

Designing Your Own PC Board

Thomas C. Wilson, Jr. • thomas.wilson@stonybrook.edu
Instrument Laboratory • School of Marine and Atmospheric Sciences
Stony Brook University

RVTEC 2020
Zoomland – ver 1.3



Stony Brook University
*School of Marine and
Atmospheric Sciences*



Standard Disclaimers

Unless otherwise noted:

- Mention of a company in this presentation does not constitute an official endorsement by the State of New York, the State University of New York, or the School of Marine and Atmospheric Sciences.
- The presenter has no ownership interest in any commercial entity mentioned.

Special Open Ocean Rated “No Chop-Busting” Disclaimers

- The presenter has never dated anyone connected to any mentioned company, nor is this ever likely. Neither have those folks plied him with treats, trinkets, or fancy food and drink.

Anyone who implies otherwise is asking for trouble.

SAFETY, SAFETY, SAFETY

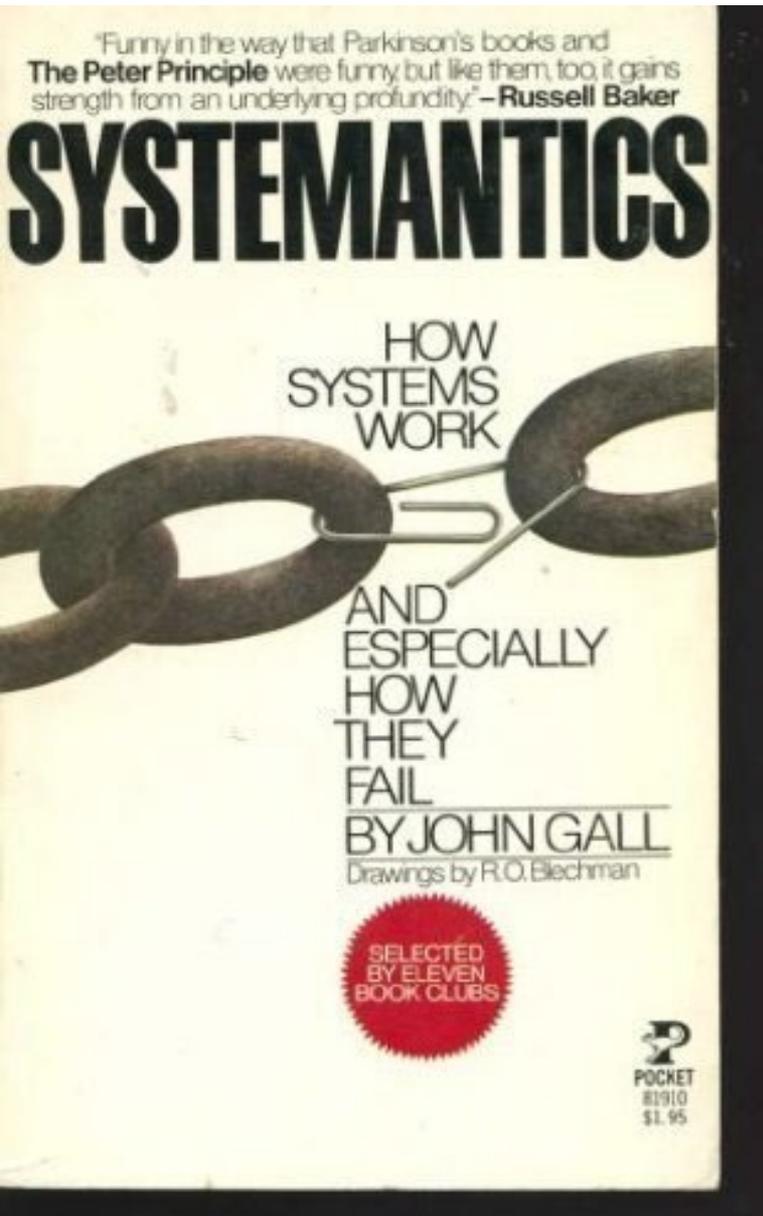
TEST all designs through as wide a range of input conditions as possible.

USE CAUTION with high voltage and high power.

CONSIDER redundant dumb limit controls and other fail safe features.

NOT for human safety applications.

GALL'S LAW



“A complex system that works is invariably found to have evolved from a simple system that worked.”

-John Gall (1925-2015)
Pediatrician and author of “Systemantics” (1975)

Large and complex electronic systems are combinations of smaller and less complex electronic systems.

This has many parallels to programming. Well designed electronic subsystems are like programming subroutines, with defined inputs and outputs (and hopefully sanity checks and safety limits).

Electronic troubleshooting is like code debugging: work down from the total system to locate the specific subsystem that is malfunctioning.

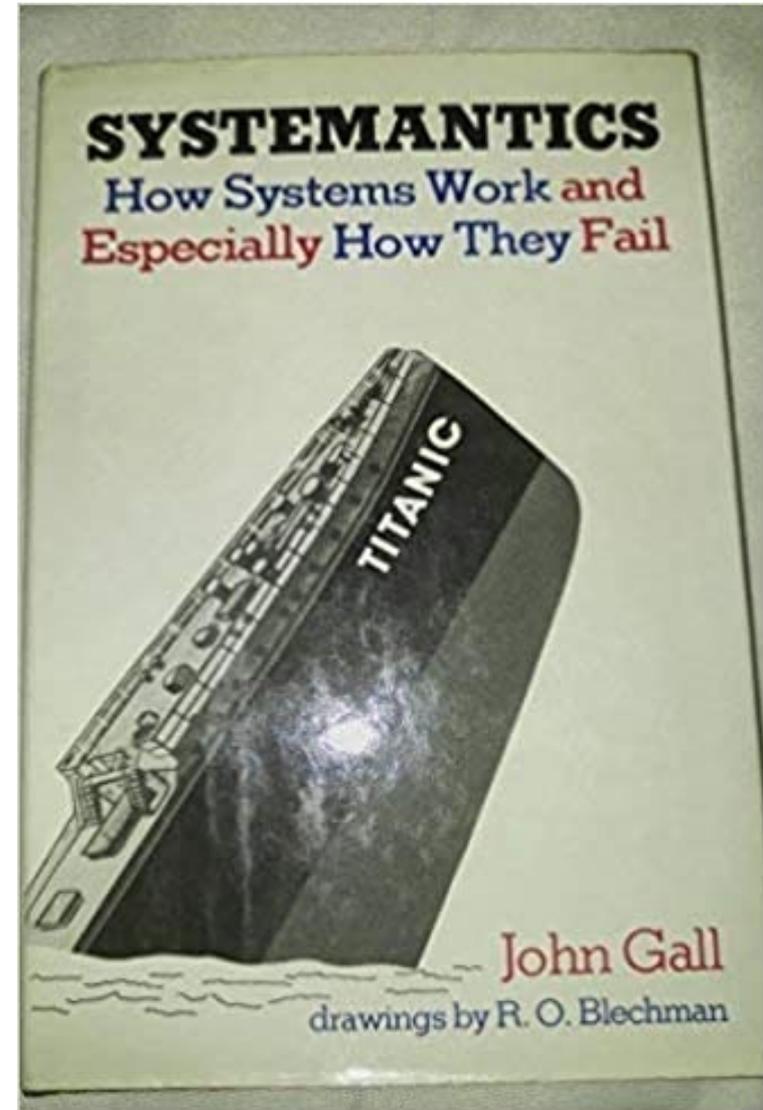
Electronic design is like code writing: design simple, robust, tested subsystems, then combine them to achieve a large and complex result.

**If you can learn to write good programs,
You can learn to design electronics!**

GALL'S LAW (part 2)

15:00

“A complex system designed from scratch never works and cannot be patched up to make it work. You have to start over, beginning with a working simple system.”



