similar existing one
  - Use hotkeys – see separate file on the ECE492/3 page

I use it and did not encounter any runtime problems
Other PCB Design Software

- Be very careful when selecting other software (!!!)

- There are restrictions on size, complexity, and components
- Sometimes these restrictions are not stated explicitly
- Free software frequently will not give you Gerber and Excellon files
- Such software companies may require you fabricate a PCB using their services or partners
PCB Material

- **FR4**
  - Fire-Retardant Class IV; Woven glass / Epoxy
  - Do not use any other material of lower grade

- **Available in the ECE Fabrication Lab – for free**
  - FR4 0.059” thick with 1oz copper clad (double-sided)
  - FR4 0.028” thick with 1/2oz copper clad (double-sided)
  - If a different board is needed, you have to buy it yourself

- **Board sizes**
  - Maximum blank board size: 9”x12”
  - Gives maximum usable area: 8”x11”
General PCB Guidance

- In-house PCB Fabrication is done through precision milling

- Two-layer or four-layer PCB?
  - By default: two-layer PCB
  - You rarely need to build a four-layer board for a typical project
    - This is a very time consuming process
    - Consult with me first before deciding to take this route

- Your board should be small, in general, however ...

Spacing between components must provide an easy access during soldering, testing, and potential rework

This is not a ground plane
- Copper trace width
  - Typically 10 mils minimum, but better if wider than 10 mils
  - Down to 5 mils is possible, but you need to justify such a need
  - Check current rating of a copper trace (on-line calculators)
    or a simple guidance: [http://armisteadtechnologies.com/trace.shtml](http://armisteadtechnologies.com/trace.shtml)

- Copper trace spacing
  - Typically 10 mils minimum
  - 8 mils is still possible, if you need it
  - 5mil (only on 1/2oz copper clad) but is not reliable – avoid it (!)

- Smaller sizes are challenging when soldering/rewiring (!)
PCB Design Guidance: Holes

- **Hole sizes**
  - Minimum diameter of 0.6 mm
  - Drills installed on the machine include: 0.6, 0.7, 0.8, 0.9, 1.0, 1.2, 1.5, 2.0, and 3.0 mm
  - Few smaller drills are available but only when justified

- **Mounting holes**
  - Mechanical holes and cut-outs must be defined on a separate layer – and stored in a file as ‘non-THT’ holes (!)
  - Use #4 screw size for typical PCB mounting – it requires the 3mm mechanical hole
  - Larger holes are milled out automatically

- **Consult first for any different setup from the recommended (!)**
PCB Design Guidance: Vias

- **Default via size setup in PCB software is too small**

- This is not a problem:
  - Define hole size: 0.6mm
  - Define larger via pad size: 1.4mm
    - More copper area improves wetting and access – so, it will be easier to solder them by hand

- Vias to a ground plane will not have thermal relieve
  - Use a bigger iron tip or wait longer for copper to heat up
PCB Design Guidance: Pads

- Check pad sizes. You do not want them to be too small.
- Larger pad makes soldering by hand easier
  - This is due to expanded wetting surface for solder
- Allow for:
  - At least 0.4mm copper belt around a hole
  - At least 0.4mm pad length extension for SMT package

\[ D \geq d + 0.8 \text{ [mm]} \]

Pads large enough
Learn about ground loops and remedies to avoid them.

In complex designs, ground plane is not always possible for a 2-layer PCB. Then, you may need to design a 4-layer PCB.

- Or use top-bottom-top trace routing – **BUT BE CAREFUL**
- Adjust ‘separation’ default setup

**Change default separation setup to 0.4mm (not larger)**

If the separation is too large, then tiny copper shavings will remain.
What is rubout?

