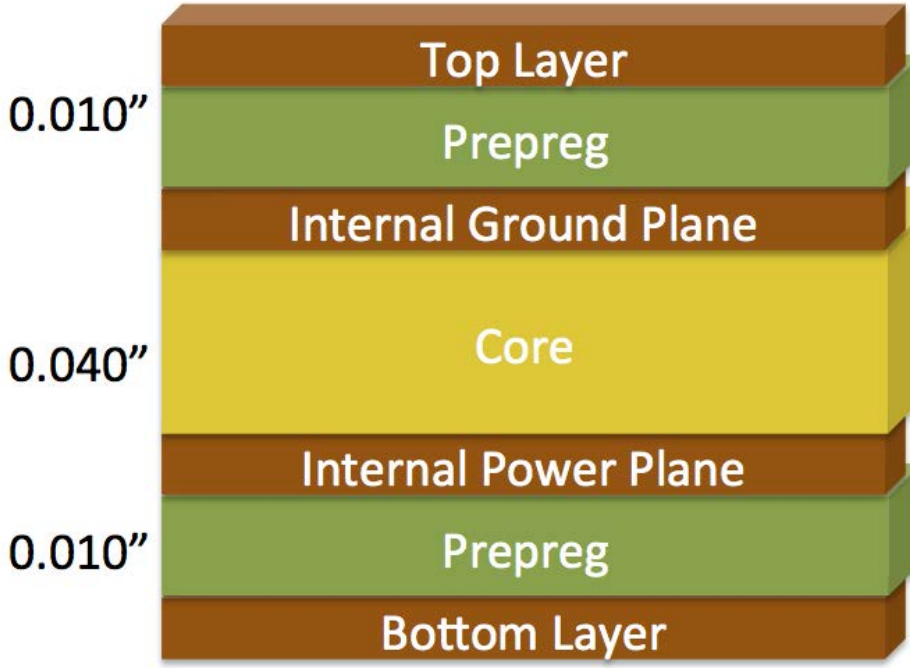
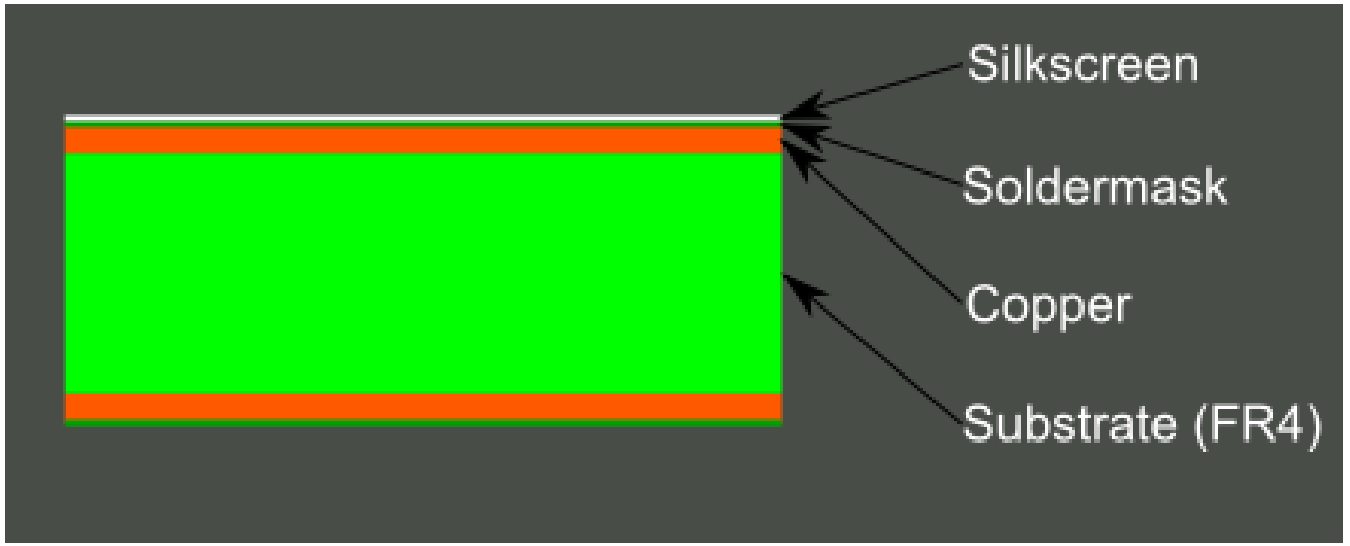
Board Layout
ECE 3400
Leah Forrest

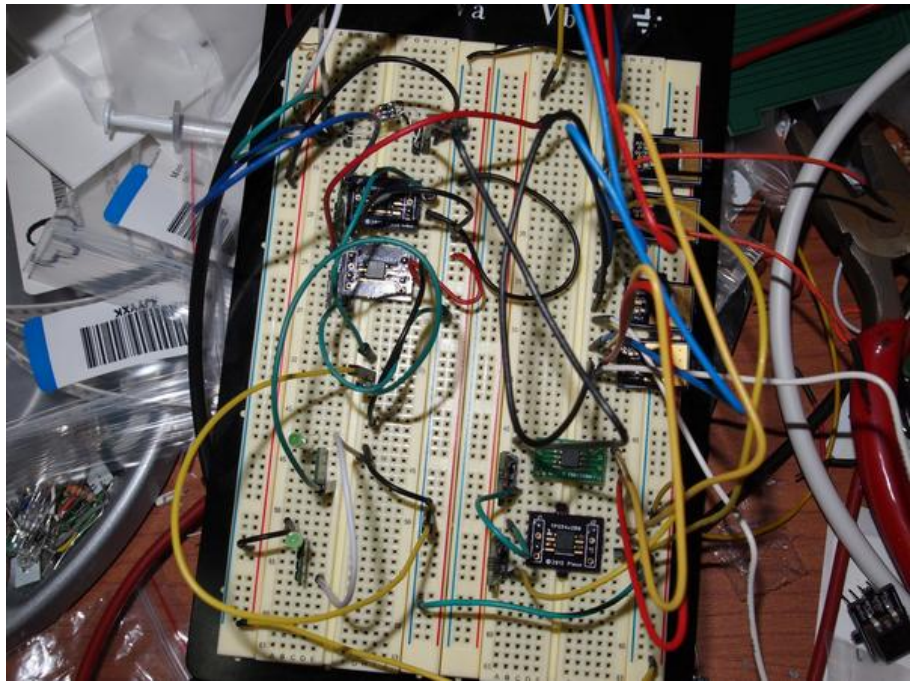
What is a PCB?...

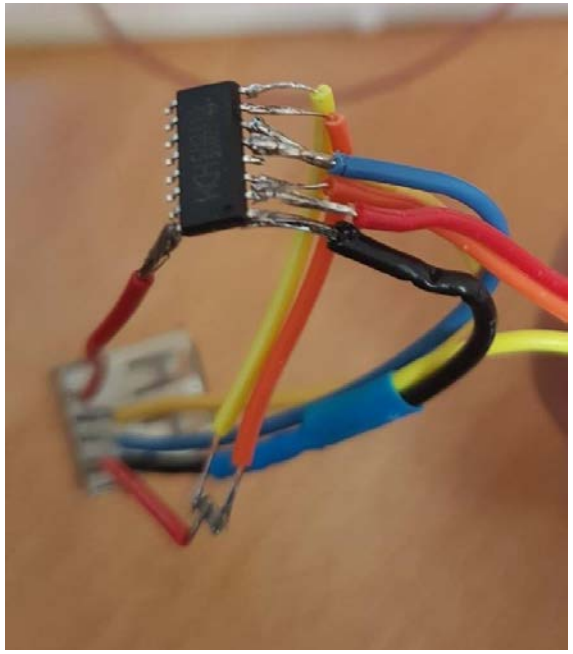
- “Printed Circuit Board”
- Substrate: made of FR4- glass epoxy/fiberglass
- Copper: thin foil (#layers = #coppers)
  - High power boards need thicker copper
- Soldermask: insulates copper traces
- Silkscreen: adds text, symbols

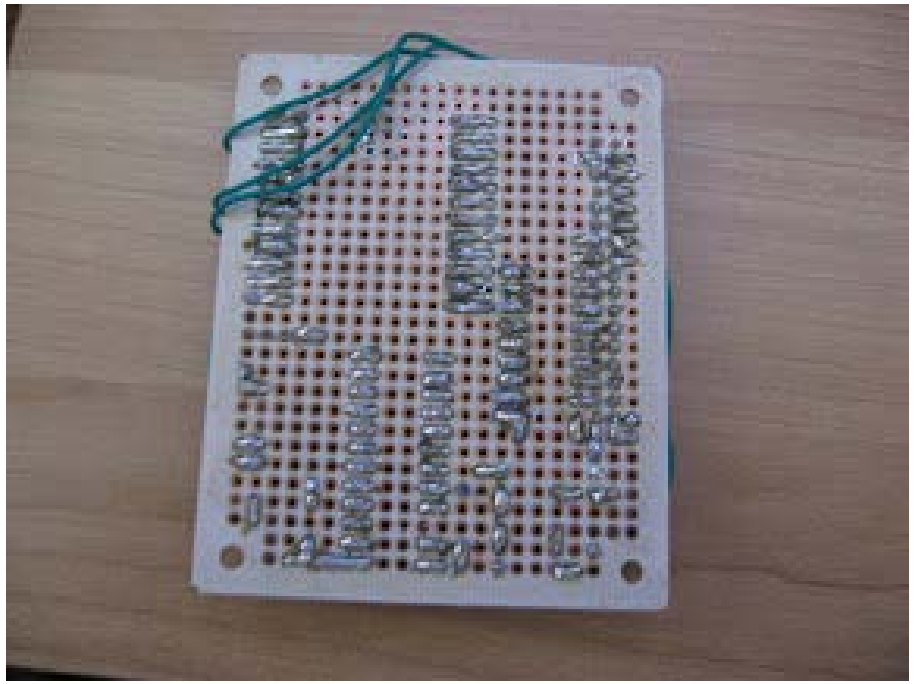




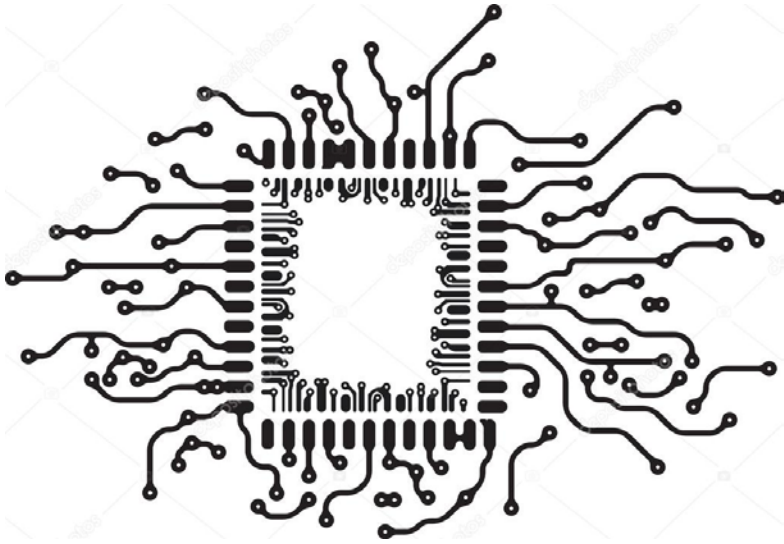
…and why do we need them?







