

Prototype To Production

Ideas to tangible reality

A Process Overview

Hamcom 2015

June 13th, 2015

Mike Malone, KD5KXF

Eric Wolf, N5EBW

<http://www.baconfatlabs.com>

THE BEST use of fat SINCE bacon!

About Me

- Computer Programmer by trade

- Hardware “programmer” by hobby

- Other hobbies include:

- Ham Radio (go figure)

- Woodworking (I still have ten fingers, see bullet point #1)

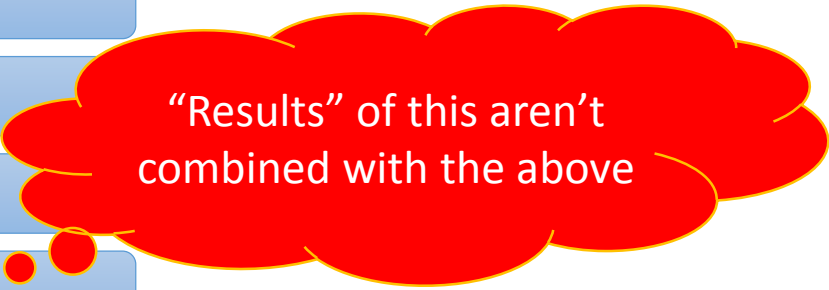
- Metalworking

- Shooting, reloading, etc., etc. I’m all about the pew, pew, pew lifestyle.

- Homebrewing

- Angering family members with the amount of hobbies to time ratio I have. 😊

- Too many more to mention for fear of further angering family members. 😊



“Results” of this aren’t combined with the above

As you might imagine, circuit board design and programming are a natural “glue” that bonds some of the above hobbies together.

Assumptions

Me.....

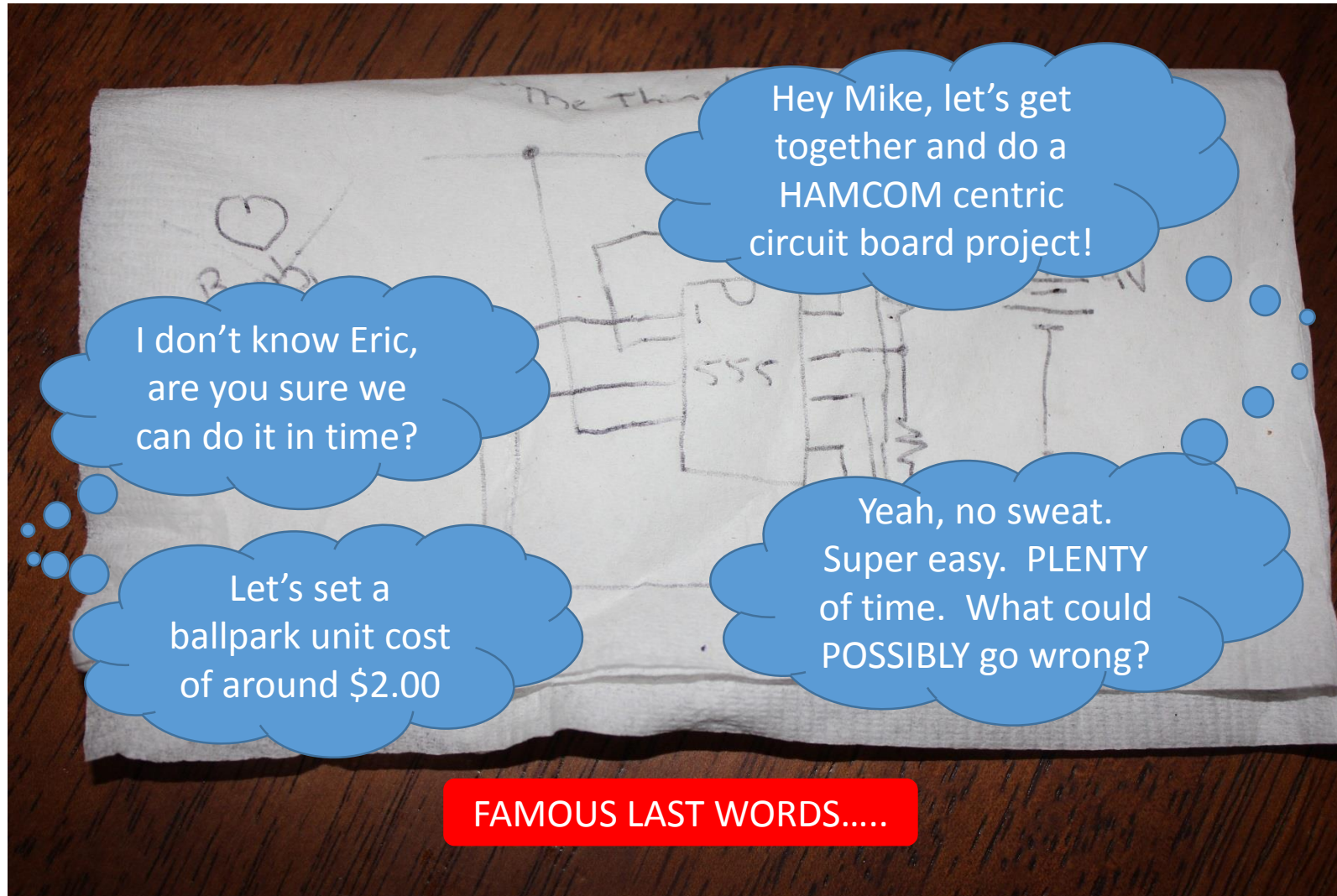
- I am not perfect. In fact, you might say I fail a lot. Fail early and fail often. That's how you learn.
- I am not a formal engineer. I am, however, well versed in the smoke test.
- For the purposes of my designs, tolerance criteria includes:
 - "Did it catch on fire?"
 - "Is it getting hot? Because that might eventually catch on fire."
 - "It functions, but is it pretty?" (Seriously, who wants a product that's ugly?)
- Methods and tools used in this presentation are a reflection of my experiences. YMMV.
- Many other people wrote the book on this stuff. I just read it and am in no way an expert.

You.....

- Have seen Part 1 of this presentation (Mike's pre-prototyping overview).
- Knowledge of electronics/ideas you wish to bring to fruition, or at least a willingness to learn.
- Restraint necessary to not throw things at me if you disagree with this presentation.

So you have an idea....where it all starts.

Here we see the abundant “bevnep” in it’s native environment, where it will not remain for long.



Schematic Capture and Board Layout

Fancy trade name is
“Electronic Design
Automation” (EDA)

