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 feature tracks.
 - 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



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



PCB Process Flow



7

What is EAGLE?



- <u>Easily Applicable Graphical Layout Editor</u>
 - 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
- <u>http://www.cadsoftusa.com/download-eagle/</u>
- 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
 - <u>https://github.com/sparkfun/SparkFun-Eagle-Libraries</u>
 - C:\Program Files (x86)\EAGLE-6.5.0\lbr
- Adafruit
 - <u>http://github.com/adafruit/Adafruit-Eagle-Library</u>
 - C:\Program Files (x86)\EAGLE-6.5.0\lbr
- Component order
 - <u>http://www.digikey.com</u>

EAGLE User Interface – Control Panel



12

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

🖊 Con	trol Panel - C:\U	sers\Feng\Documents\	eagle\flasher - EAGLE 6.5.0 Professional 🛛 🗕 🗙
File View O	ptions Window H	Help	
Name Description			Empty Project
Libraries Libraries Design Rules User Language Programs User Language Programs Script Files CAM Jobs CAM Jobs CAM Processor Jobs Projects agale		Libraries Design Rules User Language Progr Script Files CAM Processor Jobs	Use the context menu to create new schematic or board files within this project.
	Close Project	Empty Project	
	New •	Schematic der	
	Rename Copy Delete Edit Description Use all Use none Search in folder	Board Library CAM Job ULP	
		Script Text	
		Folder Project	

Save and Rename of New Schematic

- [Schematic]File->Save as... (flasher.sch)
- Note: Do not create a board file yet.

File Minus Options Mill 1	Late				<u> </u>				
File View Options Window F	lelp								
Name Description					Empty Project				
Libraries		Librar	Libraries		Use the context menu to create new schematic or board files within this project.				
Design Rules		Desig	n Rule	es					
User Language Programs		User L	angu	age Progr					
Scripts		Script	Files						
CAM Jobs		CAM	Proce	ssor Jobs					
Projects									
a eagle flashor		Empt		4. C - h h		> E			
		Empty		i Schemati	c - C:\Us	ers\F	eng\Documents\eagle\flas – 🕒 🗖		
P m play P m play P m play		Exam	File	e Edit Drav	v View	Tools	Library Options Window Help		
		EXam		New	Ctr	I+N	📖 🔍 🔍 🔍 🔍 💷 🗠 🖓 💷 🕷 🐭 🕬		
			-	Open	Ctr	1+0			
				Open recent		•	1)		
					<u> </u>				
				Save	Ctr	1+5			
				Save as					
				Save all			-		
				Print setup					
			6	Print	Ctr	I+P			
			2	CAM Proces	sor				
			-				-		
			ŏ	Switch to be	ard		-		
				Import		+			
				Export		•	>		
			SCR CAC	Execute Scri	pt				
			ULP	Run ULP					
				Class	Cha				
				Close	Ctr	1+r4	16		

Schematic Editor Window

Schematic Editor Window Command Tools

Finding and Add a Component

• [Schematic] Add

Search for a Component

• Enter "*555*" in the Search field

- Find "NE555"
- Click "OK "

1 So	chematic - (C:\Users\Fen	ig\Documents\eagle\flasher\fl	asher.sch - EAGLE 6.5.0	Prof –
ile E	dit Draw V	/iew Tools L	ibrary Options Window Help		
🖙	in 🚑 📑 🕺 🕹 🕹	'1 🔹 💔 🔛	📖 🔍 🔍 🔍 🔍 🔍 🗠 🛛 💷 🕷	design - II LTC -	
		88			
	Sheets & ×	0.1 inch (-0.6 4.	.0)		~
i 📀					<u> </u>
ŧ ĽĽ				AD	DD ×
₽ 88	1				
4 E			Name	Description ^	
1			 micro-philips 	Philips Microcontrol	
5			▷ PCA9555	16-bit I2C and SMB	
K D-			micro-siemens	Siemens Microcontr	, ∧co , ≤)>VALUE
T 6+8	3		▷ BTS555	Smart Highside Hig	
+v ¥¥ ⊳			Misc	LT-Spice library gen	
Ð			NE555	An idealized 555 ti	2 TRI GND 1- 🔍 🔍 🔍 🖤
			▲ solpad	Solder Pads/Test Poi	>VALUE
" r			LSP10	SOLDER PAD drill 1	
288			SparkFun-AnalogIC	SparkFun Electronic	General purpose bipolar Timer
/ т			> 556	Dual timer circuit (2	Package: DIL-08
			P INESSS	General purpose bip	Dual In Line Package
<u> </u>			SparkFull-Poweric	MAY1555 Lithium C	-
			4 st-microelectronics	ST Microelectronics	
J			NE555	General purpose bin	
- ABC			4 trafo	Transformers	
5 6			▷ BV054*	ERA EI 54/18.8 16 VA	
			▲ transistor	Transistors	Attribute Value
		1	▷ *-NPN-	NPN Transistror	MF Arrow Electronics
	~		▲ transistor-neu-to92	NPN Transistors	MPN NE555N
			▷ *-NPN-	NPN Transistror	OC_FARNELL 1467742
			✓ Pads ✓ Smot	ds 🗸 Description 🖌 Preview	OC_NEWARK 89K1486
			Search 🖾 *555*	v	
			Attributor 🕞		
			Autibules 🖬	•	
					OK Drop Cancel
					20