Step 1: Make a Design

Getting Started in Electronic Design

Thomas C. Wilson, Jr. • thomas.wilson@stonybrook.edu
Instrument Laboratory • School of Marine and Atmospheric Sciences
Stony Brook University

INMARTEC 2018
Woods Hole MA – v1.2



Stony Brook University
School of Marine and
Atmospheric Sciences



“THIS IS IT - THE TIME TO FINISH YOUR NOVEL”

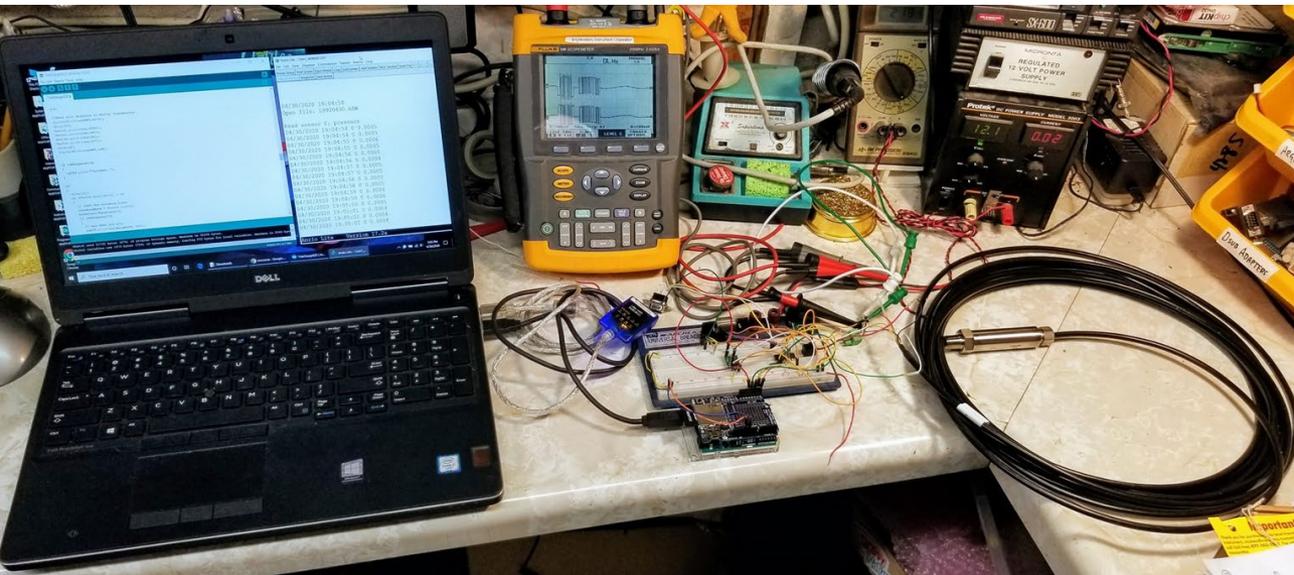
Caption of *The New Yorker* Daily Cartoon, April 29, 2020 by Hilary Allison

Tom’s Pandemic Novel “The Really Useful Datalogger Board”

High Level Specifications

- Arduino based (no single point of failure in the supply chain).
- Real time clock.
- Temperature and humidity sensor to monitor enclosure.
- SD card data storage.
- Wide range input voltage.
- Battery check circuit.
- Operable from solar power.
- Multiple RS232 interfaces.
- RS485 interface with switched sensor power.
- Switched power channels for external equipment (e.g. telemetry).
- 100% Parts I could actually get during a pandemic.
- Completed on budget and in time for inclusion in 5 new environmental stations deploying in summer 2020.

April 2020 in my basement workshop - aka "The Dungeon"

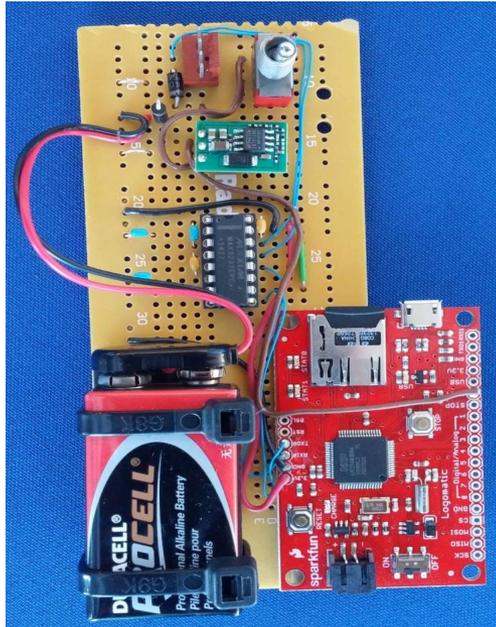


**Protoboard
Proof of Concept**

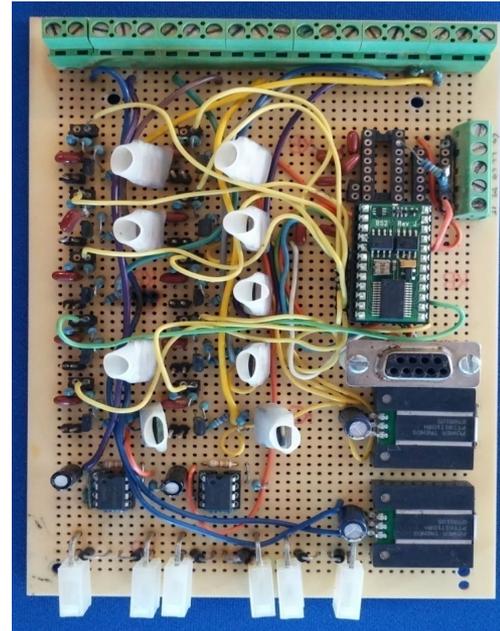
10:00

Option 2a - Hand Wiring

GPS logger



Met buoy power control



Do not disrespect hand wiring.

It's not a prototype, it's a one-off!

But the problem comes when you need a second "one-off" (field damage), and then a third, and so on.

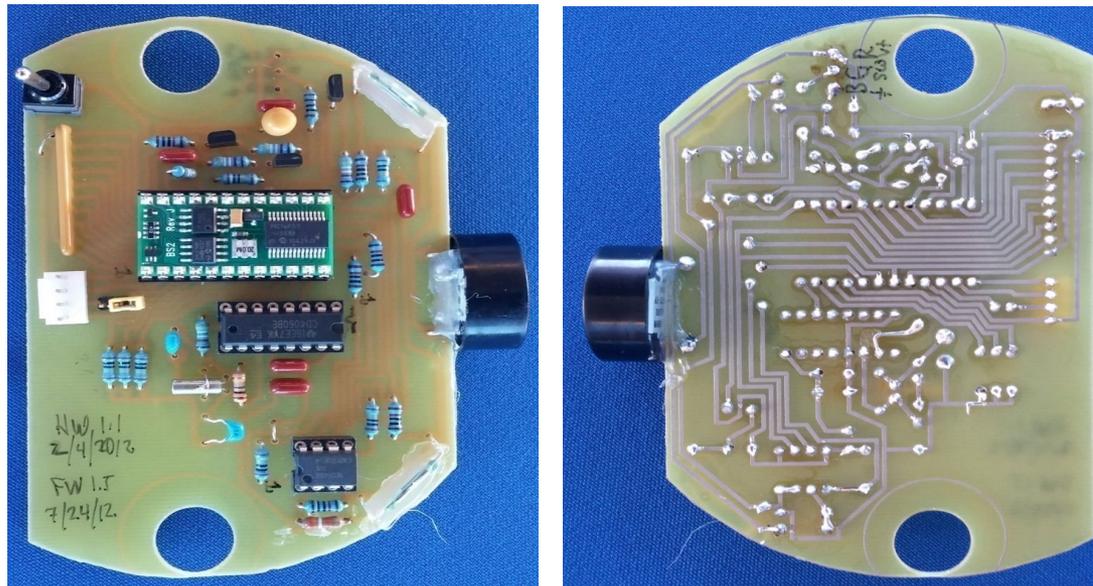
In my experience, the breakeven point for designing a PCB versus hand wiring is somewhere between 2 and 3 boards.

Option 2b - In-house PC Boards

“Desktop PCB Fabrication” – RVTEC 2004

<https://www.unols.org/sites/default/files/200411rvtap37.pdf>

Challenger deep sea pump replacement timer board



In-house PCBs time has passed.

Double sided boards are a pain.

It takes chemicals, time, and practice to develop the skill set.

You also need to work at odd hours so your spouse/significant other doesn't see what you are doing in the kitchen sink.

Option 2c - Commercial Board House

Many manufacturers now offer online ordering, low minimums, and quick turnaround.

Multilayer boards, plated through holes, solder mask, and silk screen.



Today's lesson: 2 layer bare boards, through hole components (mostly).
SMT and assembled boards left as an "exercise for the student."

Electronics Design Automation (EDA) Software

Schematic Capture

With the schematic editor you can create your design without limit; there are no paywalls to unlock features. An official library for schematic symbols and a built-in schematic symbol editor help you get started quickly with your designs.