Free

DipTrace*

Eagle*

FreePCB

Fritzing

gEDA

KiCAD

OrCAD*

Oregano


Purchase

Altium

DipTrace*

Eagle*

OrCAD*

 =Windows

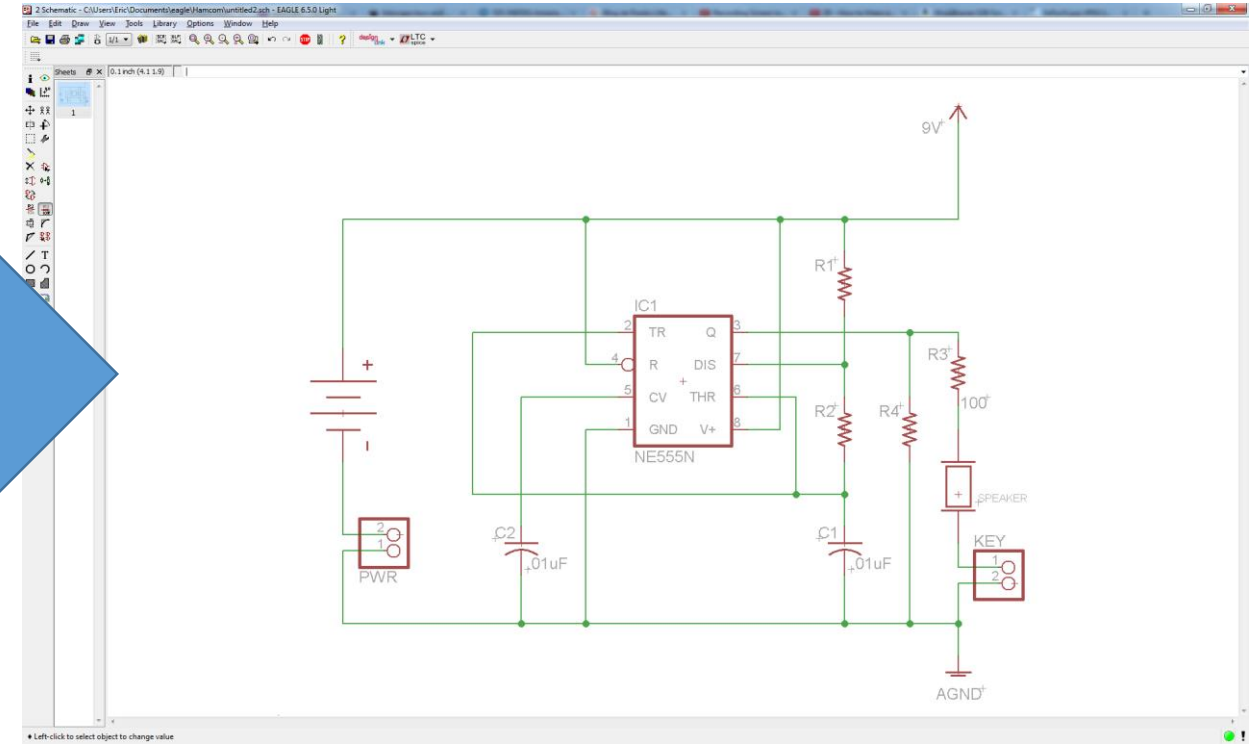
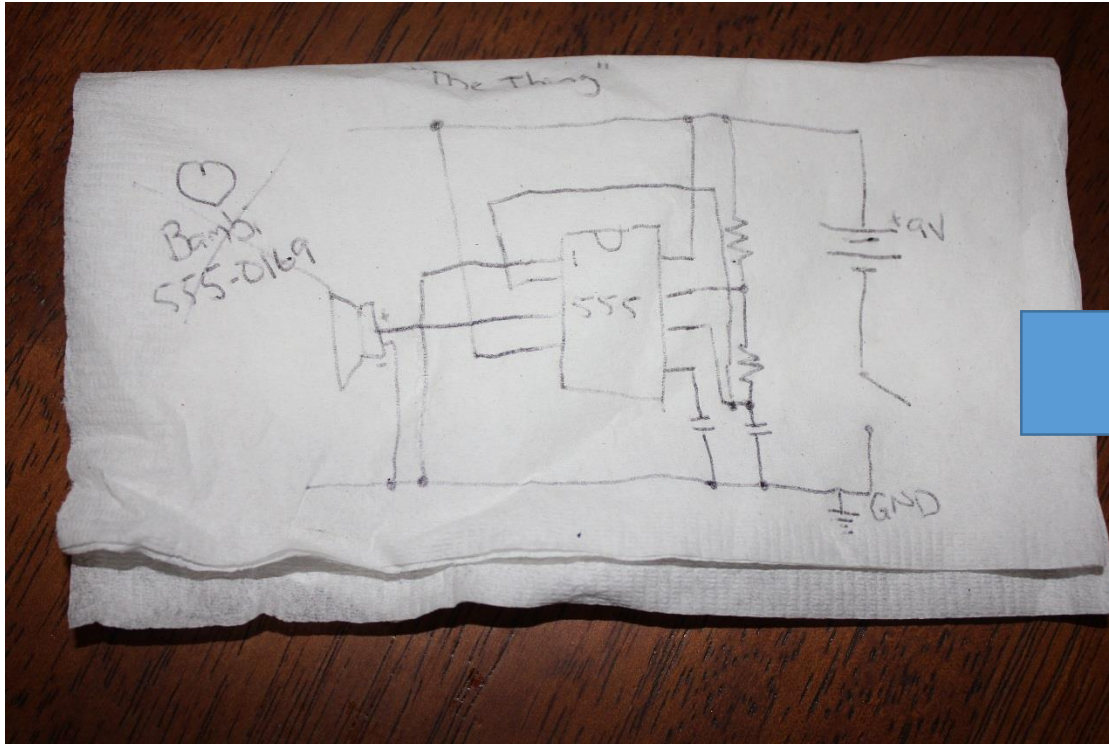
 = Unix/Linux

 = Both

This is not an exhaustive list, by far, just a few of the “heavy hitters”. We will be focusing on CadSoft’s EAGLE for the purposes of this session. It’s what I use, and as far as I can tell, the most prevalent hobbyist layout editor on the market today. Development is active, support is abundant, and the user communities are phenomenal.

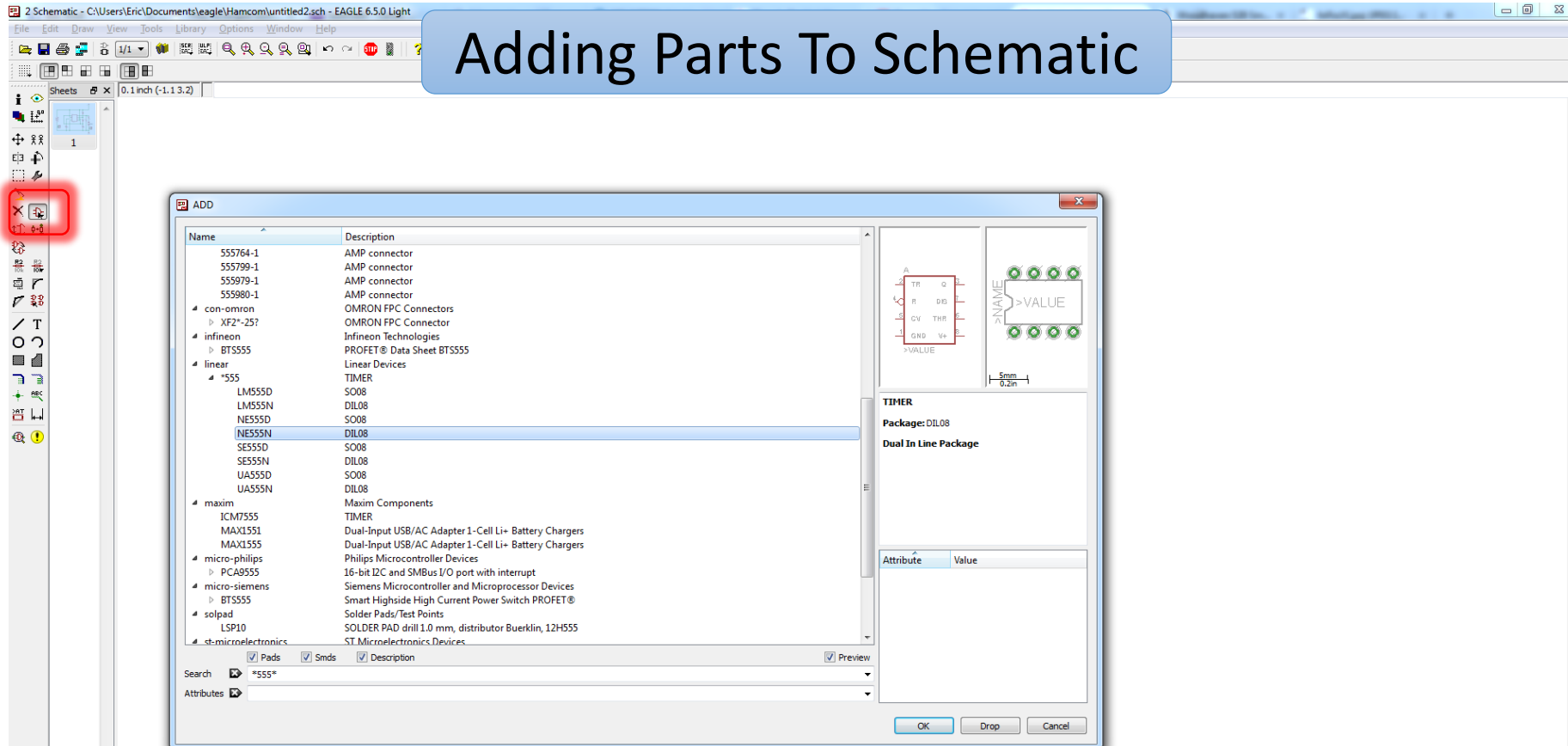
* Software suite contains a “free” version, usually limited functionality and limited terms of use (IE: Noncommercial)

EAGLE - Schematic Capture



This looks nice, but I assume you'd like to see how we got here, with less magic fairy dust.