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

🖭 1 Schematic - C:\l	Users\Feng\Documents\eagle\flasher\flasher.sch -	EAGLE 6.5.0 Prof	📟 1 Schematic - C:\Users\Feng\Documents\eag	le\flasher\flasher.sch - EAGLE 6.5.0 Prof 🗕 🗖	× • ×	
File Edit Draw View	/ Tools Library Options Window Help		File Edit Draw View Tools Library Options Window Help			
😂 🖬 🎒 🚰 📅 1/1	▾ 🗰 🕮 💐 🔍 🄍 🄍 🌒 🗠 ལ 🚥 🛔 🛛 ? 🕬 🐘	k V LTC V	🛛 🚘 🖬 🚭 🚅 🔒 1/1 🔹 🗰 📖 🔍 🔍 🔍 🔍	က ာ 💷 🎆 🛛 ? design _{ink} 🖛 🌌 spice 🖛		
) III 🖪 🖪 🗃 🖪 🖪	Đ				is project.	
	AI AI	Control of the second	Image: Street of a x 0.1 inch (1.5-0.1) i Street of a x 0.1 inch (1.5-0.1)<	Name Description 4 rd Resistors, Capacitors, In • EL- BIPOLAR ELECTROLY • resistor Resistors, Capacitors, In • EL- BIPOLAR ELECTROLY • CAP_POL Capacitor Polarized • CAP_POLPTH2 CPOL-RADIAL-100UF • CAP_POLPTH2 CPOL-RADIAL-100UF • CAP_POLPTH4 CPOL-RADIAL-100UF • CAP_POLPTH4 CPOL-RADIAL-100U • CAP_POL Capacitor Polarized	Is project. DO X Image: Specific Constraints of the spec	
 ▲ Left-click to place 'NE 	RESISTORPTH-1/4W AXIAL-0.1EZ RESISTORPTH-1/6W 1/6W-RES RESISTORPTH-1/W AXIAL-0.6 RESISTORPTH-2W AXIAL-0.8 Pads Smds Ø Description Ø Preview Search Ø *resistor*	Attribute Value	Left-click to place 'RESISTORAXIAL-0.3'	✓ Pads Smds ✓ Description ✓ Preview Search ✓ electrolytic capacitor ✓		
	Attributes 🖾 🗸 🗸	OK Drop Cancel		Attributes 🗗 🗸 🗸	OK Drop Cancel	

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...

🕮 1 Schematic - C:\Users\Feng\Do	ocuments\eagle\flasher\flasher.sch - EAG	LE 6.5.0 Prof 🗕 🗖 🗙
File Edit Draw View Tools Library	y Options Window Help	
🚘 🖶 🎒 🚍 🔓 1/1 🔻 🗰 🗮 🗮 🍼	🔍 🔍 🔍 😰 🗠 🗠 💷 📓 🗌 💡 🔤 designa 🗸 🖊	LTC v
	я. <u> </u>	
Sheets		×
■ 比 1 2	++	
		45
		≸ ¥⊡
2	IC1	1
× 🕰		
\$_ 0 • 0	₩2 /RES 4	· ☆₹
	-7 DIS +OUT 3	
	CON 5	
m 99	2 TRI GND 1	
	Ϋ́Τ	[≩] 1
	<u>+ C</u> 1	4
	\uparrow	** <u>-</u>
× <		>
◆ Left-click to select object to move		0 ! .:

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

Net classes

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

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.

