Senior Design Workshop

PCB Design with EAGLE

Dr. Feng Li

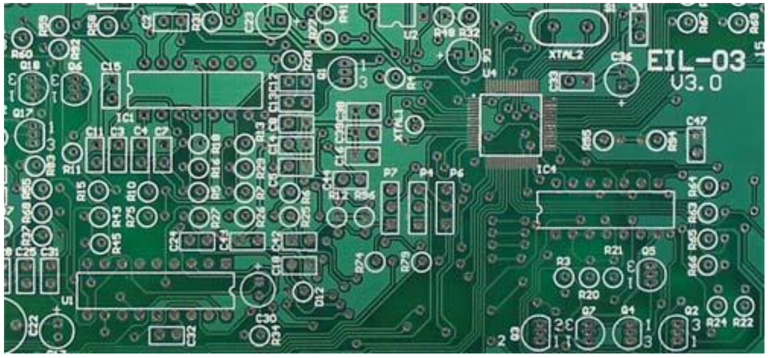
Dept. of Electrical & Computer Engineering

University of Idaho

Outline

- PCB introduction
- CadSoft EAGLE introduction
- Schematic design
- Board layout and routing
- CAM processor export
- DFM
- Fabrication
- Assembly

What is PCB?



- Printed Circuit Board
  - Mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from coper sheets laminated onto a non-conductive substrate.
- Two purposes:
  - A place to mount the component
  - Provides the means of electrical connection between the components
- PCBs can be
  - Single sided (one copper layer)
  - Double sided (two copper layers)
  - Multi-layer

PCB Terminology – Component Packages

- Through-hole
  - Soldered to the opposite side of the board
- SMD/SMT (surface mount device/technology)
  - Soldered in the same side of the board
  - Can be mounted on both sides of the PCB
  - Smaller than the thru-hole type-smaller & denser PCB
  - Some common form factors
    - 0805 (means 0.08”x0.05”)
    - 1206
    - 1210

http://www.pcb.electrosoft-engineering.com/04-articles-custom-system-design-and-pcb/01-printed-circuit-board-concepts/printed-circuit-board-pcb-concepts.html

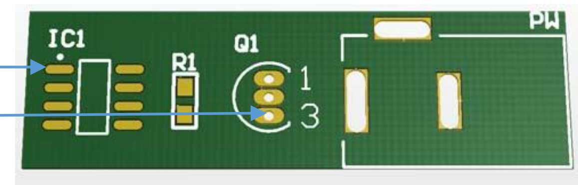
PCB Terminology

- Pad: a small surface of copper where component will be soldered to the board
- Via: plated hole that allows the current to pass thru the board
- Track (trace): conductive path connecting 2 points (pads, vias)
- Soldermask: a layer of insulating lacquer covering both surfaces of the board to prevent the solder to short-circuit two tracks from different nets.
- Silkscreen (overlay): letters printed on the final board, e.g., solder mask

PCB Terminology

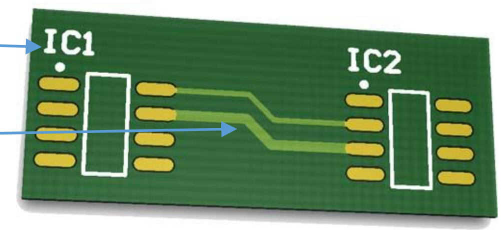
SMD pads

Thru-hole pads



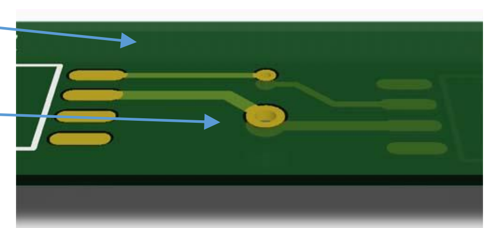
Silkscreen

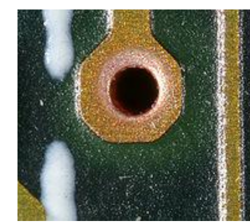
Copper tracks



Soldermask (stop mask)

Via

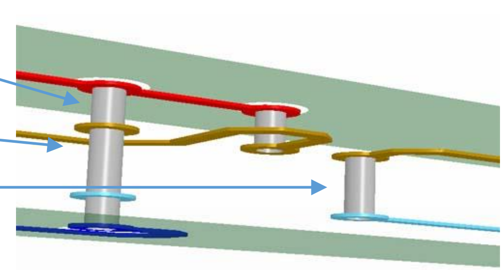




Thru-hole (full stack) via

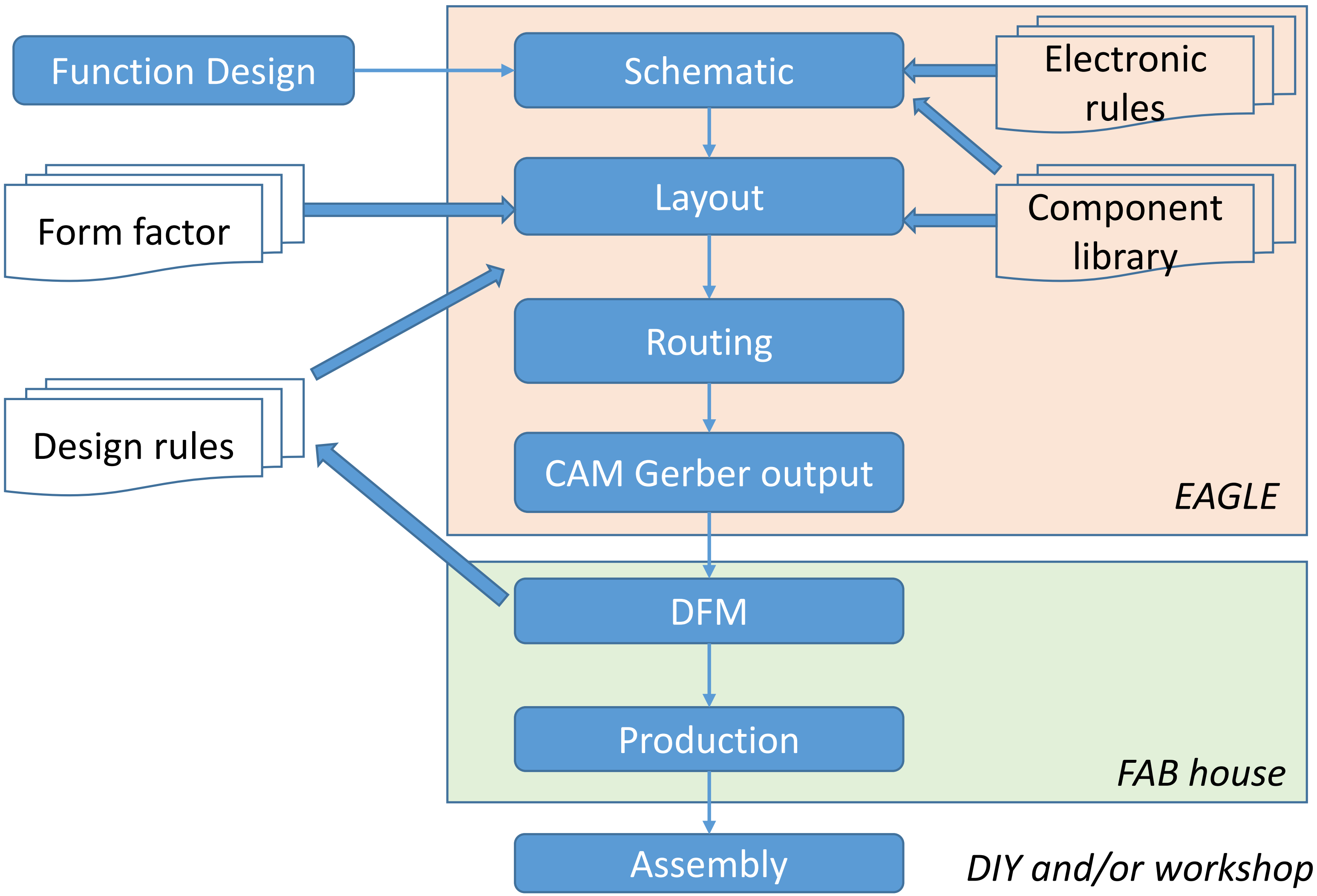
1-2 blind via

2-3 buried via



http://www.pcb.electrosoft-engineering.com/04-articles-custom-system-design-and-pcb/01-printed-circuit-board-concepts/printed-circuit-board-pcb-concepts.html
http://en.wikipedia.org/wiki/Via_(electronics)

PCB Process Flow



What is EAGLE?

- Easily Applicable Graphical Layout Editor
  - Easy to use CAD tool for designing PCBs
  - CadSoft
- Consist of three main modules:
  - Schematic editor
  - Layout editor
  - Autorouter
- It runs under Windows, Linux, and MacOSX
- It comes with an extensive library of components

Three Versions

- EAGLE light
  - Limitations:
    - 1 schematic sheets;
    - 2 signal layers (top & bottom)
    - 100x80mm routing area
  - It can be used for free

- EAGLE standard
  - 99 schematic sheets
  - 4 signal layers
  - 160x100mmrouting area

- EAGLE Professional: full featured version
  - Up to 1600x1600mm
  - Up to 16 routing layer
  - Up to 999 sheets per schematic

Download and Installation

- Current edition: 7.1.0
- http://www.cadsoftusa.com/download-eagle/
- Professional version (6.5.0) available in the Senior Design Lab
  - Contact John Jacksha (J.J.)

- 7.1 and 6.5 may have compatibility issues
  - Begin with 6.5 light version and use Pro version in lab
  - Begin with 7.1 light version and order your standard or professional version

Installing Optional Libraries

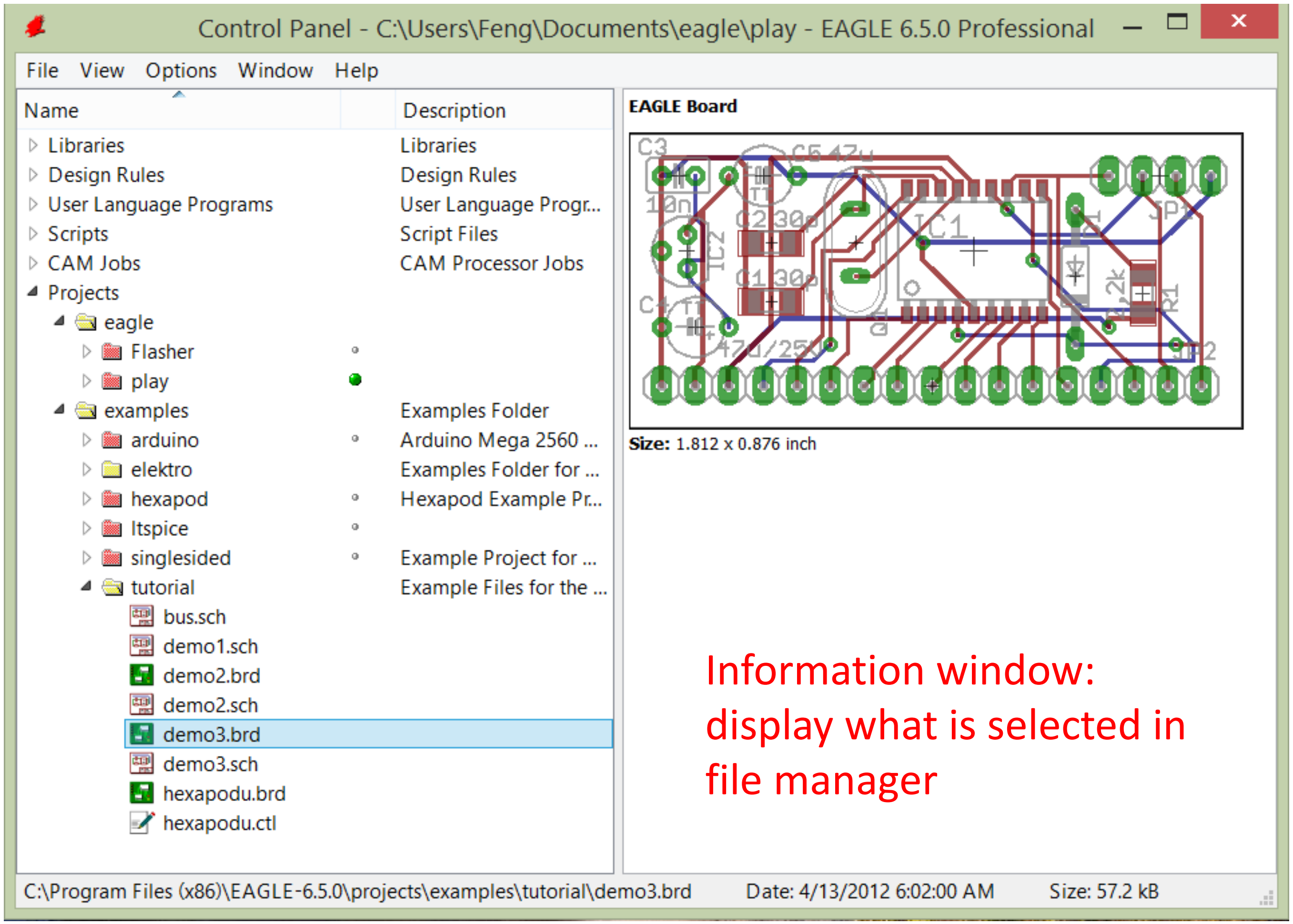
- SparkFun library
  - contains footprints for many common devices
  - helps narrow the choices
  - https://github.com/sparkfun/SparkFun-Eagle-Libraries
  - C:\Program Files (x86)\EAGLE-6.5.0\lbr
- Adafruit
  - http://github.com/adafruit/Adafruit-Eagle-Library
  - C:\Program Files (x86)\EAGLE-6.5.0\lbr
- Component order
  - http://www.digikey.com

EAGLE User Interface – Control Panel

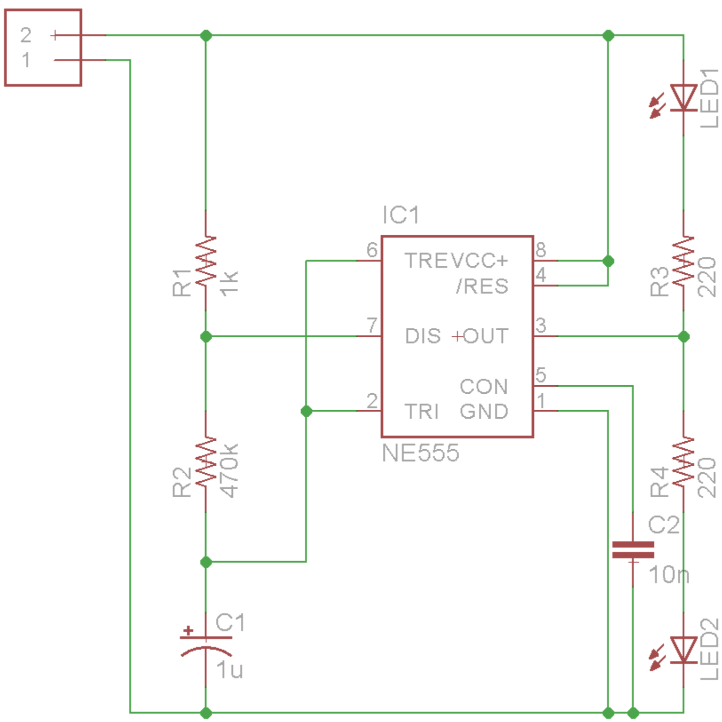
Menu bar

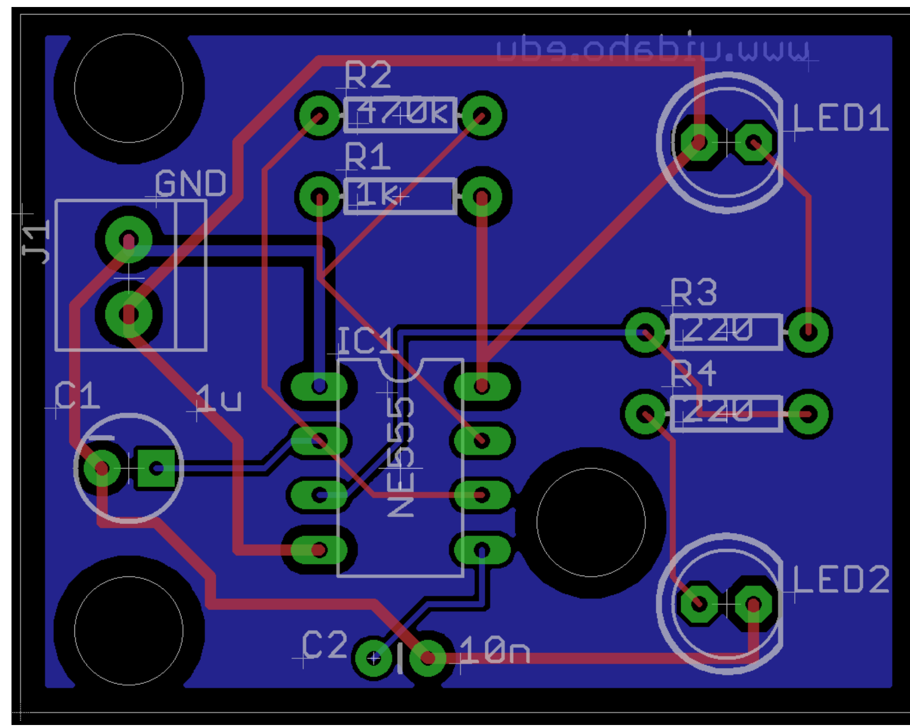
File manager

Status line



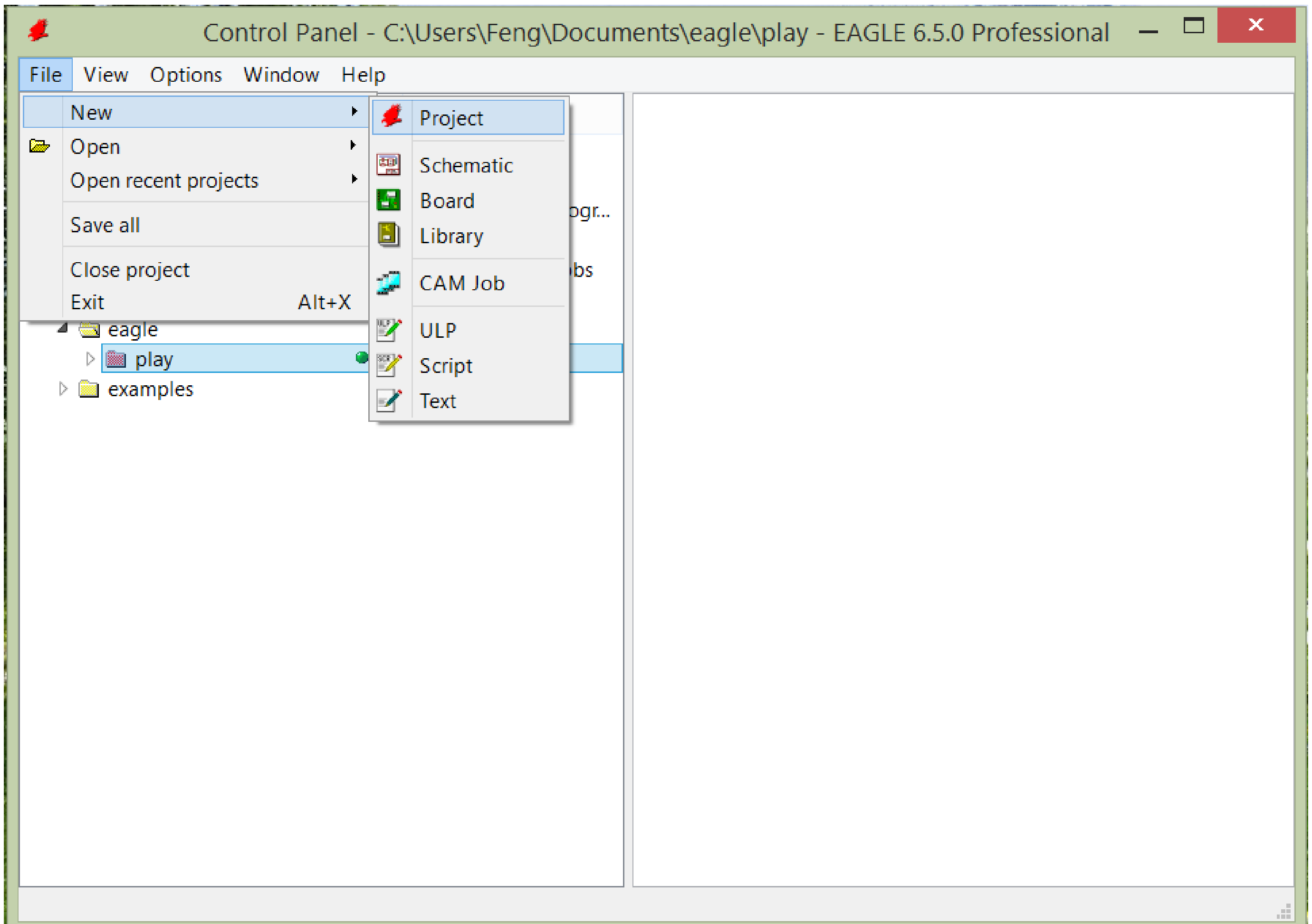
Example Project:
LED Flasher Using 555 Timer





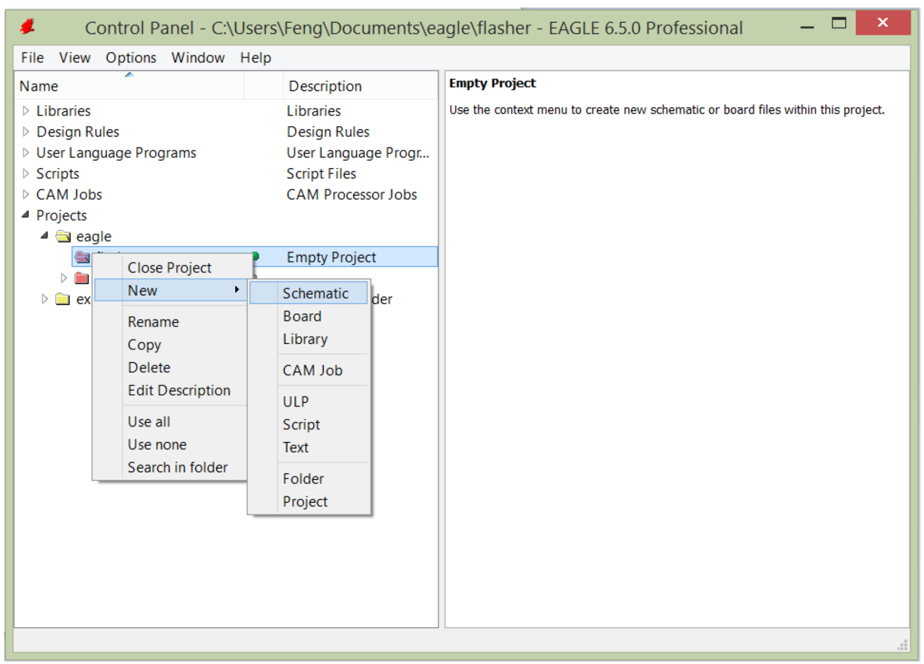
Creating and Naming a New Project

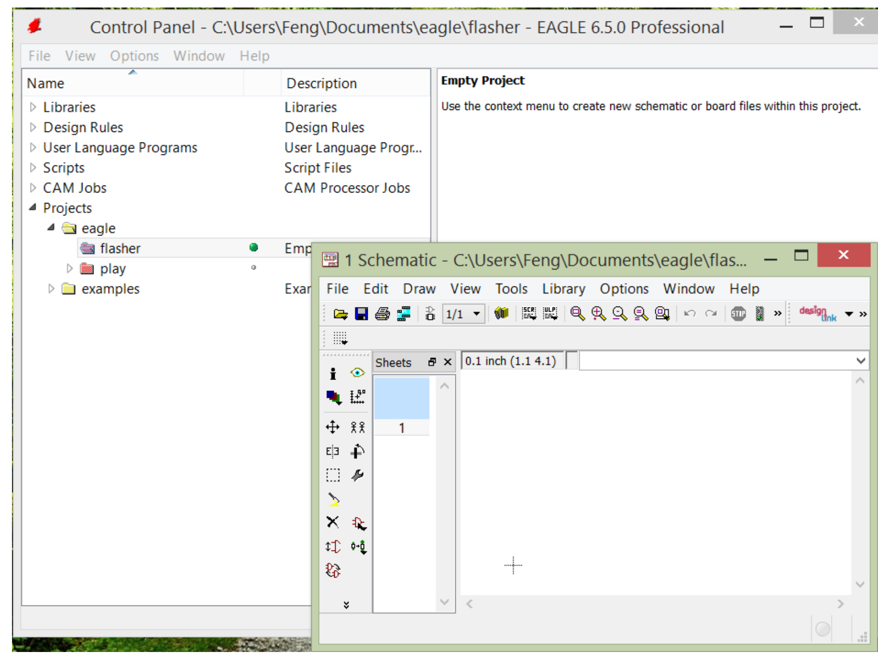
- File->New->Project (you can name the folder as flasher)