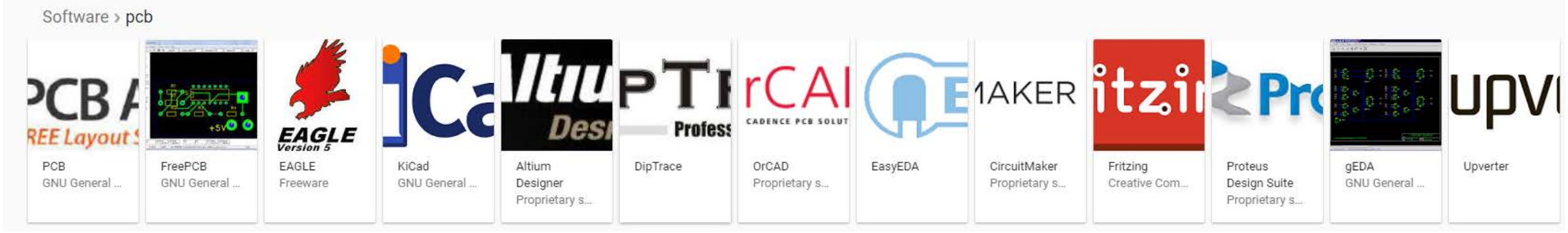
Useful Terms:



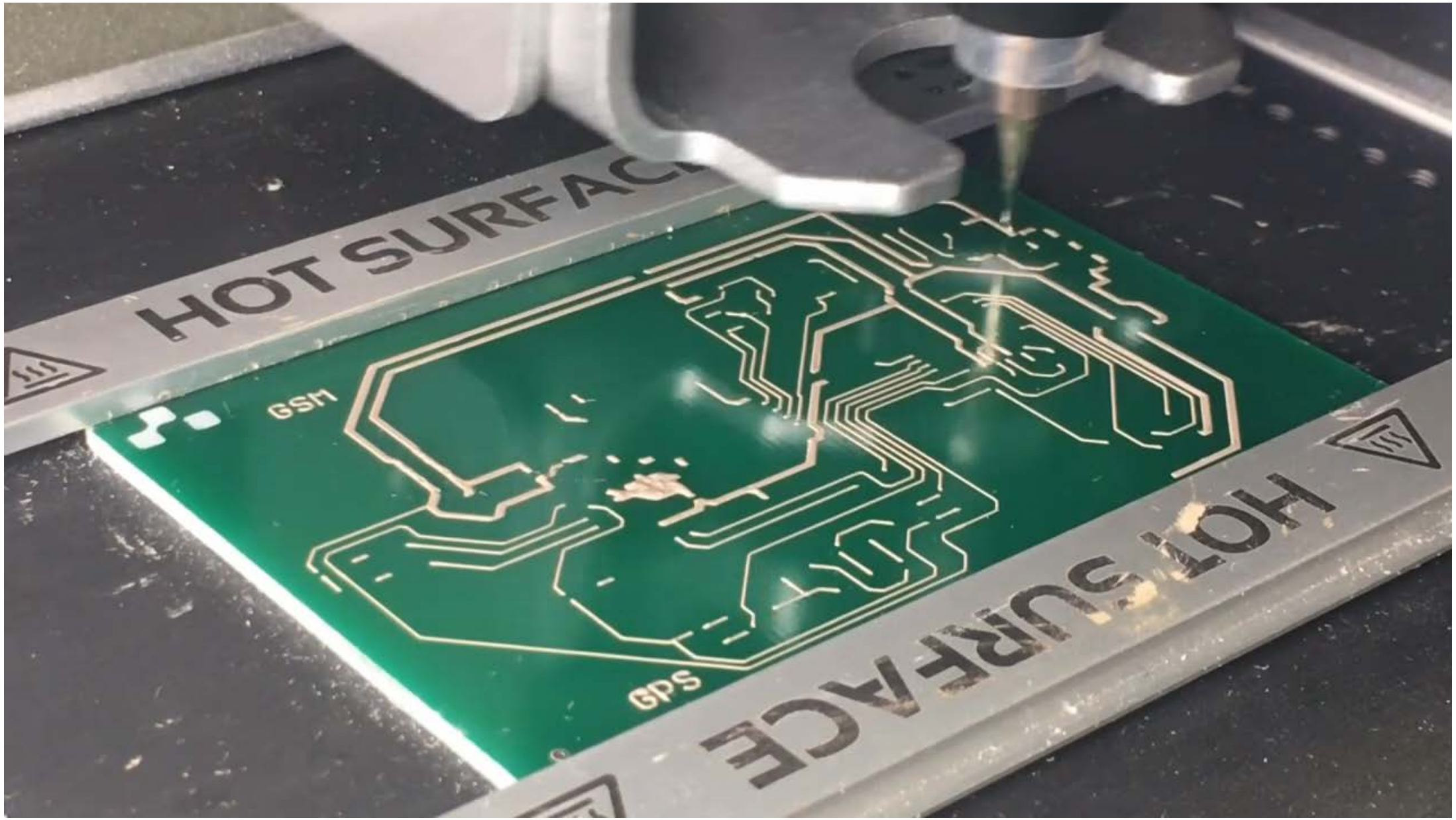
- Pad- exposed metal on a board where a component is soldered.
- Via- a hole in the board used to pass signal from _____
- Plated through hole- a hole on board with annular ring, plated through entire board.
- Trace-a continuous path of _____to connect two points
- _____- package that allows components to be connected on the top/bottom of a board without holes.

Software Tools

- Eagle CAD (https://www.autodesk.com/products/eagle/free-download)
- Altium (free 15 day trial http://www.altium.com/free-trial)
- PCB Artist (http://www.4pcb.com/free-pcb-layout-software/)

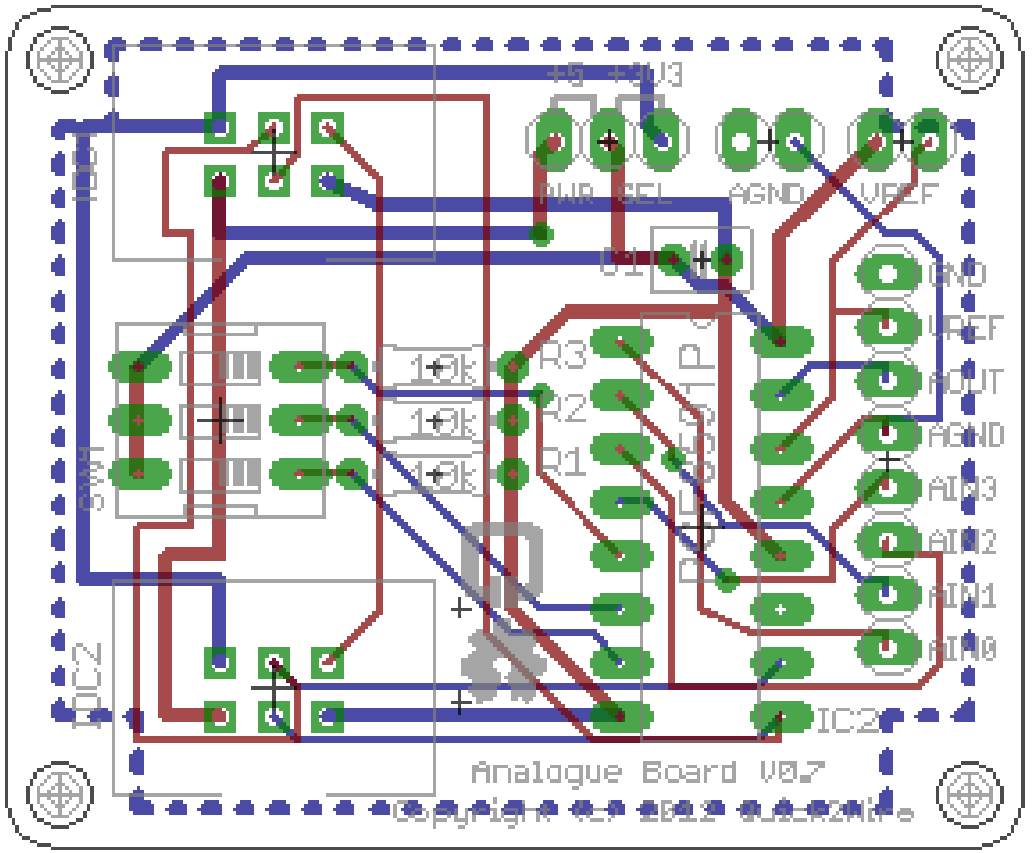


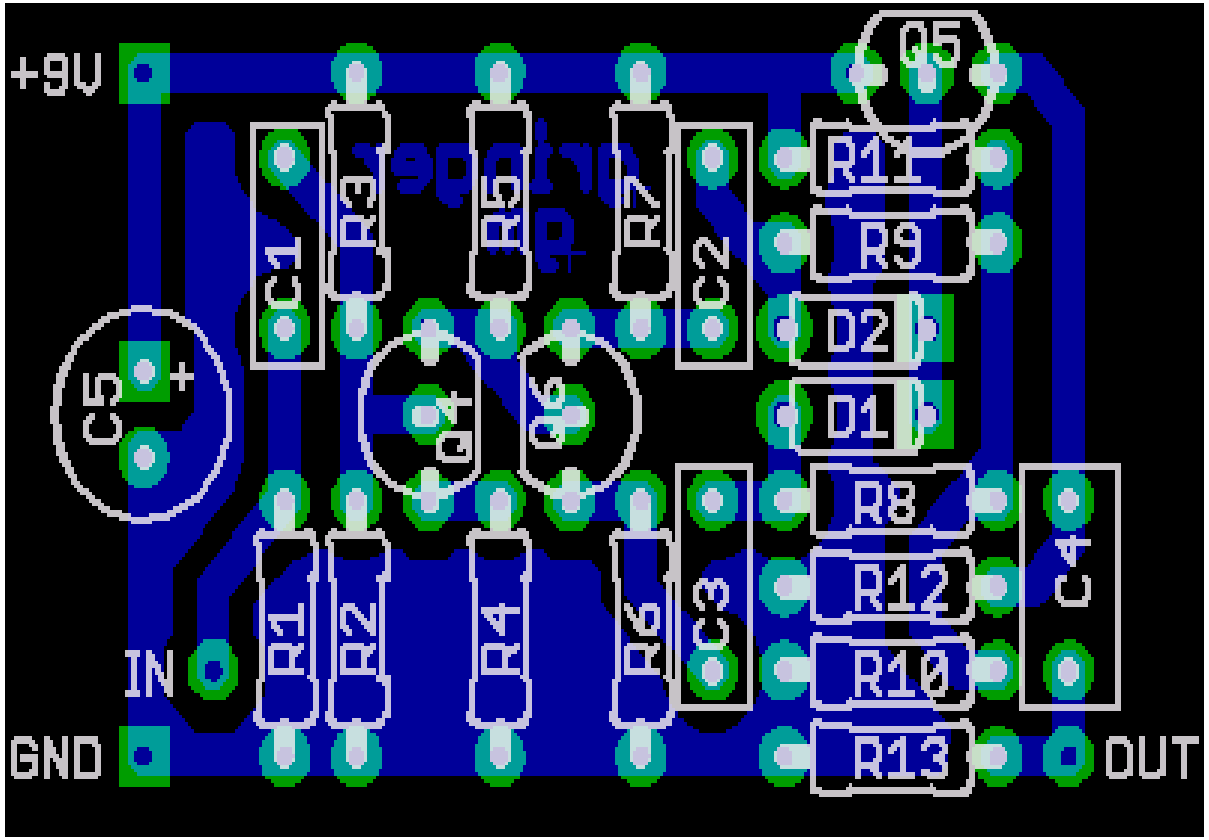
PCB Printing:



PCB Pick and Place:

Ok- done with background! Lets move on to Eagle!

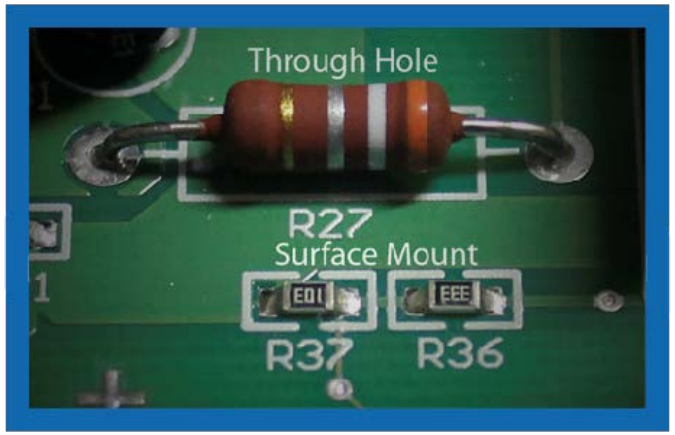


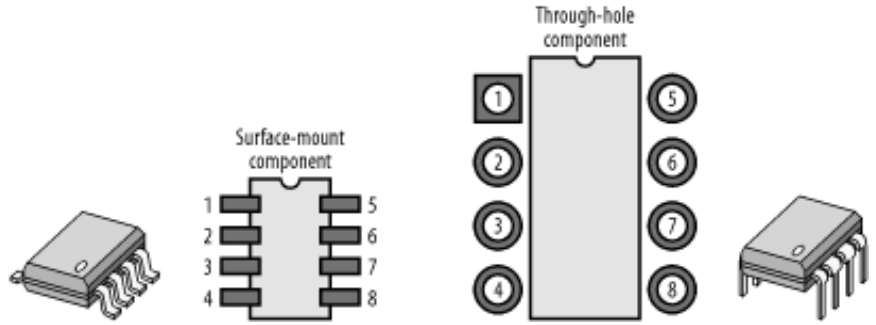


Steps:

1. Design your circuit on paper
2. Import/create component libraries
3. Layout the circuit on the schematic
4. Transfer schematic to board layout
5. Decided component positioning, layer stack manager
6. Layout (autoroute?)
7. Export gerber files
8. Print/Assemble at fab house