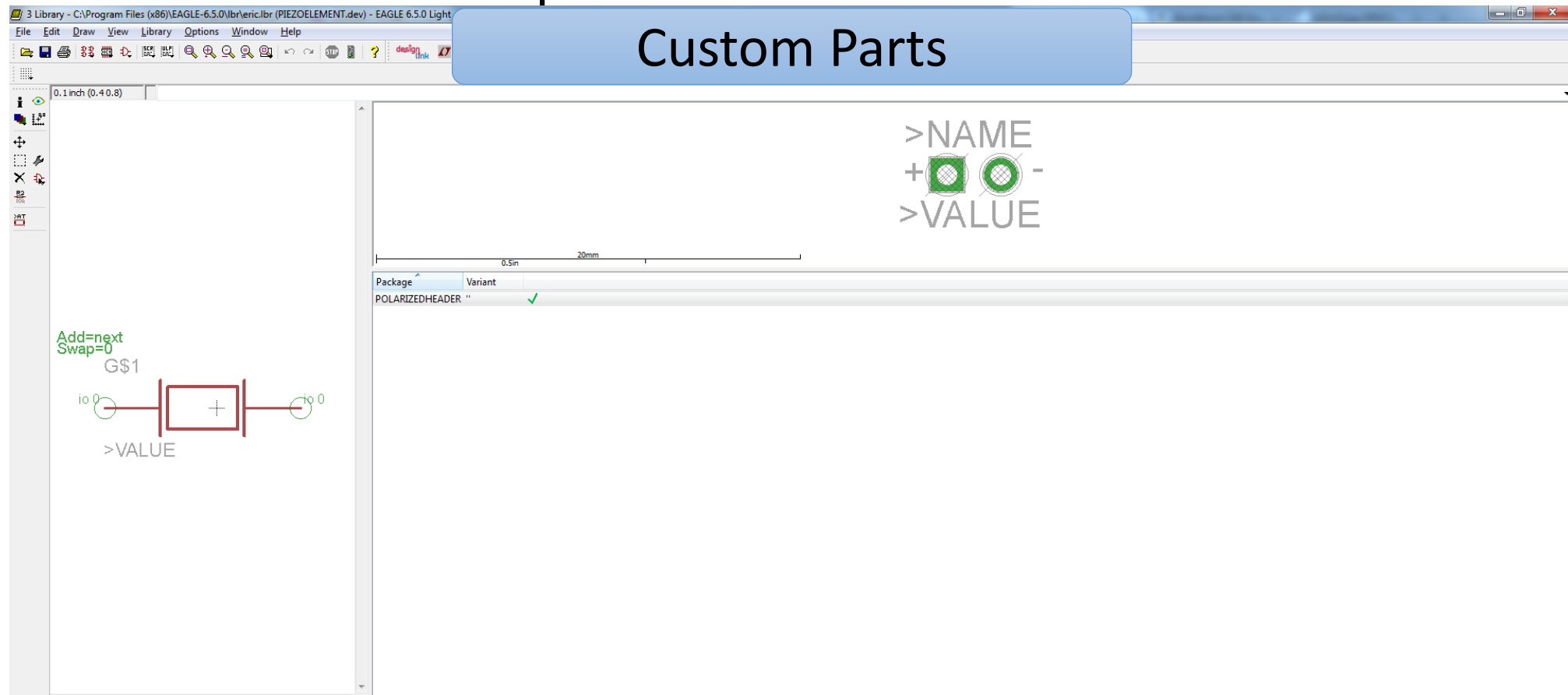
EAGLE - Schematic Capture



By clicking on the “Add” button in an Eagle schematic, you can choose from an exhaustive list of components in organized libraries. These are parts that are a mix of releases from CadSoft, actual vendors of the parts (IE: Microchip and Analog Devices), and also the user community.

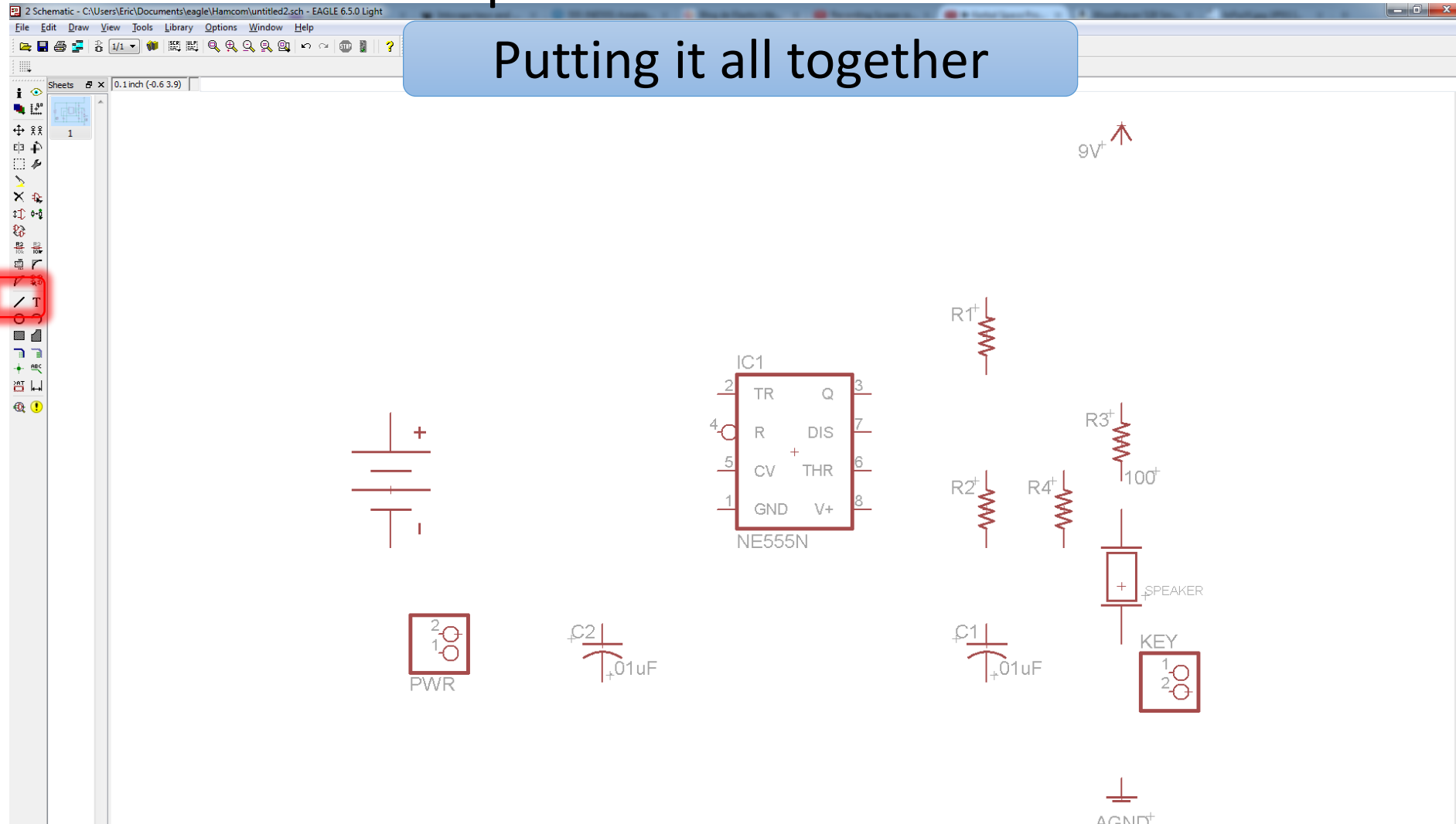
Also included are common “componentless” pieces like symbols for voltage supply and ground. You merely have to click on the part you want to add and click “OK” to drop it on the schematic.