Creating a New Schematic

- Right click folder “flasher”->New->Schematic

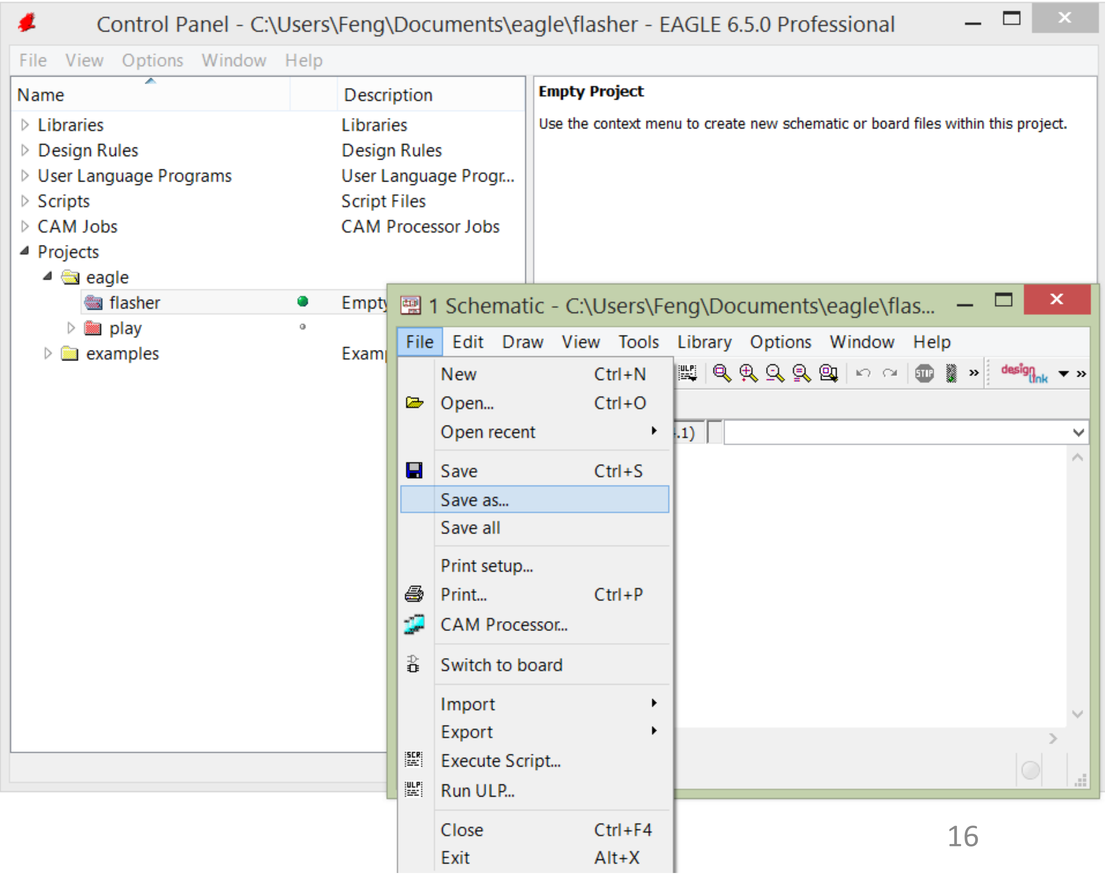




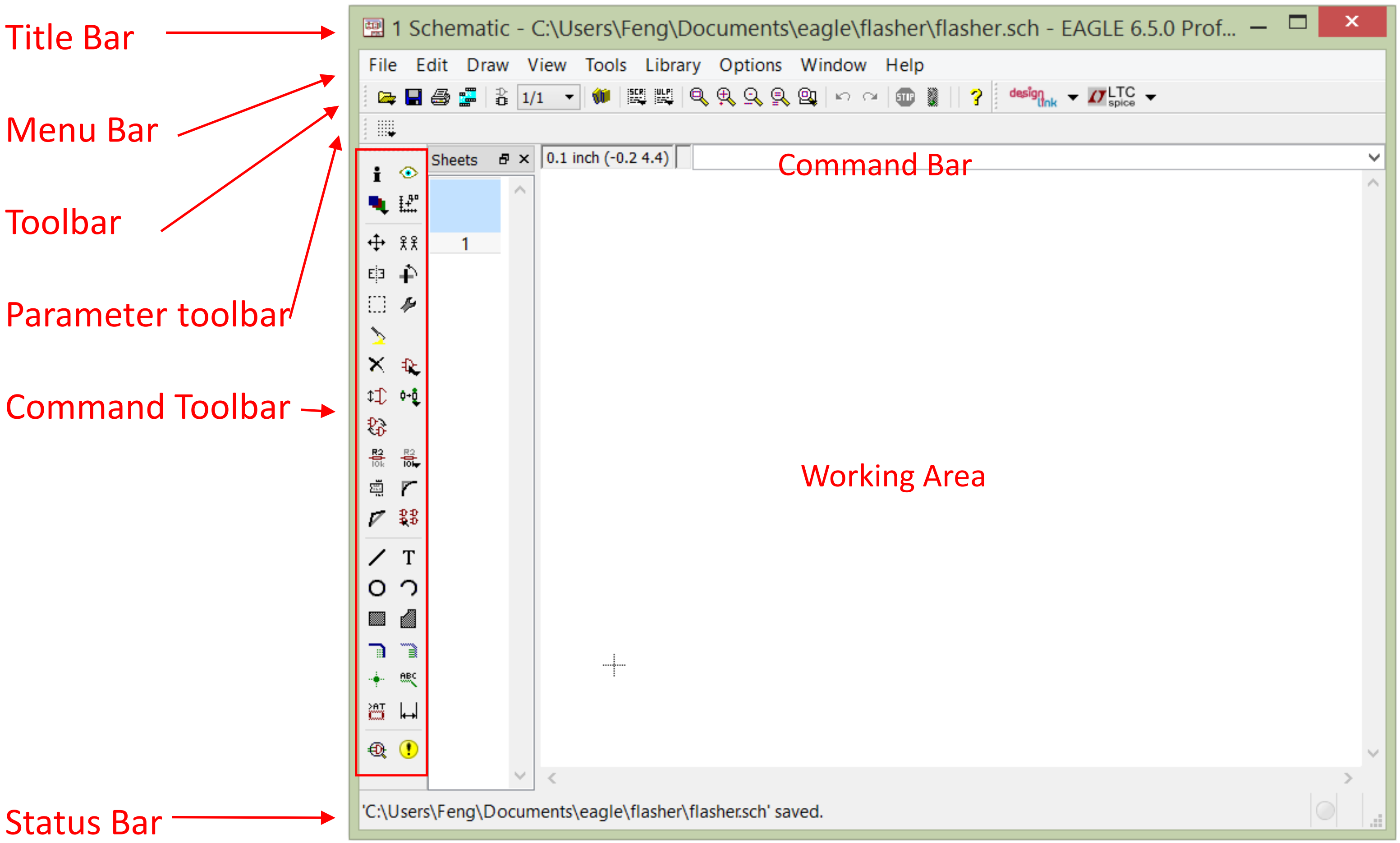
Save and Rename of New Schematic

- [Schematic]File->Save as... (flasher.sch)

- Note: Do not create a board file yet.



Schematic Editor Window



Schematic Editor Window Command Tools

Info – Info on a part

Layer settings – Change Layers

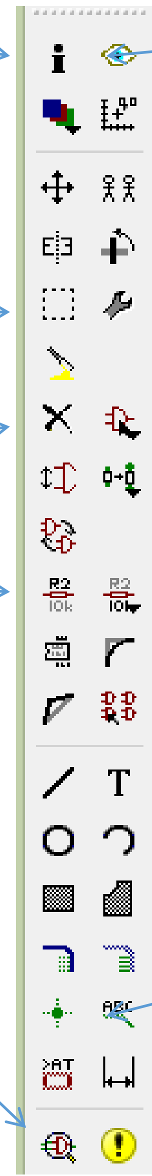
Move

Group – Define a bunch of parts to a group to move.

Delete – Delete a part

Name – Name a part

ERC - Electrical Rule Check



Show – Highlight a part, wire, or trace. Very Useful

Rotate

Add – Add a part from the library

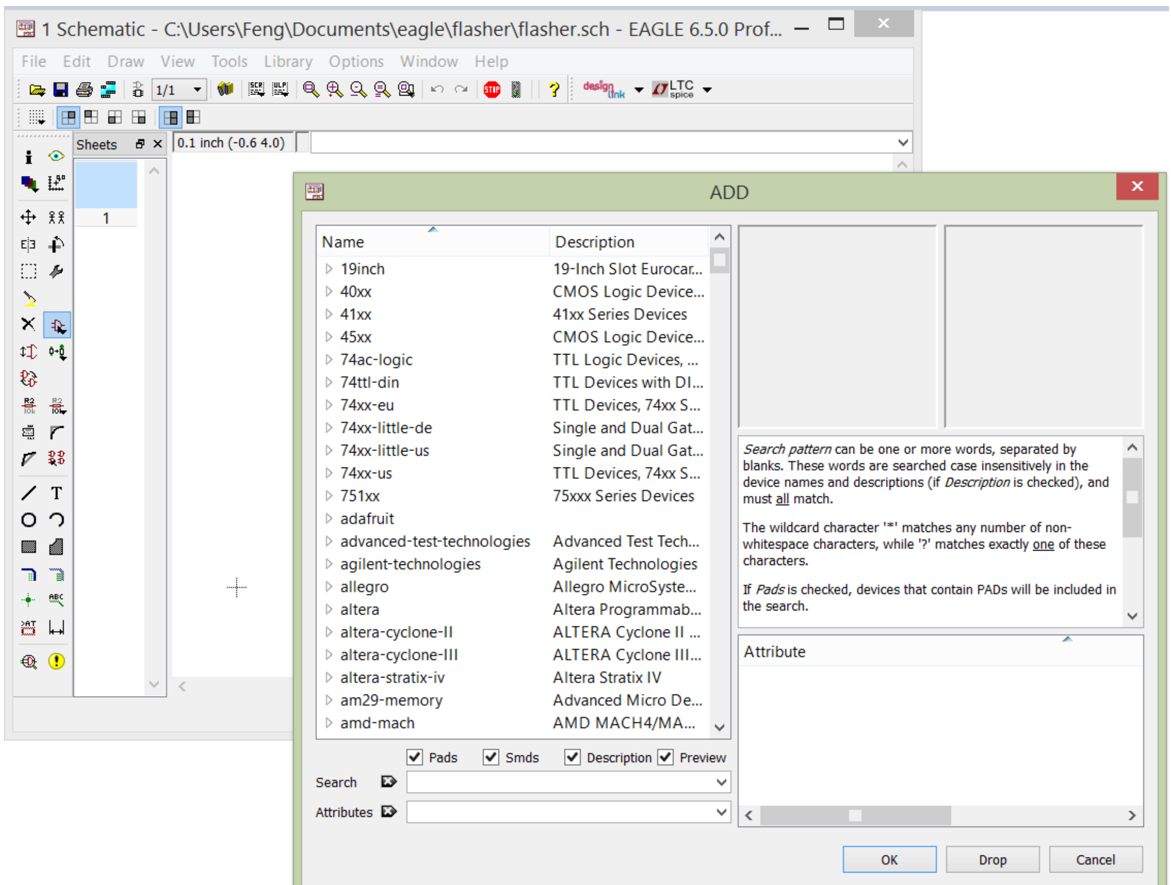
Value – Define a value (20 Ohms, etc)

Net – Connect component

Label – Label items

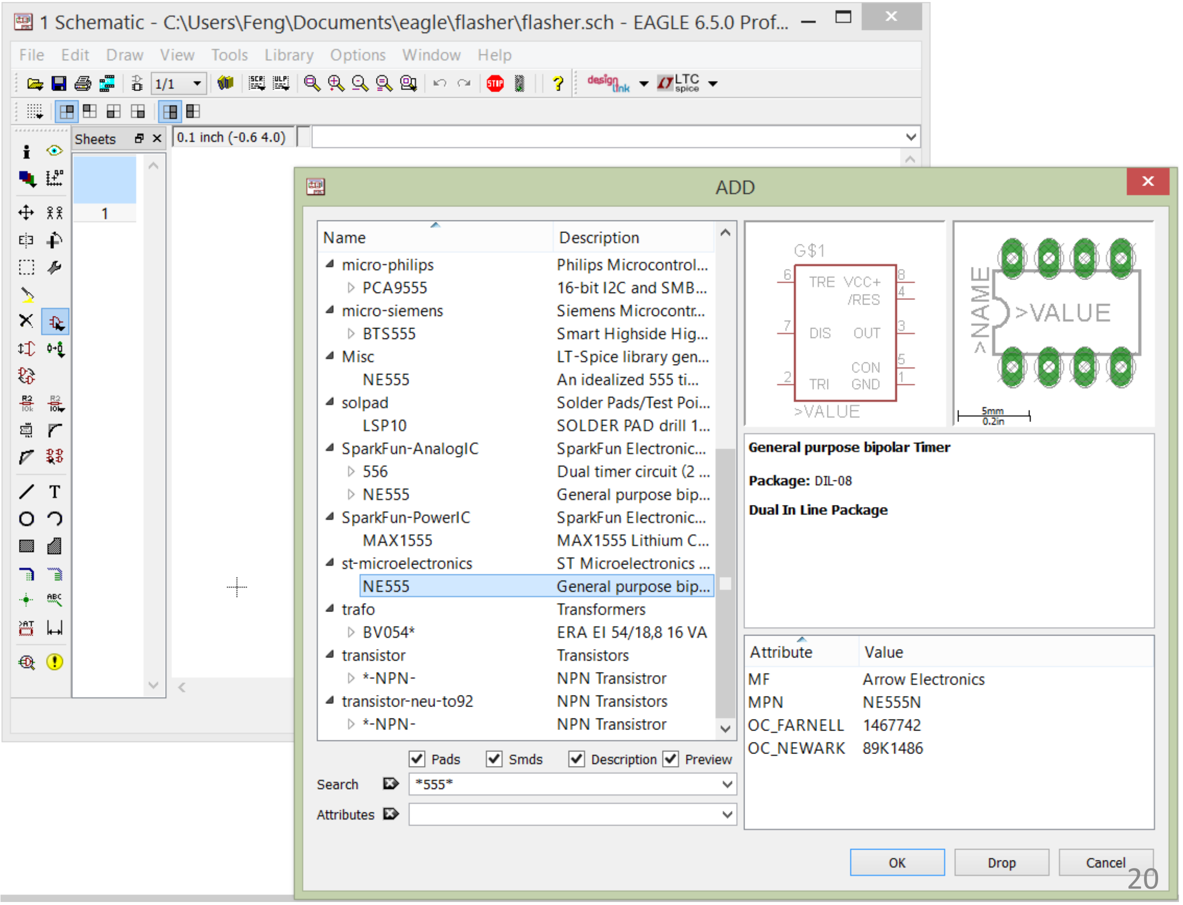
Finding and Add a Component

- [Schematic] Add



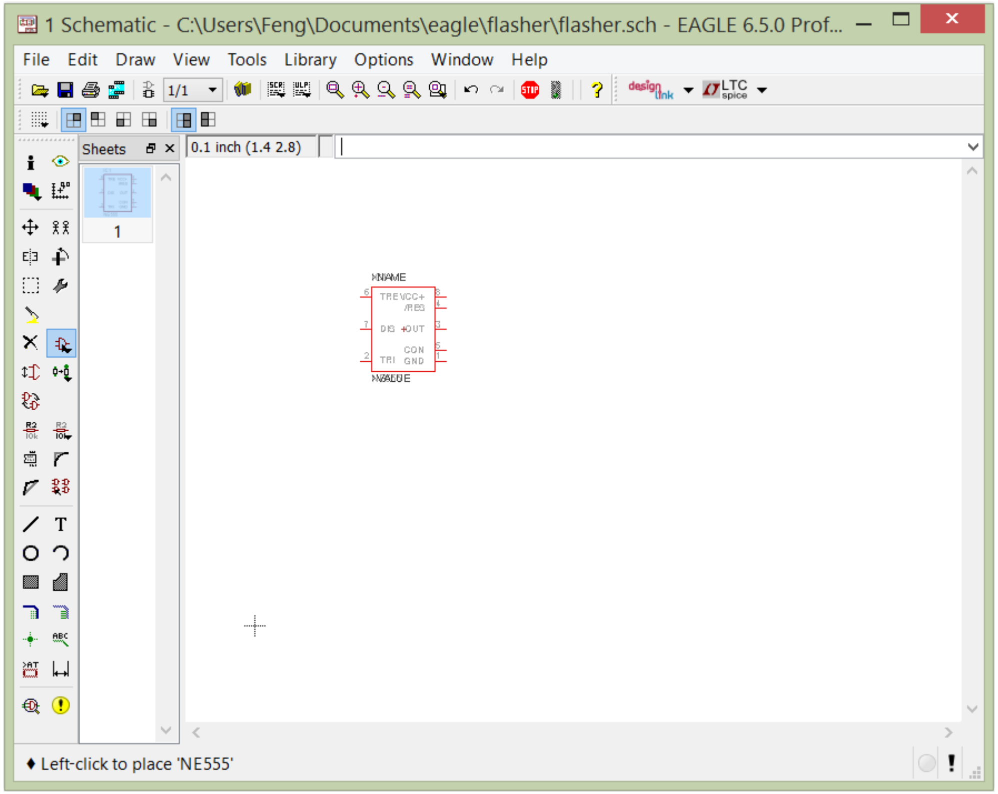
Search for a Component

- Enter “*555*” in the Search field
- Find “NE555”
- Click “OK “



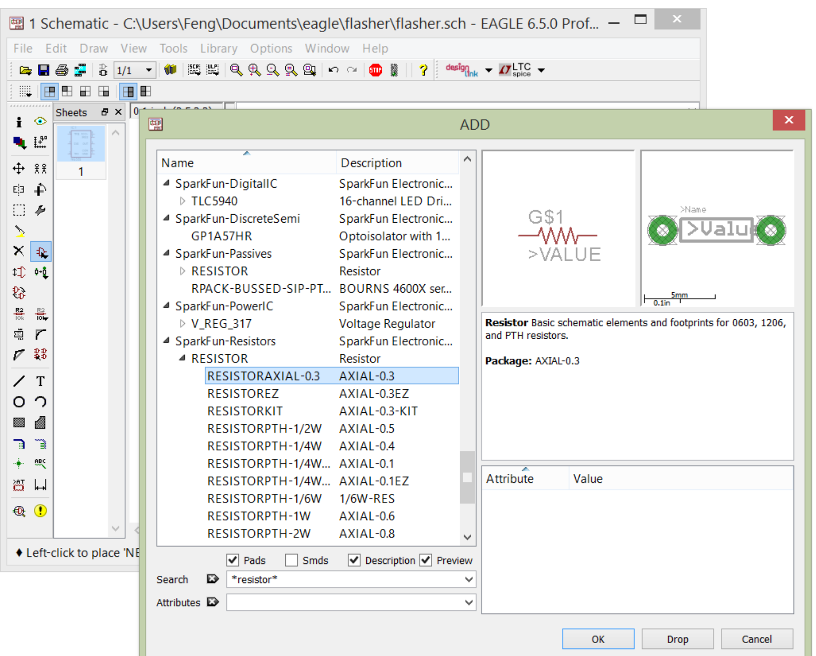
Drop a Component

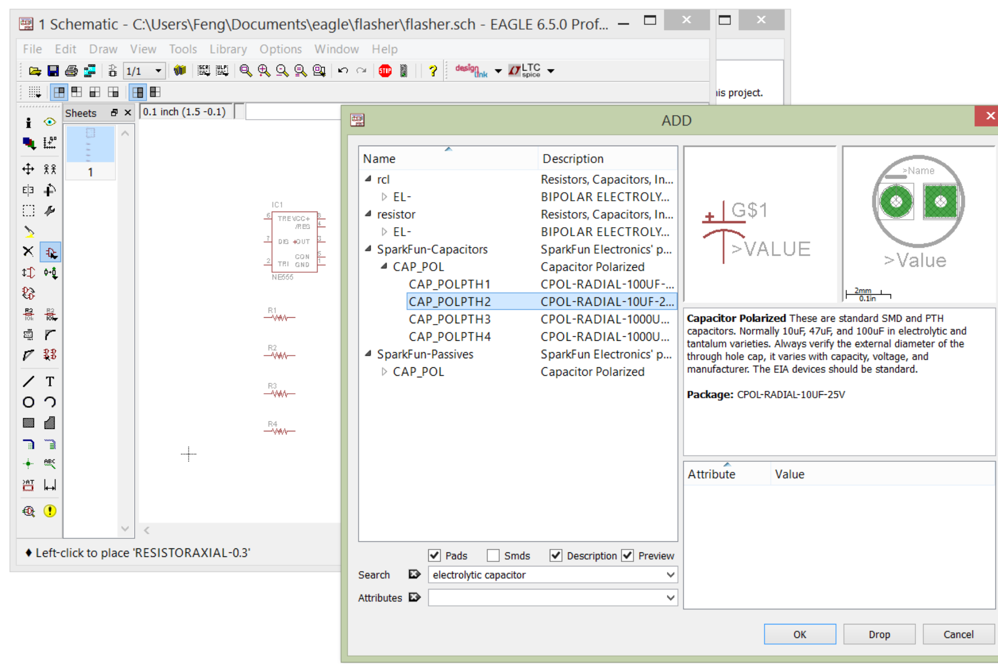
- Move the cursor toward the middle of the schematic window
- Click to drop the component
- Press “ESP” key to escape from adding component;
- Click “Cancel” to escape from the adding Window.
- You can delete a Component by clicking “Delete”



Add Components – Resistors and Capacitor

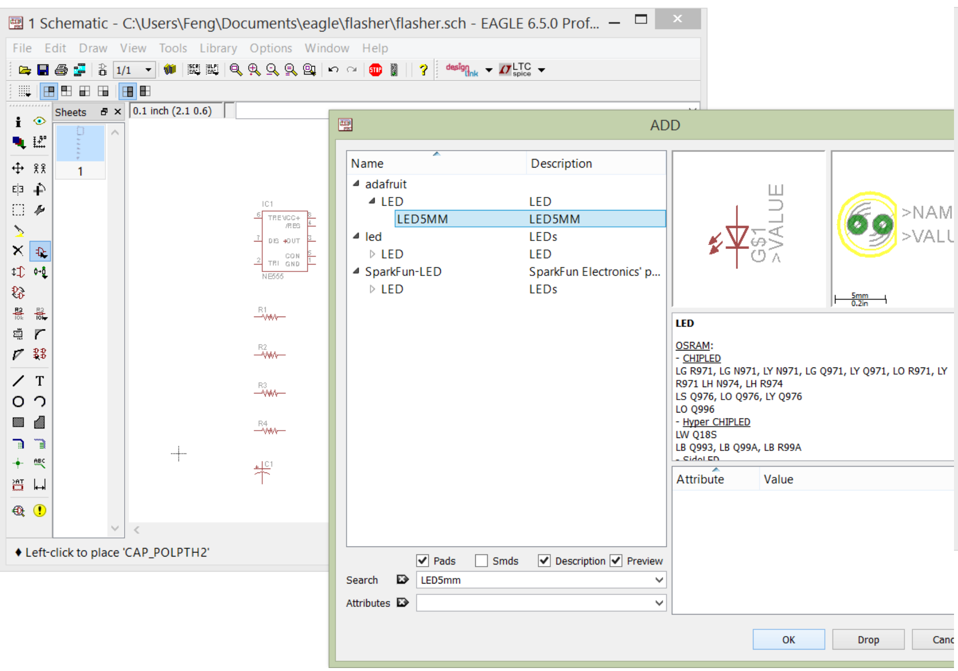
- SparkFun-Resistors: RESISTORAXIAL-0.3 (x4)
- SparkFun-Capacitors: CAP_POLPTH2