Choose Components



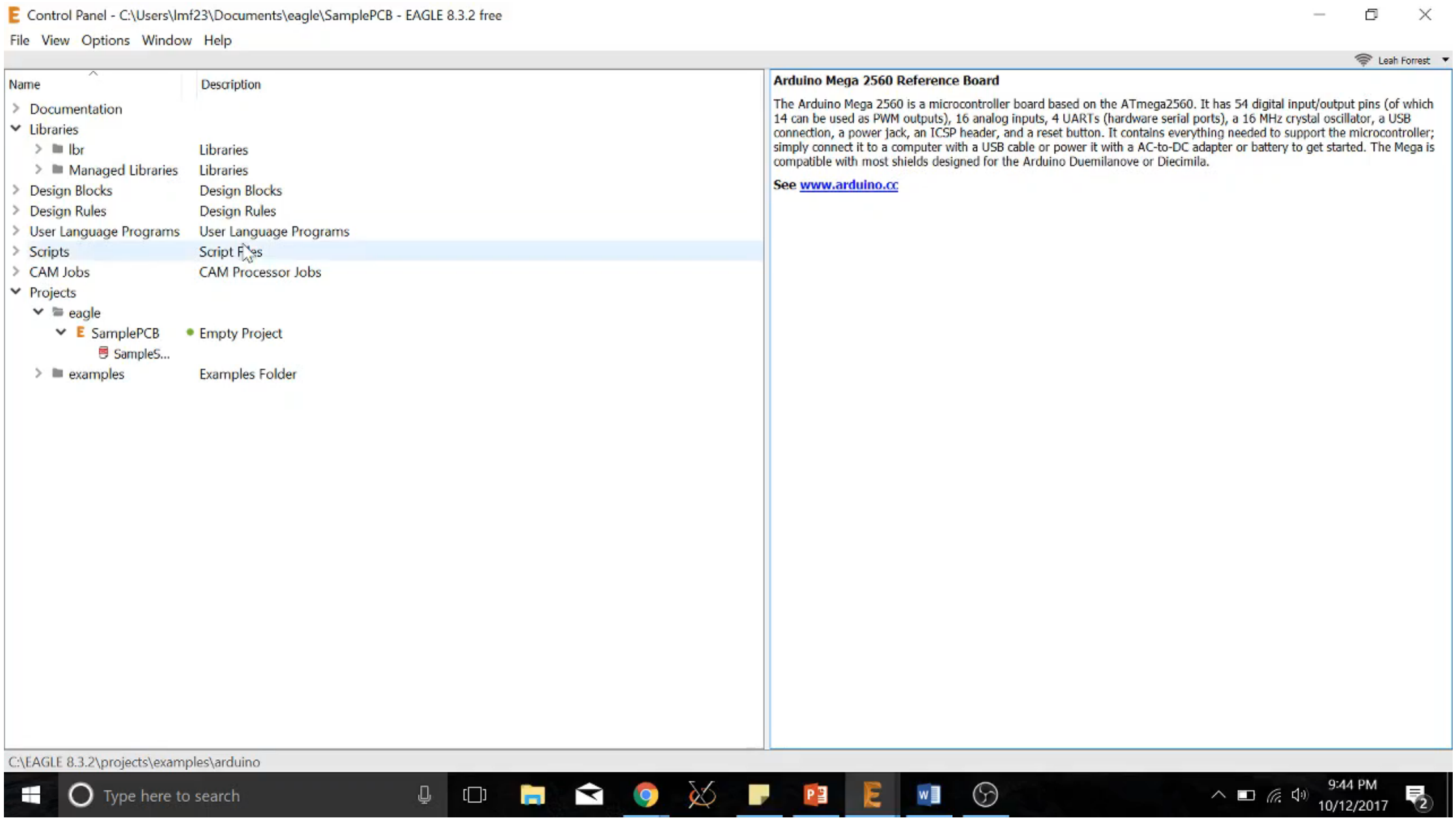


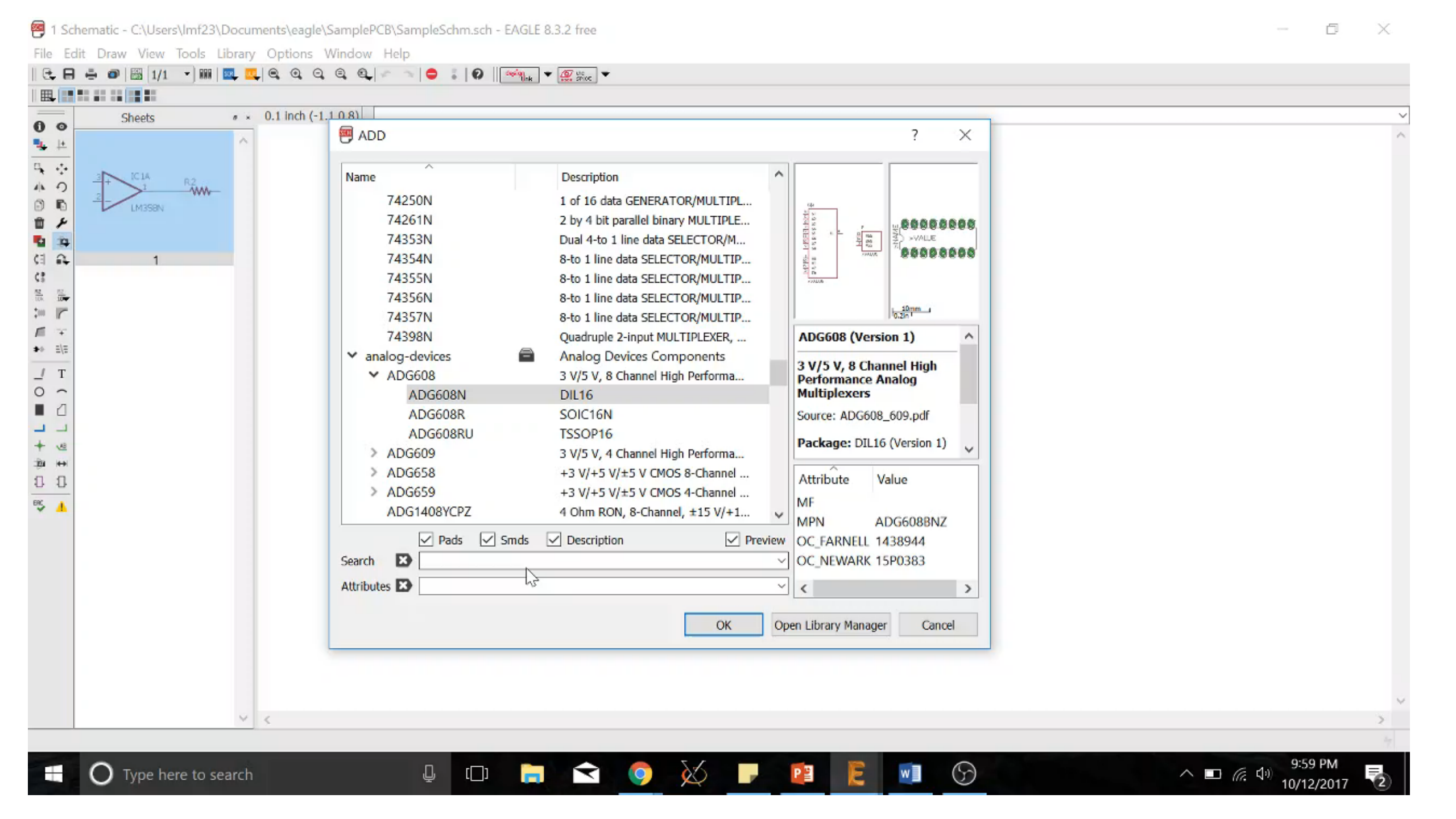
Rectangular Connectors - Board Spacers, Stackers (Board to Board) (143850 items
Rectangular Connectors - Contacts (7081 items)
Rectangular Connectors - Free Hanging, Panel Mount (15635 items)
Rectangular Connectors - Headers, Male Pins (196707 items)
Rectangular Connectors - Headers, Receptacles, Female Sockets (125752 items)
Rectangular Connectors - Headers, Specialty Pin (3766 items)
Rectangular Connectors - Housings (16331 items)

http://www.molex.com/molex/products/family?key=kk_254_rpc_connector_system

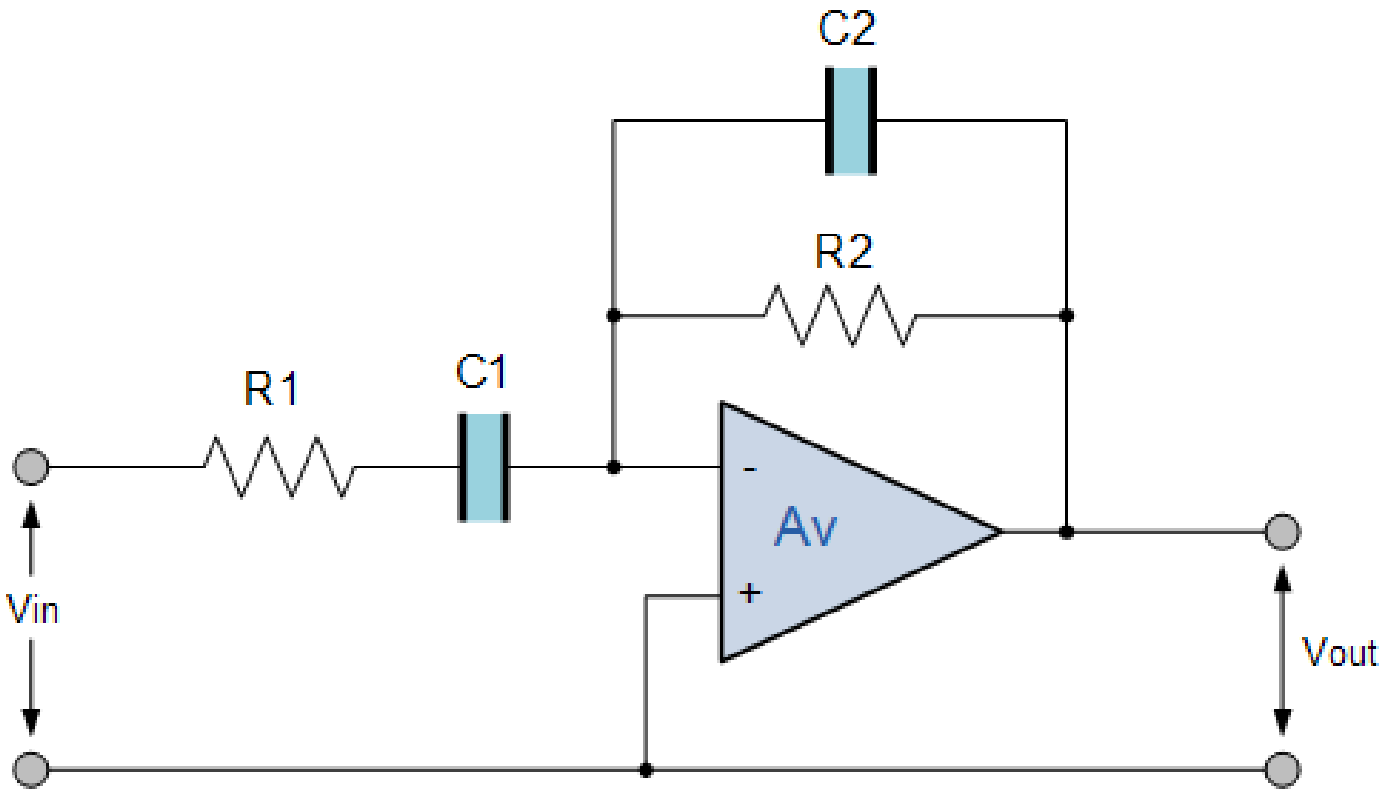
http://www.molex.com/molex/products/family?key=picoblade&channel=products&chanName=family&pageTitle=Introduction