[LEARN MORE](#)

PCB Layout

Make professional PCB layouts with up to 32 copper layers. KiCad now has a push and shove router which is capable of routing differential pairs and interactively tuning trace lengths.

[LEARN MORE](#)

3D Viewer

KiCad includes a 3D viewer which you can use to inspect your design in an interactive canvas. You can rotate and pan around to inspect details that are difficult to inspect on a 2D view. Multiple rendering options allow you to modify the aesthetic appearance of the board or to hide and show features for easier inspection.

[LEARN MORE](#)

Latest Blog Posts

- Development Highlight: PCBNew Layers, DRC and Toolbar Palettes
Sun, Oct 4, 2020
- KiCad 5.1.7 Release
Wed, Sep 30, 2020
- Development Highlight: 3D Viewer Improvements
Sat, Sep 26, 2020
- Development Highlight: CADSTAR PCB Importer
Tue, Sep 22, 2020
- KiCad and the Open Ventilator Project
Wed, May 20, 2020

KiCAD: kicad-pcb.org

Cross platform, open source, community supported, FREE.

EAGLE: from AutoDesk.com

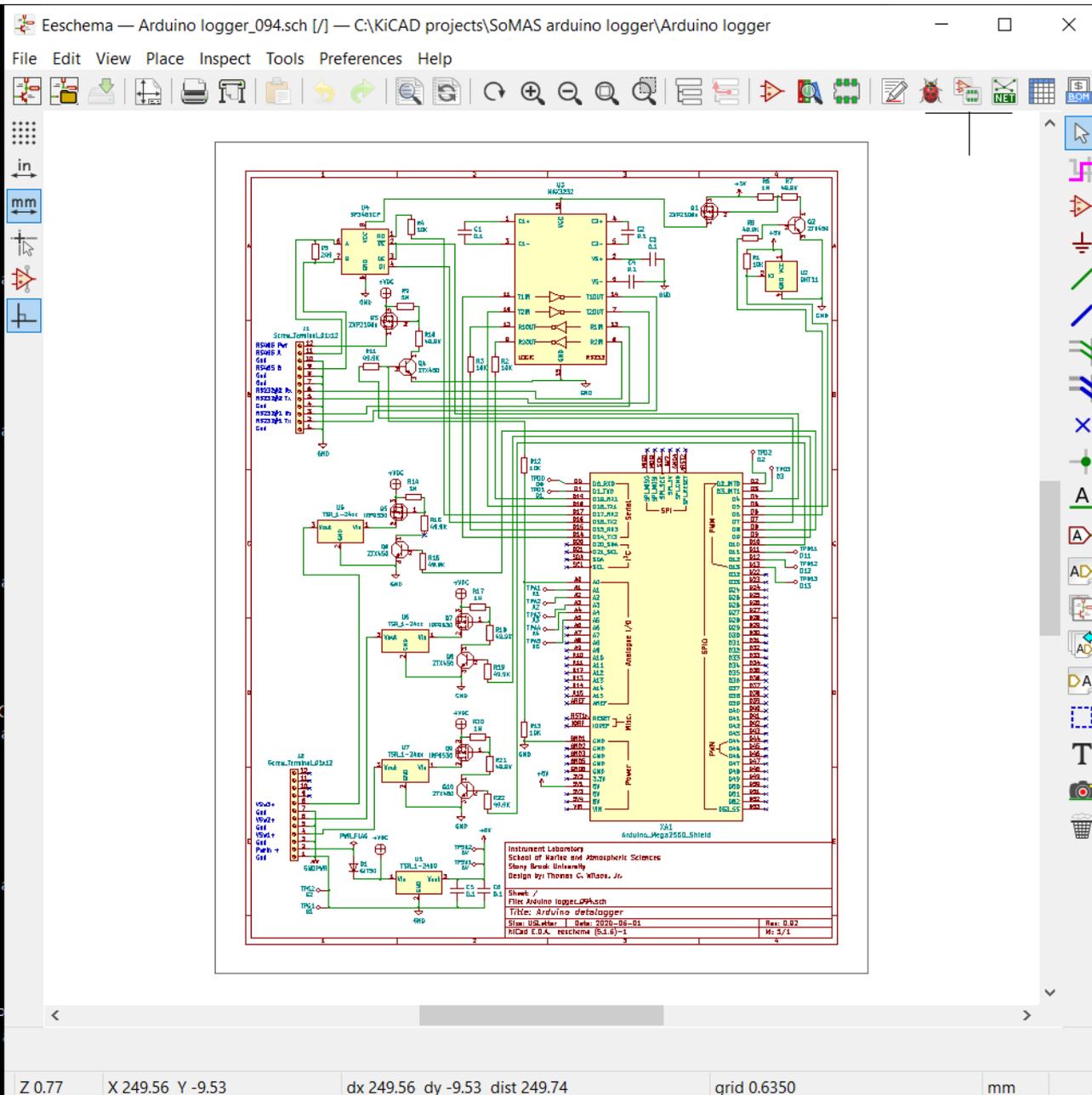
Commercial standard

Free version: 2 layers, 12 in² board

Full version: \$60/month, \$495/yr.

Beware of "free PCB design" software offered by a PCB vendor - it usually traps your files with that vendor.

Schematic Layout

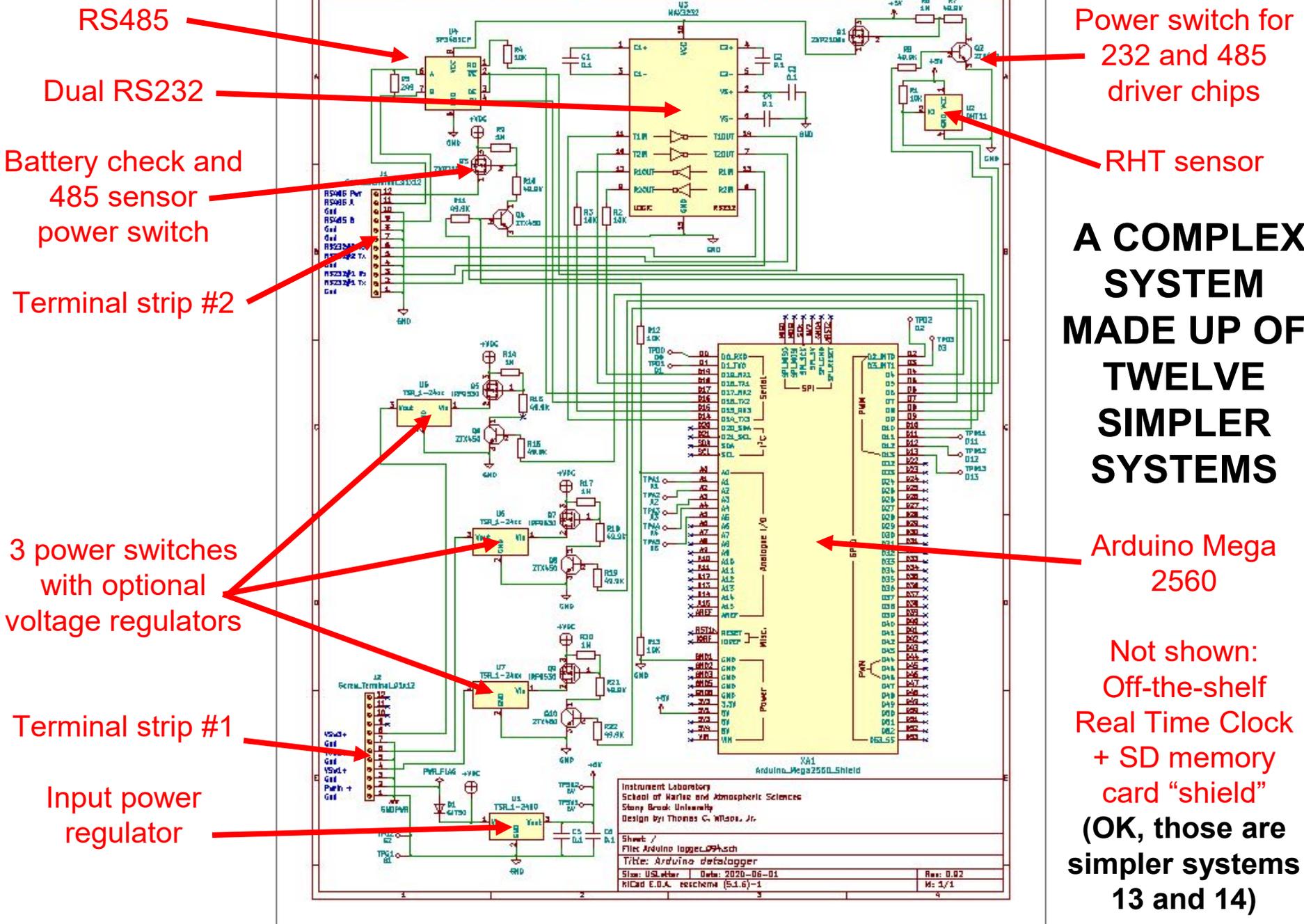


Guaranteed to impress your supervisor!