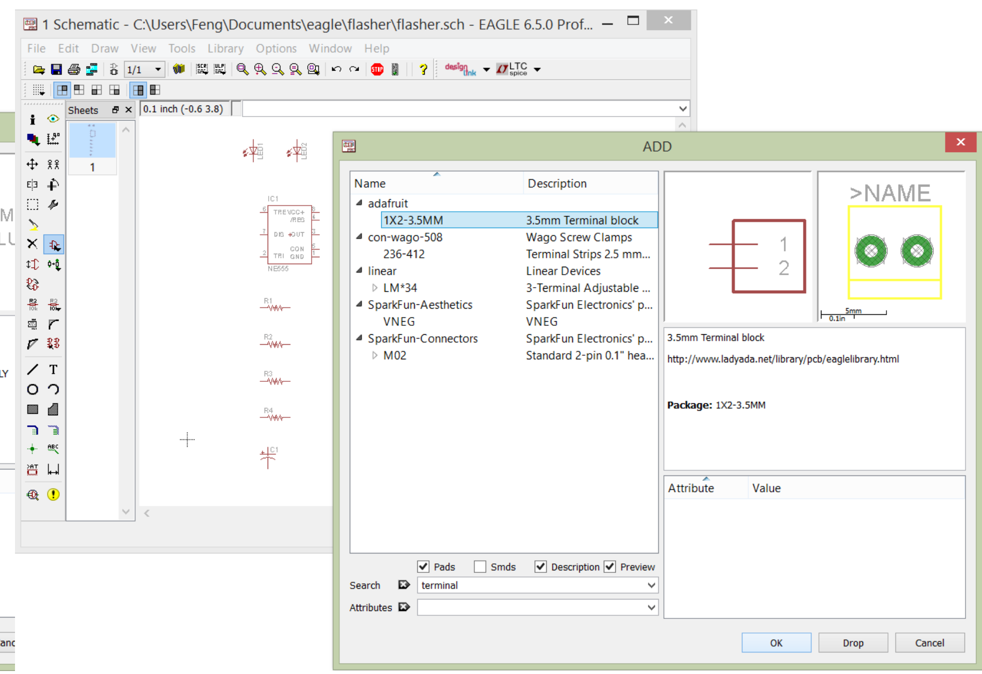




Add Component – LED and a Screw Terminal for Power Supply

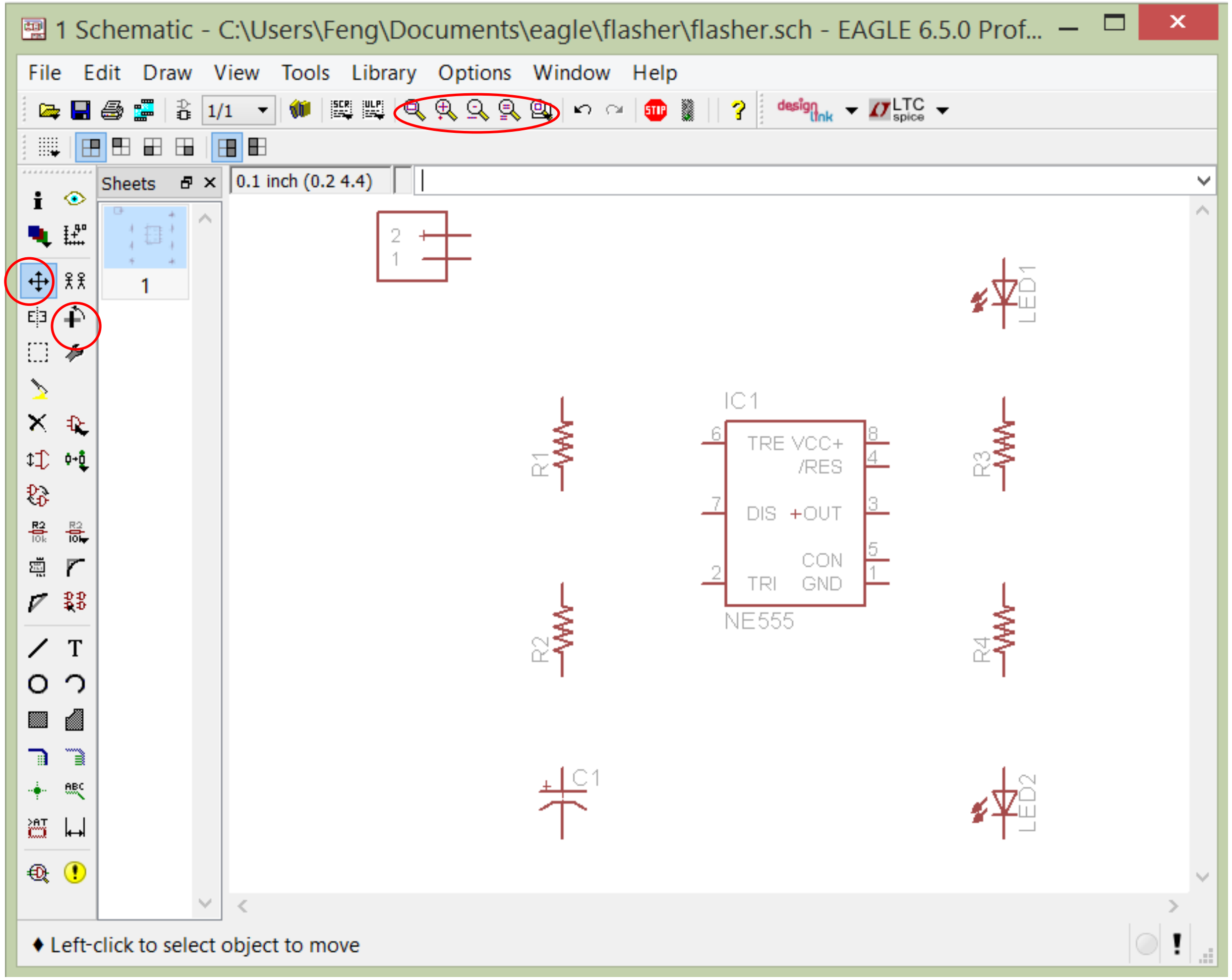
- Adafruit: LED5MM (search “LED5mm”)
- Adafruit: 1X2-3.5MM (search “terminal”)