- By default, double separations are made around all pads.
- You rarely need a full rubout – it is expensive in time and tooling costs.
Partial Rubout

Partial rubout is needed for easier soldering of:

- Multiple-pin SMT components
- Very small SMT components such as QFN packaged

Use separate Gerber file to declare rubout areas

- In KiCad: User1 or User2 layer

Useful case

- Pitch 0.85
- QFN SMD package

Absolutely needed

- No need here
- 1206 SMD package
PCB Design Guidance: Other

- **Text/graphics**
  - No silkscreen capability
  - If you really need some markings, then
    - Very short text (one or two words) or small graphics can be cut on the bottom or top copper layer
    - But, do not insist on rubout around the text

- Always mark chip and polarization orientations (!)
  - Print silkscreen layer on a hardcopy and use when populating your PCB

- Check ECE PCB Guide for more information and any changes to the requirements, rules, capabilities, and the fabrication process
Through Hole Plating Capability

- **Two-layer board**
  - Only when truly justified – it’s a lengthy, messy, and expensive process (includes 30 min baking)
  - Two methods for avoiding TH plating:
    1. Soldering THT components on both layers when needed (Problems: Missed joints, Difficult rework)
    2. Soldering all THT components on the bottom layer, designing vias to route traces to the top layer when needed, and connecting via pads using a thin wire

- **Connection resistance:** ~250mΩ per hole

- **Four-layer board**
  - You must use through hole plating
  - Plating must be done very carefully
Use thermal relief pads on copper planes

The purpose – to minimize heat leakage from a pad during soldering

They can have different patterns
You must define board contour on a separate layer (!!!)

- PCB design software has a separate layer dedicated to this job

Your board is cut out from a larger blank board through routing

- This operation adds holding tabs
- How this outline will be cut?
- Can you suggest a modification?
By an appointment only
- Please respect my limited time availability
- If you are not sure then ask first (before you send files)
- No last minute jobs – plan ahead
- I will not review your designs – Please, ask your faculty advisor (!)

You need to plot your PCB design to separate files

Files required (for a two-layer PCB):

1. Top copper layer file (Gerber format)
2. Bottom copper layer file (Gerber format)
3. Drill file (Excellon format)
4. Board contour file (Gerber format)

Optional:
- Mechanical holes/cutouts file (as non-plated holes: Excellon)
- Rubout file for each layer (Gerber format)
Double check your files by a different Gerber viewer (!!!)
(e.g., http://gerbv.geda-project.org/)

Cost:
- Free for reasonable jobs, but
- No repetitive jobs due to design errors

Notes:
- Properly annotate your files by layer ID (top, bottom, etc.)
- If you do not have a contour file, your board will not be cut (!)
- If you email additional files that are not needed in the process, your board will not be cut (!)
- If you include (more than allowed) text/graphics or additional symbols (dimensions, etc.), your board will not be cut (!)
Component Placement

- Place components on one-side only
  - On two, if you have a real problem with space or routing

- Allocate some space around components
  - Do not pack them as seen on professional boards
  - Space is needed to solder components by hand
  - More space may be needed to rework your PCB
  - Space is needed to test and debug your PCB and embedded system
Routing Traces: Tips

- You MUST have a complete and accurate schematic diagram before routing traces (!)

- Two-layer PCB
  - You need to mix signal, power and ground traces to achieve an acceptable result

- Four-layer PCB
  - Separate traces into: Signal – Power – Ground – Signal layers

- Always look from the top of a PCB

- Try to avoid auto-routing option (and auto-placement)

- For more information read an excellent PCB Design Tutorial:
Basic Routing

Power routing

Signal routing

GOOD

BAD

GOOD

BAD
Trade Secret: Zero Ohm Resistor

- Use 0Ω resistors to jump traces
  - SMT resistor of a larger size will jump more than one trace
  - The same applies to the THT 0Ω resistor
  - Be sensitive to price – certain packages can be expensive

- Helps in simplifying trace routing
- You can combine jump resistors but do it only in a series (no more than 45° angle between them)
If you understand a PCB structure then you will understand how to handle heated PCB

Melting point:
~180°C for the solder
~150°C for the glue

Do not press too heavily on the pads/traces when soldering
- Pads can detach, lift, twist, and break
- Traces can break easily

Do not move components when applying an iron
Preparation Before Soldering

- Clean milled PCB from dust, oil residues, and oxidation
  - This will greatly improve solder wetting
  - Isopropyl alcohol and a toothbrush
  - Do not use acetone in the ECE lab
  - Remember, re-oxidation starts shortly

- Place your board in a vise
  - Understand how you can gain access from different directions
  - Understand how you can adjust vise to improve the access