(presuming they said you could work on it)

Does it look a little scary? Remember Gall's Law!

15:00



RS485

Dual RS232

Battery check and
485 sensor
power switch

Terminal strip #2

3 power switches
with optional
voltage regulators

Terminal strip #1

Input power
regulator

Power switch for
232 and 485
driver chips

RHT sensor

**A COMPLEX
SYSTEM
MADE UP OF
TWELVE
SIMPLER
SYSTEMS**

Arduino Mega
2560

Not shown:
Off-the-shelf
Real Time Clock
+ SD memory
card "shield"
(OK, those are
simpler systems
13 and 14)

Instrument Laboratory
School of Marine and Atmospheric Sciences
Stony Brook University
Design by Thomas C. Wilson, Jr.
Sheet /
File: Arduino_logger_094.sch
Title: Arduino_defatlogger
Size: 55.5kbit Date: 2020-06-01 Rev: 0.02
KiCad E.D.A. reschema (5.1.6)-1 W: 3/1

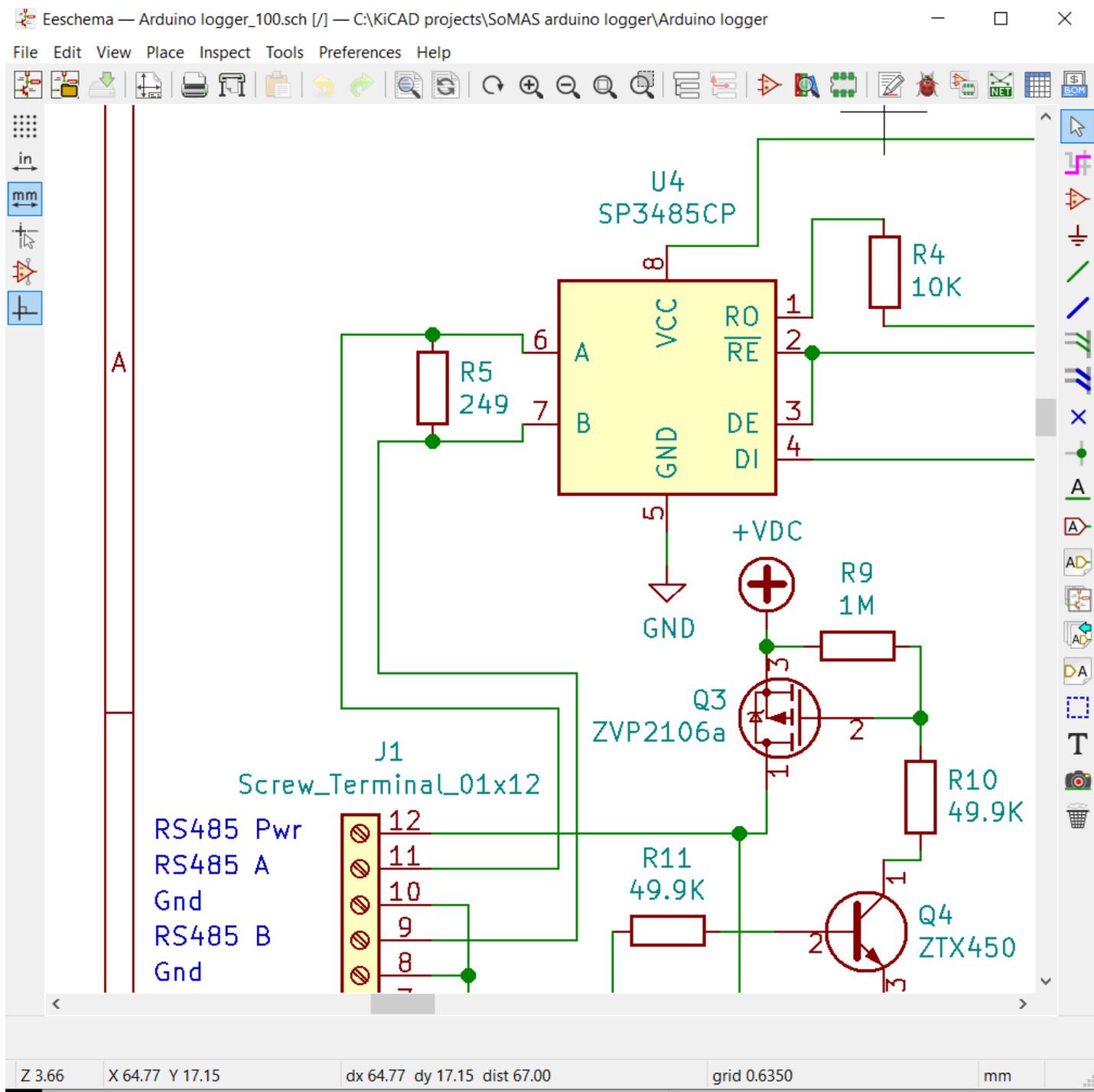
Each component includes

Designator (U4, Q3, R9, R10)

Identification or value (SP3485CP, ZVP2106a, 1M, 49.9K)

If appropriate, pin definitions (Vcc, GND)

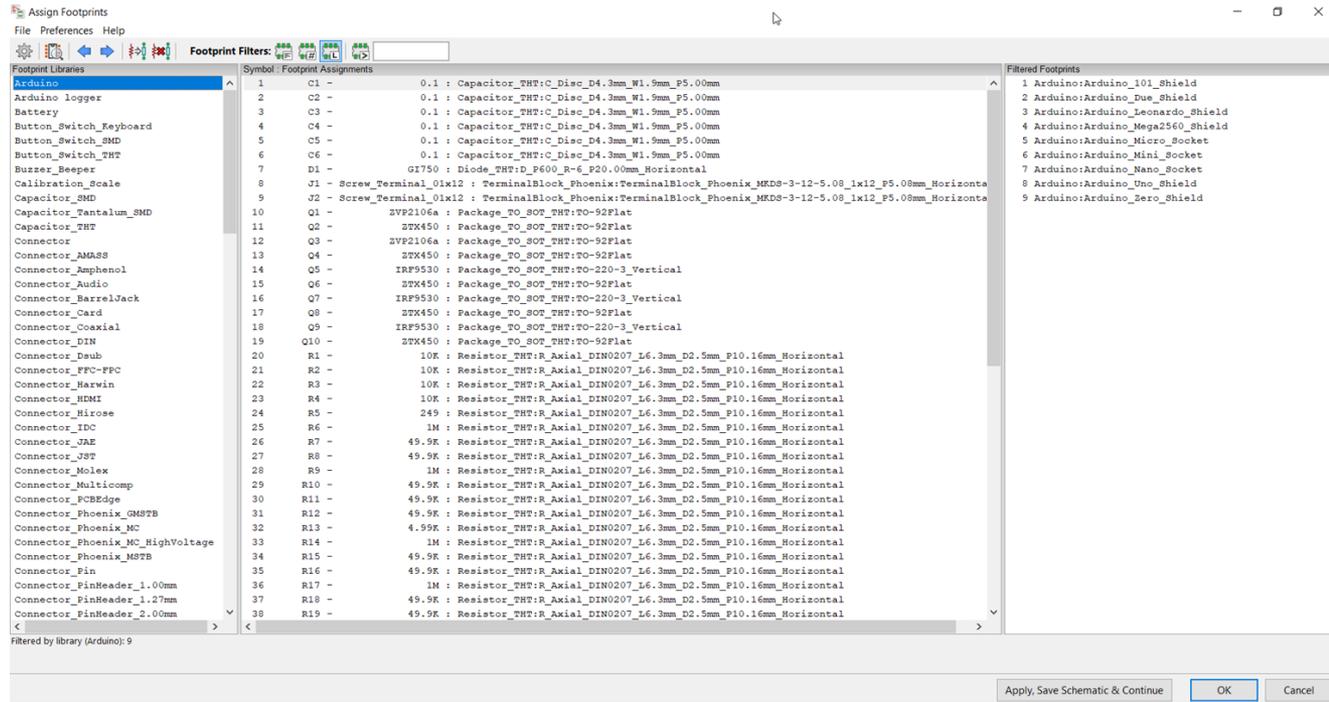
Select components from the standard libraries or define your own



