PCB Design (with EAGLE tutorial)

TA: Robert Likamwa
ELEC 424, Fall 2010
Printed Circuit Boards

- What are they?
- How can I make one?
- 424 Project description
- Eagle Tutorial

http://www.electronicmanufacturers.co.za/
What is a Printed Circuit Board?

• “A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or signal traces.” (Wikipedia)
How does it work?

- Drilling (vias and holes)
- Patterning (etching)
  - Subtractive process to remove copper cover from a preimpregnated substrate
  - Silk-screen printing of etch-resistant inks
- Lamination
  - Multilayer PCBs
- Coating (Solder and Solder mask/resist)
- Printing text and symbols
Why do I need to know PCB Design?

• Create your own embedded devices.
  ▫ More robust than breadboard. Won’t fall apart.
  ▫ Can use surface mount chips
  ▫ Light in weight and size
  ▫ Production quality devices

• Put it on your resume... Recession-proof skill!
Steps to Design a PCB

- Figure out Functional Design
  - Identify components to be used

- Design schematic

- Design PCB Layout and Routing
Function Design

- Schematic
  - Layout
    - Routing
      - Production
        - Assembly
          - Any local workshops

- Component library
  - PCB design tool
    - PCBexpress.com

- Form factor constraints
- Design rules
Function Design

- What’s your device supposed to do?
- What sensors do you need to achieve your tasks?
- How is everything going to be powered?
- Will it fit in the provided space?
Component Selection

- Which Integrated Circuit chips can perform your task?
- Do they play nicely with each other?
- Are they available from the distributor?
Through-hole components

- Transistors, Resistors, Capacitors
- DIP (Dual In-Line Package) Packages

http://www.wikipedia.org/
Two-terminal SMD Packages

- Surface Mount Devices
- Resistors, Capacitors, LEDs, etc.
- Usually given in hundredths of an inch
  - Careful, they can be given in metric, also.
- Some common form factors:
  - 0805 (means 0.08” x 0.05”)
  - 1206
  - 1210
  - 1806

http://www.digikey.com/
IC Form Factors

- Surface Mount Device (SMD) Chip form factors:
  - Small Outline IC (SOIC) (variants - TSOP, SSOP, TSSOP)
  - Quad Flat Package/No-lead QFP, QFN
  - Ball Grid Array (BGA)

http://www.digikey.com/
Decoupling (Bypass) Capacitors

- Remove noise by shunting noise.
- 22-100uF for board (electrolytic or tantalum)
- 10nF for each IC (ceramic)
- Put capacitors as close as possible to ICs.

http://wikipedia.org
Steps to Design a PCB

- **Figure out Functional Design**
  - Identify components to be used

- **Design schematic**

- **Design PCB Layout and Routing**

*Use Eagle!*
Project Description

- Build a PCB to control the QuadRotor Helicopter.
- Figure out tilt of board, control motors to balance.
  - Use gyroscope and accelerometer sensors.
- Control altitude
  - Use ultrasound rangefinder
- Offer user-control of movement via bluetooth.
- Use MSP430 as the CPU (the “brain”)

Eagle Schematic Design
Schematic Project Considerations

- Gyroscope MUST be connected to I2C pins on MSP430

- Accelerometers and Rangefinder MUST be connected to ADC pins on MSP430
Add parts

Move
Clone
Delete
Mirror
Rotate
Group objects
(try right-clicking!)
Schematic Exercise! Part 1

- New Project
- New Schematic
- Save it inside the project folder.

- Use library “ricemobile.lbr” (Library->Use)
- Add Part MAX604 (MAX604)
- Add Part MSP430 (F16X---PM64)
- Add Part KXM52 Accelerometer (KXM52)
- Add Ground, Add VCC (From Supply Library)
Schematic Exercise! Part 2

- Connect GND wires on MAX604.
- Clone GND, connect it to GND. Connect VCC to IN.
- Add Electrolytic Capacitors (1206) to IN and OUT.
  - Rotate by right-clicking while moving it
  - Make sure minus side of capacitor is pointed to GND
- Value the Capacitors appropriately (10 uF).
Schematic Exercise! Part 3

• Draw lines to connect
  ▫ OUT_Y, OUT_Z on KXM52
  ▫ A1, A2 on MSP430

• Connect “wirelessly”
  ▫ Draw line sticking out of OUT_X.
  ▫ Draw line sticking out of A0.
  ▫ Name both lines ACC1_X.
  ▫ Label both of them.