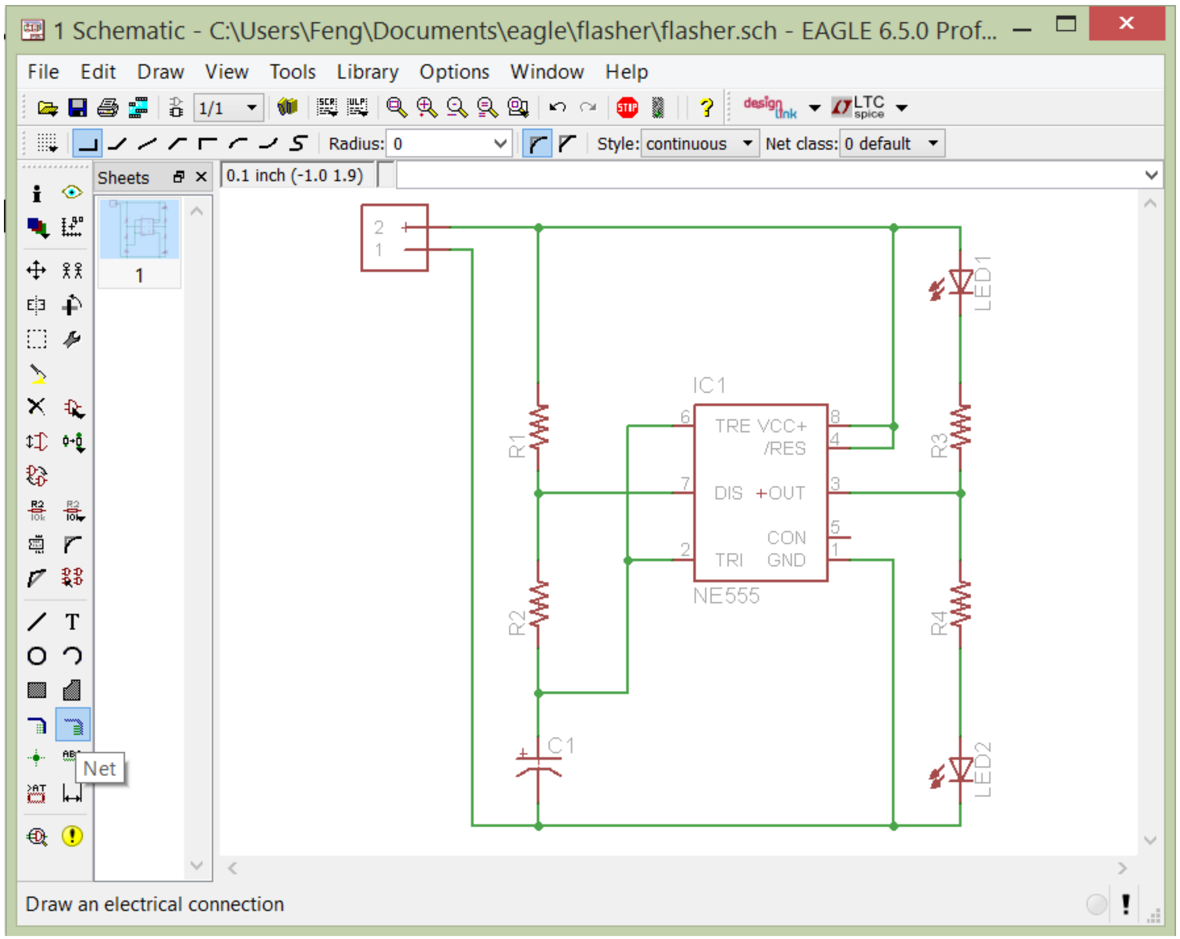
Rotating and Moving Components

- Rotate
- Move
- Zoom…



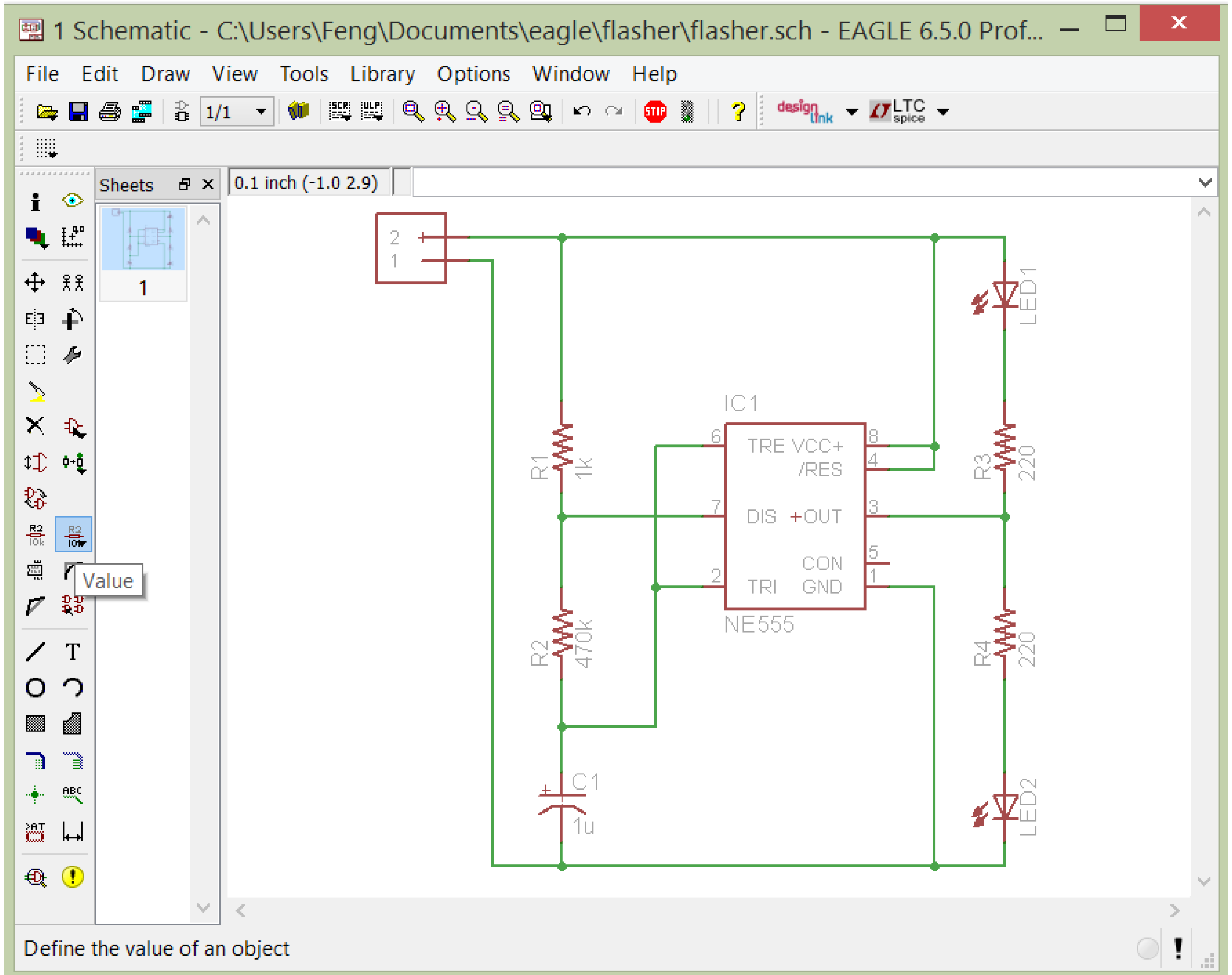
Connecting the components

- Net



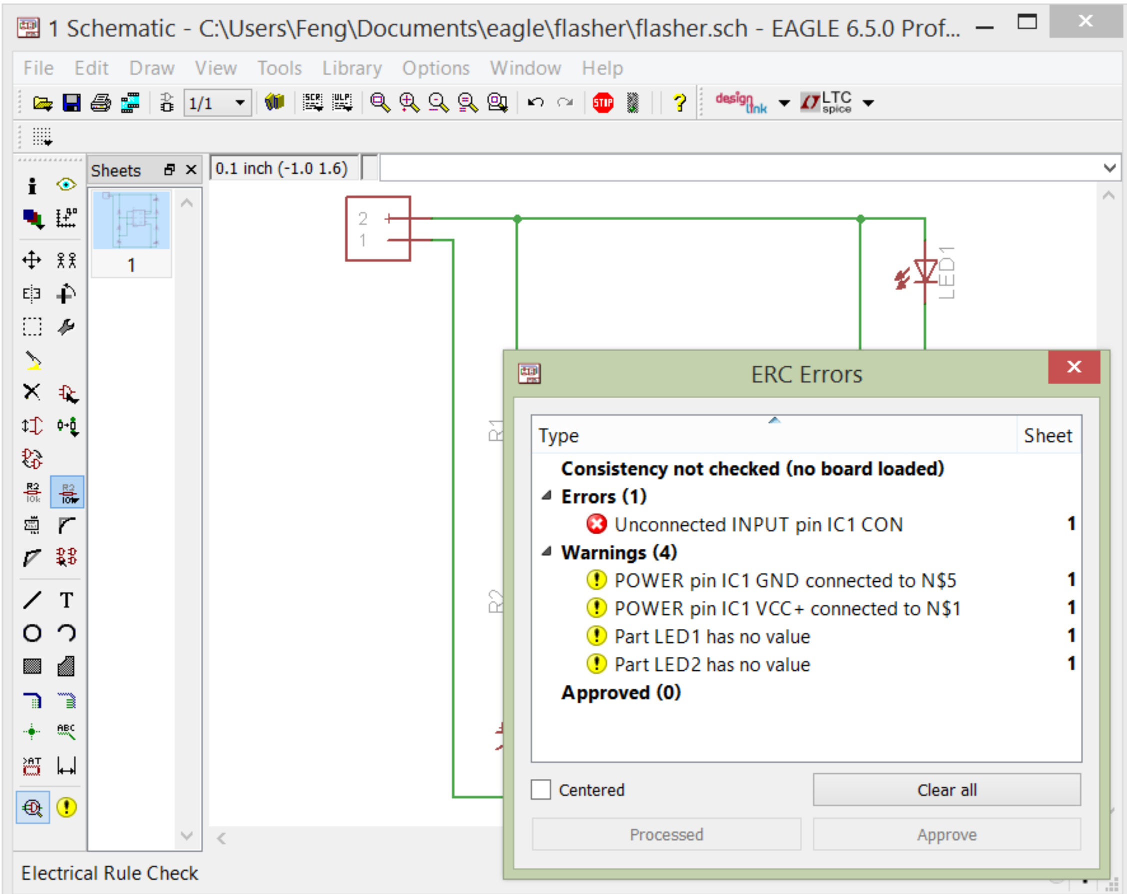
Setting Component Values

- Value

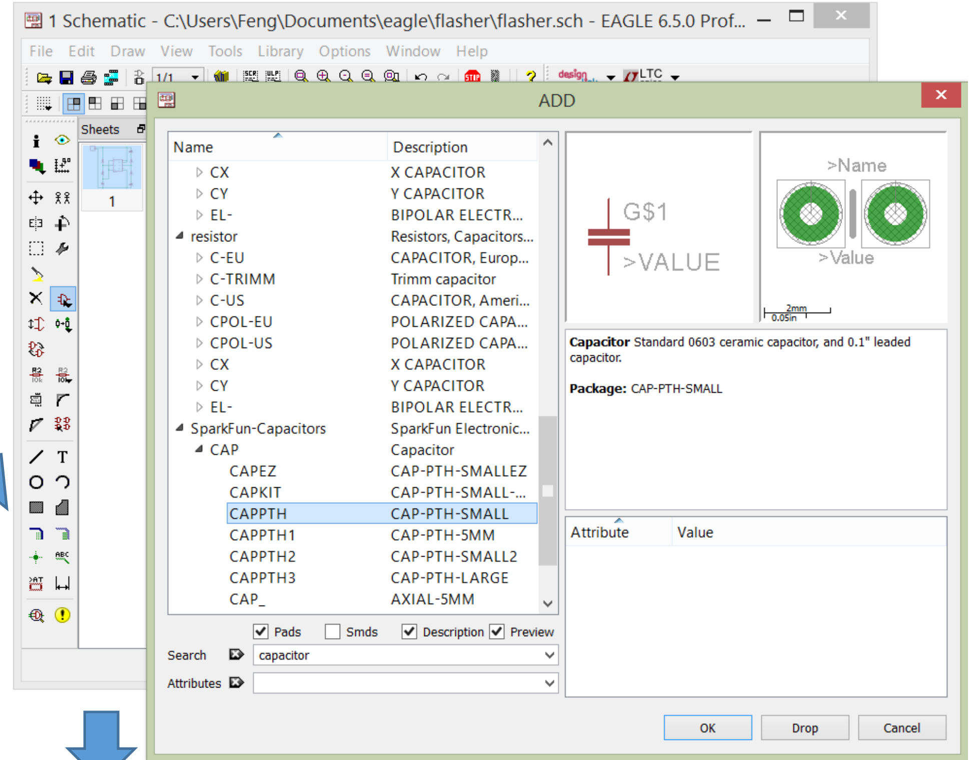


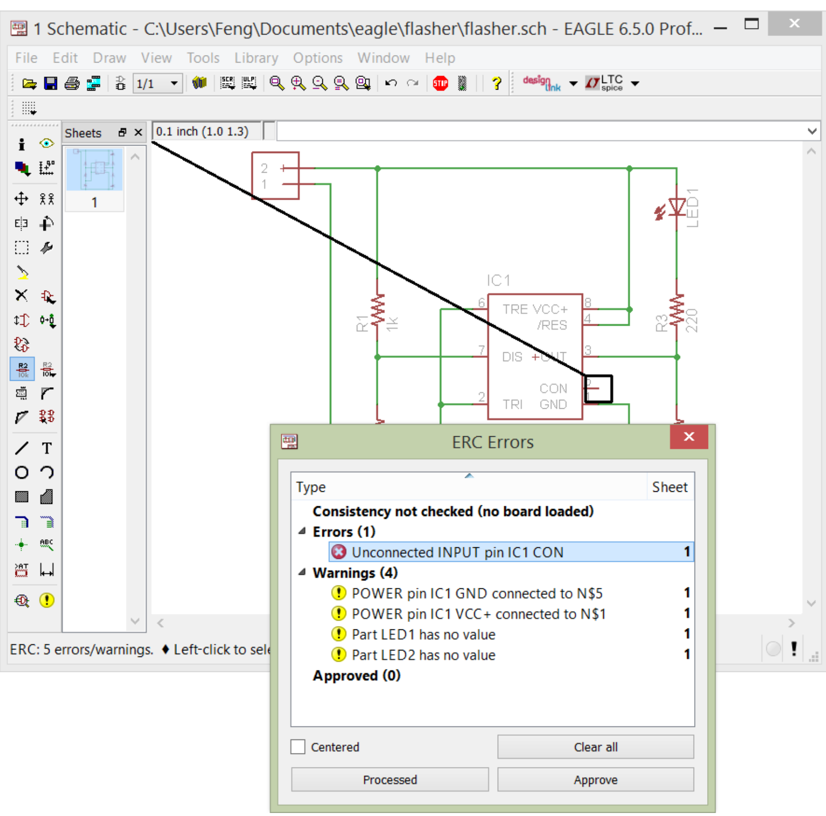
Electrical Rule Check

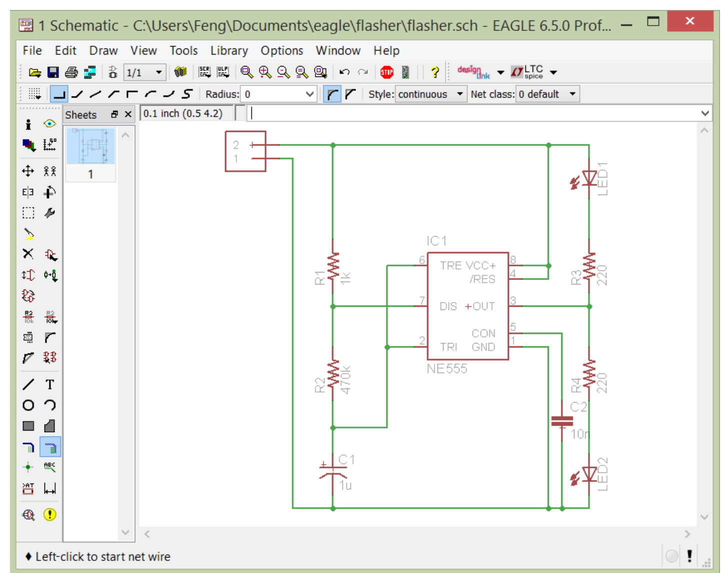
- ERC

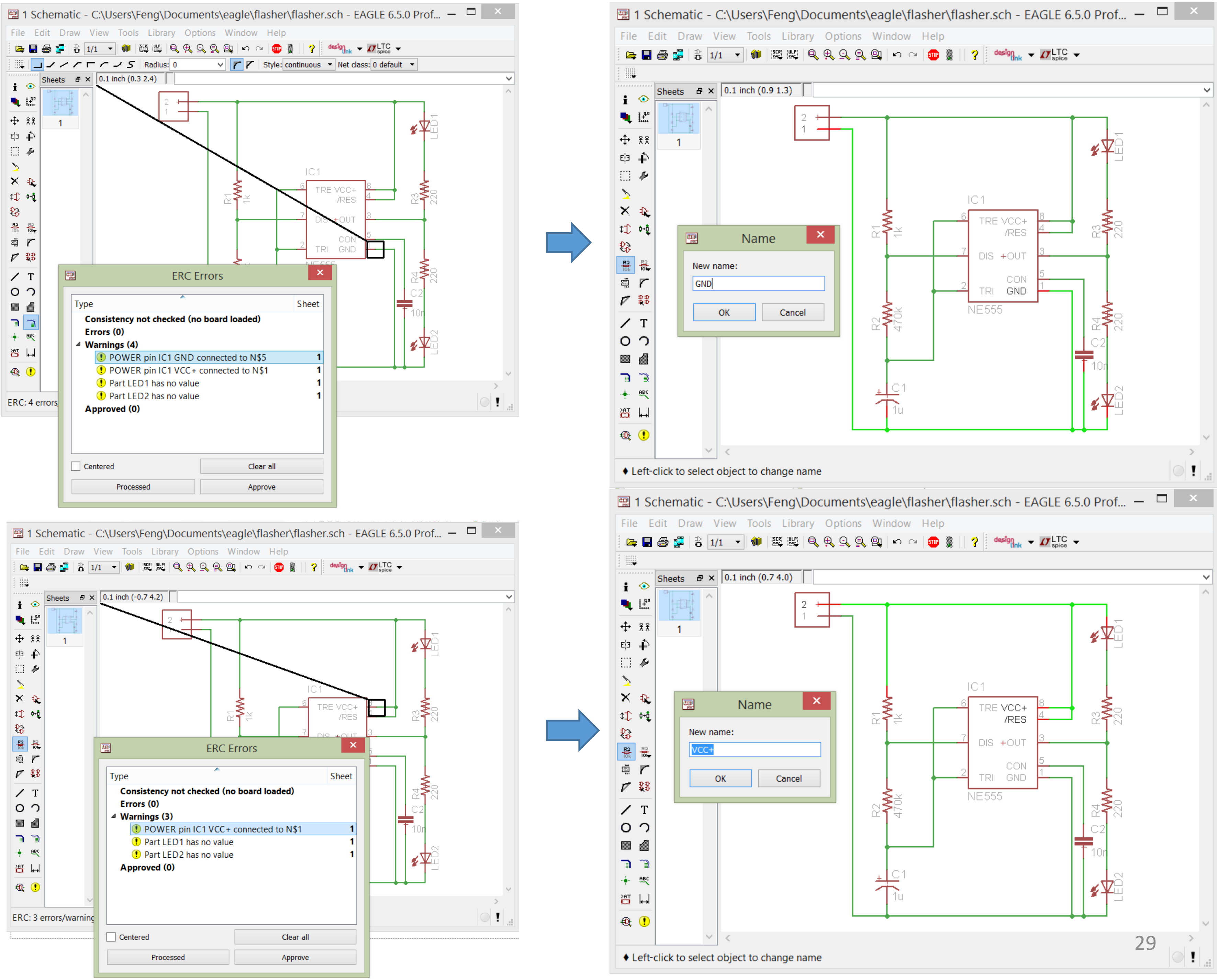


- Use “Show” to show net name
- Search for “capacitor” and find CAP-PTH-SMALL
- Add the capacitor and connect it to Pin5 and the ground



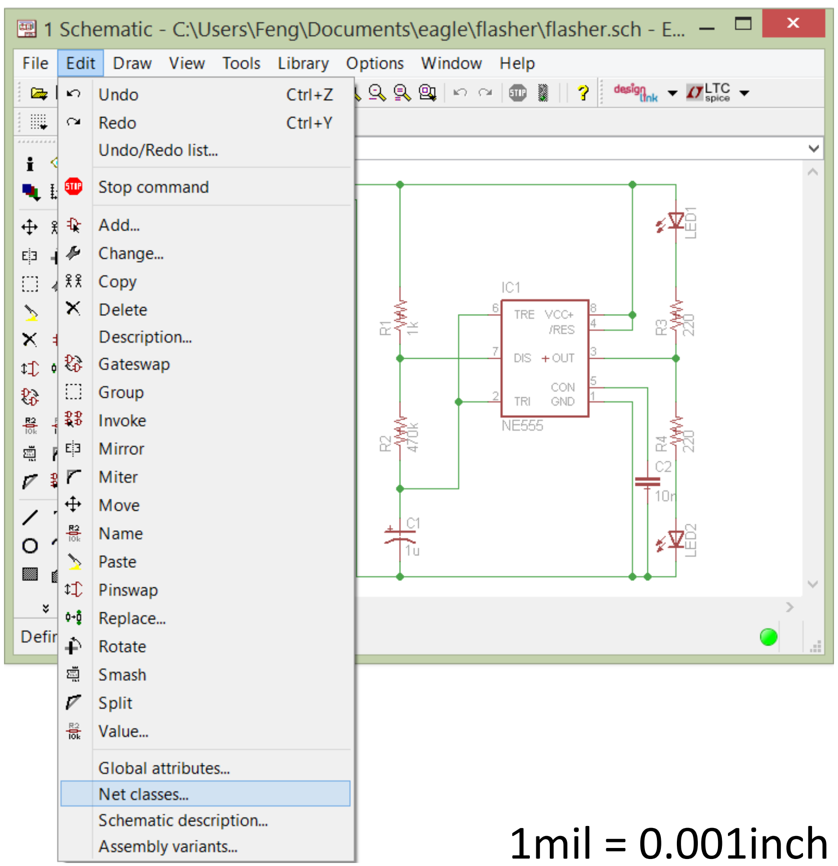


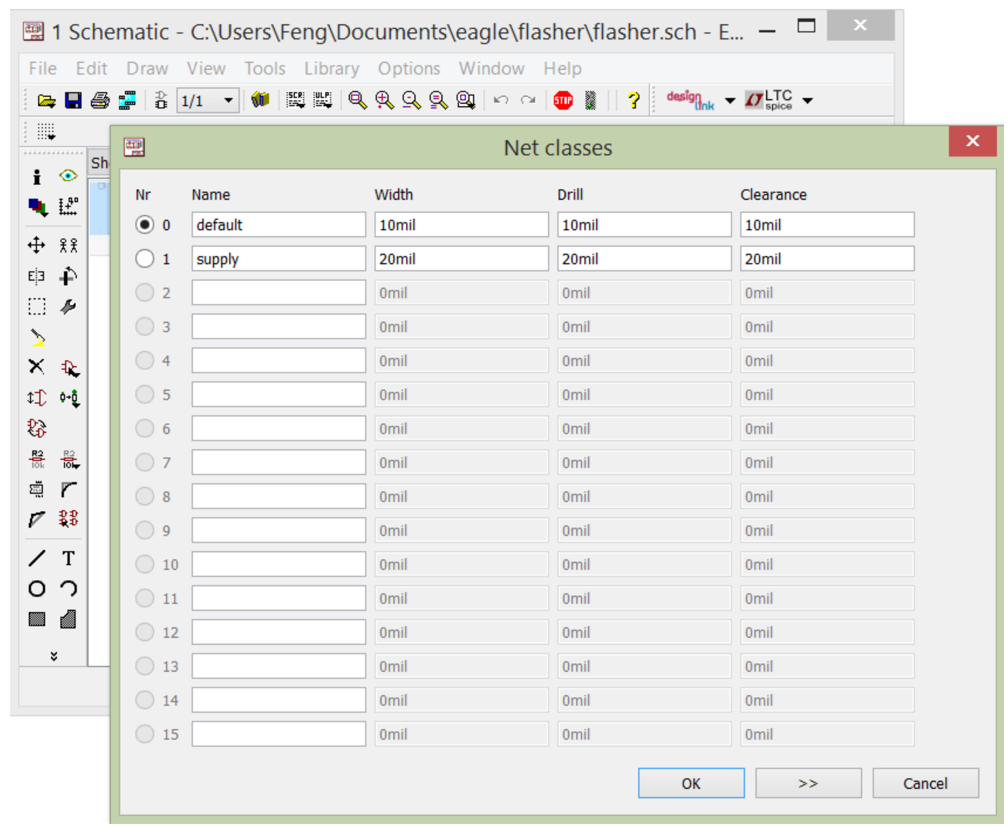



1 Schematic - C:\Users\Feng\Documents\eagle\flasher\flasher.sch - EAGLE 6.5.0 Prof...
File Edit Draw View Tools Library Options Window Help
ERC Errors
Type
Sheet
Consistency not checked (no board loaded)
Errors (0)
Warnings (4)
POWER pin IC1 GND connected to N$5 1
POWER pin IC1 VCC+ connected to N$1 1
Part LED1 has no value 1
Part LED2 has no value 1
Approved (0)
Centered
Clear all
Processed
Approve
ERC: 4 errors
Name
New name:
GND
OK
Cancel
Left-click to select object to change name
ERC Errors
Consistency not checked (no board loaded)
Errors (0)
Warnings (3)
POWER pin IC1 VCC+ connected to N$1 1
Part LED1 has no value 1
Part LED2 has no value 1
Approved (0)
ERC: 3 errors/warning
Name
New name:
VCC+
OK
Cancel
Left-click to select object to change name
IC1
NE555
TRE VCC+
/RES
DIS +OUT
CON
TRI GND
R1 1k
R2 470k
R3 220
R4 220
C1 1u
C2 10n
LED1
LED2

Net classes

- [Schematic]Edit->Net classes
- Can also be done in Layout Editor later

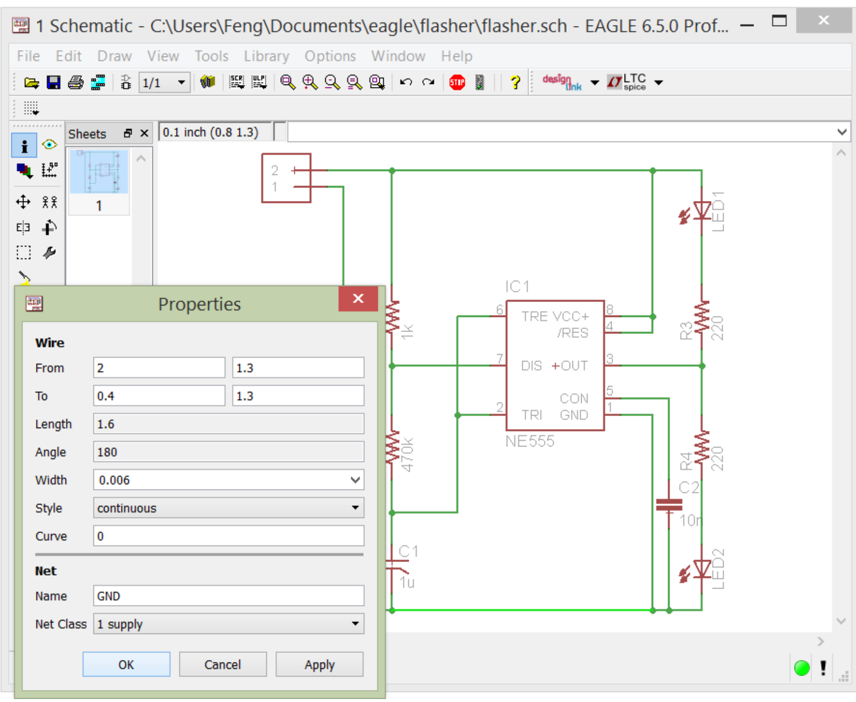


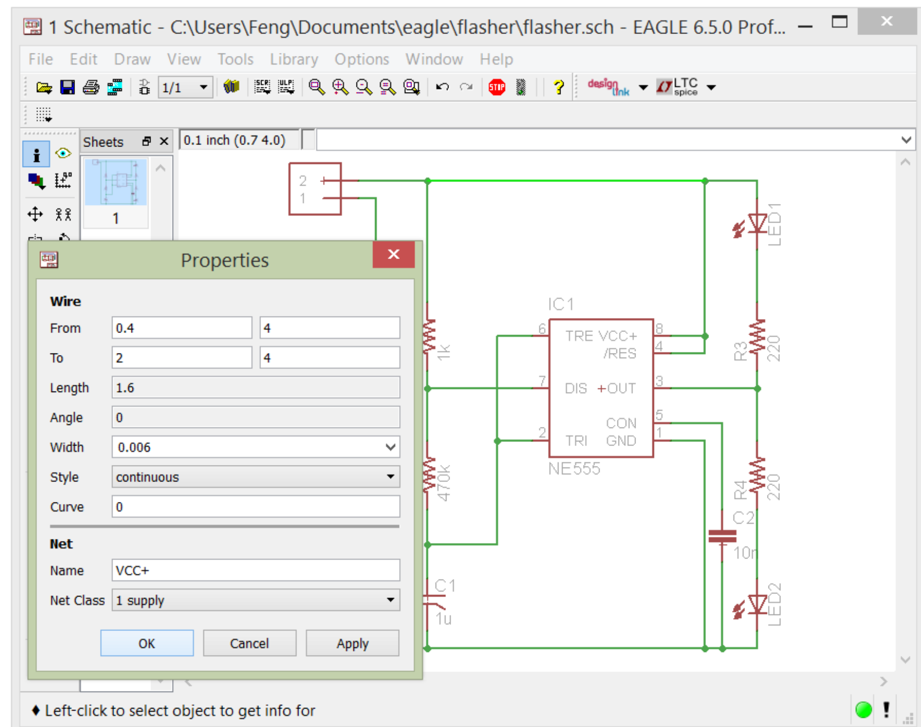


1mil = 0.001inch

Net Classes

- Info



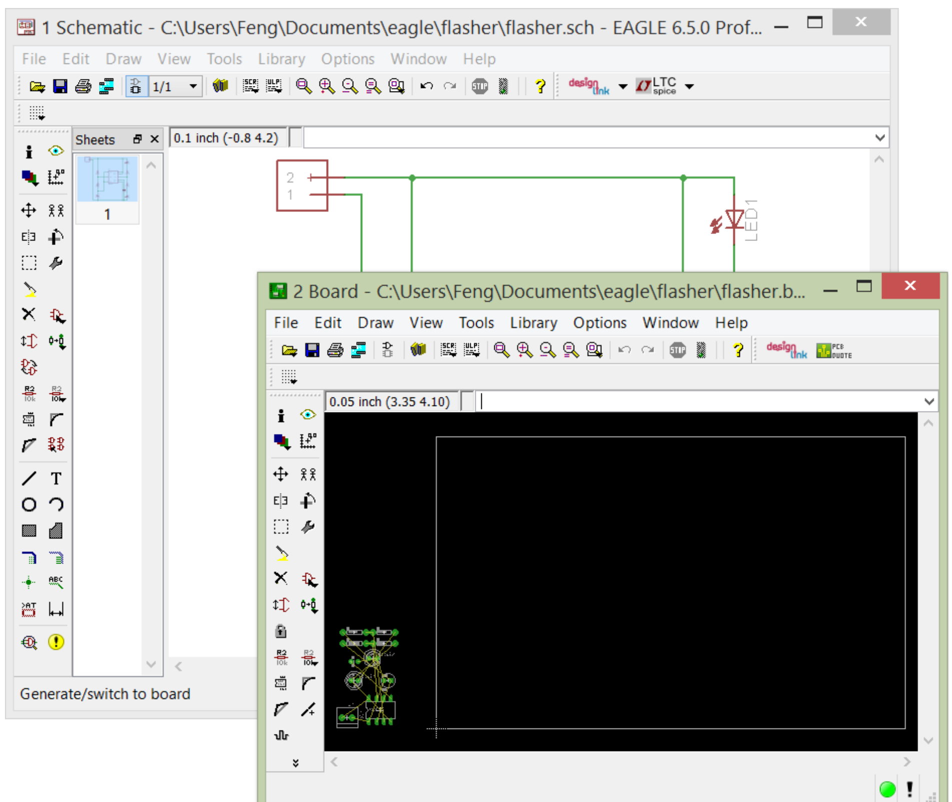


Laying Out the Board

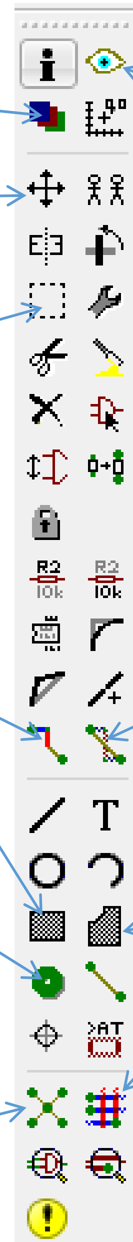
- [Schematic]File->Switch to board
- “This is no board, so would like to create one from the schematic?” – “Yes”
- Yellow lines: airwires-connections that will have to be converted into tracks
- Rectangle: borders

Note: 1. Do NOT close either schematic or board window. They must both remain open while working. Change in on editor window will lead to change in the other window.

2. Change background color: [Board]Options->User interface...->Background->White



Board Tools



Layer Settings – Change layers

Move – Select to Move components

Group – Group multiple components to move at once.

Route – Create a trace (this is the actual wire that carries the electrical signal)

Rectangle – Draw and define a rectangle

Via – Brings a trace from the top layer to the bottom layer

Ratsnest- Calculates the shortest possible airwires and polygons

Show – Highlight a trace or part. Useful to see what connects where.

Ripup – Remove Traces

Text – Add text

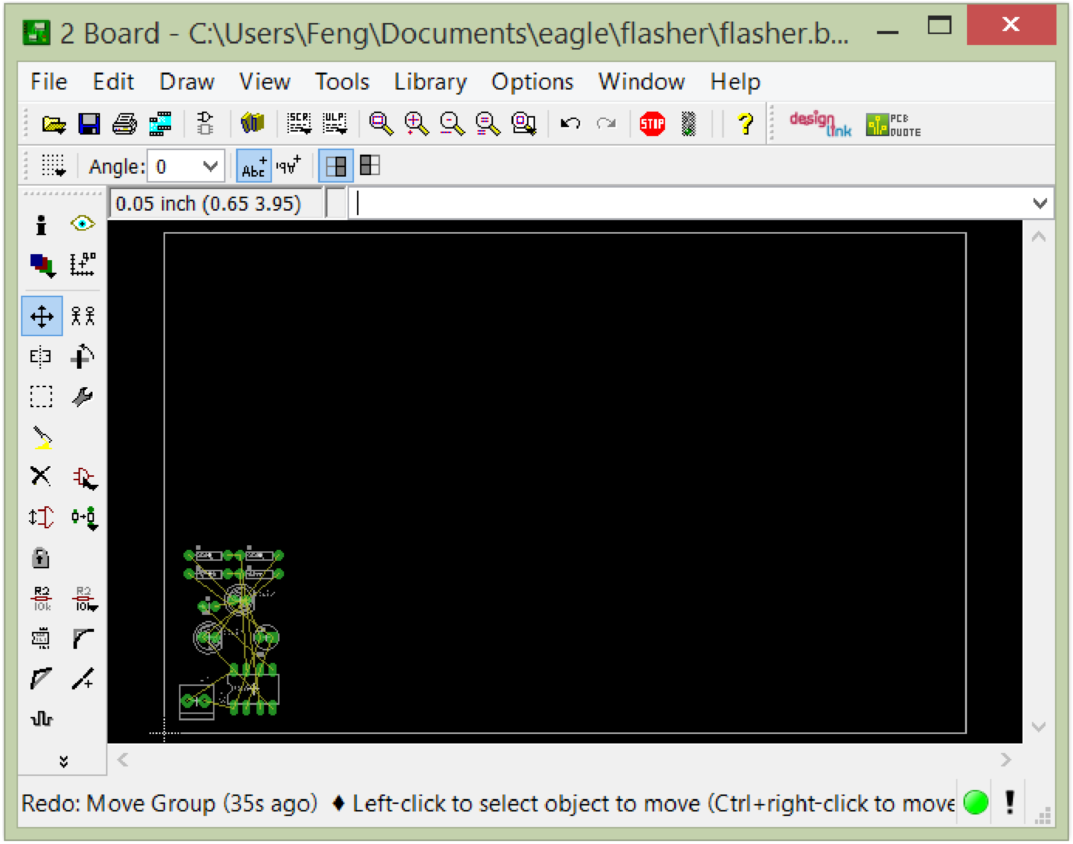
Polygon – Draw and define a polygon.