Assign Footprints

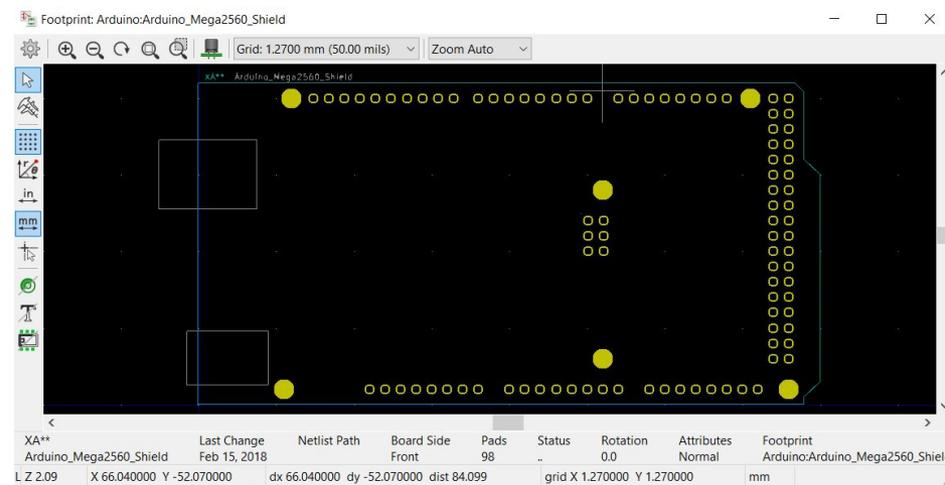
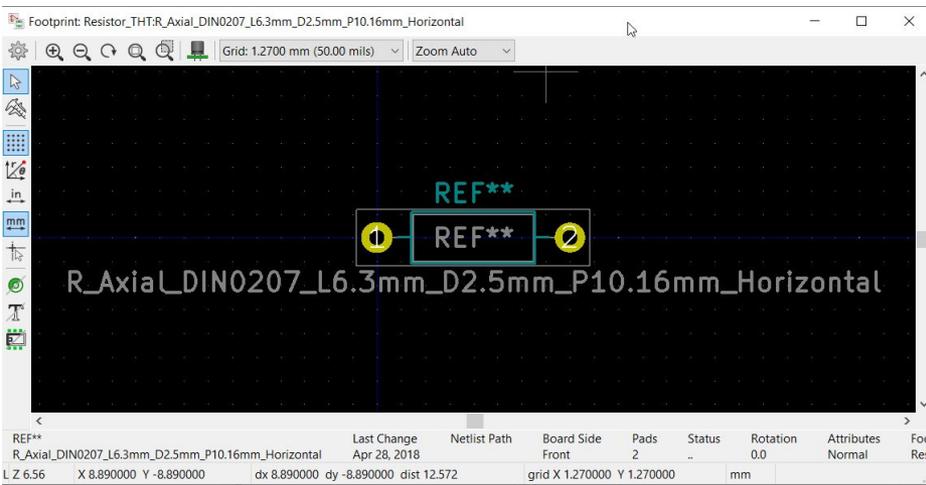
Is that 10K resistor
 $\frac{1}{4}$ watt or 1 watt,
surface mount or through
hole?

Assign footprints for each
component from the
standard libraries
or define your own.
Footprints,
like components, can be
simple or complex.



$\frac{1}{4}$ watt axial resistor, through hole, horizontal

Arduino Mega 2560 shield



Generate the Netlist

- The Netlist is a file listing all components, the associated physical footprints, and all interconnections.
- Export the Netlist, save your schematic (you have been doing that all along, haven't you?), and exit the schematic editor with a feeling of accomplishment.

Import the Netlist

- Start the PCBoard editor.
- Take a deep breath, intone “Fear is the mind killer”, and
- **IMPORT THE NETLIST.**

Welcome to the Rat Nest

Pcbnew — C:\KiCAD projects\SoMAS arduino logger\Arduino logger\Arduino logger_100_test.kicad_pcb

File Edit View Place Route Inspect Tools Preferences Help

Track: 9.84 mils (0.250 mm) * Via: 31.5 / 15.7 mils (0.80 / 0.40 mm) * Grid: 10.00 mils (0.2540 mm) Zoom Auto

Layers Manager

Layers	Items
<input checked="" type="checkbox"/>	F.Cu
<input checked="" type="checkbox"/>	B.Cu
<input checked="" type="checkbox"/>	F.Adhes
<input checked="" type="checkbox"/>	B.Adhes
<input checked="" type="checkbox"/>	F.Paste
<input checked="" type="checkbox"/>	B.Paste
<input checked="" type="checkbox"/>	F.SilkS
<input checked="" type="checkbox"/>	B.SilkS
<input checked="" type="checkbox"/>	F.Mask
<input checked="" type="checkbox"/>	B.Mask
<input type="checkbox"/>	Dwgs.User
<input checked="" type="checkbox"/>	Cmts.User
<input checked="" type="checkbox"/>	Eco1.User
<input checked="" type="checkbox"/>	Eco2.User
<input checked="" type="checkbox"/>	Edge.Cuts
<input type="checkbox"/>	Margin
<input type="checkbox"/>	F.CrtYd
<input type="checkbox"/>	B.CrtYd
<input checked="" type="checkbox"/>	F.Fab
<input checked="" type="checkbox"/>	B.Fab

Sheet No.	Date	Rev.
KiCad E.D.A. pcbnew (5.1.6) - 1		10:1/1

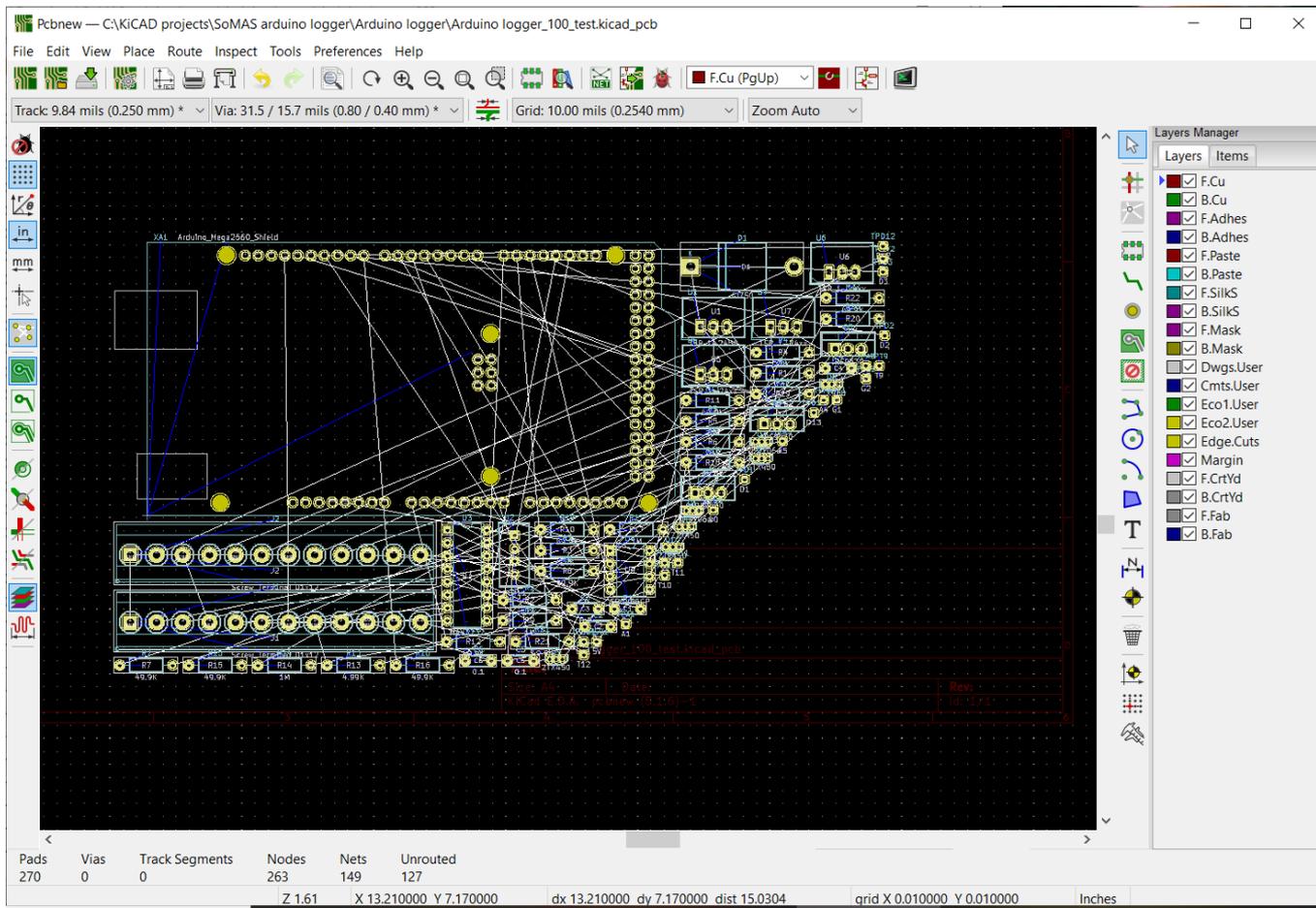
Pads	Vias	Track Segments	Nodes	Nets	Unrouted
270	0	0	263	149	127