• By the way, you’re not finished here. There are resistors and capacitors that need to be placed around the KXM52. Always check the datasheets!
Eagle PCB Layout Design
PCB Layout Considerations

• Positions of the following need to be EXACT:
  ▫ 4 mounting holes (1.75” square pattern)
  ▫ Accelerometer (1.5”, 1.5”)
  ▫ MSP430 (1.5”, 1.0”)

(For accelerometer and MSP430, we’re sending in a stencil for PCB Assembly)

• Board size needs to be 3”x3”
Layout Exercise! 1: Resize Board

- Use “Move” tool
- Type in (4.0 2.0)
  - This will select the rightmost border as if you clicked exactly there.
- Type in (3.0 2.0)
  - This will move the cursor to that position, resizing the board to exactly 3”x3”
- Then move all of your components in.
Poke holes in the right places

• 1.75” apart in a square pattern on a 3” x 3” board

\[
3” - 1.75” = 1.25” \text{ extra} \\
1.25”/2 = 0.625” \text{ clearance} \\
0.625” + 1.75” = 2.375”
\]

(0.625, 0.625)  \quad (0.625, 2.375)

(2.375, 0.625)  \quad (2.375, 2.375)
Layout Exercise! 2: Precision Layout

• Draw 4 holes.
  ▫ Use hole tool. Type: drillsize (x y)
    • 0.193 (0.625 0.625)
    • 0.193 (0.625 2.375)
    • 0.193 (2.375 0.625)
    • 0.193 (2.375 2.375)
• Move KXM52 to its place (1.5 1.5)
• Move MSP430 to its place (0.5 1.5)
Layout Exercise! 3: MAX604 on bottom

- Put MAX604 and its capacitors on bottom by using Mirror tool.

- Yellow lines are “Airwire” lines
- Use Route Manually tool to turn Airwires into traces
  - Turn off Grid (View->Grid, Finest Grid)
    - Or just change grid spacing to what you want it to be.
  - Change line width as necessary at top of screen
Layout Exercise! 4: Connect

- From KXM52, connect pin 6 to MSP430.
- From KXM52, connect pin 9 to MSP430.
- From KXM52, connect pin 7 to MSP430.
  - Go from top to bottom by selecting “Bottom” at the top-left of the screen. This will create a via.
  - Try to make your bottom traces as short as possible.
Layout Exercise! 5: Create GND Plane

- Use Polygon tool to draw a GND Plane. (Make sure not to draw the plane beneath your Bluetooth antenna)
  - Use the name tool to make it GND.
  - “Ratsnest” to see the result

- Create GND vias near GND pins on KXM52 and MSP430.
  - Place Vias, then “Name” them GND.
- Route the vias to the chips.
Create your own part

- Some parts don’t have an Eagle footprint associated with them.

- Let’s create our own gyroscope part.
Part Creation Tutorial 1

- Go to Control Panel (Window->Control Panel)
- File->New Library
  - Save library as 424parts.lbr

- Library->Symbol
  - Call it GYRO-BREAKOUT
- Use “Draw a Pin” to add pins. Change names
  - SCL, SDA, CLK, INT, GND, VLOGIC, VDD
Part Creation Tutorial 2

- Library->Package
  - Call it GYRO-BREAKOUT
- Draw 7 Pads in a row. Space them by 0.100”
  - Drill size set to ~0.043307
  - Rename the pads if you want to.
- Draw a box around the 7 pads with “Draw Lines”
Part Creation 3

- **Library->Symbol**
  - Call it GYRO-BREAKOUT
- **Add a Part**
  - Select GYRO-Breakout, lay it down
- **Click “New” button (on right side)**
  - Select GYRO-Breakout
- **Click Connect**
  - Assign pins to the symbol to pins on the device appropriately.
    - SCL, SDA, CLK, INT, GND, VLOGIC, VDD
- **Now if you want to use the part, all you have to do in a schematic is Use Library & Add Part.**
Where to go from here:

- **Design Rule Checking**
  - Find the Sunstone design rules online.
    - DFM Add-ons
  - Follow its provided instructions to check your design rules.

- **Create Gerber Files**
  - Gerber: standard file format for patterns on PCB – used by most fabrication houses

- **Send Gerber Files to PCBExpress**
Create Gerber Files

- Open your board
- Click on ULP then select “drillcfg.ulp”
- Click on CAM then select “excellon.cam”
- Click on CAM then select “xLPlus-Sunstone.cam”
  - $x = \text{number of layers}$
  - Note which layers you want for each file
  - Dimension layer (20) should be selected in all files
- Important: always check your Gerber files afterwards
Send files to PCBExpress

- Outline: .oln
- Drill hole locations/size: .drd/.drl
- Copper layers: .l1, .l2, .l3, .l4
- Top/bottom solder mask: .smt/.smb
- Top/bottom silkscreen: .slk/.slb
- Top/bottom soldering stencils: .tps/.bps

(May be different files for you)
And that’s the tutorial!

• Now you know how to Layout a PCB. The rest comes from experience!

• Just remember to always read the datasheets for all components.

• Further project specifications will be provided.