Auto Router – Somewhat unreliable

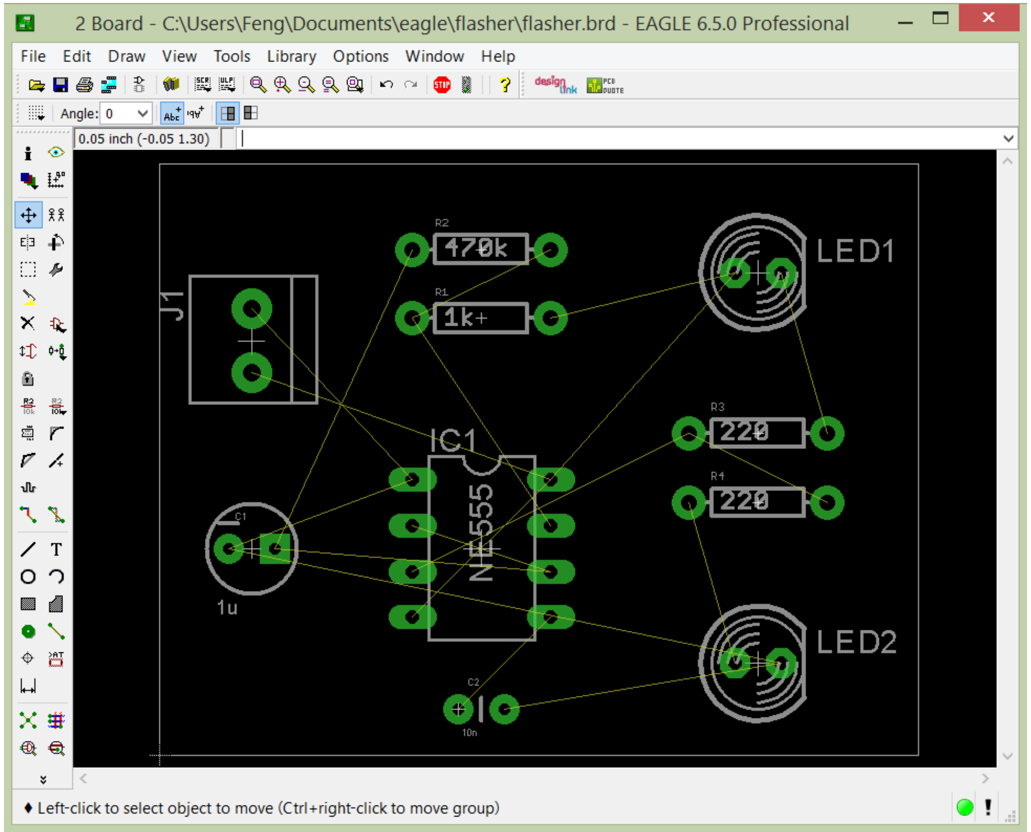
Design Rule Check – Very Important! More Later

Move

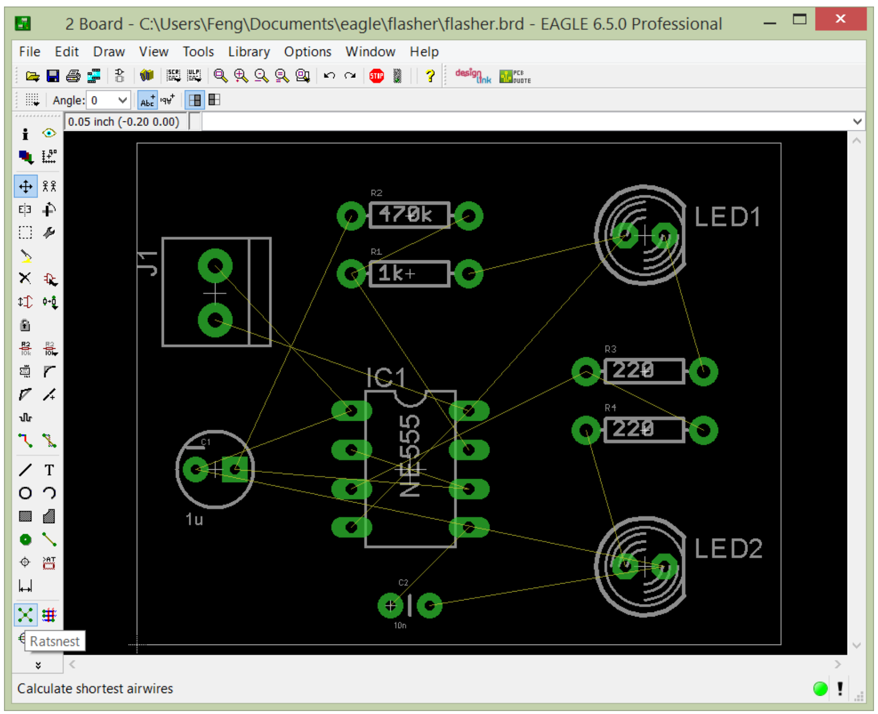
- Move
- Group
- Group move

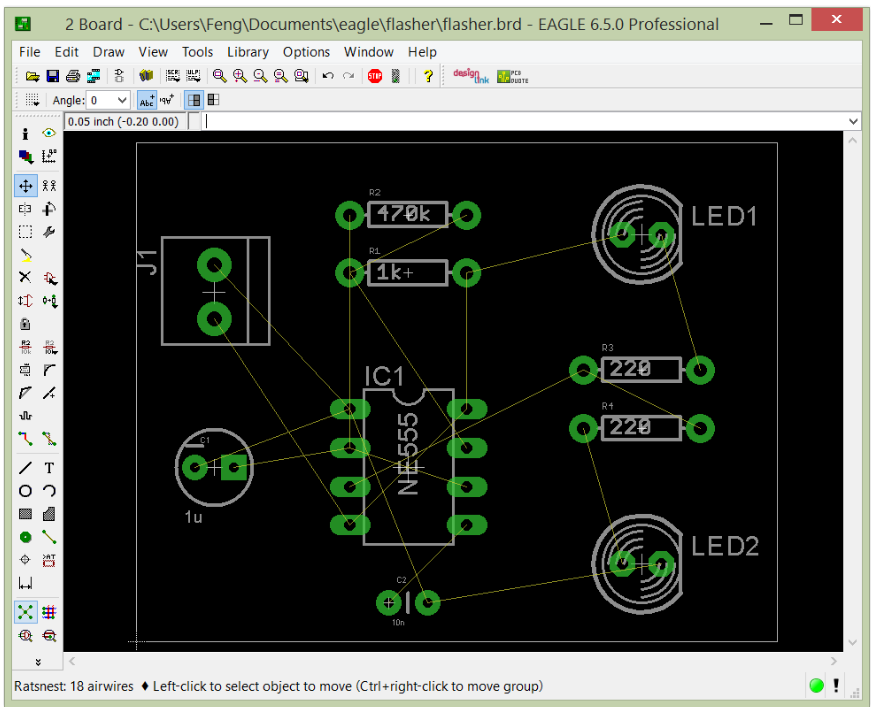


- Move
- Rotate
- Move
- (resizing the board)
- Zoom to fit

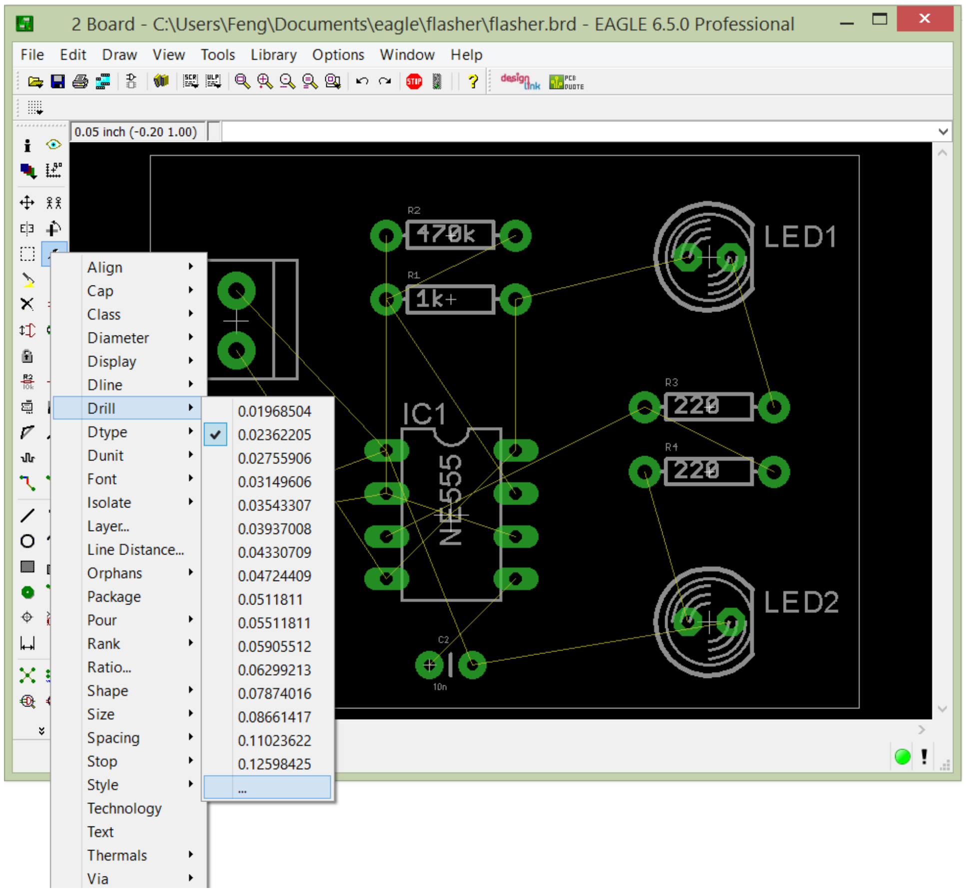


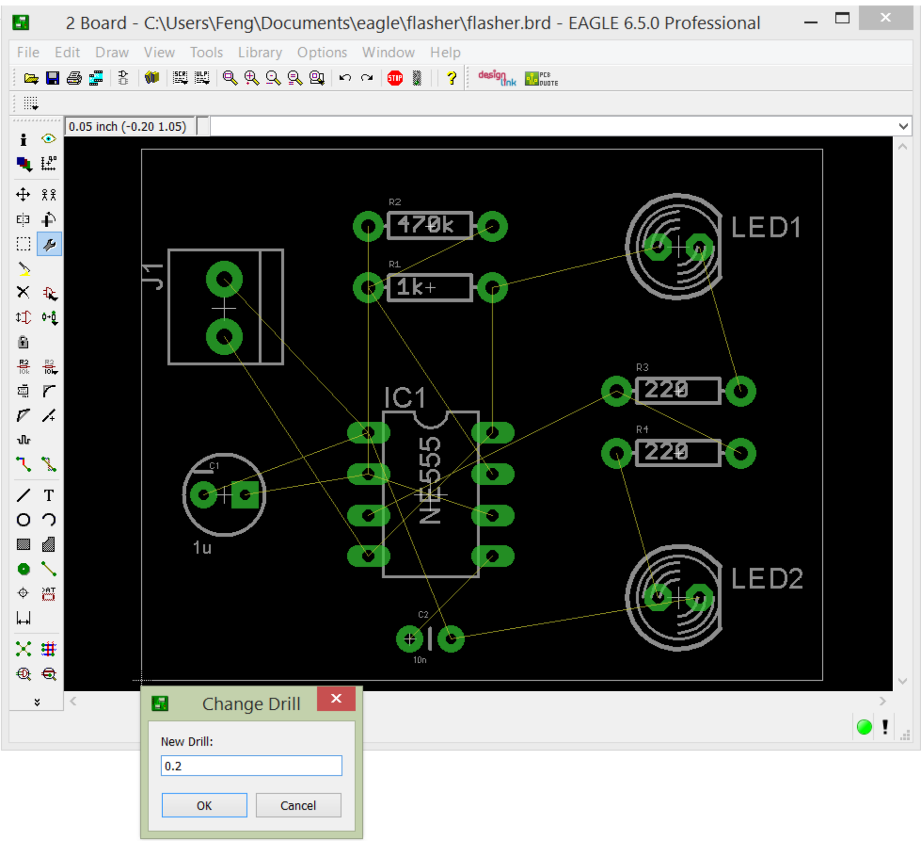
Ratsnest – calculate the shortest airwires (and polygons)



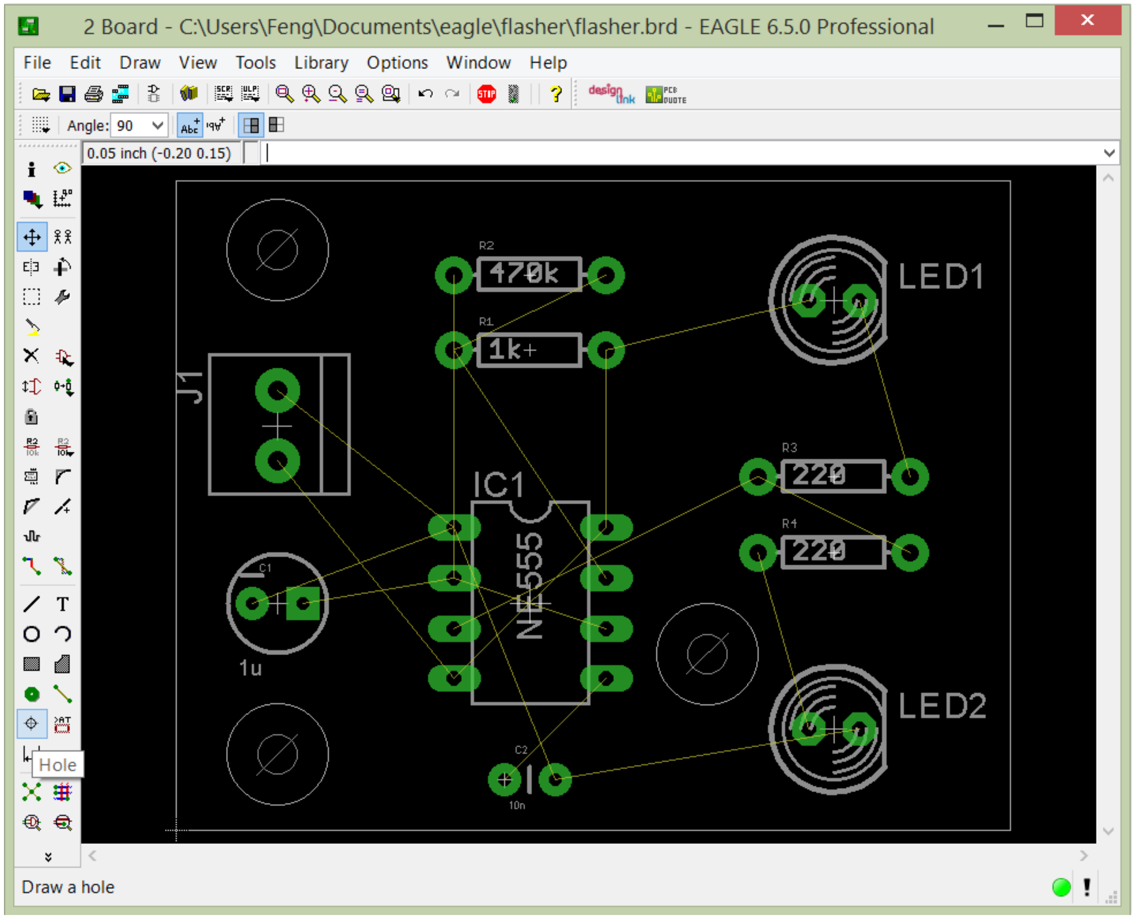


Add Mounting Holes



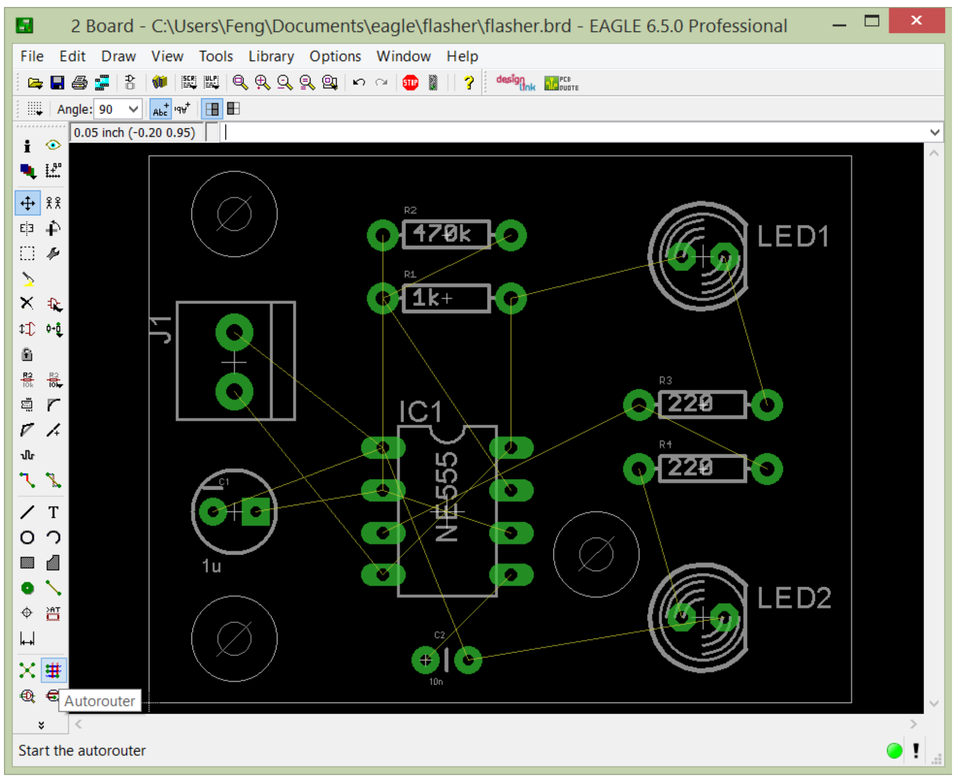


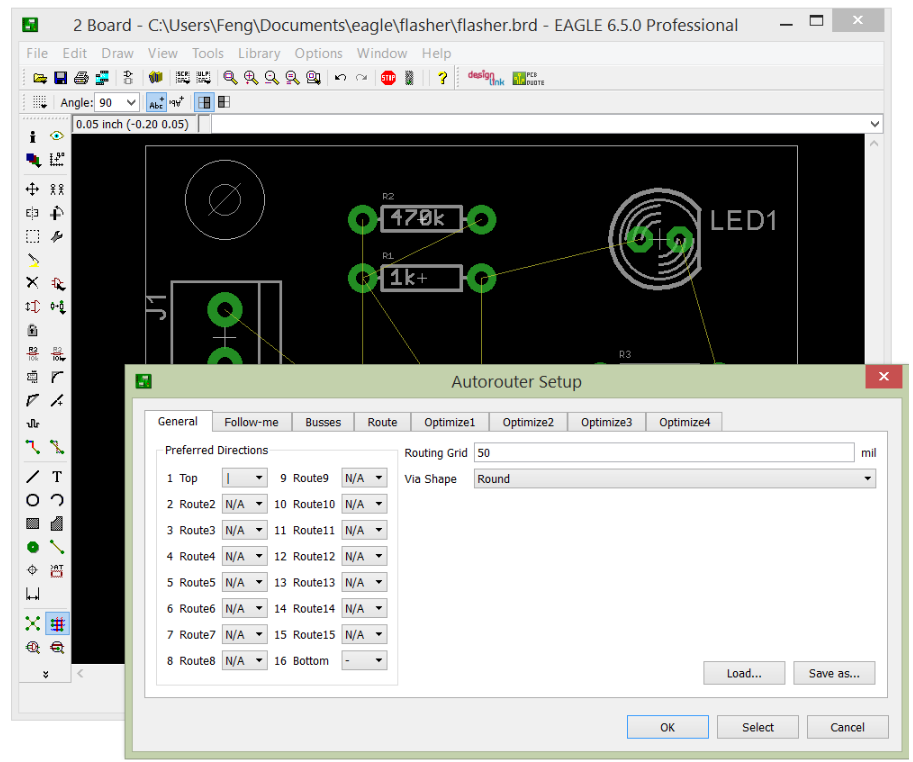
Add Mounting Holes



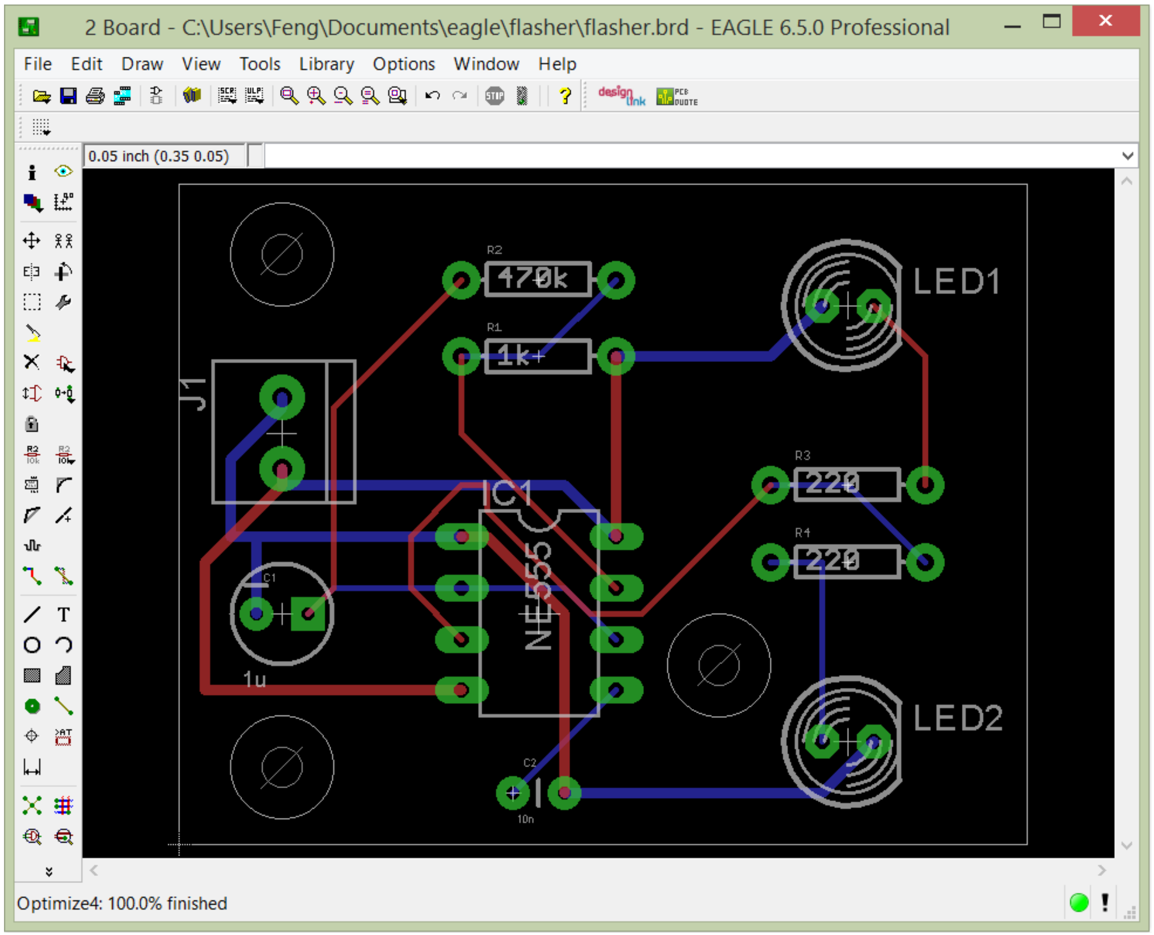
Autorouter

- [Board] Autorouter



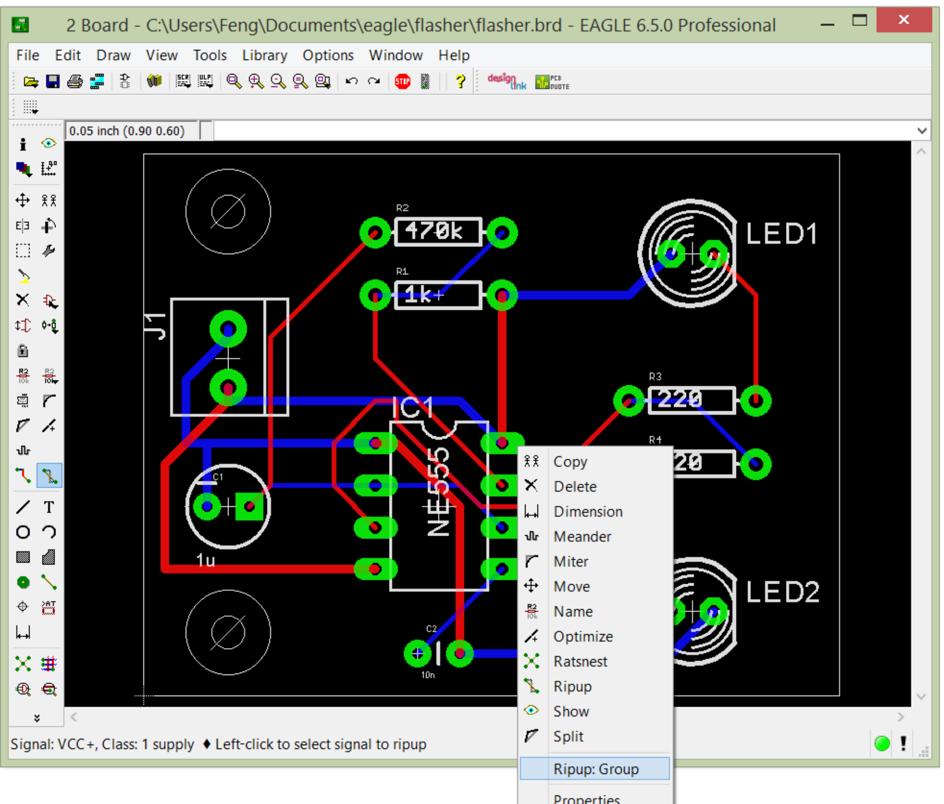


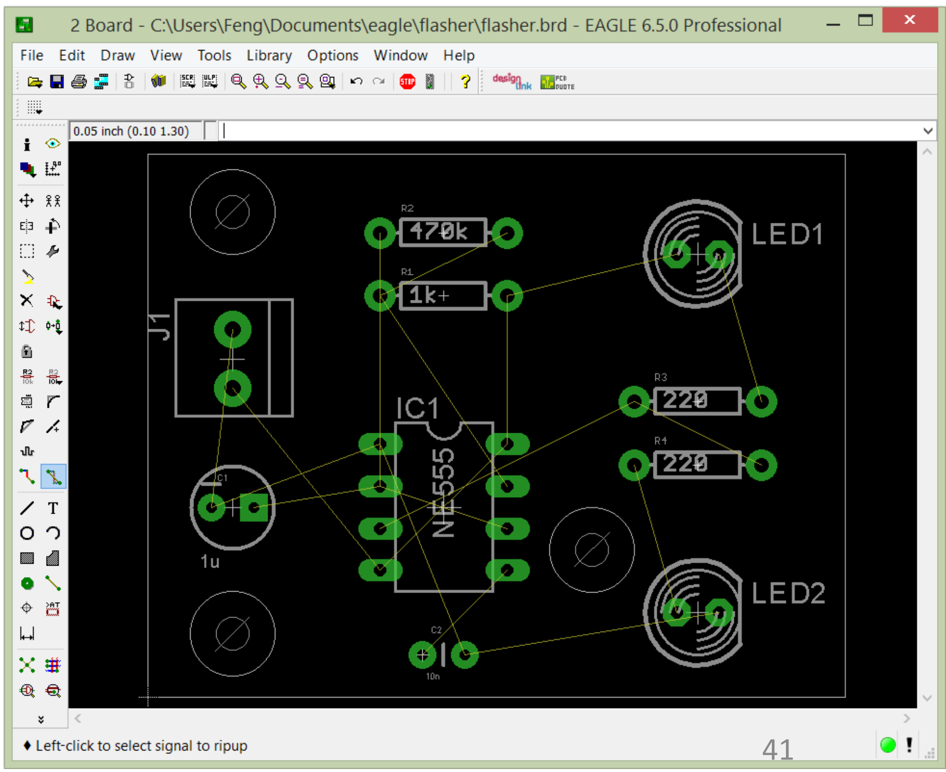
Board Layout



Rip Up the Tracks (Optional)