Z 1.61 X 13.210000 Y 7.170000 dx 13.210000 dy 7.170000 dist 15.0304 grid X 0.010000 Y 0.010000 Inches

20:00

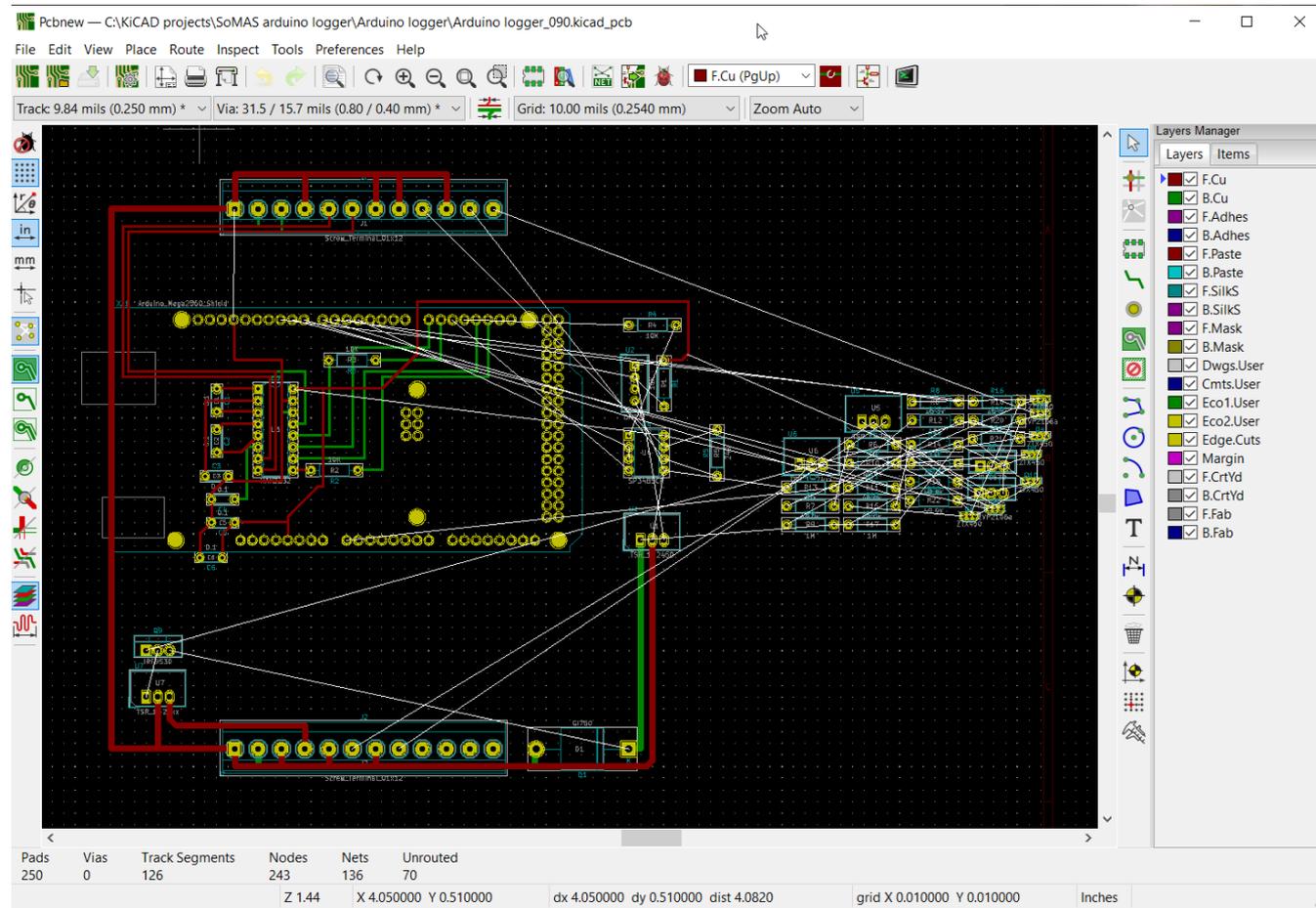
Welcome to the Rat Nest

- The PCB editor just jams all the components onto the board, then draws a “rat nest” of little white lines between all the pads that need connections.
- YOU get to figure out where to place 69 components, then how to route 127 traces between them.
- Yes, some PCB software purports to “auto place” and “auto route.” Those with the money to buy these high end packages tell me they don’t work so well.
- HEAR ME, HUMANS - this is something we can still do better than a computer!



Layout

- Put on some groovy music - Parliament / Funkadelic works for me.
- If you have a board size constraint, draw “edge cuts”, otherwise leave that to the end.
- If you have components that have to go somewhere (buttons, displays, terminal strips) place them first.
- Group related components together, then start routing traces.
- If two traces have to cross, go from top to bottom to top using a component pad, or add a “via.”



Layout Tips

- Line width and spacing - I like wide traces and lots of spacing.
- Supposedly the program will not let you inadvertently short two nets together. If you cannot make a trace you know needs to be there, go back and double check your schematic. If you made a schematic error, go back and correct it in the schematic editor, generate a new netlist, then update the netlist in your PCB.
- **SAVE A LOT, SAVE NEW VERSIONS A LOT** so if you mess up you can back up, not start over.