EAGLE - Schematic Capture



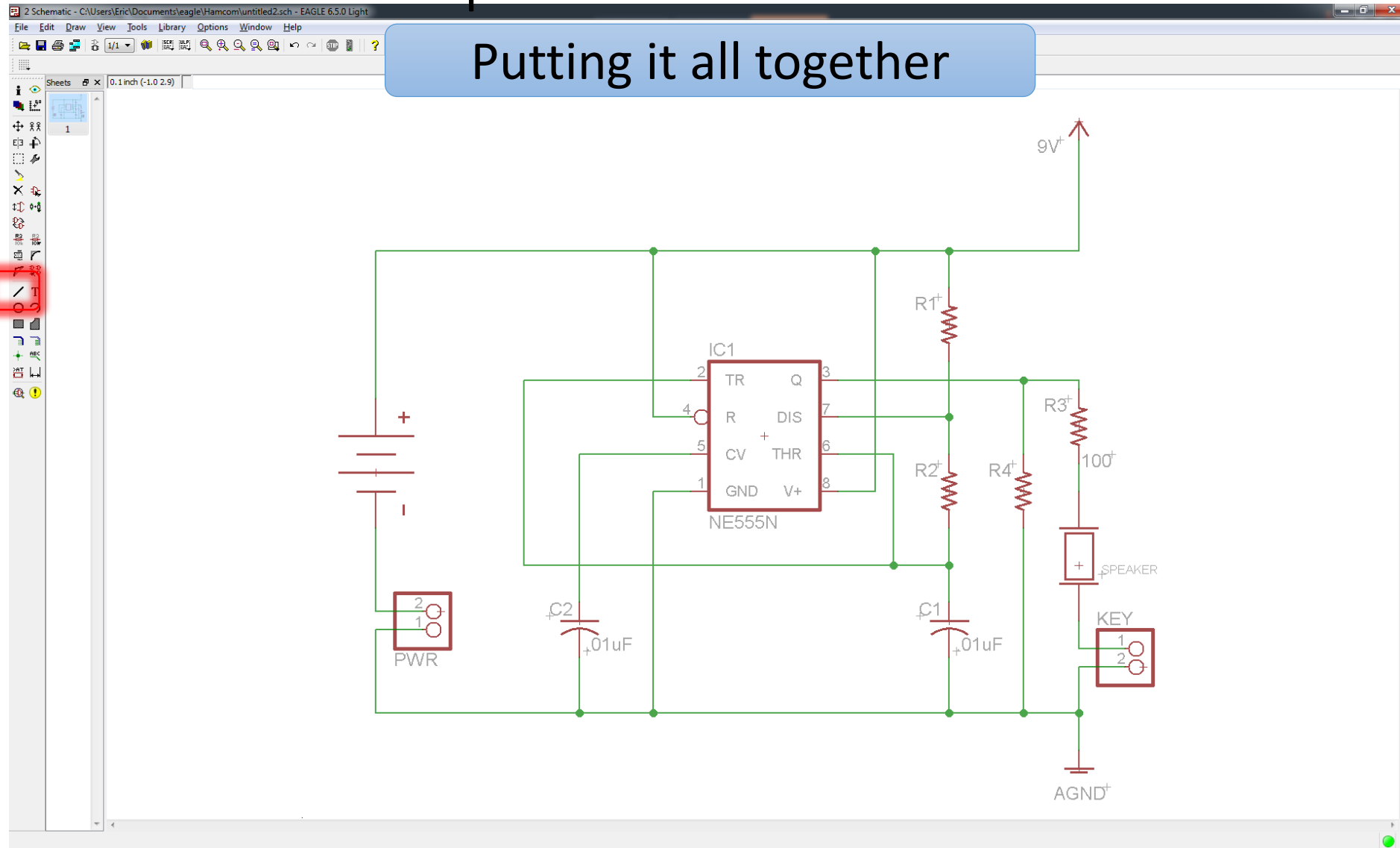
If you don't have an existing footprint in EAGLE, feel free to create your own and place it in the relevant library. This is done by drawing a schematic symbol with "pins", a footprint with "pads" and linking them together in a device that routes the correct pin to the pad. Initially, you will experience MUCH frustration with this, but hang in there, it gets easier.
.....maybe.

EAGLE - Schematic Capture



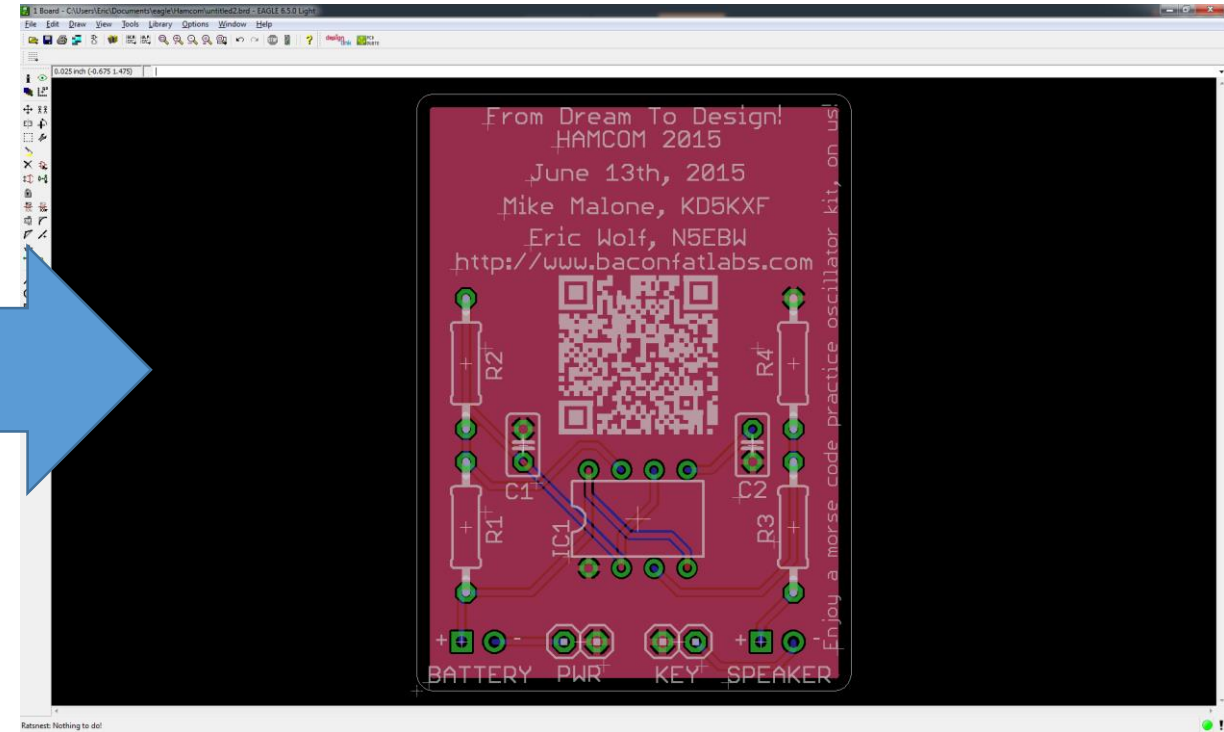
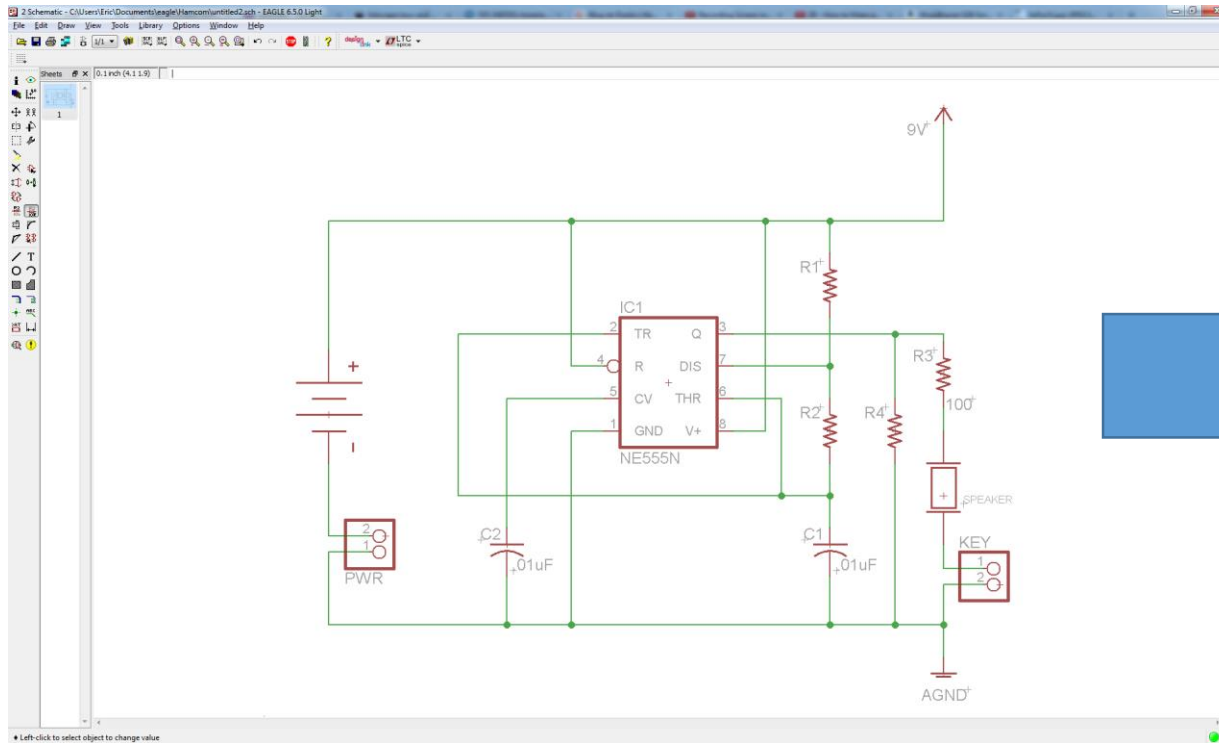
Once you have components placed where you want them (likely using the “rotate” and “move” functions on the menu, you can utilize the “wire” tool to connect them together, forming “Nets”. It’s as simple as point and click, but you can get more complex by naming them, and even routing them through the “Bus” tool. (I’ve only used that once)