Component libraries

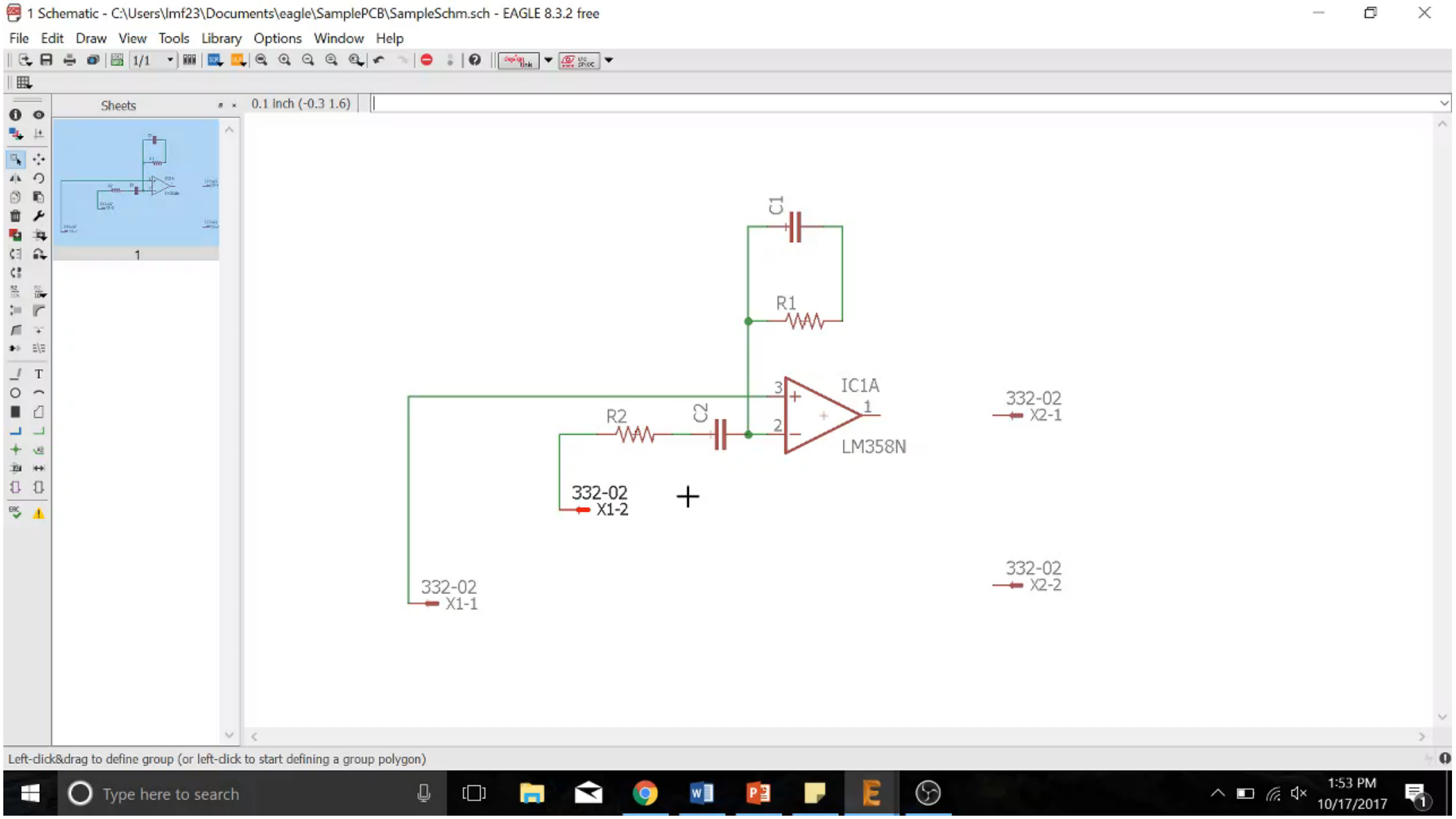



1 Schematic - C:\Users\lmf23\Documents\eagle\SamplePCB\SampleSchm.sch - EAGLE 8.3.2 free
File Edit Draw View Tools Library Options Window Help
Sheets
0.1 inch (-1.1 0.8)
ADD
Name
Description
74250N 1 of 16 data GENERATOR/MULTIPL...
74261N 2 by 4 bit parallel binary MULTIPLE...
74353N Dual 4-to 1 line data SELECTOR/M...
74354N 8-to 1 line data SELECTOR/MULTIP...
74355N 8-to 1 line data SELECTOR/MULTIP...
74356N 8-to 1 line data SELECTOR/MULTIP...
74357N 8-to 1 line data SELECTOR/MULTIP...
74398N Quadruple 2-input MULTIPLEXER, ...
analog-devices Analog Devices Components
ADG608 3 V/5 V, 8 Channel High Performa...
ADG608N DIL16
ADG608R SOIC16N
ADG608RU TSSOP16
ADG609 3 V/5 V, 4 Channel High Performa...
ADG658 +3 V/+5 V/±5 V CMOS 8-Channel ...
ADG659 +3 V/+5 V/±5 V CMOS 4-Channel ...
ADG1408YCPZ 4 Ohm RON, 8-Channel, ±15 V/+1...
ADG608 (Version 1)
3 V/5 V, 8 Channel High Performance Analog Multiplexers
Source: ADG608_609.pdf
Package: DIL16 (Version 1)
Attribute Value
MF
MPN ADG608BNZ
OC_FARNELL 1438944
OC_NEWARK 15P0383
Pads Smds Description Preview
Search
Attributes
OK Open Library Manager Cancel
Type here to search
9:59 PM
10/12/2017

Design your circuit on paper



Schematic


1 Schematic - C:\Users\lmf23\Documents\eagle\SamplePCB\SampleSchm.sch - EAGLE 8.3.2 free
File Edit Draw View Tools Library Options Window Help
Sheets
0.1 inch (-0.3 1.6)
C1
R1
R2
C2
IC1A
LM358N
3
1
2
332-02
X1-2
332-02
X1-1
332-02
X2-1
332-02
X2-2
Left-dick&drag to define group (or left-dick to start defining a group polygon)
Type here to search
1:53 PM
10/17/2017

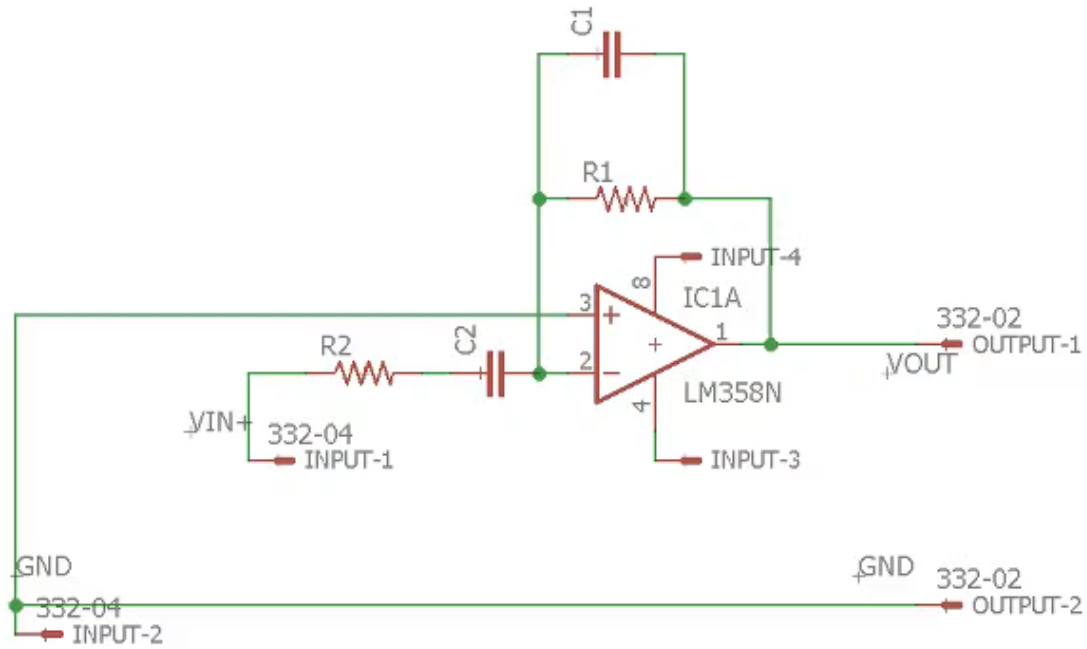
1 Schematic - C:\Users\lmf23\Documents\eagle\SamplePCB\SampleSchm.sch - EAGLE 8.3.2 free

File Edit Draw View Tools Library Options Window Help

Sheets

1

0.1 inch (-3.2 3.6)