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

Board Tools

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.

Display							
Layers:							
Nr	Name	^					
1	Тор						
16	Bottom						
17	Pads						
18	Vias						
19	Unrouted						
20	Dimension						
21	tPlace						
22	bPlace						
23	tOrigins						
24	bOrigins						
25	tNames						
26	bNames						
27	tValues						
28	bValues						
29 💋	tStop						
30 🔊	bStop						
31 💋	tCream						
32 🔊	bCream						
33 💋	tFinish						
34 📉	bFinish						
35 💋	tGlue						
36 🔊	bGlue						
37	tTest						
38	bTest						
39	tKeepout						
40	bKeepout						
41	tRestrict						
42	bRestrict						
43	vRestrict						
44	Drills						
45	Holes						
46	Milling						
47	Measures	~					
		Del					
New	Change						
New	Change	None					

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)	стр	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. <u>https://github.com/sparkfun/SparkFun_Eagle_Settings/tree/master/cam</u> 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->sfegerb274x.cam

File Edit Draw View Tool	ls Library Options	Window Help				options Window ବ୍ଲୁ ଷ୍ଟ୍ରା ନ ଦା 🌚		
	3 CAM P File Layer Open Open recent Save job Assembly varia Close Exit Exit Device File Offset X Oinch	Yrocessor - EAGL ow Help Ctrl+S nt Ctrl+S Ctrl+F4 Alt+X GERBER_RS274X	Board Schematic Drill rack Job	yle Mirror Rotate Upside down Pos. Coord Quickplot Optimize Fill pads	Nr 1 16 17 18 19 20 21 22 23 24 25 26 27	3 CAM Processo er Window Help n • • • • • • • • • • • • • • • • • • •	or - EAGLE 6.5.0 Professional → P Open CAM	I Job ✓ ♂ Search cam ✓ ♂ Search cam I Job ✓ ♂ Search cam II Job II Job ✓ ♂ Search cam III Job III Job
	Y Dinch Proce	ess Job Process Sect	tion Description	Add	28 29 v < >	eng\	dows (C:) v c File name: sfe-gerb274x.cam	✓ CAM Processor Job Files (•িচুচি) Canc

Gerber File Generation – CAM Processor

• [CAM Processor] Process Job

👠 💽 🚯 = I	flasher		-					
File Home Share Vie	W			~ ?				
$$ $$ \checkmark \uparrow $$ $$ $$ This PC	▷ Documents ▷ eagle ▷ flasher	✓ 🖒 Searc	h flasher	Q				
👢 Uldaho 🔨	Name	Date modified	Туре	Size				
a llomograun	flasher.dri	11/8/2014 4:51 PM	DRI File					
- Fong Li	✓ flasher.GBL	11/8/2014 4:51 PM	GBL File	3(
	✓ flasher.GBO	11/8/2014 4:51 PM	GBO File					
🐚 This PC	✓ ☐ flasher.GBS	11/8/2014 4:51 PM	GBS File					
	📄 flasher.gpi	11/8/2014 4:51 PM	GPI File					
	Ilasher.GTL	11/8/2014 4:51 PM	GTL File	9				
Downloads	✓ flasher.GTO	11/8/2014 4:51 PM	GTO File	5(
Music	flasher.GTP	11/8/2014 4:51 PM	GTP File					
Dictures	✓ flasher.GTS	11/8/2014 4:51 PM	GTS File					
Videos	🕑 📄 flasher	11/8/2014 4:51 PM	Text Document					
Windows (C)	flasher.brd	11/8/2014 4:37 PM	BRD File	3!				
CD Drive (D:) 112 Synta	eagle.epf	11/8/2014 4:37 PM	EPF File	3(
CD Drive (D.) 03 Syste	flasher.pro	11/7/2014 4:17 PM	PRO File	-				
Cruzer (L.)	flasher.b#1	11/7/2014 4:10 PM	B#1 File	3!				
Aletwork	flasher.sch	11/7/2014 4:09 PM	SCH File	154				
Network	flasher.s#1	11/5/2014 8:57 PM	S#1 File	1				
~	✓ <							
16 items 7 items selected 92	2.8 KB							

Gerber Review, Submit and Order

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

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