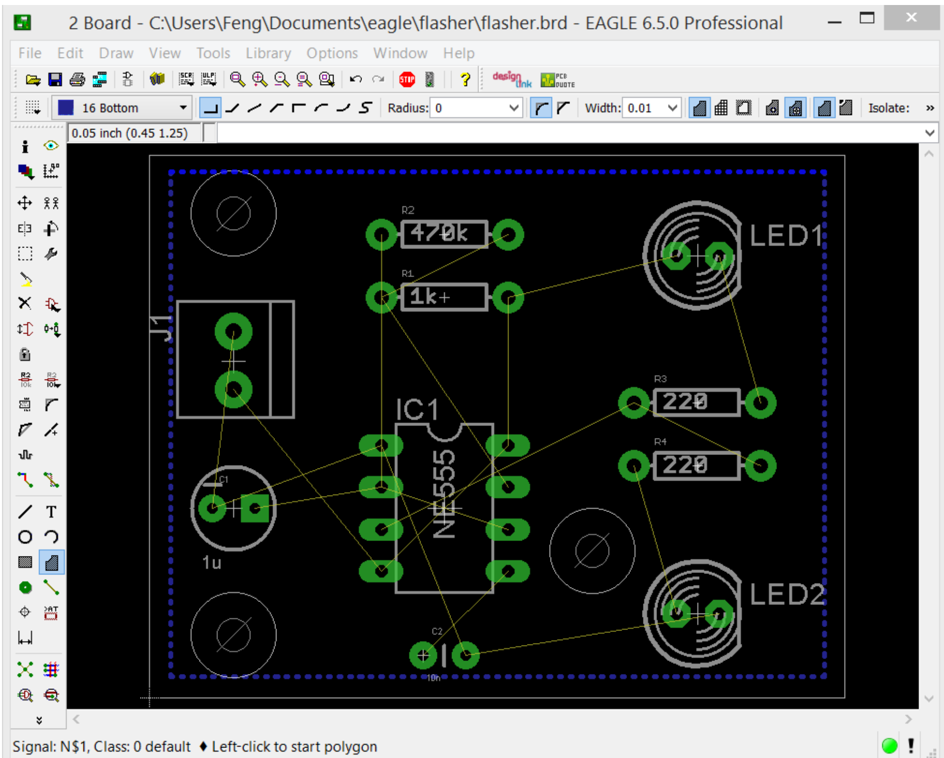
- “Rip up”
- “Group”
- “Ripup: Group”

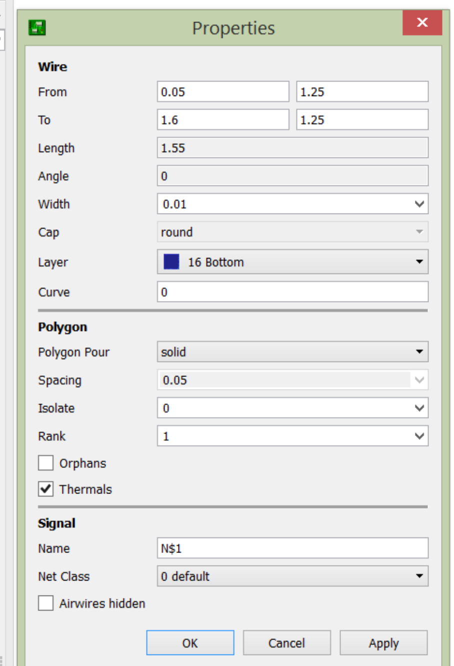




Tweak the Result – Add a Ground Plane

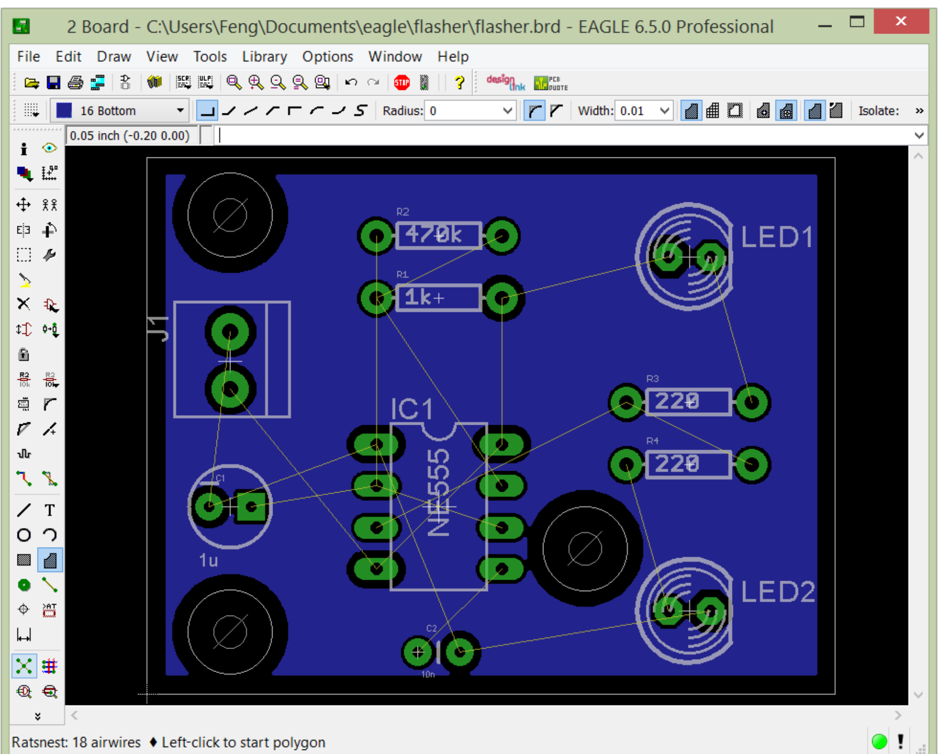
- Polygon – draw a square around the outline of the board
- (Right click the border of the polygon)Properties – Layer: 16 Bottom, Width 0.01

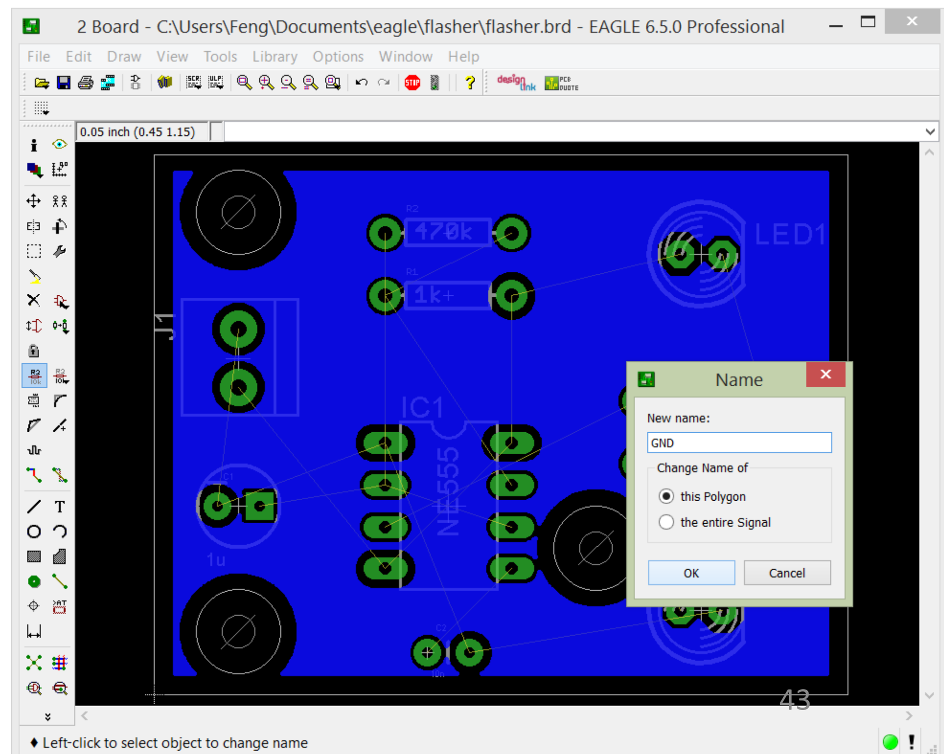




Add a Ground Plane

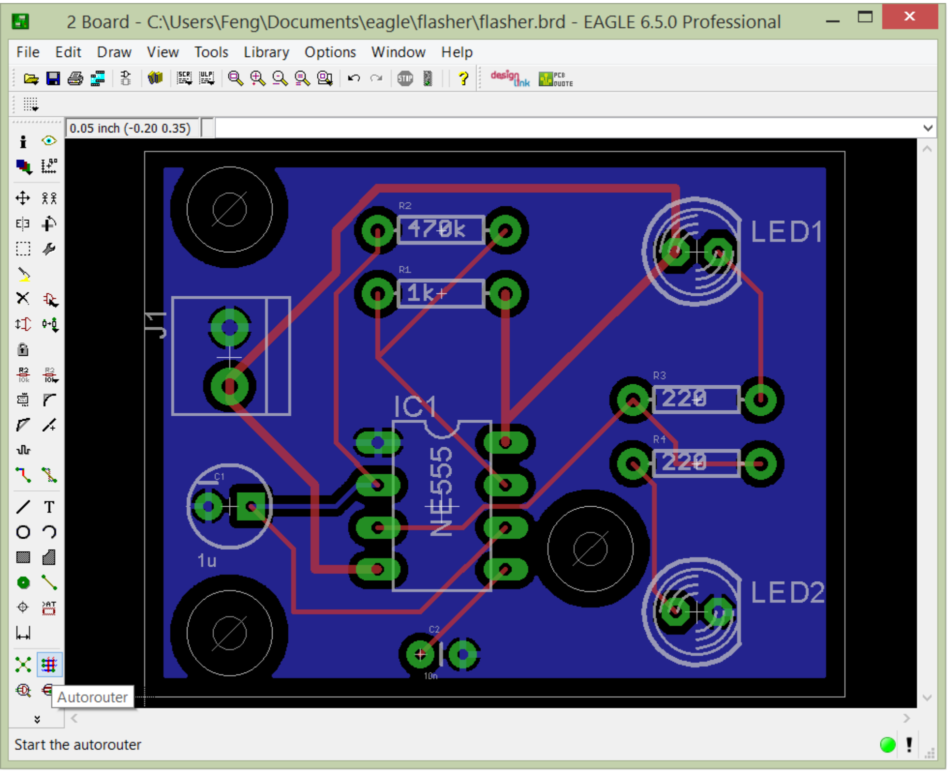
- “Ratsnest”- the polygon fill with blue
- “Name” - GND

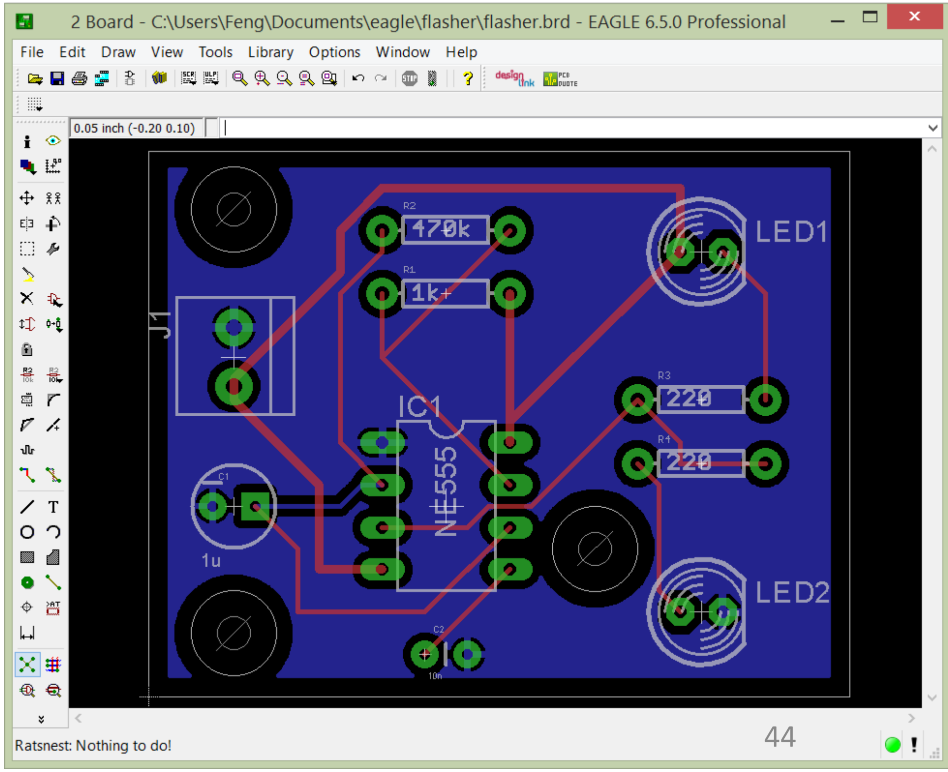




Add a Ground Plane

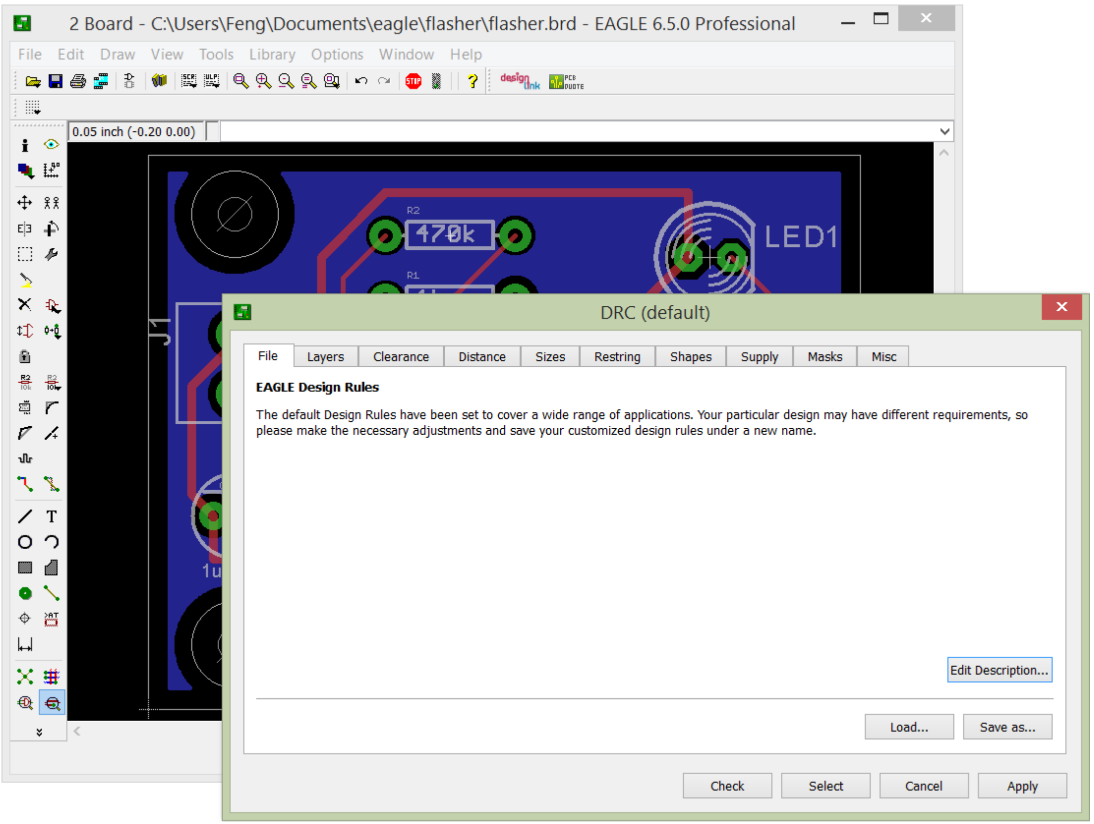
- Autorouter
- Ratsnest: Nothing to do!





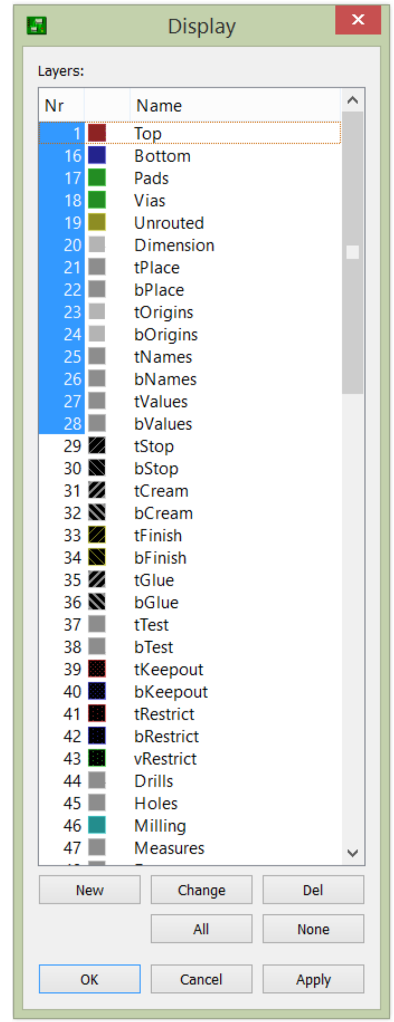
Design Rule Checker

- [Board]DRC



CAD Layers

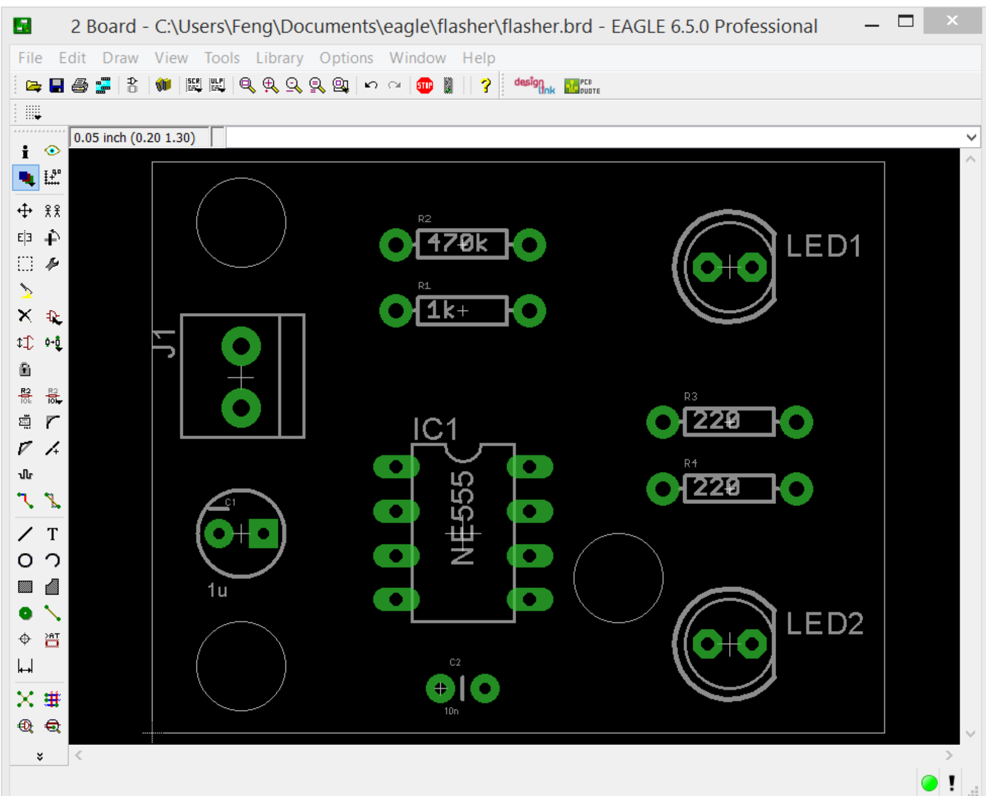
- 1 Top (top tracks)
- 16 Bottom (bottom tracks)
- 17 Pads
- 18 Vias
- 19 Unrouted
- 20 Dimension
- 21 tPlace (top silkscreen)
- 22 bPlace (bottom silkscreen)
- 23 tOrigins
- 24 bOrigins
- 25 tNames
- 26 bNames
- 27 tValues
- 28 bValues
- 44 Drills (for pads (of through hole components) and vias)
- 45 Holes (for mounting holes)

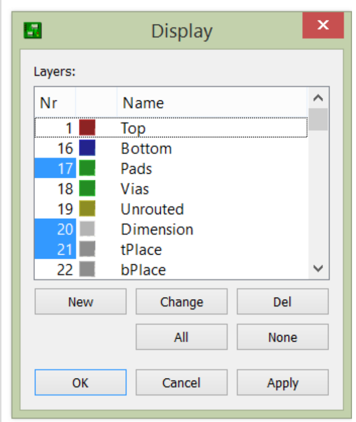


Note: CAD layer is different from (conductive) layer.