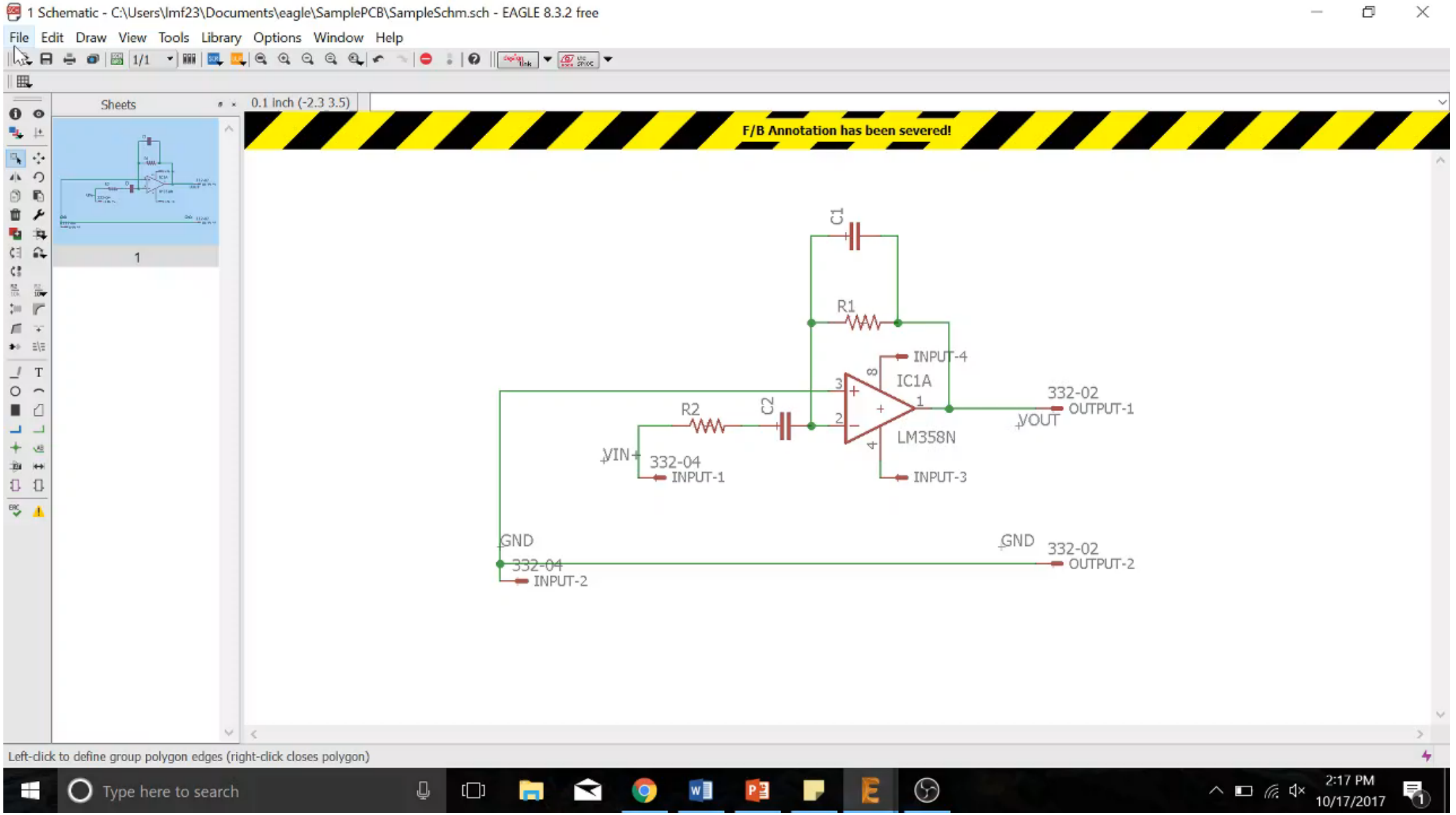


ERC: 8 errors/warnings. Left-click to select object to change name

Type here to search

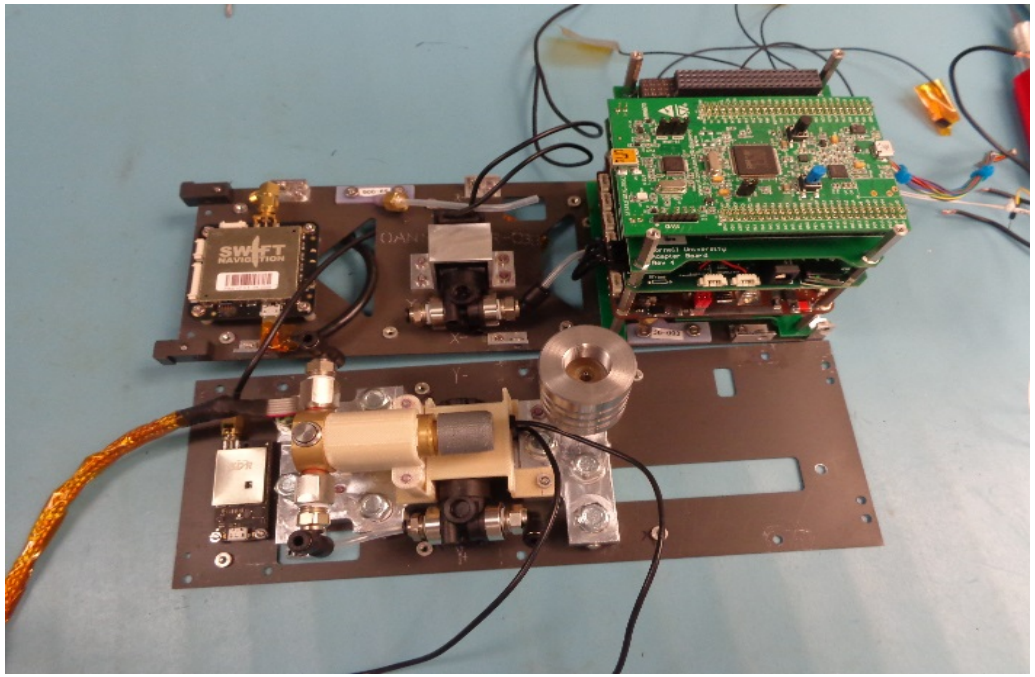
2:15 PM
10/17/2017

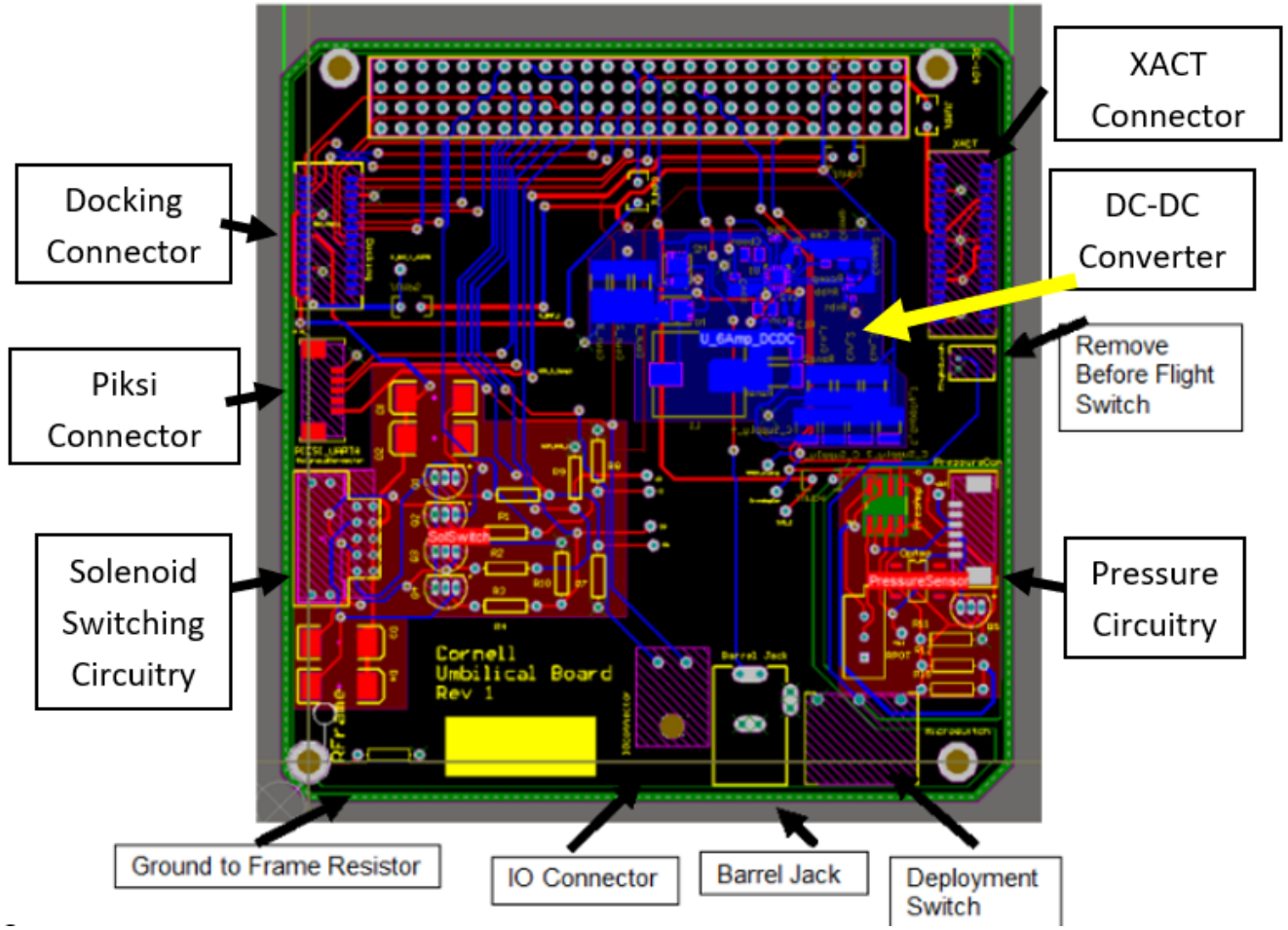
Transfer Schematic to Board



Decide Physical Parameters

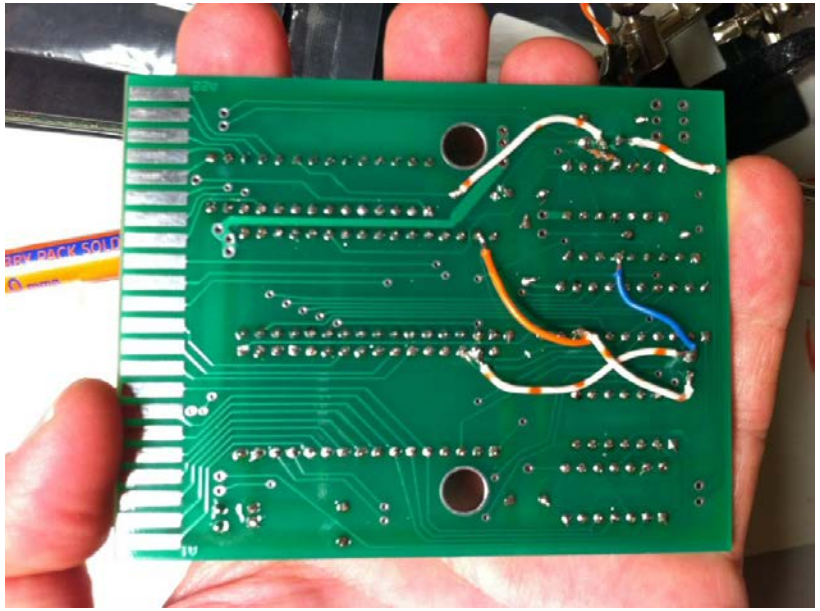
- Component Positioning
- Layer Stack Manager
- Board Size/Outline

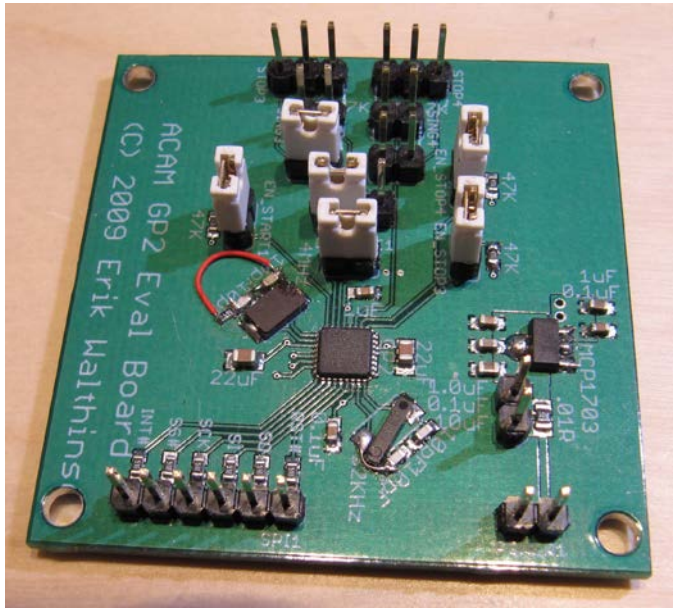




Layout

- Trace Width: http://www.4pcb.com/trace-width-calculator.html
- Jumpers
- Debugging Pads





Inputs:

Current	.5	Amps
Thickness	2	oz/ft^2 ▾

Optional Inputs:

Temperature Rise	10	Deg C ▾
Ambient Temperature	25	Deg C ▾
Trace Length	1	inch ▾

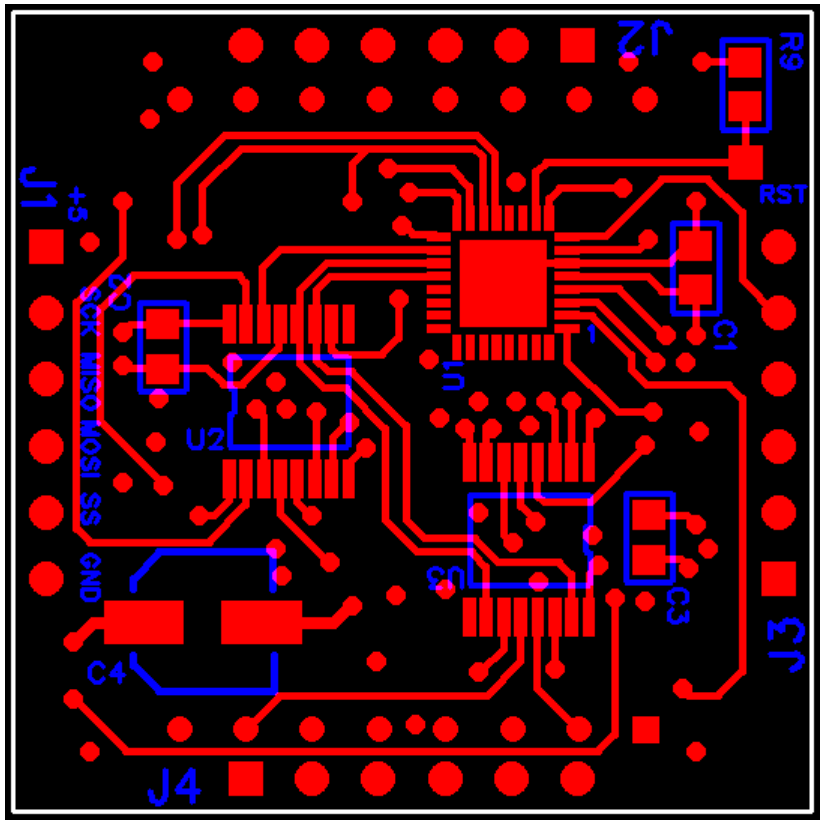
Results for Internal Layers:

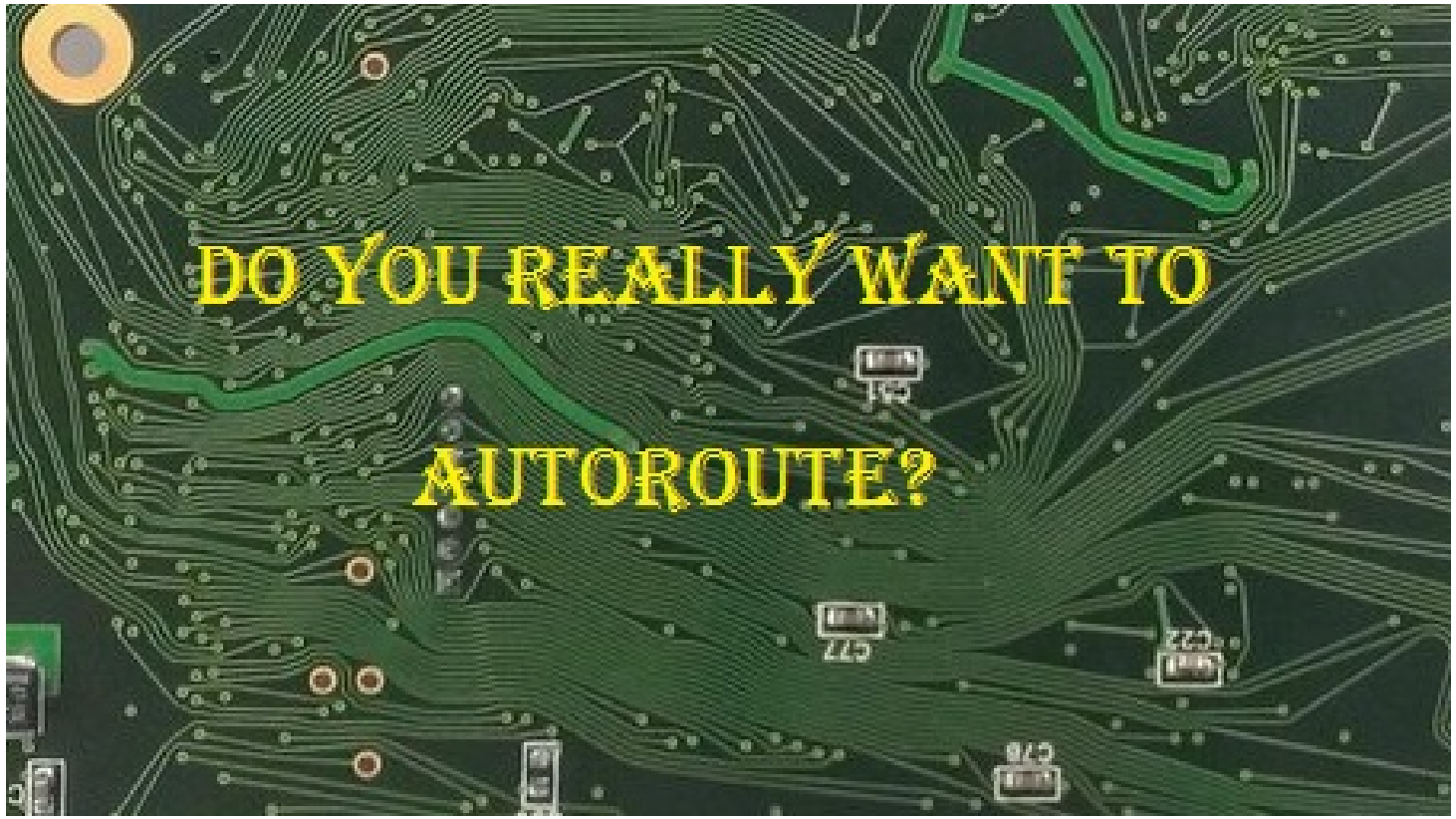
Required Trace Width	5.91	mil ▾
Resistance	0.0427	Ohms
Voltage Drop	0.0213	Volts
Power Loss	0.0107	Watts

Results for External Layers in Air:

Required Trace Width	2.27	mil ▾
Resistance	0.111	Ohms
Voltage Drop	0.0555	Volts
Power Loss	0.0278	Watts

Should I use Autoroute?



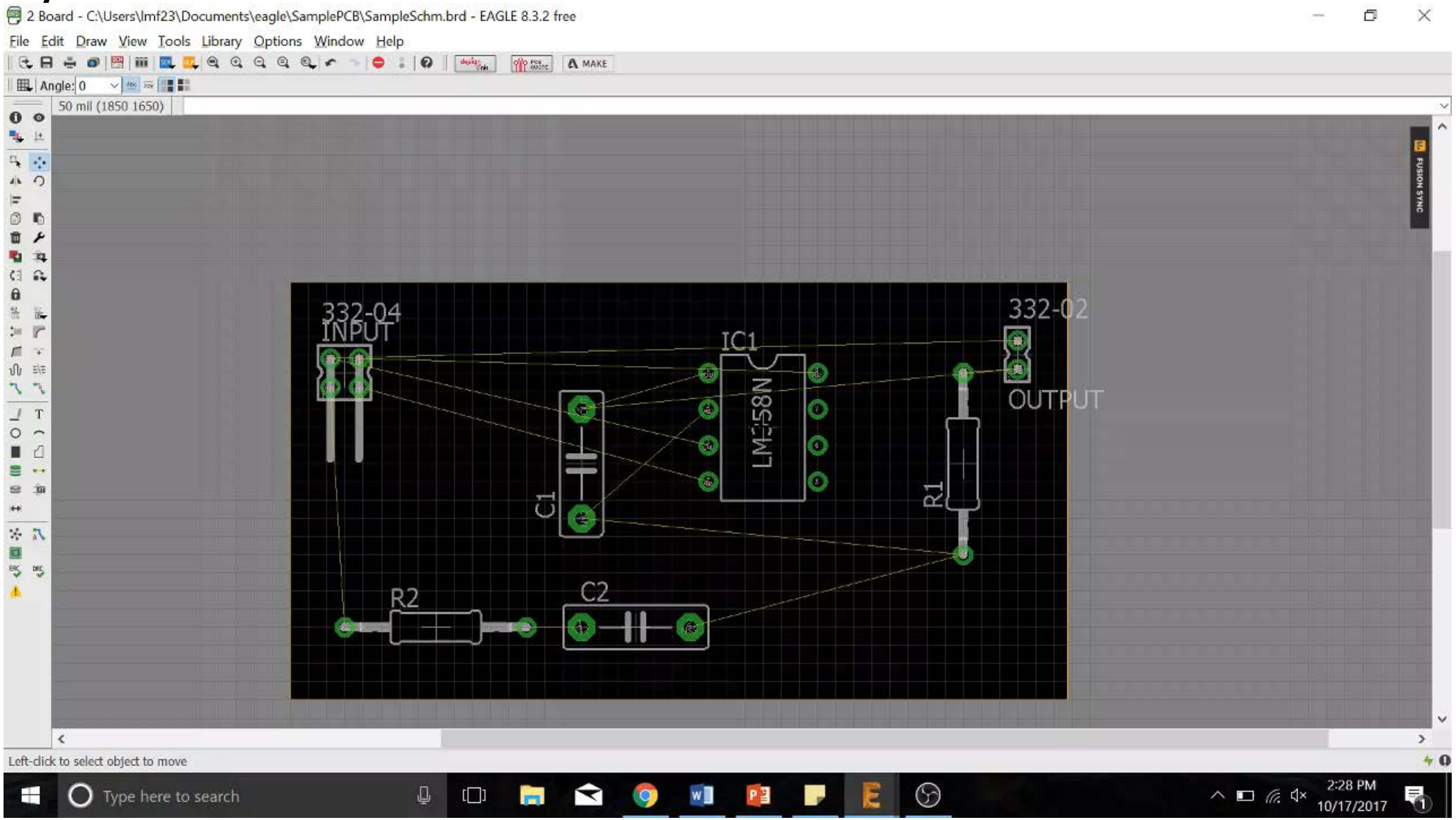


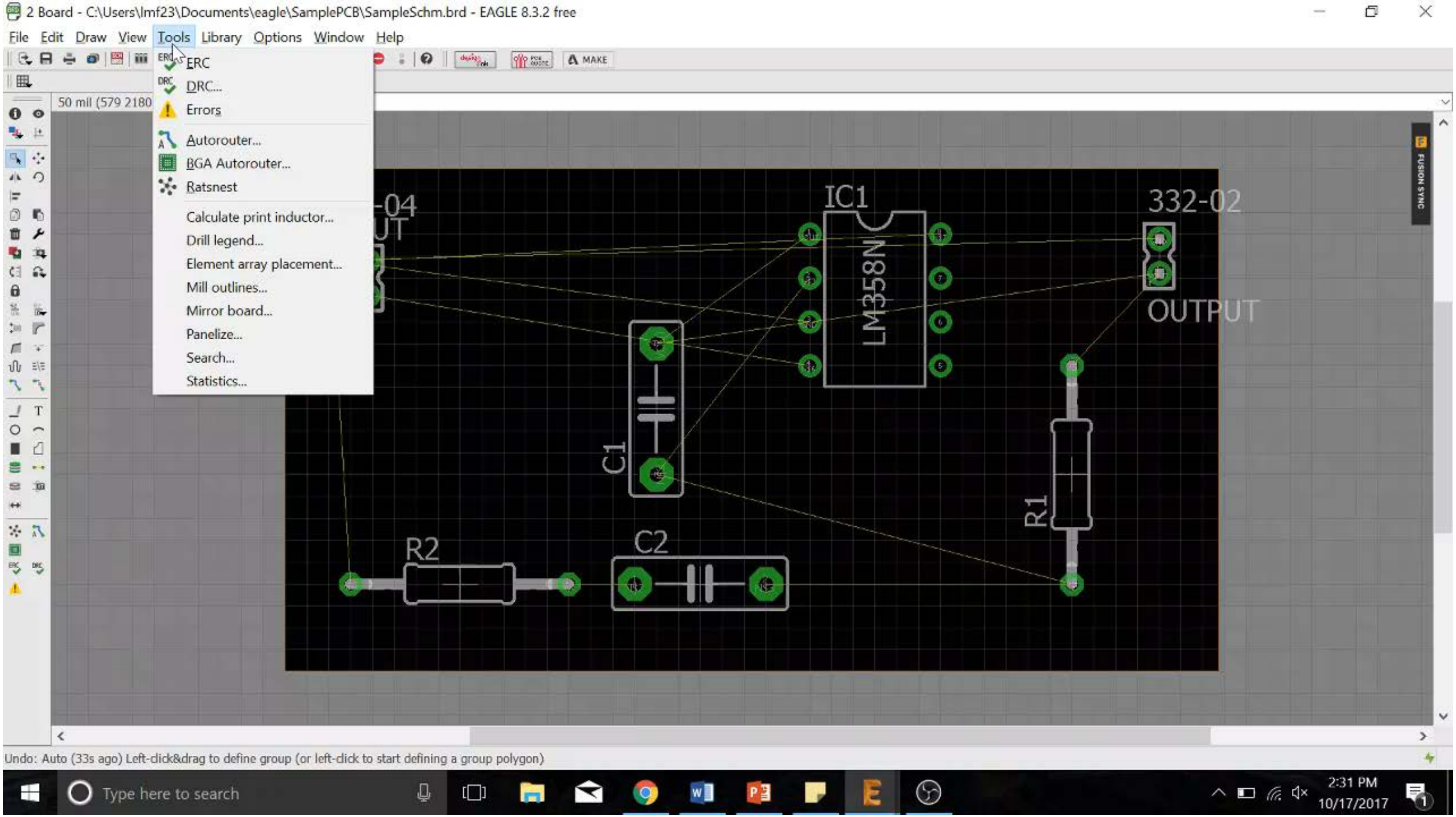
Layout Tips:

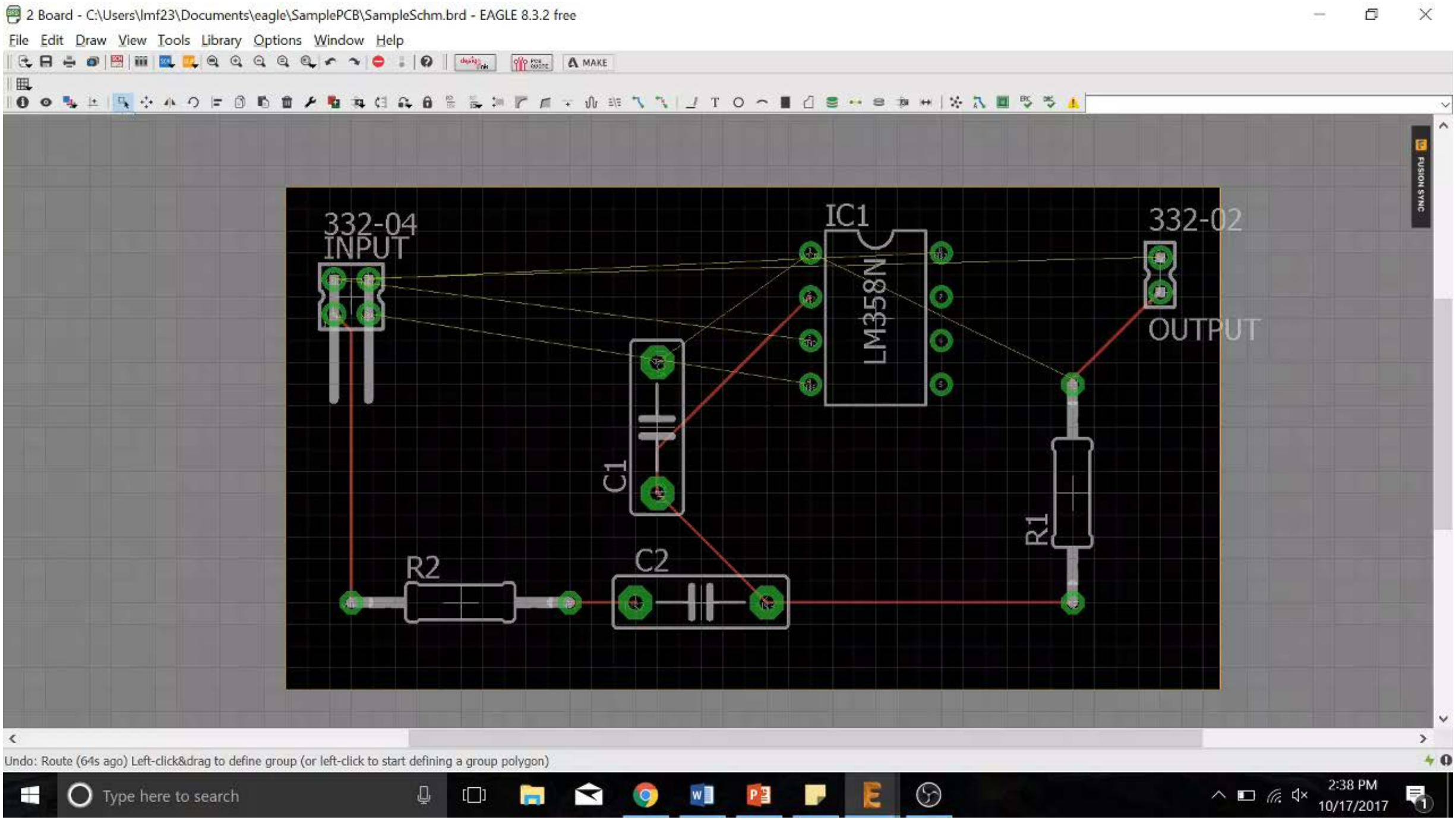
- Keep digital and noisy traces away from analog signal traces
- Keep analog and digital grounds separate (for multi layer boards- isolate each ground plane!)
- Beware of loops- especially for traces that have high frequency signals!