- Prepare the iron
  - Define temperature to work with (!!!)
Pay attention to post-soldering cleaning of your PCB
- Shorter distances greatly influence current leakage
- Tight spaces hold flux better – some flux may be left there
- Water-based glue will absorb moisture

How to clean
- Soak in warm water+soap for a longer time; brush thereafter
- Isopropyl alcohol bath and brush
- Use nylon toothbrush (never use a metal brush)

- Ultrasonic cleaner is the best (water+soap)
Use an AmScope microscope
- It’s for inspection only
- Store it always under the cover

Inspection
- Hold your board in hand at an angle
- There are two knobs to work with to achieve different magnification level and focus – work with them very slowly

Never move the microscope around the lab (!)
Never solder under the microscope (!)
Camera is connected to the adjacent PC
- Open an *AmScope* application
- Select camera
- Camera focus is adjusted at camera tube
- Save a picture, if needed
Testing Tips (1)

- Traces, vias, THT connections should be tested before a PCB is populated with components
  - Called as bare-board testing
  - At least, check Power-to-Ground resistance
- Use current limiting power supply or fuses on PCB to limit potential damages
- At the moment you power your populated PCB:
  - Look for smoke first – disconnect immediately if it is
  - You need to limit catastrophic failures to your board and components
- Another approach – for larger boards w/ many components
  - Partially populate your board (areas of single functionality) and test before adding additional chips/functionality
Testing Tips (2)

- Touch chips when board is powered up (low voltages only!)
  - Any significantly hot chip may be an indication of a problem
  - Design problem/error
  - Wrong chip rating
  - Current leakage or ... (???)
  - This may cause connection problems if under prolonged high temperature exposure
  - Solder fracture
  - Pad/trace detachment

- Verify quality of power supply – observe voltages on oscilloscope
- Verify quality of clock signals frequency and wave form shape
- Finally, test functionality of a single module at a time
Testing Tips (3)

- Test leads for an oscilloscope/multimeter must have sharp fine point-ends
  - So you can sink them slightly into the solder
  - Spring loaded leads are the best
  - Typical leads are not suitable
  - Circuits can be easily damaged by touching more pins at the same time
    - This can be devastating to your effort (see next slide)

- Be warned:
  - Probe holder, vise and other probe holding devices do not help
  - Assume that circuits, cables and everything around can move unpredictably at an instance (!!!)
Debugging Tips

- Identify essential signals in your schematics
- Add test pins/connectors to these signals
- Group test pins for easier navigation over PCB
  - 0.1” pitch TH pins are the best
  - Can be mounted from the bottom of PCB
- Connect test pins to oscilloscope through female test leads
Embedded System Debugging Tips

- Developing an embedded system you need additional connectors for testing and debugging of your software (!!!)
  - This has an influence on:
    - Your design (schematics and PCB), and
    - Selection of microcontroller (w/ more pins)
- At minimum, include in your design four test pins connected to four freely available MCU pins
  - You can flag events and tasks from the software
  - Use 0.1” pitch TH pins
  - Connect test pins to oscilloscope through female test leads
- If needed, fan-out MCU/device pins
- Use LEDs only for flagging low-frequency events
PCB Revision Tips

- If you need to revise your PCB after it has been built, then WAIT (!)
  - Do a complete debugging
  - Write a list of fixes
  - Next, redesign and build

- Do not be tempted to build another PCB based on a single problem (!)
  - Try to fix a PCB by rework before final revision
  - Rework may include adding a “fix” module

- Test every functionality you can before final revision
  - Learn as much as possible from a prototype

- Understand that RF, EMI, and ESD problems can be very difficult to diagnose and fix – plan accordingly
Conclusions

- Do not forget: “Think ten times, solder once”
- There are many tricks that help in designing PCB. Use them.
- What next? There is a lot to learn:
  - Design for manufacturability
  - Design and prototyping high speed digital circuits (PCBs)
  - Design and prototyping mixed signal circuits (PCBs)
  - PCB design for EMC compliance (!!!)
  - RF PCBs

- Do not forget: Your PCB is a circuit on its own
  - Especially at higher frequencies
Questions