EAGLE - Schematic Capture



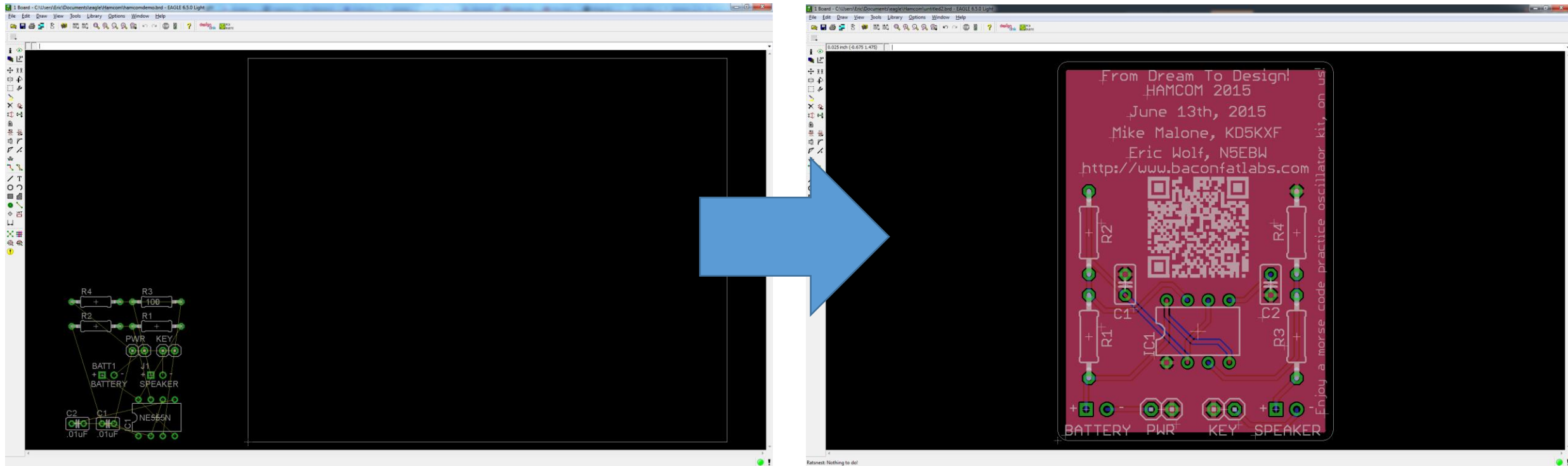
Now we're getting somewhere!

EAGLE – Board Layout from Schematic



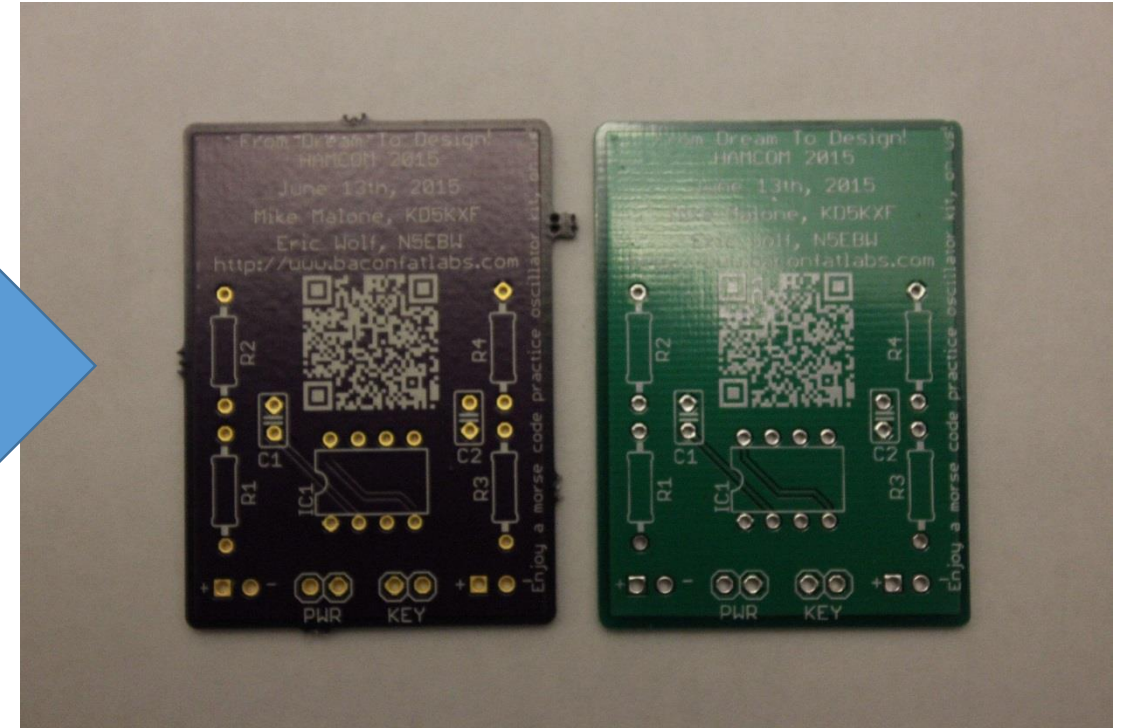
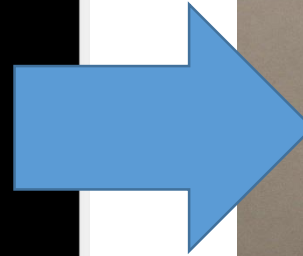
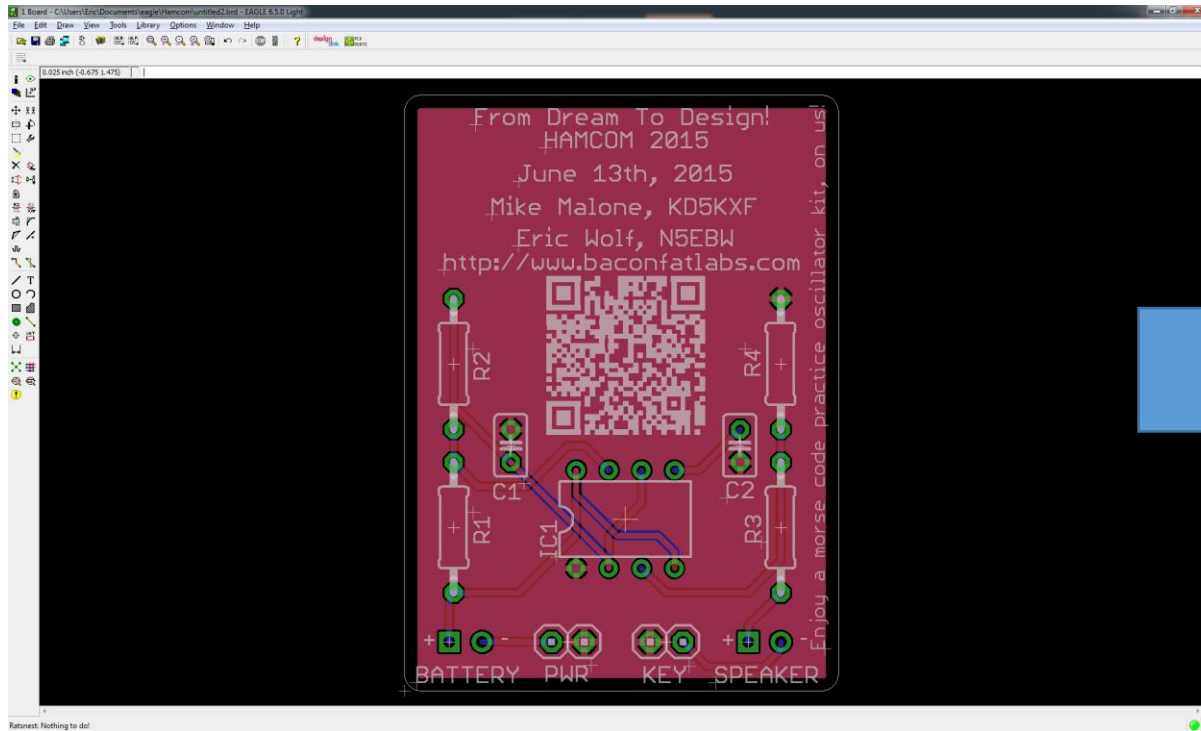
The board layout module in EAGLE is a virtual view of all layers, routed traces, component location, dimensions, silkscreen, labels and MANY other things. Of course, it doesn't come out looking that pretty right off the bat. There's a few things you have to do. Like, everything.

EAGLE – Board Layout from Schematic



In fact, this is much more representative of the transformation. An entire day could be taken showing you the ins and outs of board layout in EAGLE. Unfortunately, we only have 50 minutes. Maybe next year 😊

EAGLE – Physical Circuit Boards from Layout



Now that we have a virtual layout of what we think our circuit board should look like, let's get them produced!