  smaller loops-> smaller V=L*(di/dt), so less voltage spike

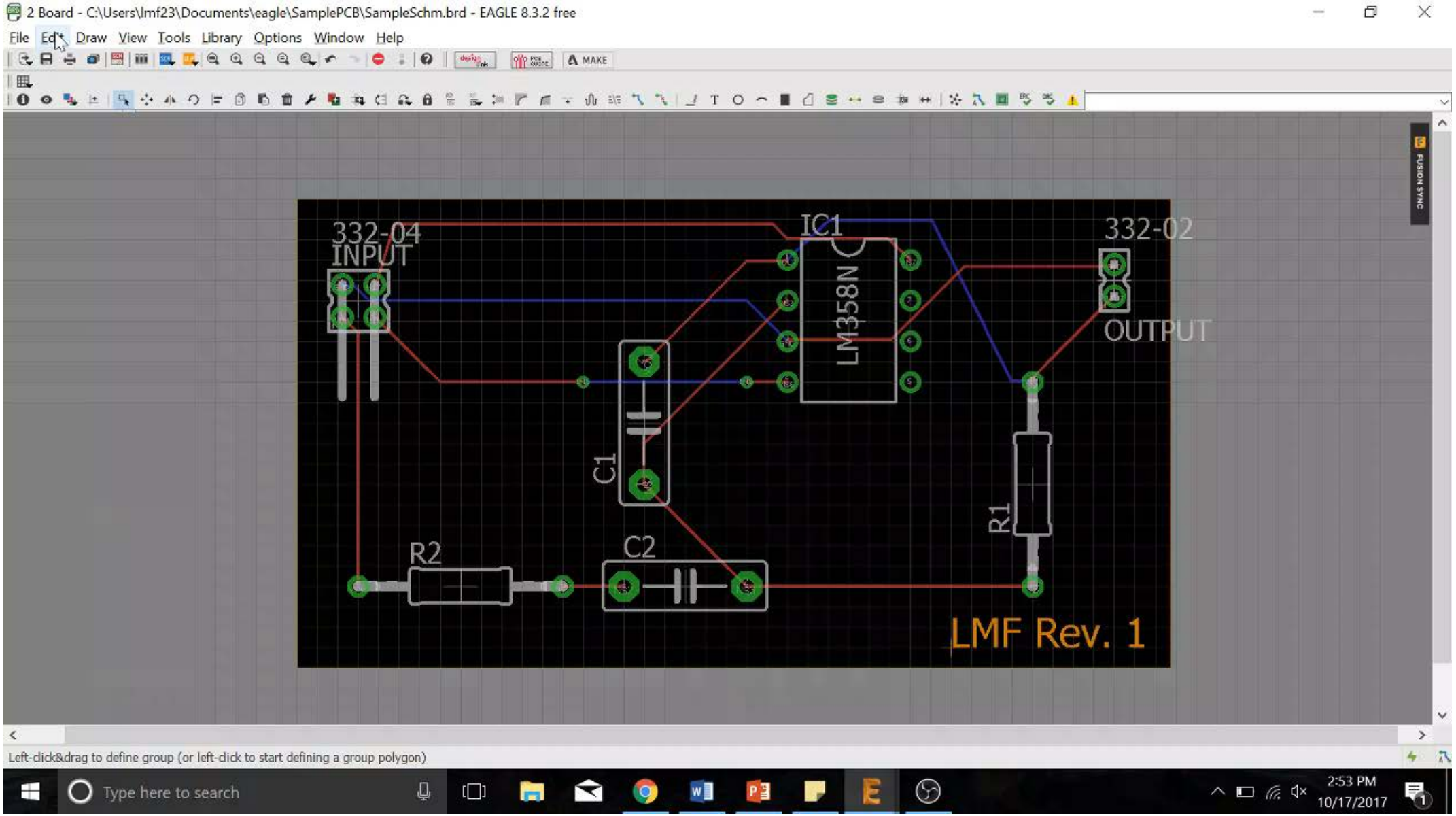
Layout


2 Board - C:\Users\lmf23\Documents\eagle\SamplePCB\SampleSchm.brd - EAGLE 8.3.2 free
File Edit Draw View Tools Library Options Window Help
Angle: 0
50 mil (1850 1650)
332-04
INPUT
IC1
LM358N
332-02
OUTPUT
C1
R1
R2
C2
Left-click to select object to move
Type here to search
2:28 PM
10/17/2017


2 Board - C:\Users\lmf23\Documents\eagle\SamplePCB\SampleSchm.brd - EAGLE 8.3.2 free
File Edit Draw View Tools Library Options Window Help
ERC
DRC...
Errors
Autorouter...
BGA Autorouter...
Ratsnest
Calculate print inductor...
Drill legend...
Element array placement...
Mill outlines...
Mirror board...
Panelize...
Search...
Statistics...
50 mil (579 2180
IC1
LM358N
332-02
OUTPUT
C1
C2
R1
R2
Undo: Auto (33s ago) Left-click&drag to define group (or left-click to start defining a group polygon)
Type here to search
2:31 PM
10/17/2017

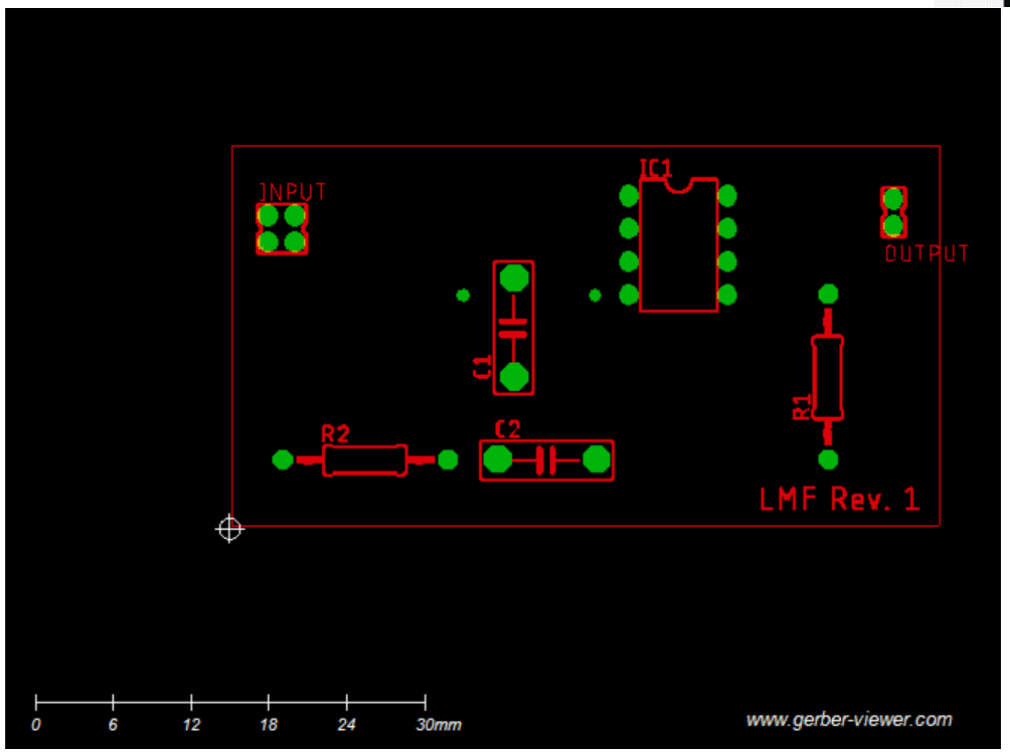

2 Board - C:\Users\lmf23\Documents\eagle\SamplePCB\SampleSchm.brd - EAGLE 8.3.2 free
File Edit Draw View Tools Library Options Window Help
MAKE
332-04
INPUT
IC1
LM358N
332-02
OUTPUT
C1
C2
R1
R2
Undo: Route (64s ago) Left-click&drag to define group (or left-click to start defining a group polygon)
Type here to search
2:38 PM
10/17/2017

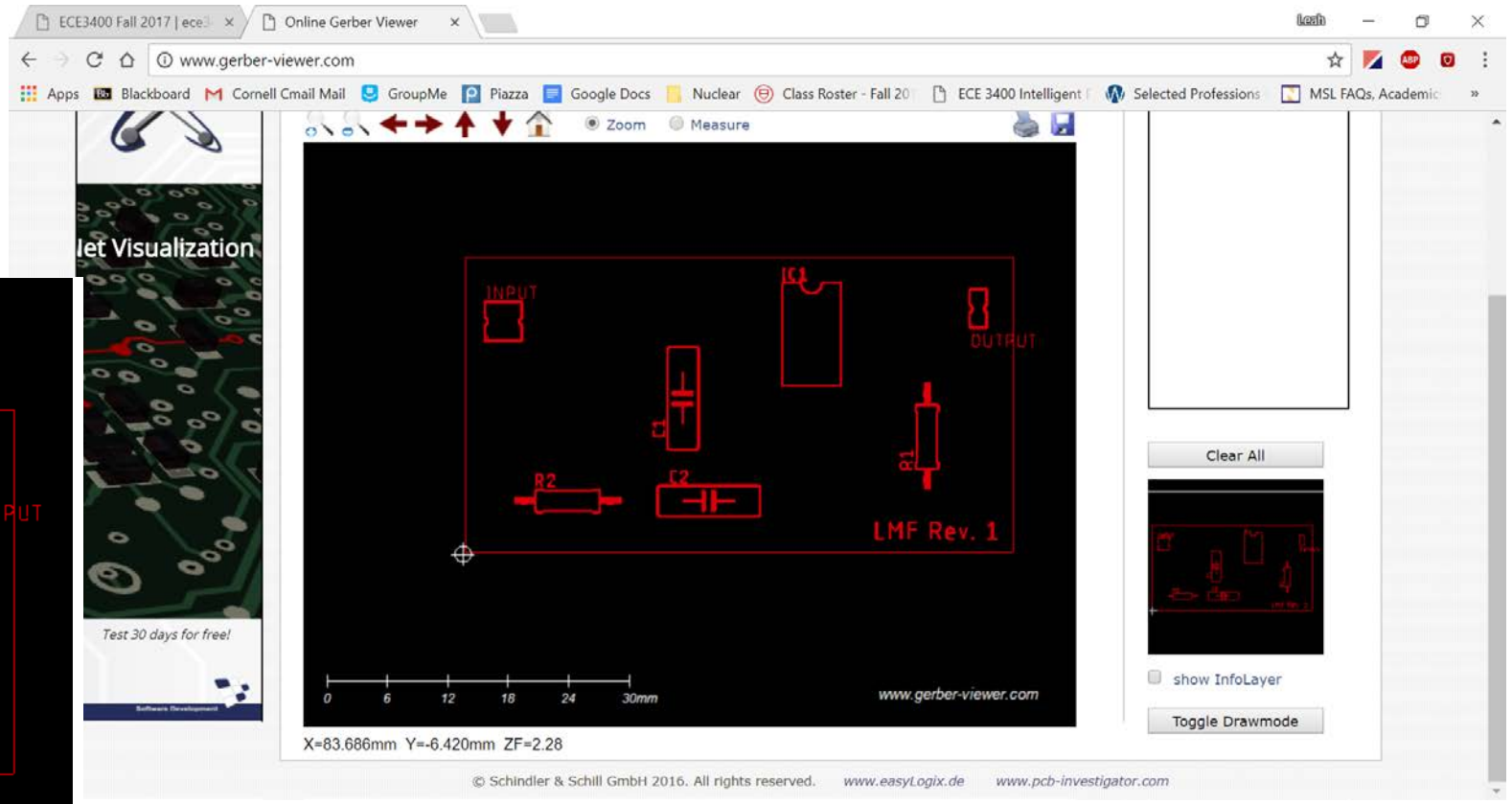
Gerber Files



Exporting Gerber Files

- https://sourceforge.net/projects/gerbv/
- http://www.gerber-viewer.com/





Suppliers/Fab Houses/Do I need assembly?