Add Text on the Top (Silkscreen)

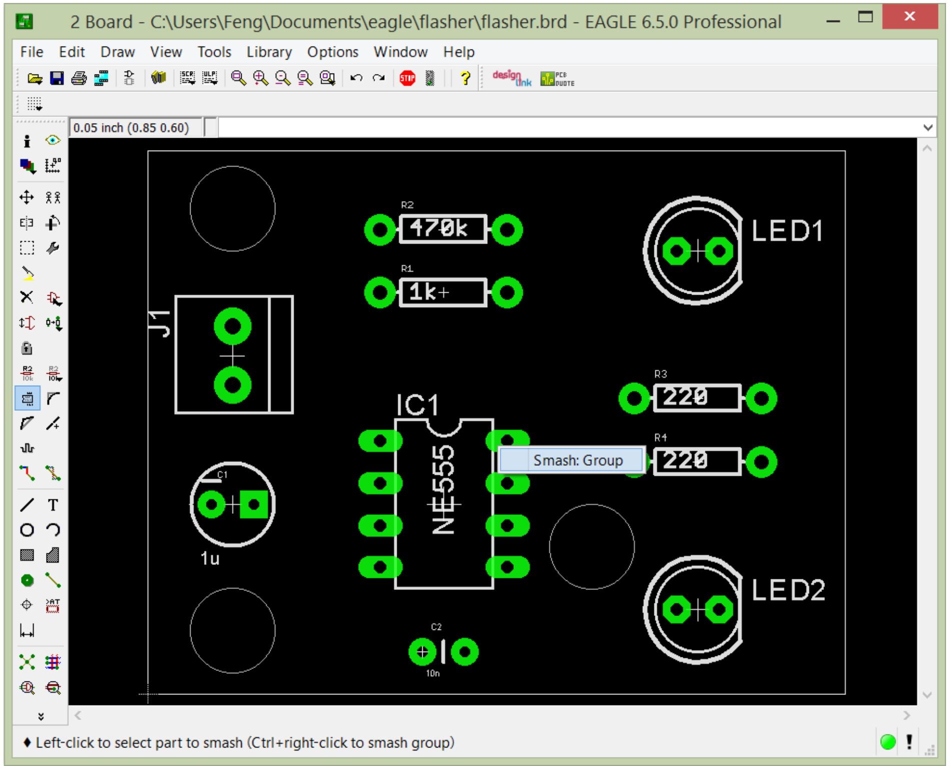
- [Board]Layers – Pads, Dimension, tPlace, tOrigins, tNames, tValues





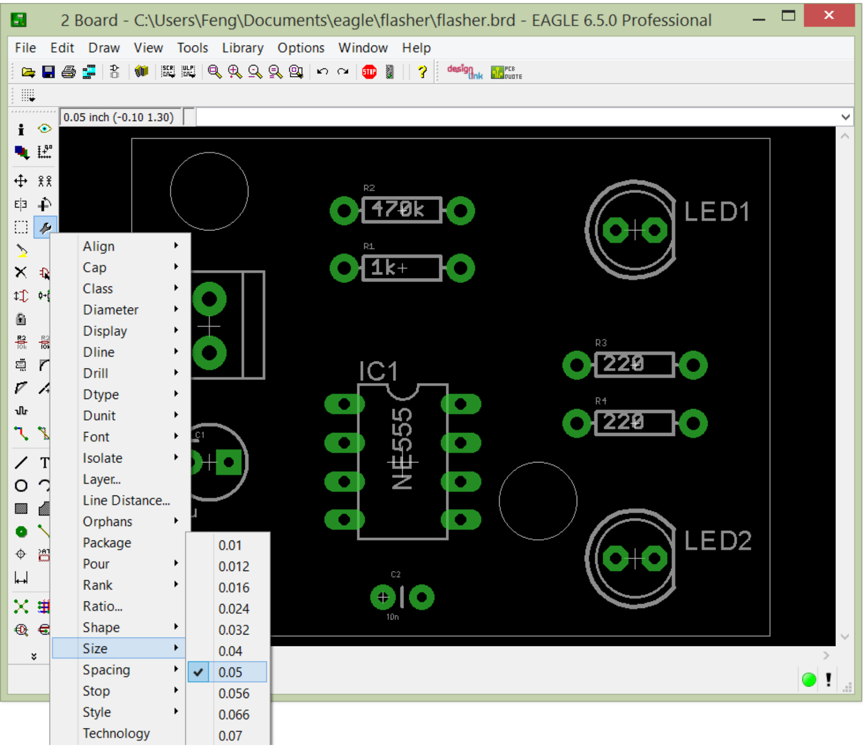
Smash – Separate the Text From Devices

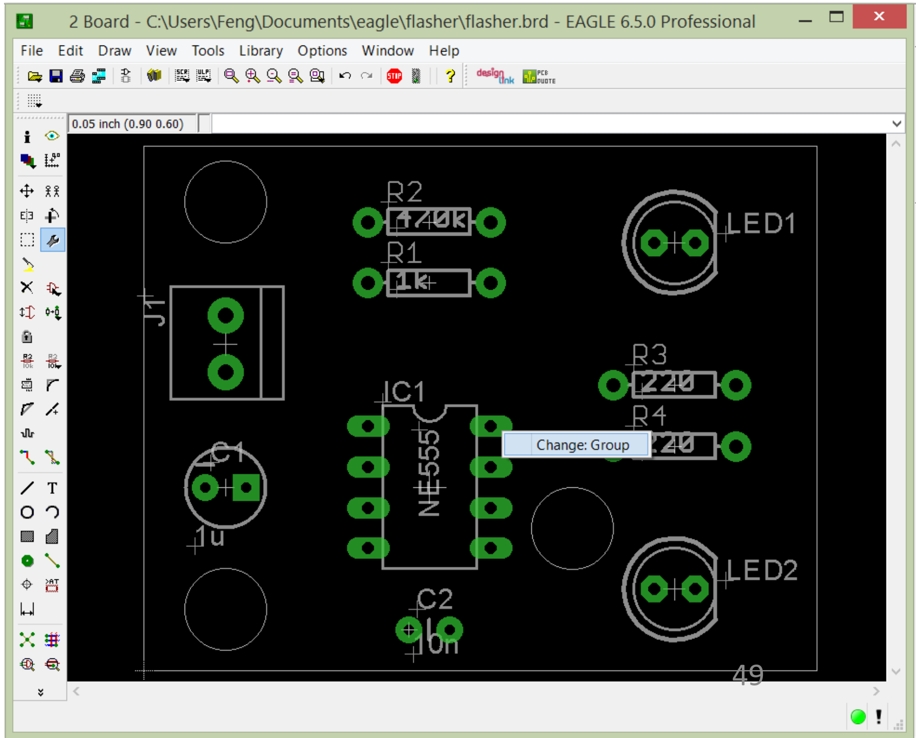
- [Board]Smash
- Group
- Smash: Group



Change Size

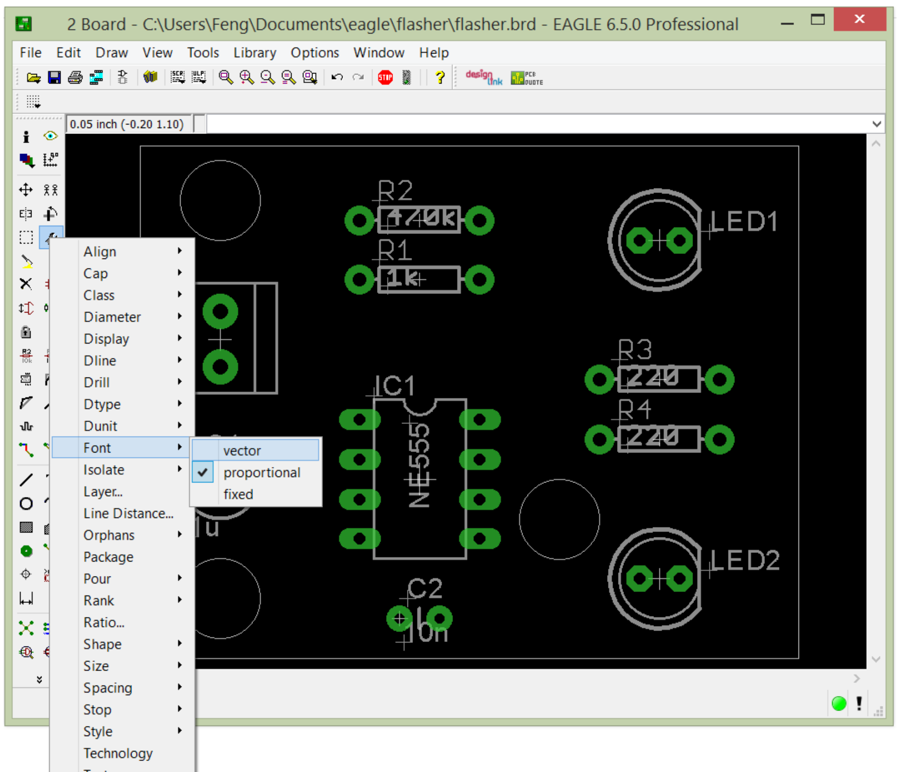
- Change->Size->0.05

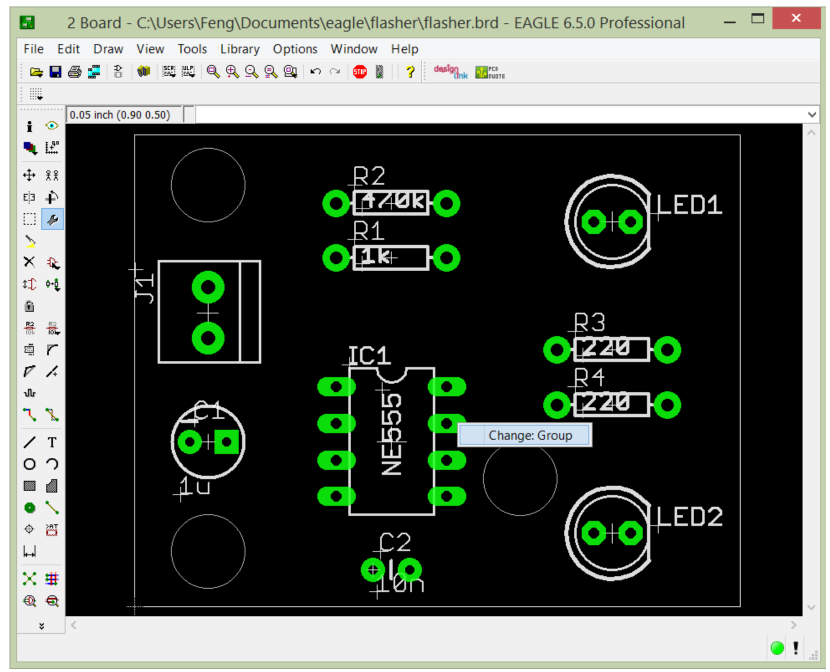




Change Font

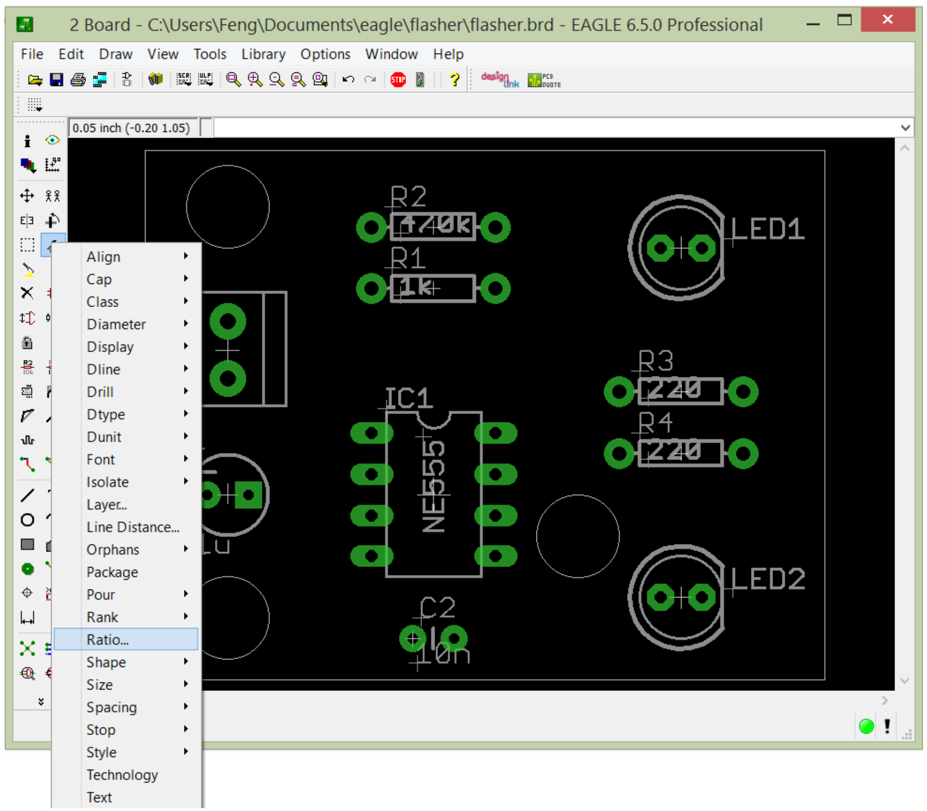
- Change –>Font->Vector
- Group- Change:Group

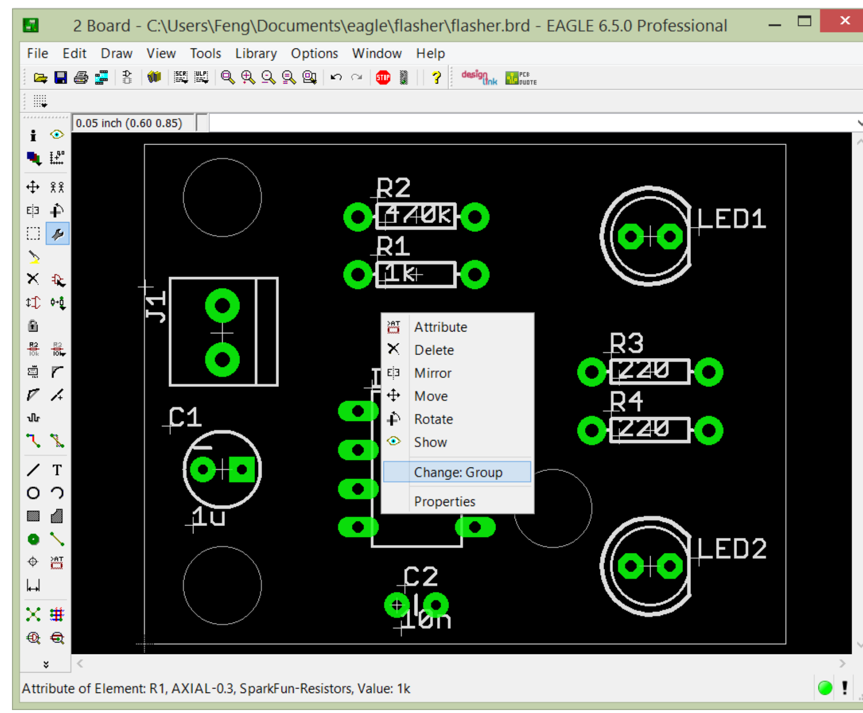




Change Ratio

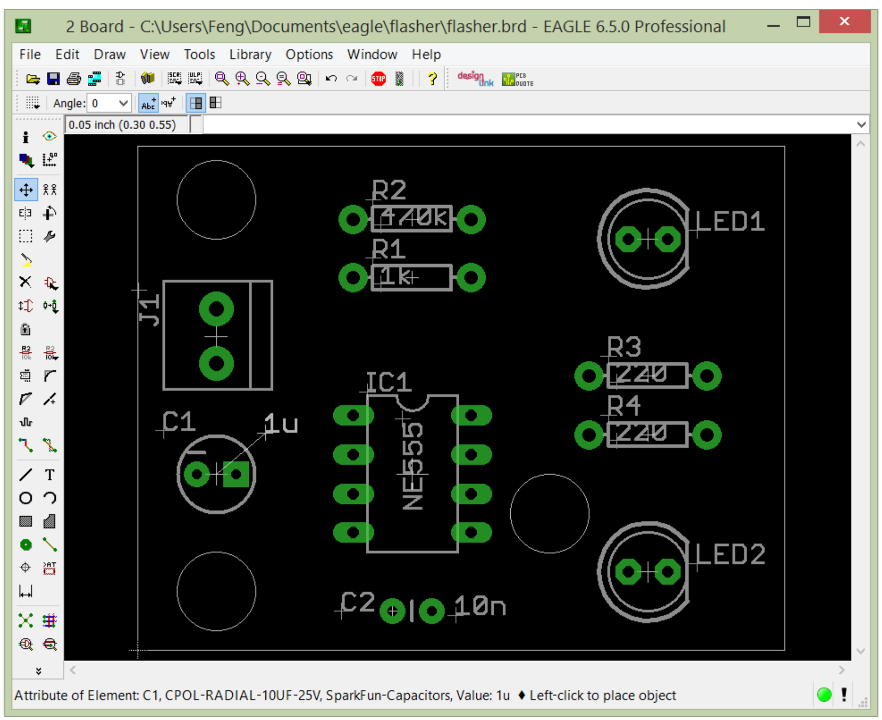
- Change->Ratio->12
- Group – Change:Group

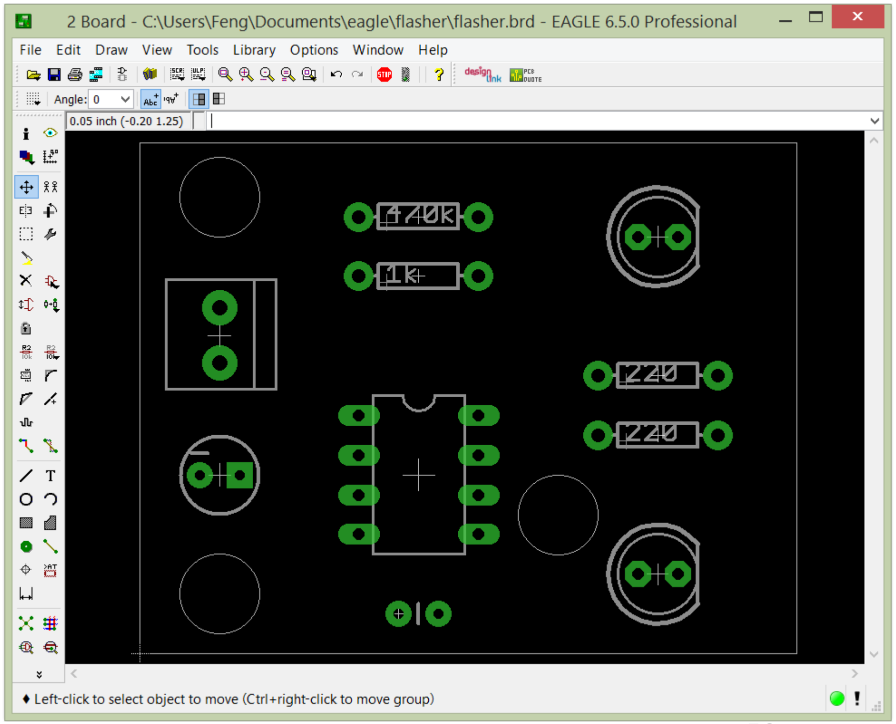




Move Labels

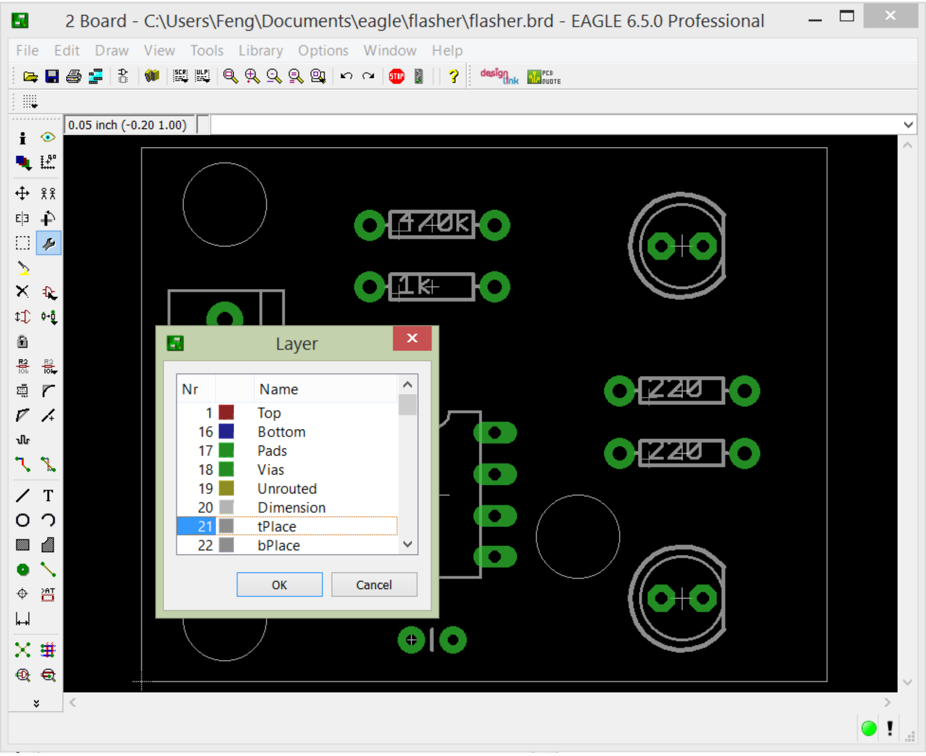
- Move (labels)
- Layers – Pads, Dimension, tPlace





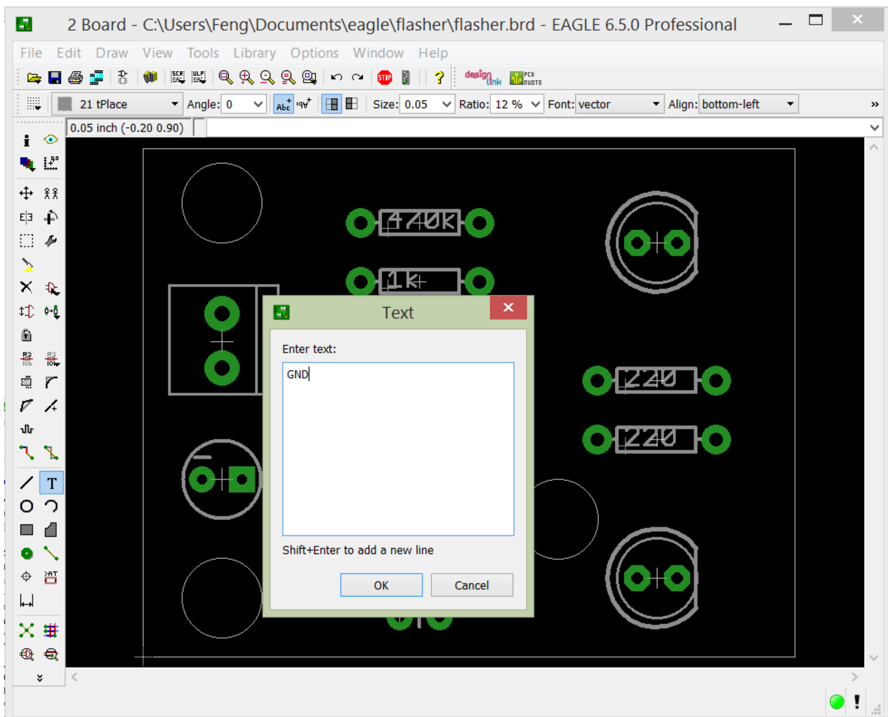
Add Text to the Top (silkscreen)

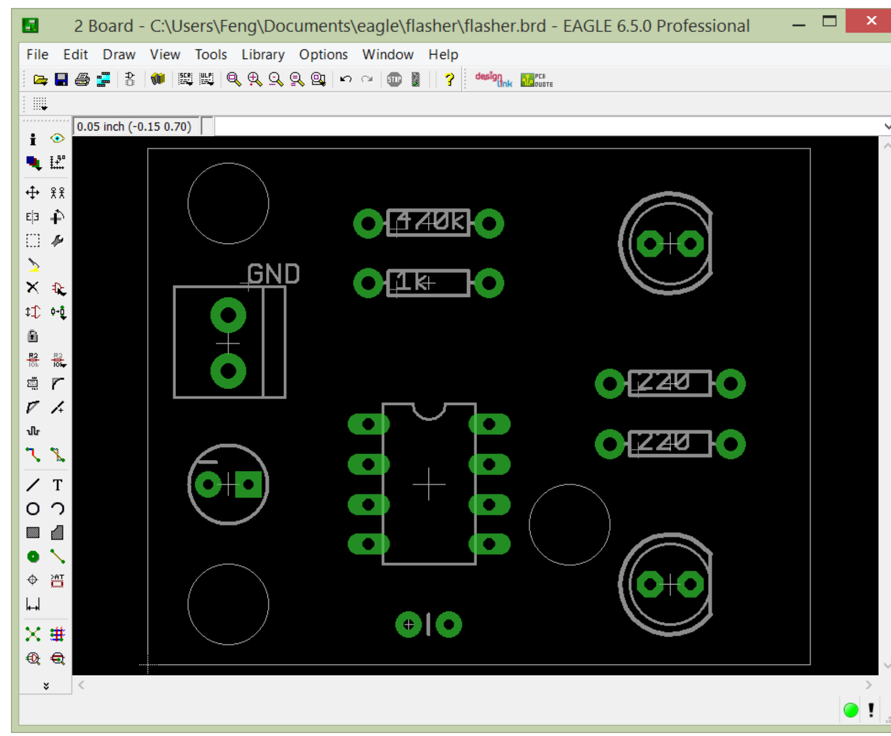
- Change-Layer-tPlace
- Change-Size-0.05
- Change-Font-Vector
- Change-Ratio-12