Obtaining a prototype

Prototyping Services

ExpressPCB

Gold Phoenix

OSHPark

Screaming
Circuits

Just like the vast amount of choices available with EDA software, there are literally hundreds of circuit board prototyping services available, and these are only a few of the more commonly known ones. Each have very different pricing models, design rules, and accepted design files.

Although all of the above have a certain “following” in the hobby circuit board realm, the clear favorite as of this presentation is OSHPark, which is also what I use for my designs.

Obtaining a prototype



OSHPark

OSHPark grew out of a small prototyping service called LaenPCB, which was started by James Neal (Laen is Neal spelled backwards) to service the prototyping needs of the DorkbotPDX robotics group. The service has grown by several orders of magnitude. OSHPark also absorbed BatchPCB when they parted ways with Sparkfun.

Fixed cost - \$5.00 dollars per square inch for three copies of a circuit board, with free shipping.

Quick turnaround – usually around 2 weeks from submission of design until delivery.

User interface- Intuitive panel makes managing and ordering designs easy.

Multiple file types for board layouts accepted.


You can have it in any color you want, as long as it's purple with gold traces.

Obtaining a prototype


OSHPark

OSH Park

PCB Order - Upload your design



You can upload your design as an Eagle board file, or a ZIP file containing Gerber CAM files. See our [design submission guidelines](#) for more information.

 Processing file...

Great, while we're waiting for that to upload and process why don't you give us some additional information about your project? This is useful if you decide to share your project, or to help you remember things about your orders.

Project Name


Description

Email

Designed and developed by [Resistor](#)

OSH Park

PCB Order - Upload your design



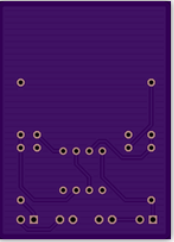
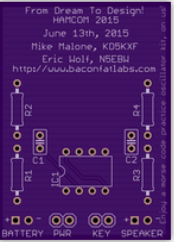
Detected 2 layer board of 1.33x1.82 inches (34x46mm). \$12.05 for three.

Your upload has finished processing. Enter the project details below and we'll move on to checking all the individual layers to make sure that they're correct.

Project Name

Description

Email



Designed and developed by [Resistor](#)

The only thing you have to do is upload a Eagle board file (file with extension .brd in the directory where your project exists), or if you prefer, gerber files in a .zip archive from whatever layout tool you are using. The web ordering panel creates a 2D front and back rendering of what your circuit boards will look like. When you hit "Continue" you will be prompted to order and pay via Paypal. Three prototype boards for \$12.05 isn't bad at all!

Look what came in the mail!



Our prototype run of three boards arrived in the mail. OSHPark always sends their boards in purple bubble mailers. Here you see one of the raw boards already populated with components.

Gerber what?

Eagle

File Extension	Description
.cmp	Top Copper
.sol	Bottom Copper
.stc	Top Soldermask
.sts	Bottom Soldermask
.plc	Top Silkscreen
.pls	Bottom Silkscreen
.drc	NC Drill

OrCad

File Extension	Description
.top	Top Copper
.bot	Bottom Copper
.smt	Top Soldermask
.smb	Bottom Soldermask
.sst	Top Silkscreen
.ssb	Bottom Silkscreen
thruhole.tap	NC Drill

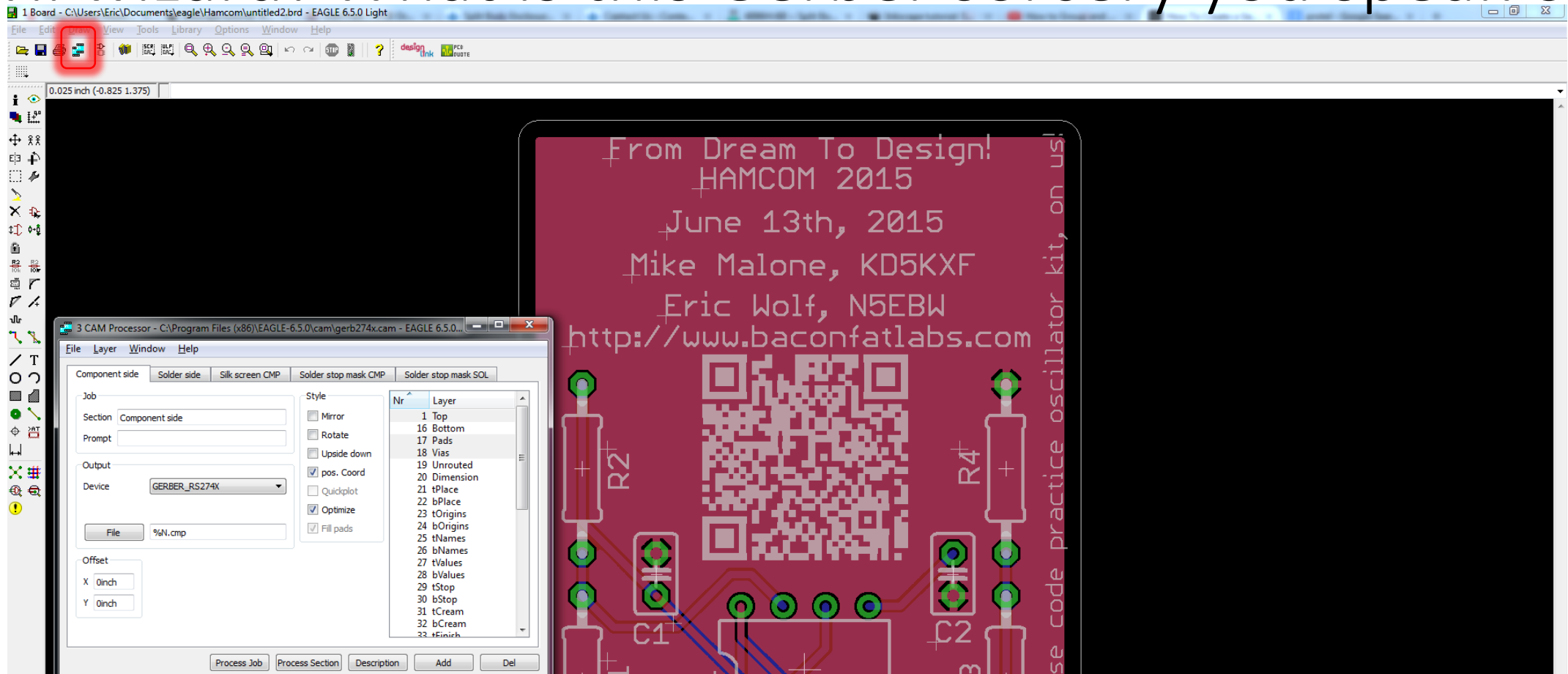
Altium

File Extension	Description
.gtl	Top Copper
.gbl	Bottom Copper
.gts	Top Soldermask
.gbs	Bottom Soldermask
.gto	Top Silkscreen
.gbo	Bottom Silkscreen
.drl	NC Drill

A few examples of file format differences between EDA suites. Isn't that special?!?

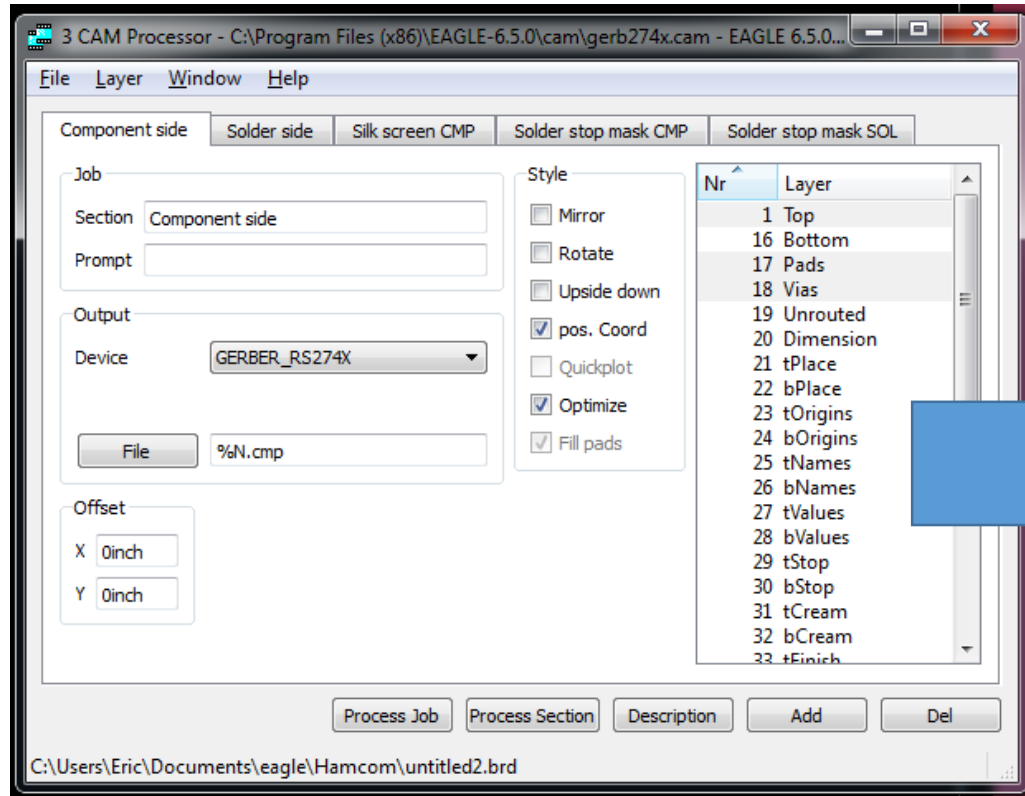
The Gerber format is an open 2 dimensional binary vector image file format. It is the universal industry standard method to describe printed circuit board layouts using several files. Many circuit board manufacturers are starting to accept EAGLE .brd layout files, but that is far from standard and only Gerber format files are universally acknowledged. The extension of a file in a Gerber archive denotes exactly what that file does, but these can be vendor specific. The only real way to know is to have special Gerber viewing software analyze it.