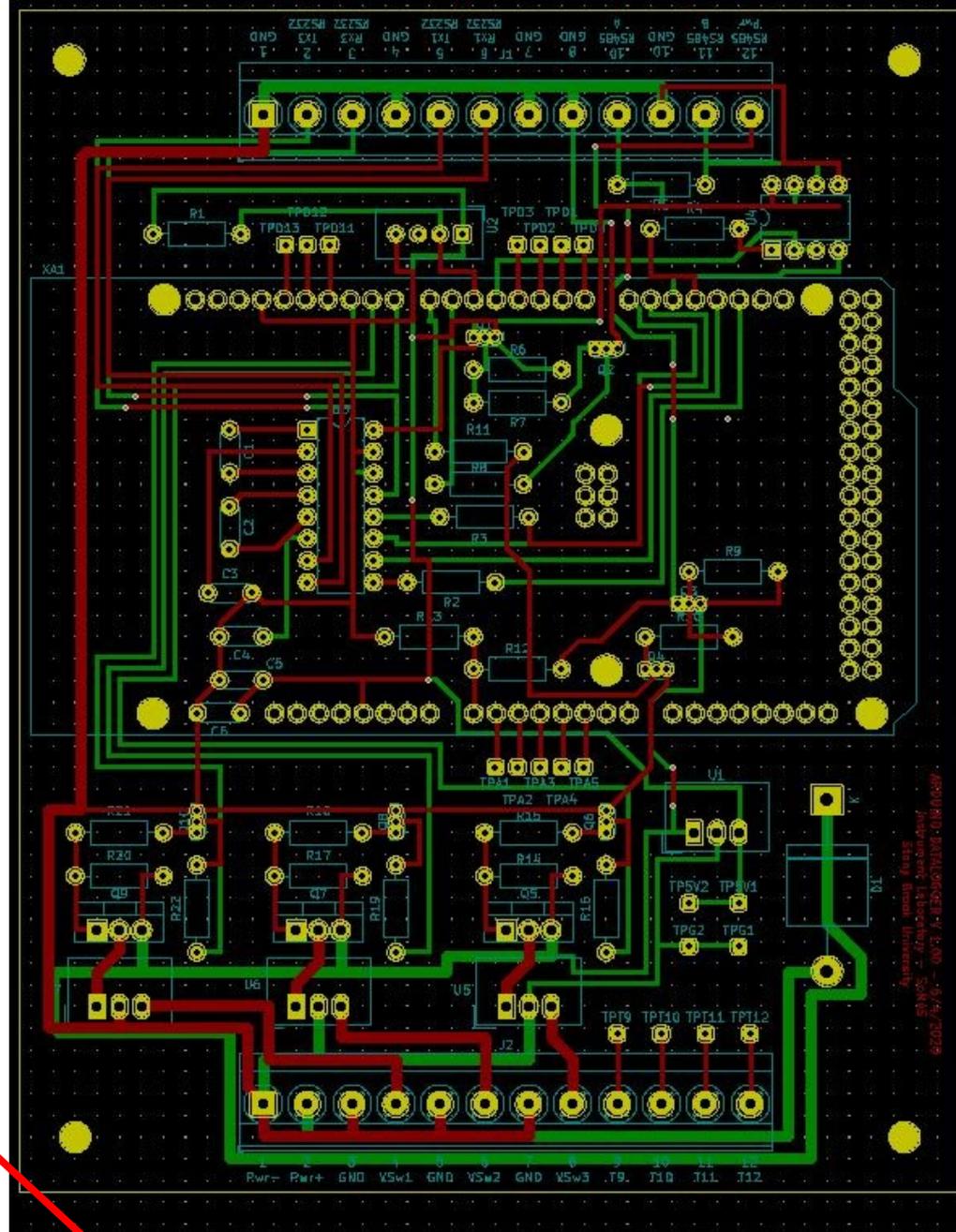
Every successful trace lowers the “Unrouted” count toward the magical number of ZERO.

The screenshot shows a PCB layout software interface with a dark background. The main workspace displays a complex circuit board design with various components and traces. The status bar at the bottom shows the following data:

Pads	Vias	Track Segments	Nodes	Nets	Unrouted
250	0	126	243	136	70

A red arrow points from the text "ZERO" in the text block to the "Unrouted" count of 70 in the status bar.

**Eventually
Nirvana (version 1.00)
is
Reached!**



Nets	Unrouted
136	0

Export Manufacturing Files

Warning - do NOT select "extended X2 format."

The screenshot shows the Pcbnew Plot dialog box. The 'Plot format' is set to 'Gerber' and the 'Output directory' is 'Gerber/'. The 'Included Layers' list includes F.Cu, B.Cu, F.SilkS, F.Mask, and B.Mask. The 'General Options' section has several checked items: 'Plot footprint values', 'Plot footprint references', 'Exclude PCB edge layer from other layers', 'Exclude pads from silkscreen', and 'Check zone fills before plotting'. The 'Gerber Options' section has 'Generate Gerber job file' checked and 'Use extended X2 format' unchecked. A red arrow points to the 'Coordinate format' dropdown menu, which is set to '4.6, unit mm'. The 'Plot' button is highlighted with a blue border.

Manufacturing Files List

SoMAS arduino logger > Arduino logger > Gerber > Gerber >

Name	Date modified	Type	Size
 Arduino logger_101.zip	8/20/2020 10:59 PM	Compressed (zipp...	52 KB
 Arduino logger_101-B_Cu.gbr	8/20/2020 10:49 PM	GBR File	47 KB
 Arduino logger_101-B_Mask.gbr	8/20/2020 10:49 PM	GBR File	20 KB
 Arduino logger_101-Edge_Cuts.gbr	8/20/2020 10:49 PM	GBR File	1 KB
 Arduino logger_101-F_Cu.gbr	8/20/2020 10:49 PM	GBR File	70 KB
 Arduino logger_101-F_Mask.gbr	8/20/2020 10:49 PM	GBR File	20 KB
 Arduino logger_101-F_SilkS.gbr	8/20/2020 10:49 PM	GBR File	129 KB
 Arduino logger_101-job.gbrjob	8/20/2020 10:49 PM	GBRJOB File	3 KB
 Arduino logger_101-NPTH.drl	8/20/2020 10:49 PM	DRL File	1 KB
 Arduino logger_101-PTH.drl	8/20/2020 10:49 PM	DRL File	5 KB

*.gbr = Gerber photoplot format:

B_Cu = back copper, F_Cu = front copper,

Edge-cuts = board edges, B_mask = back solder mask, F_mask = front solder mask,

F_SilkS = front silkscreen, B_SilkS = back silkscreen.

*.drl = Excelon drill file:

NPTH = non-plated through holes, PTH - plated through holes

*.gbrjob = job file, not sure what this does.

ZIP IT ALL TOGETHER!

Design Rules Check @ 4pcb.com

Upload zip file for free DFM - report usually back in <2 hours.

The screenshot displays the 4pcb.com website interface. At the top, the navigation bar includes the 4PCB logo, a 'My Account | Register' link, and a phone number '1.303.557.1865'. A secondary menu contains links for Home, Products/Services, Assembly, Capabilities, Specials, Engineering/Technical, About Us, Blog, Contact Us, and a highlighted 'COVID-19 Info' link. Below the navigation, a banner reads 'FREE PCB File Check' with a 'Share | Print' option. The main content area features a large image of a computer monitor displaying 'YOUR RESULTS' with sections for '\$100 DISCOUNT CODE', 'SHOW STOPPERS', 'AUTOMATICALLY FIXED', and 'INSUFFICIENT SMT SOLDERMASK CLEARANCE'. To the right of the monitor is the 'FreeDFM' logo and a description: 'Ensure your PCB design is free of manufacturability issues. Our FREE tool is fast & easy to use.' A prominent green 'UPLOAD FILES' button is positioned below the text, along with a note: '\$ Save up to \$100 when you use it!'. On the right side of the page, a 'Products/Services' dropdown menu is open, listing options such as 'connection center', 'GET INSTANT QUOTE', 'Place PCB & Assembly Order', 'Capabilities', 'FREE PCB Layout Software', 'FREE PCB File Check', 'Student Program', 'Monthly Specials', and 'Download Presentation'. At the bottom right, there is a 'Live Chat Expert Online Ask a Question' button and a 'PCB Artist' logo.

Manufacturing @ 4pcb.com

After your design passes Free DFM, upload and place order.
\$33 special is usually cheaper until you get quantity >20.

The screenshot shows the 4pcb.com website with a navigation menu and a table of pricing options. The navigation menu includes Home, Products/Services, Assembly, Capabilities, Specials, Engineering/Technical, About Us, Blog, Contact Us, and COVID-19 Info. The table lists three pricing options: BareBones™, \$33 Each, and \$66 Each. The \$33 Each option is highlighted in green. A live chat widget is visible in the bottom right corner.