- Suppliers:
  - Barebones PCB from Advanced Circuits (http://www.4pcb.com/)
  - Cornell Maker lab (not for class stuff, but personal PCB's ok)

Specifications	Standard Specs	Custom Specs
Layer Count	0 - 10 Layers	0 - 40 Layers
Turn Time	Same Day - 5 Day	Same Day - 4 Weeks
Quantity Req.	1 - 10000+	1 - 10000+
Materials	FR-4	FR-4 / Rogers / Polyimide / Aluminum Clad / High-Temp. FR4 / *Others*
Plate Finish	Lead-Free HAL*	Electrolytic Hard Gold / Soft Gold / ENIG / Nickel / Immersion Silver / Leaded & Lead-Free HAL
Cert. / Qualifications	IPC Class 2 - A600	IPC6012 Class 2-3A / IPC6018 Class 3 / MIL-PRF-31032 / MIL-PRF-55110 / ISO 9001:2008 / AS9100C / *Others*
Board Thickness	.031" / .062" / .093" / .125"	Full Range Available
Copper Weight	1 oz. Inner / Up to 2 oz. Outer	0.5 - 4 oz. Inner / 1-20 oz. Outer
Trace / Space	5 / 5 Mils	Down to 2.75 / 3 Mils
View Full Chart	See More »	

BareBones™

2 Layer - **1 Day** Turn

12" x 16" Max Board Size

Min. Order Quantity: 1

FR-4 .062" Material

I oz. Cu.

Tin Finish

No Mask *(bare)*

No Legend

Rectangle Shape Only

Place Order »

How can you use a PCB on your robot?

- Amplifier/filter for your microphone circuit
- Amplifier for your treasure circuit
- A tidy base station voltage divider
- The whole stack…?

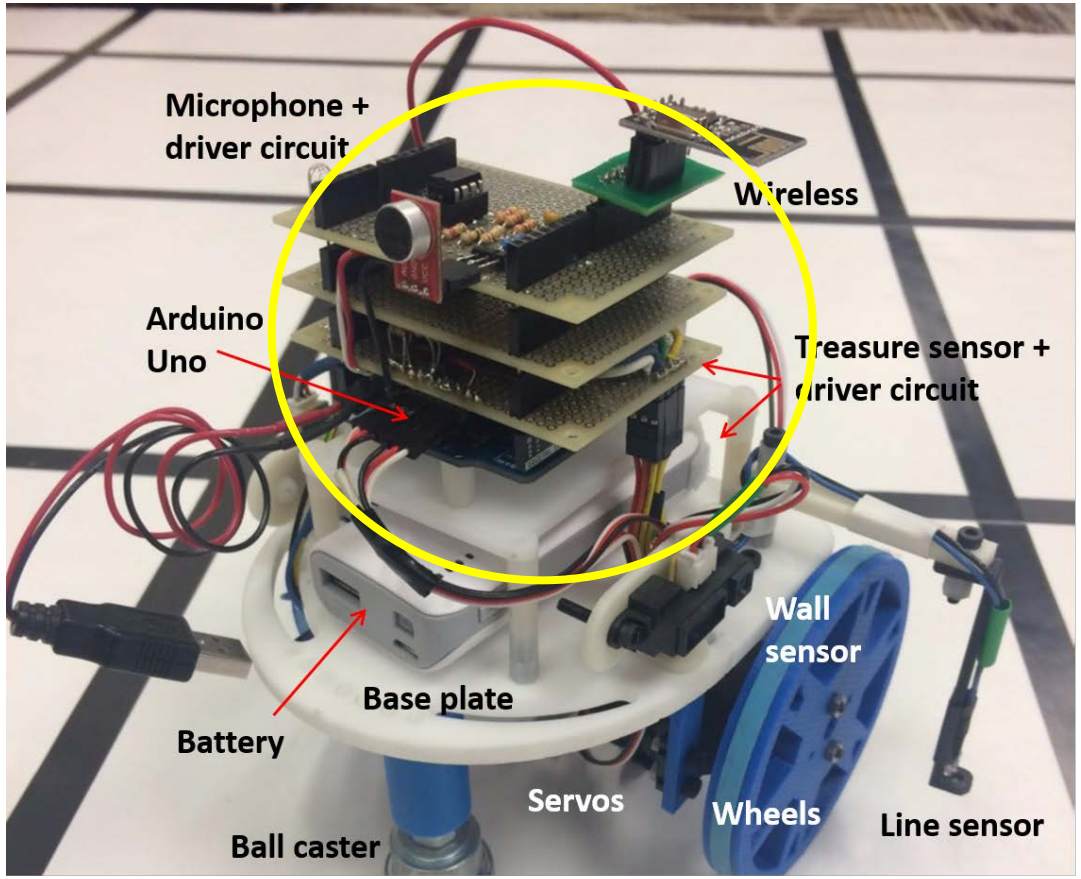
Grading of System Design and Documentation

ECE 3400, Fall 2017

The TA's will award up to 25 points for good system design.

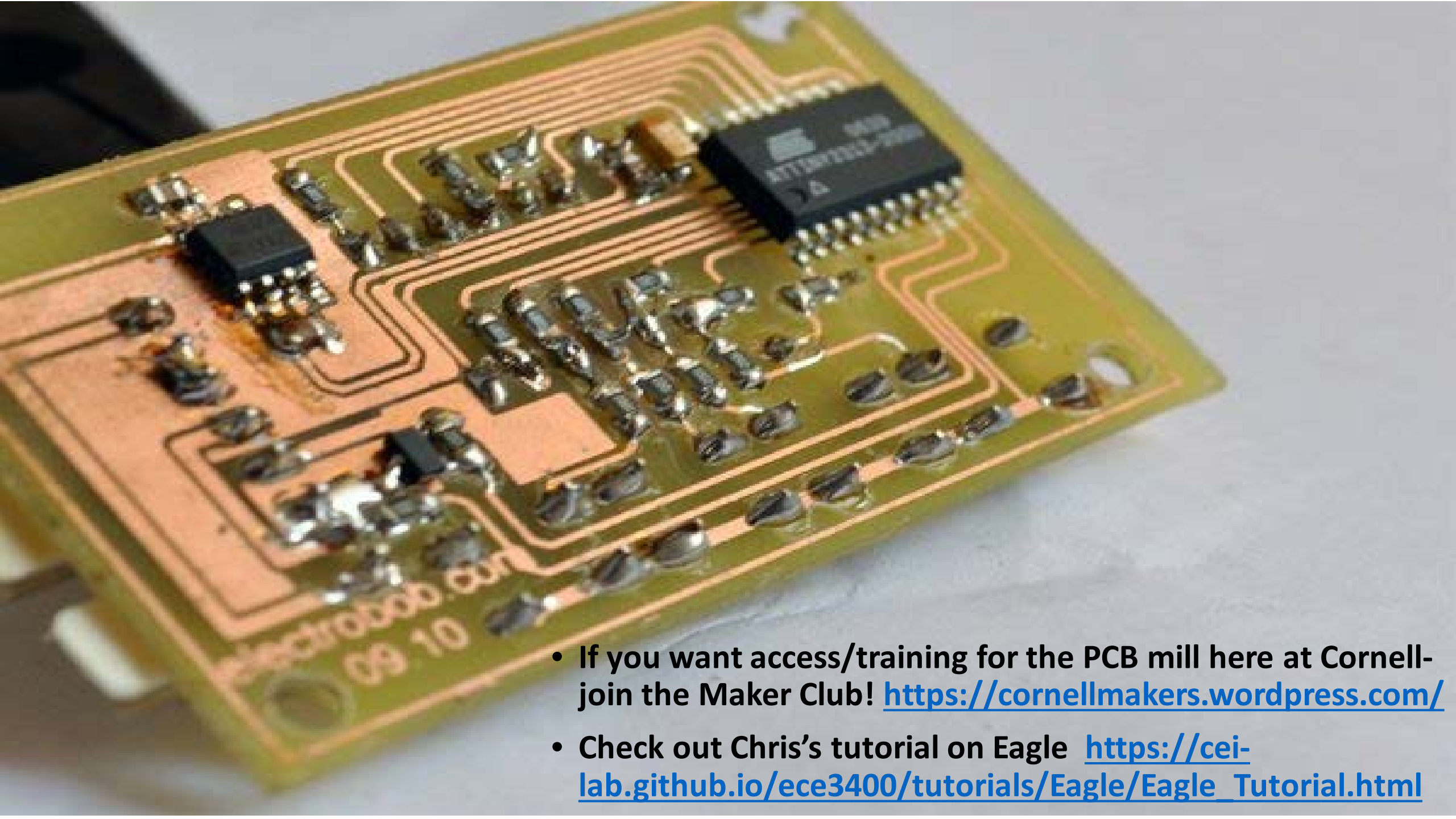
All members on a team will not necessarily be awarded the same number of points.

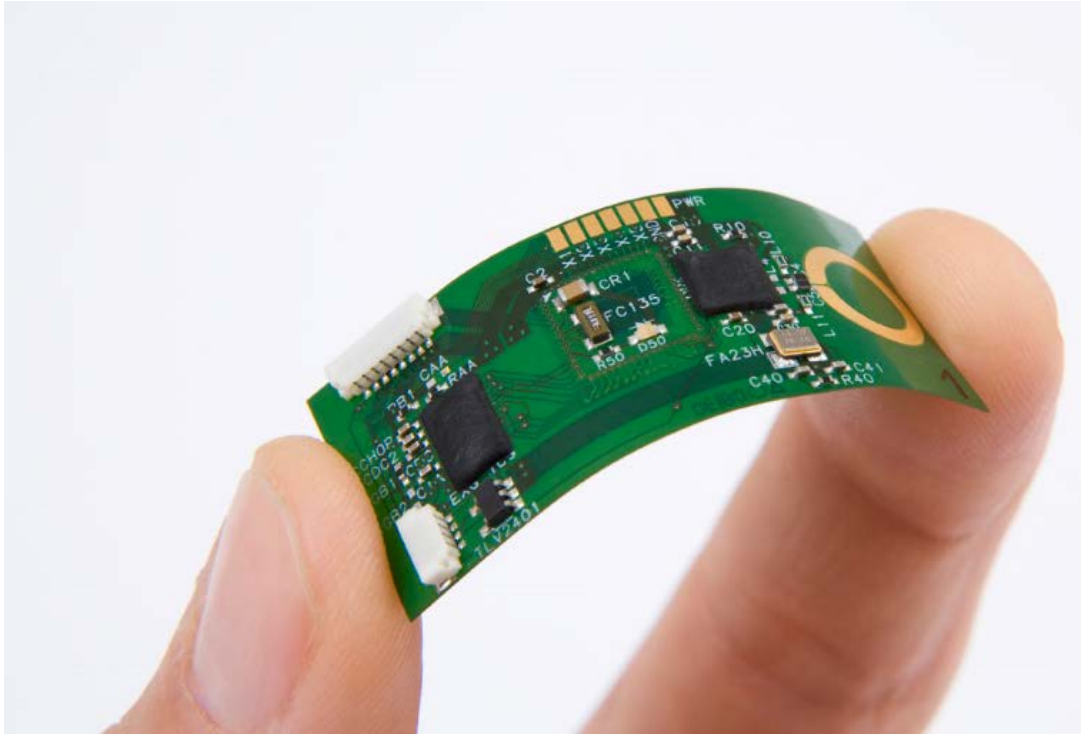
To earn these points you can for example implement electronic circuits that improve the accuracy of the maze mapping, efficient search strategies, nice FPGA implementation and a great screen display, or new mechanical components to make the robot faster or easier to maintain. If you have tons of spare time, consider using two cooperative robots to map the maze. The sky's the limit!



Tips and Tricks to PCBs

- Label each pcb you make- you can draw a blank square and write with sharpie or put a revision number label on the board.
- Put notes in your schematic- and keep a burn list!
  - https://volersystems.com/v-2010/91-pcb-layout-checklist/
- Try to think about noise and grounding



- If you want access/training for the PCB mill here at Cornell- join the Maker Club! https://cornellmakers.wordpress.com/
- Check out Chris's tutorial on Eagle https://cei-lab.github.io/ece3400/tutorials/Eagle/Eagle_Tutorial.html

Happy PCB-ing!