You evil wizard! What is this Gerber sorcery you speak of?



Every piece of EDA software either allows you to save your layout as Gerbers, or export Gerber files. More and more, things are going towards exporting them in favor of the application retaining a (usually) proprietary file format for use. In Eagle, you can generate Gerber files by using the “CAM” (Computer Aided Machining) processor. There are two jobs that usually do the trick.... “gerb274x.cam” for 2 layer boards and “gerb274x-4layer.cam” for 4 layers. These are included with the software, and if you have to tweak the output, modification is not hard.

Generated Gerber Files



Documents library

gerbers

Name	Date modified	Type	Size
untitled2.cmp	6/8/2015 9:25 PM	CMP File	24 KB
untitled2.crc	6/8/2015 9:25 PM	CRC File	1 KB
untitled2.crs	6/8/2015 9:25 PM	CRS File	1 KB
untitled2.gpi	6/8/2015 9:25 PM	GPI File	1 KB
untitled2.l15	6/8/2015 9:25 PM	L15 File	1 KB
untitled2.ly2	6/8/2015 9:25 PM	LY2 File	1 KB
untitled2.plc	6/8/2015 9:25 PM	PLC File	65 KB
untitled2.pls	6/8/2015 9:25 PM	PLS File	2 KB
untitled2.sol	6/8/2015 9:25 PM	SOL File	28 KB
untitled2.stc	6/8/2015 9:25 PM	STC File	1 KB
untitled2.sts	6/8/2015 9:25 PM	STS File	1 KB

With a few clicks of the mouse, the Gerber files are generated to our computer filesystem. From here, it is manufacturer dependent as to how they receive them. Some prefer you attach them individually while others would like for you to attach them all as one zip archive.

Production!

Production Circuit Board Manufacturers

ExpressPCB

Olimex

PCBCart

PCBExpress

Screaming
Circuits

Sierra
Circuits

Just as the case with prototypers (of which you'll notice some in this list, too), each production board manufacturer has different requirements and design rules. Examples of this might be a minimum hole size, minimum trace size, annular ring, and minimum overall size.

PCBCart is the manufacturer I leverage for production runs, and have been solidly for 8 years without a single complaint. I always get exactly what I ask for, even if I ask for something that's my mistake!

Production Quotation

Quotation

Please fill or select the following specification.

Standard PCB

Prototype PCB

Standard PCB mainly for the official products or repeat orders, the quality level is IPC2. Check Comparison Table to [learn more...](#)

Material: FR4

Layers: 2 layer

Material Details: Standard Tg 140C

Part Number: MCK

Board type: ☒ single unit ☐ panel

Board Size (width): 33.7 mm

Board Size (height): 46.3 mm

Quantity: 100 pcs

Thickness (Finished Board): 1.6 mm

Surface Finish: HASL -Hot Air Solder Leveling

Copper Weight (Finished): 1 oz

Min. Tracing/Spacing: 0.20 mm

Min. Annular Ring: 0.30 mm

Smallest Hole: 0.40 mm

PCB Qty:100 Unit Price:\$0.78 Tooling cost:\$36.87 Sub-Total:\$114.87

Material: FR4

Layers: 2 layer

Material Details: Standard Tg 140C

Part Number: MCK

Board type: single unit

Board Size (width): 33.7

Board Size (height): 46.3

Laser Stencil cost Qty:0 Unit Price:\$0.00 Sub-Total:\$0.00

Part Number:

Quantity:

Stencil type:

framework stencil prices:

Finished Thickness:

Send me a check plot:

Fiducials:

\$ 114.87

ADD TO CART


No, it's not uncommon to see typos on webpages of foreign origin.


Price Matirx				
Quantity / Leadtime		in 8 days		in 12 days
50		\$ 1.28		\$ 1.22
75		\$ 0.95		\$ 0.91
100		\$ 0.78		\$ 0.75
125		\$ 0.68		\$ 0.65
150		\$ 0.62		\$ 0.59
200		\$ 0.56		\$ 0.53
250		\$ 0.50		\$ 0.48
375		\$ 0.43		\$ 0.41
500		\$ 0.40		\$ 0.38
Quantity:0.00		Unit price:\$ 0	ToolingCost:\$ 0.00	Leadtime:0.00
Total:\$0.00				
Add to cart				


Production circuit board manufacturers have tools that allow you to obtain a quote, including associated set up costs (tooling costs). A price matrix is also a main component of the quote. As you can see, the more boards you order in quantity, the lower the unit cost. Also, longer lead times are cheaper. Many circuit board manufacturers can also assemble the components onto bare boards as well, but it is often cost prohibitive for the average hobbyist. Most production runs are either distributed in kit form, or self assembled prior to sale.

Production Quotation


Payment method


☒ 
Credit Card or Paypal (Recommend): accept Visa, Mastercard, AE, Discover, or by PayPal


☐ 
Western Union, Moneygram: Pay by Western Union or Moneygram


☐ 
Wire Transfer: Pay by Bank Wire Transfer

Deliver method

☐ 

☐ 

☒ 

☐ 

Part Number	Product	Tooling cost	Price	Qty	Total
MCK	FR4 - 2 layers Upload Gerber File	\$ 36.87	\$ 0.78	100	\$ 114.87

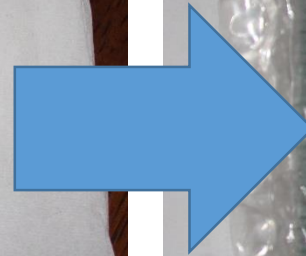
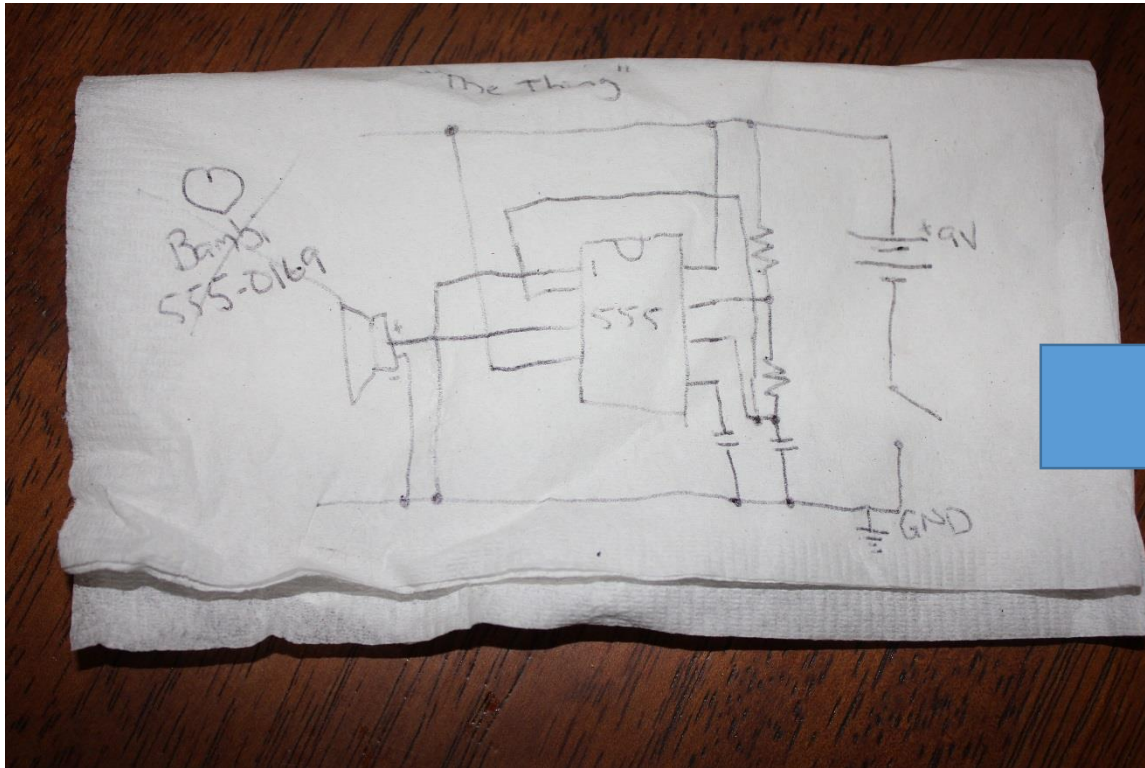
Purchase Number:

Notes:

Items subtotal: \$ 114.87
shipping cost: \$ 30.89
Total: \$ 145.76

We produced a quantity of 100 circuit boards for our project, giving us a unit cost of \$0.75. After set up fees (one time cost) and shipping are factored in, the unit cost is \$1.45 per bare board. If we reordered this, the gross cost drops to ~\$1.09.

Production In Hand!

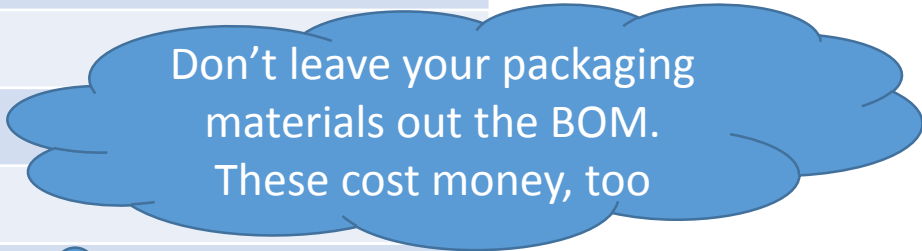


In a few weeks, a DHL package shows up at your doorstep and you now have a tangible product from your idea!

Bill Of Materials

Each board will require the following parts:

Component	Description
R1	22K Resistor
R2	22K Resistor
R3	100 ohm 1/4Watt Resistor
R4	Resistor (Not used, no procurement needed)
J1	2 position header pin, .10" pitch, power switch
JMP	2 position power switch jumper
SPK	Speaker. Utilizing Piezo element to save costs
BATT	9V battery snap
IC1	555 Timer Integrated Circuit
C1	.01uF capacitor
C2	.01uF capacitor
BAG	Antistatic bag
LABEL	Peel and Stick Label



Don't leave your packaging materials out the BOM. These cost money, too

Sourcing Parts

Digikey

Mouser

Newark

Futurlec

Ebay*

My main source of components is generally Mouser Electronics. I like them because they are based out of Mansfield. You have to pay state tax to pretty much any online electronics megasupply now, so it might as well be Texas State Tax. Also, if you select USPS Priority Mail, your parts arrive the next day for the 3 day price!

Mouser doesn't have every single thing so occasionally I will look to an alternate vendor for something I really need. That is generally Digikey, or if I can get it cheaply and don't mind the wait, I will use Futurlec. I have had a bad experience with Futurlec before where they kept pushing back my order ship date after neglecting to tell me the item was out of stock despite being in stock on their website, but mainly they are just EXTREMELY slow.

*I listed Ebay as a supply source. This is fine if you're just doing one run of something and you can get them all at the same time, but I don't recommend it if you plan on leveraging a design repeatedly due to the temporary nature of auctions and sellers. **ALSO: BE VERY CAREFUL OF KNOCKOFF PARTS!!!**

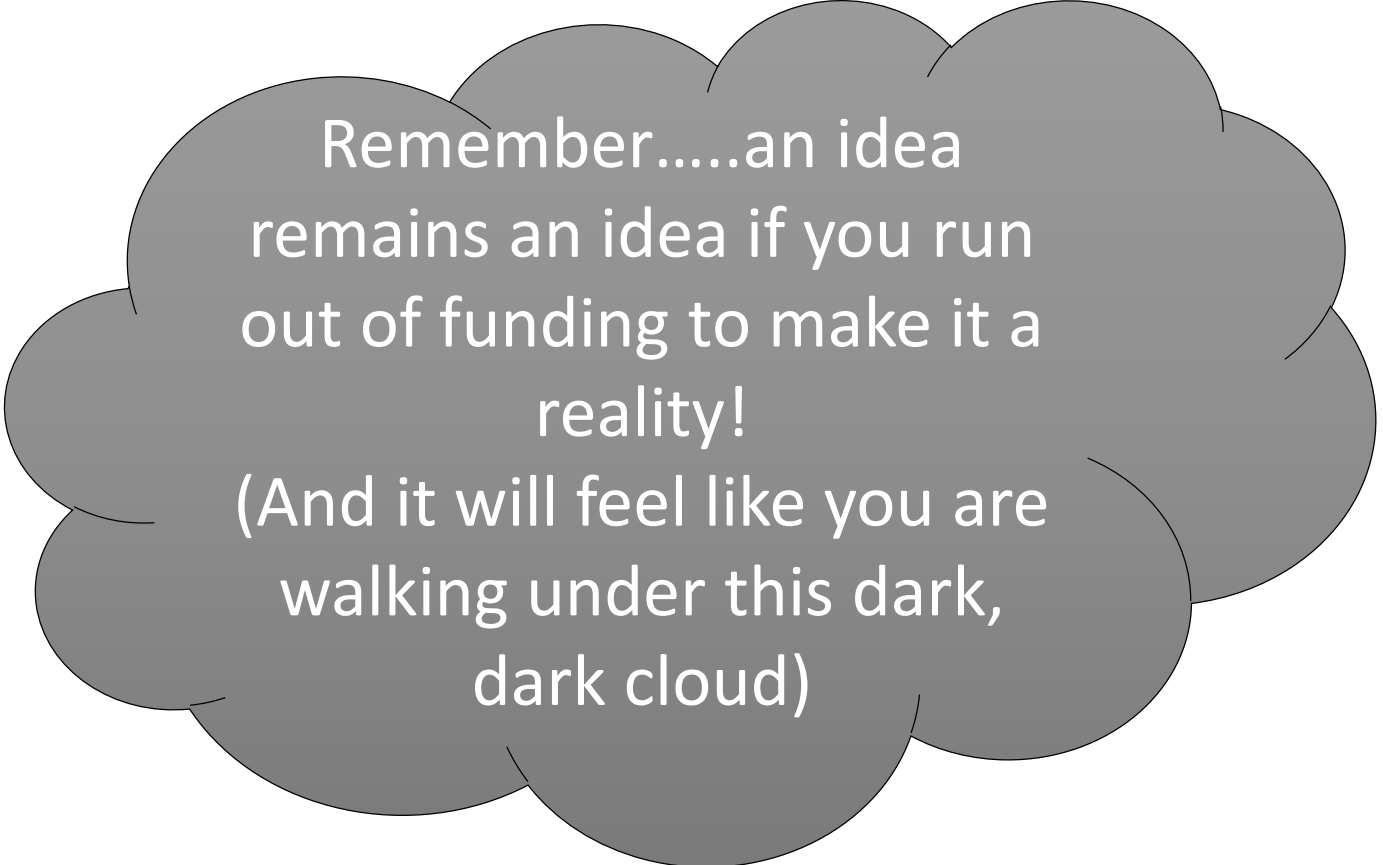
Sourcing Parts



Since this was a limited run with a very low price point achievement, our kit parts came from a mixture of Mouser and Ebay.

Last but not least, financial considerations....

A project of even this magnitude can quickly spiral out of control if you aren't watching your cash flow. Before ordering anything, take a little while to place all associated costs within a spreadsheet, or piece of paper, even the bathroom mirror with lipstick if that's your thing. Whatever works for you, but try to anticipate any costs with your product, ESPECIALLY if profit is a consideration!



Remember....an idea
remains an idea if you run
out of funding to make it a
reality!
(And it will feel like you are
walking under this dark,
dark cloud)

By the way, those kits we made? They're for you. Please obtain one on your way out and enjoy assembly as our way of saying thank you!



Our presentations as well as kit assembly instructions can be downloaded by scanning the QR code on your circuit board, and at the following URL:

<http://www.baconfatlabs.com/downloads/presentations/hamcom2015.zip>

THANK YOU!

Questions?