Add Text to the Top

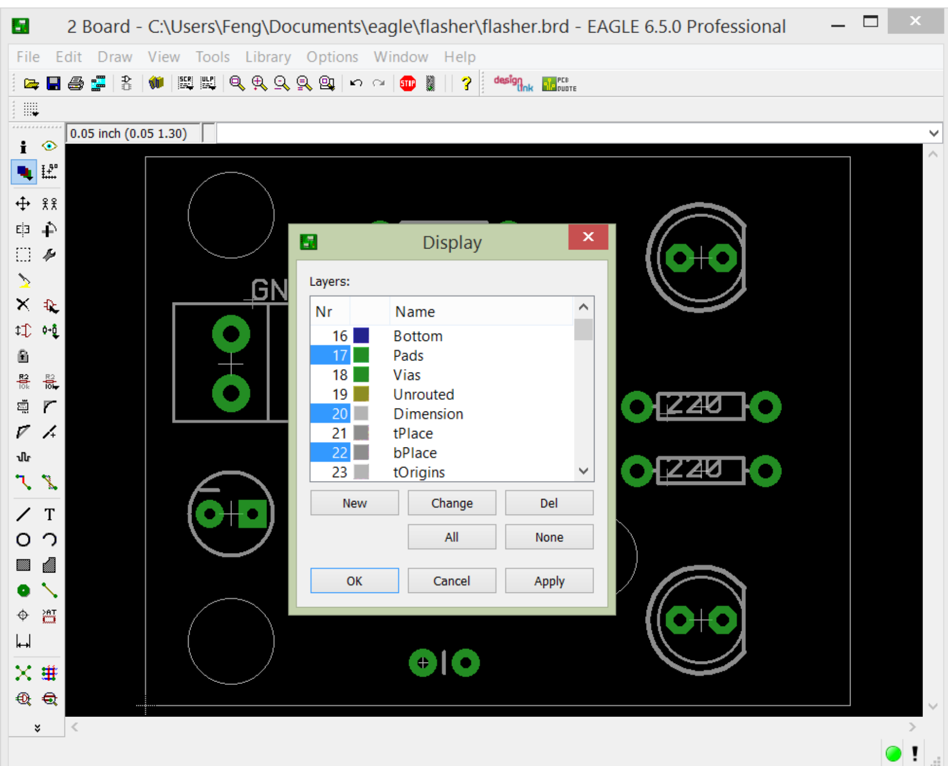
- Text->GND

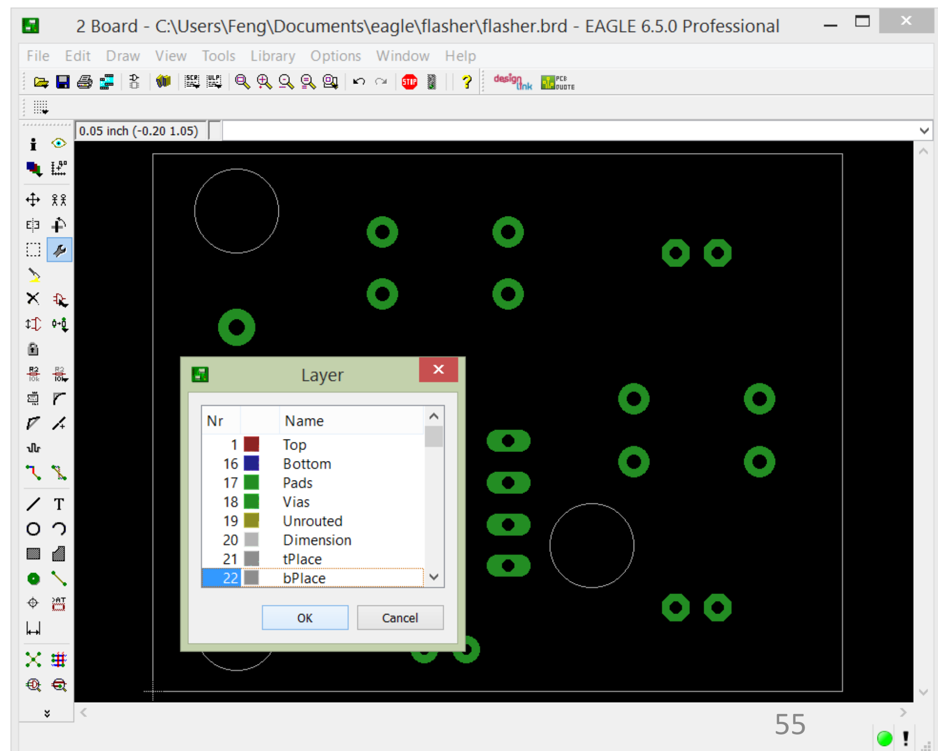




Add Text to the Bottom

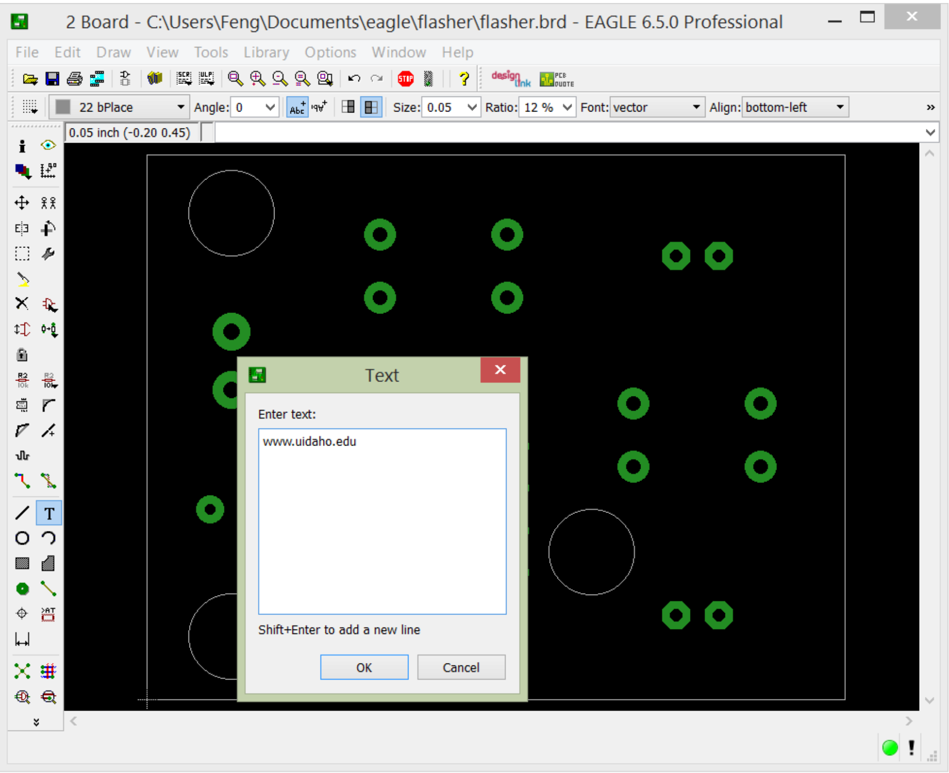
- Layer->Pads, Dimension, bPlace
- Change->layer->bPlace

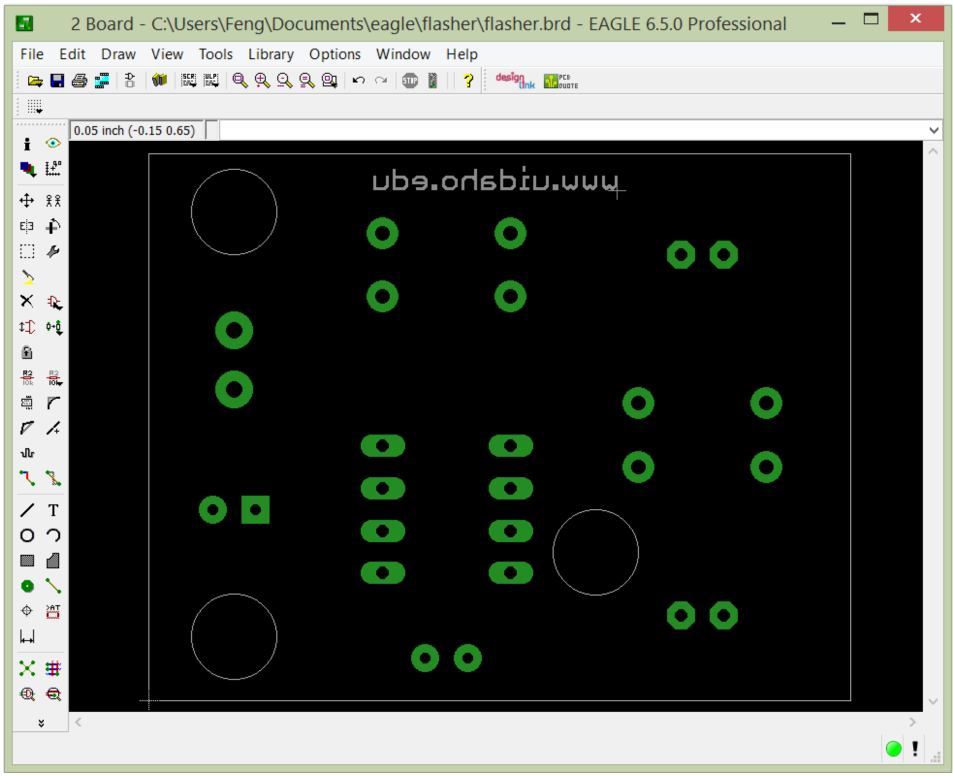




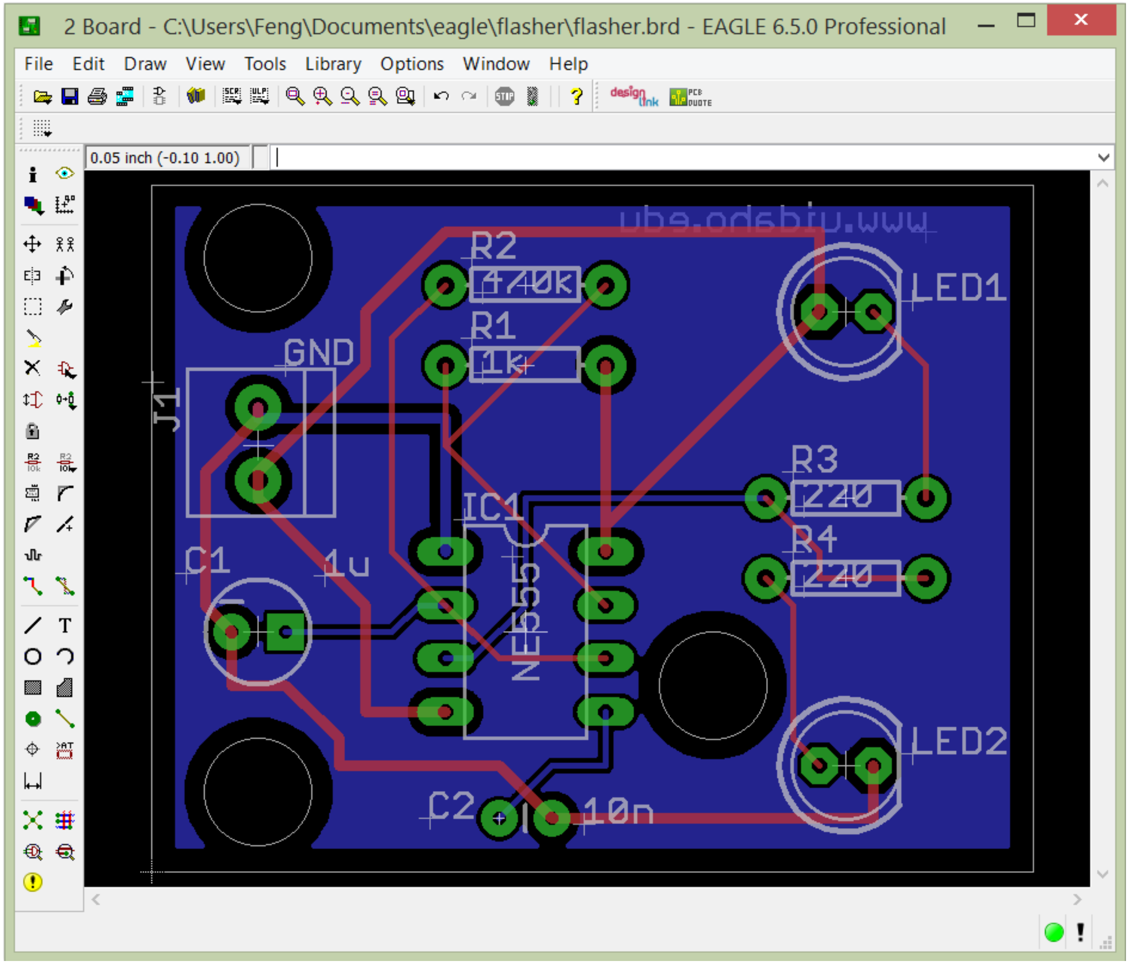
Add Text to the Bottom

- Text – www.uidaho.edu





Final Board Layout



Gerber Files

- A 2D vector image format
- Universal format and industry standard for PCB fabrication
- TXT files with coordinates that tell the PCB machines to go to location X, Y and do something (drill, expose, etch, print, etc).
- Formats:
  - RS-274X (Extended Gerber or X-Gerber)
    - Human readable ASCII format consisting of a sequence of commands and coordinates
  - RS-274D (obsolete)
- Different text files for different layers.
- Generated by the CAM (computer aided manufacturing) processor

Typical Gerber File Types

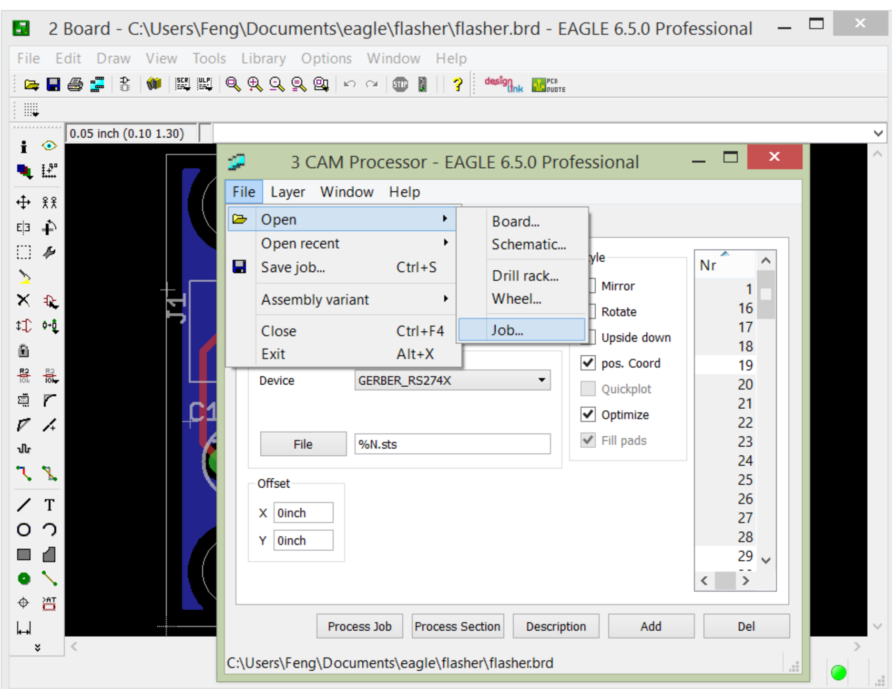
Description	EAGLE (default)	EAGLE (Sparkfun)	Orcad	Protel
Top copper layer (component side)	cmp	gtl	top	gtl
Bottom copper layer (solder side)	sol	gbl	bot	gbl
Top solder mask	stc	gts	smt	gts
Bottom solder mask	sts	gbs	smb	gbs
Top overlay (silkscreen)	plc	gto	sst	gto
Bottom overlay (silkscreen)	pls	gbo	ssb	gb0
NC Drill file (coordinate info)	drd	txt	thruhole.tap	drl

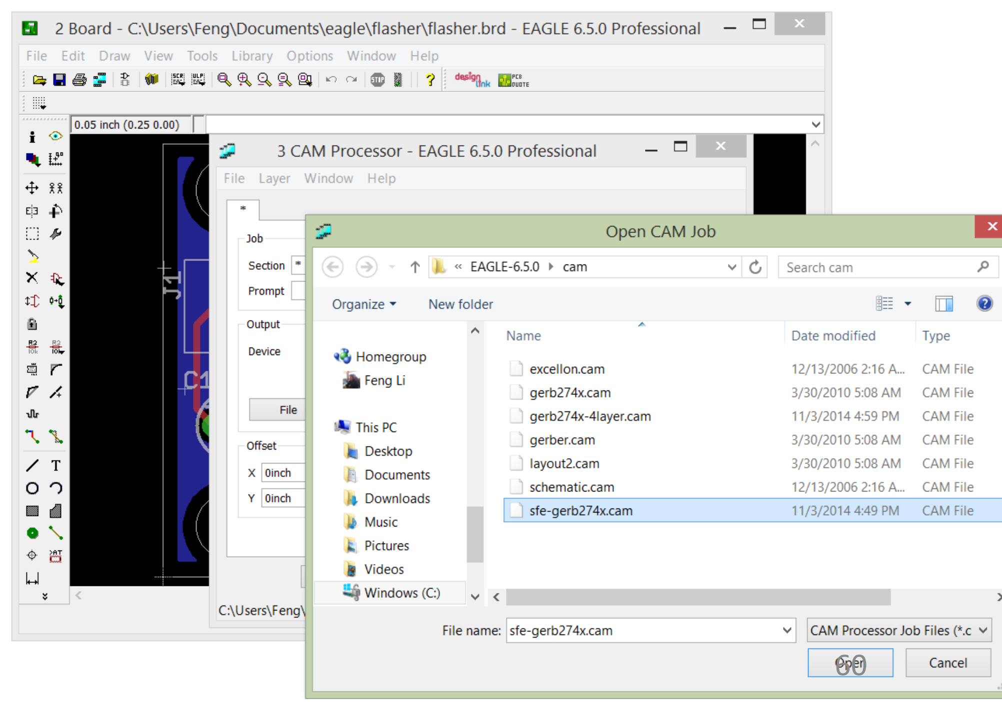
Note: Extensions are different for different PCB design tools, but information is the same.

EAGLE default CAM job needs to create drill file and gerber files separately.
SparkFun CAM job needs only one step.
https://github.com/sparkfun/SparkFun_Eagle_Settings/tree/master/cam
Download the file sfe-gerb274x.cam, and save to some folder

Gerber File Generation – CAM Processor

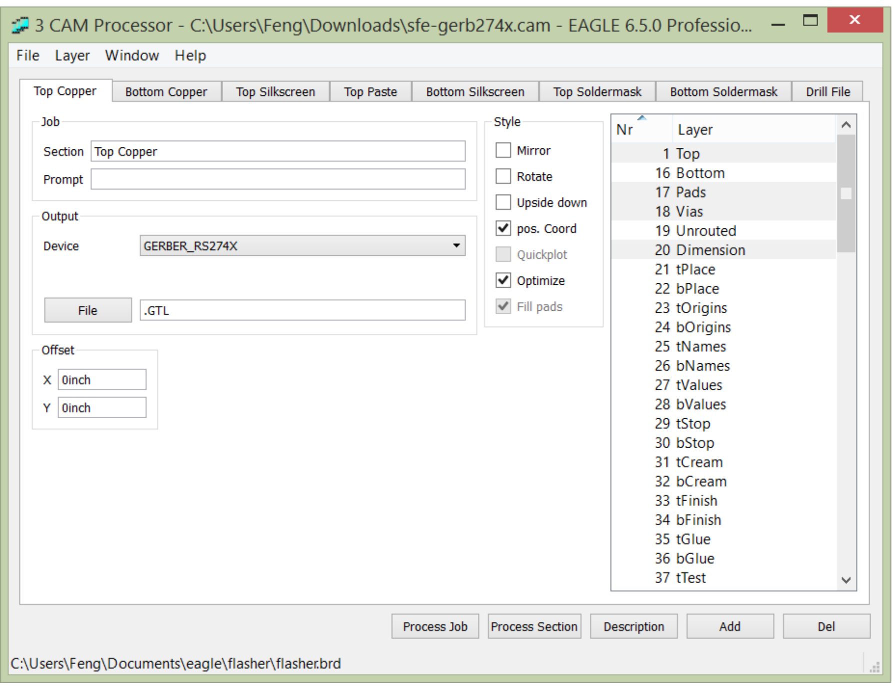
- [Board] File->CAM Processor
- [CAM Processor] File->Open->Job->sfe-gerb274x.cam

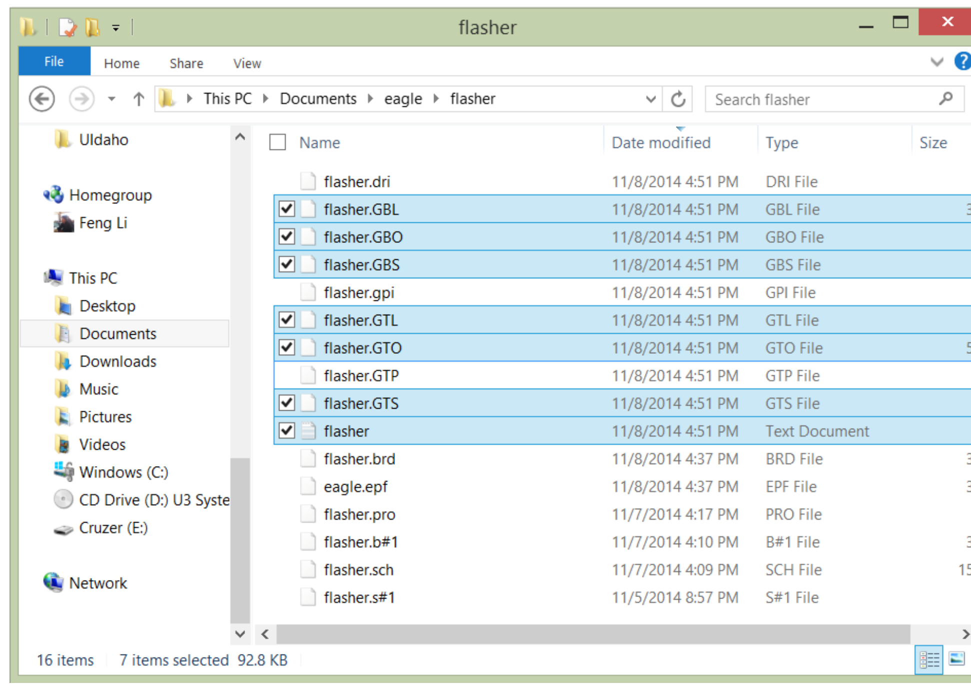




Gerber File Generation – CAM Processor

- [CAM Processor] Process Job





Gerber Review, Submit and Order

- Review: verify the CAM output in a Gerber viewer to make sure everything was positioned correctly
  - Viewplot: http://www.viewplot.com (can be downloaded and installed
  - Online viewer: www.gerber-viewer.com
- Design for manufacturability (DFM)(Files upload)
  - FreeDFM: http://www.freedfm.com https://www.my4pcb.com/net35/FreeDFMNet/FreeDFMHome.aspx
- Some PCB fab houses
  - Advanced Circuits http://www.4PCB.com
    - quick, reliable, and relatively cheap
    - $33/board with a week turn around
  - Dirty Circuits http://www.dirtycircuits.com

Assembly (Soldering)

- Through-hole components
  - DIY at ECE Senior Design Lab (BEL 216)

- Surface-mount devices
  - GJL 001 ECE lab (free)
    - < 30 components
    - Case size > 0402
    - Label pin numbers

    - Mr. Greg Klemesrud: gregk@uidaho.edu

Reference

- Simon Monk, “Make Your Own PCBs with EAGLE: From Schematic Designs to Finished Boards,” McGraw-Hill/TAB Electronics; 1 edition (May 22, 2014)

Special Thanks to

- Dr. Michael Santora
- Dr. Steven Beyelein
- Mr. John Jacksha (J. J.)
- Mr. Greg Klemesrud