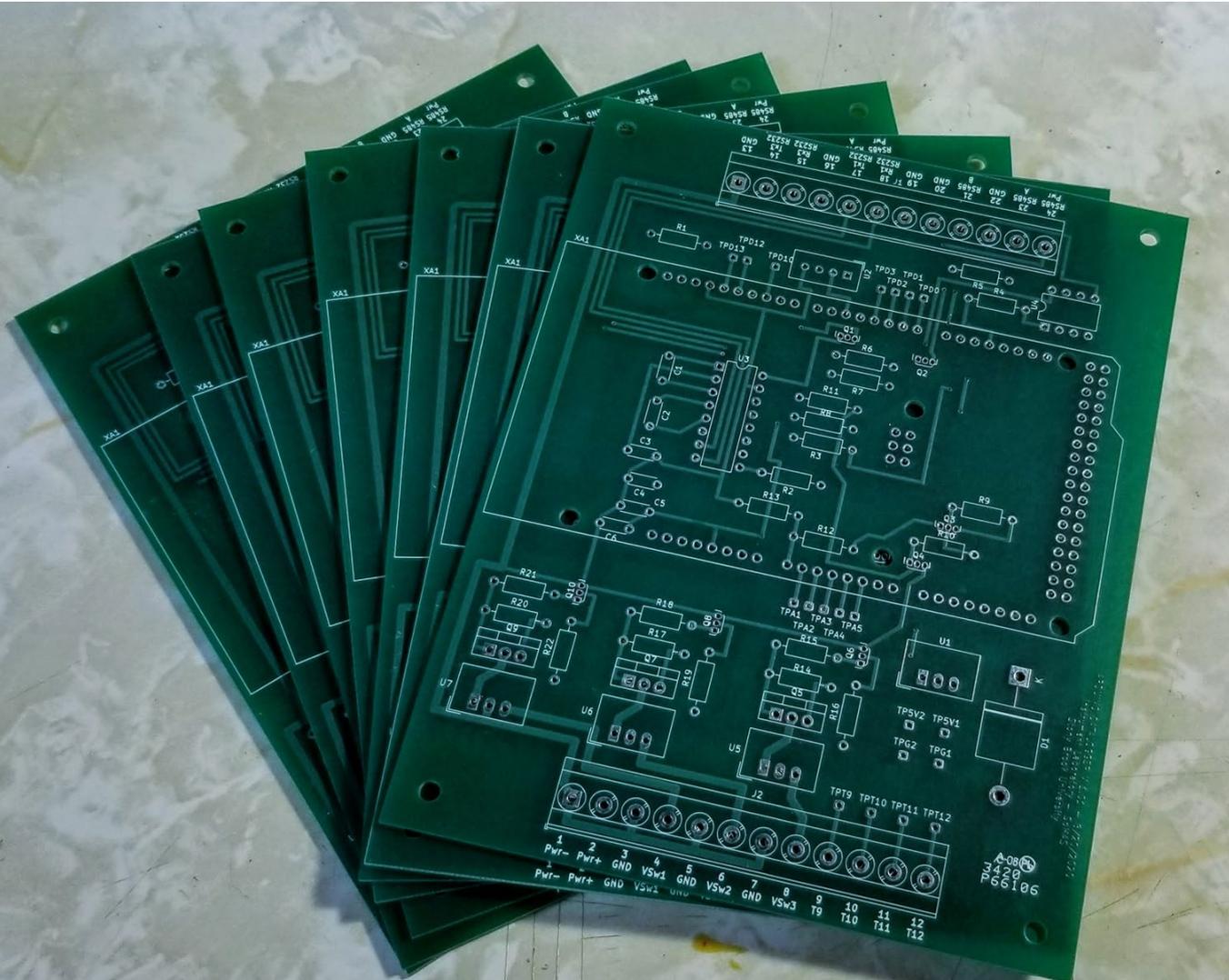
BareBones™	\$33 Each	\$66 Each
2 Layer - 1 Day Turn	2 Layer - 3 Day Turn	4 Layer - 5 Day Turn
10" x 16" Max Board Size	Max. Board Size: 60 sq. in.	Max. Board Size: 30 sq. in.
Min. Order Quantity: 1	Min. Order Quantity: 3	Min. Order Quantity: 4
FR-4 .062" Material	FR-4 .062" Material	FR-4 .062" Material
1 oz. Cu.	1 oz. Cu.	1 oz. Cu.
Tin Finish	Lead-Free Solder Finish*	Lead-Free Solder Finish*
No Mask (bare)	Green Mask	Green Mask
No Legend	White Legend (1 or 2 Sides)	White Legend (1 or 2 Sides)
Custom Shape*	Custom Shape	Custom Shape
Place Order >>	Place Order >>	Place Order >>

Products/Services

- connection center
- GET INSTANT QUOTE
- Place PCB & Assembly Order
- Capabilities
- FREE PCB Layout Software
- FREE PCB File Check
- Student Program
- Monthly Specials
- Download Presentation
- Referral Bonus

Live Chat Expert Online Ask a Question

In a Week or So...



Free
popcorn
in every
order!

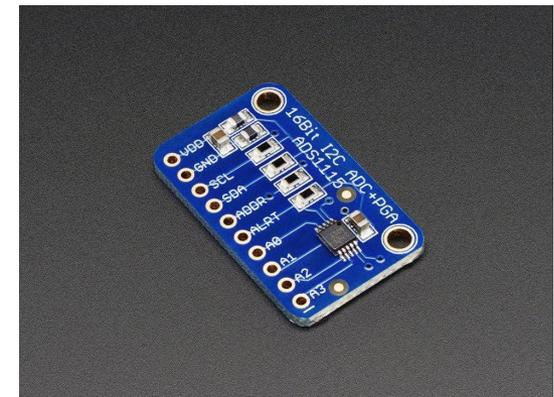
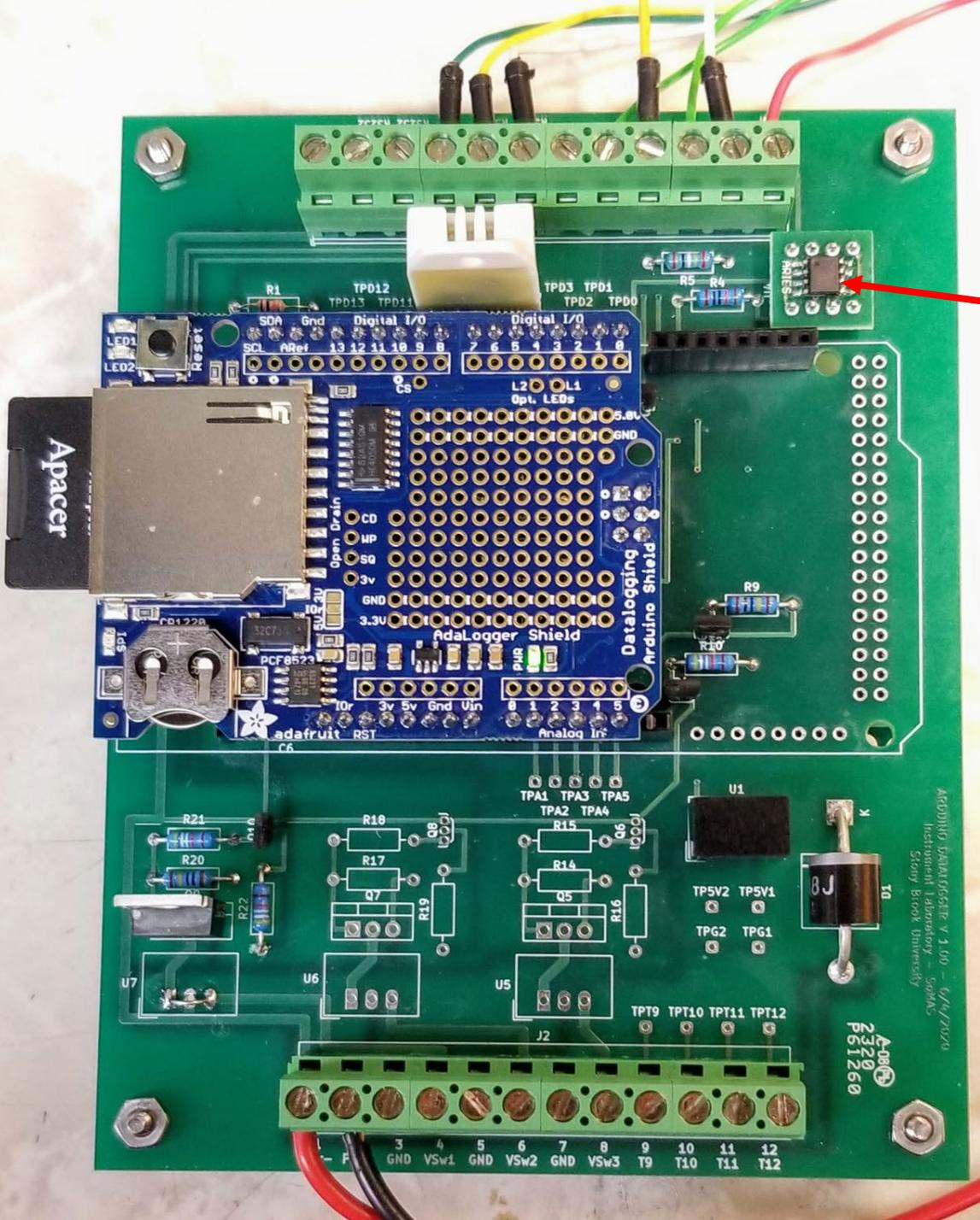
Assemble

Remember “mostly”
through hole?

Surface mount component plus
SMC/DIP adapter board.

To use surface mount components
on through hole boards, search for
“SMC to DIP adapter boards”

Searching “breakout boards”
on sites like sparkfun.com
or adafruit.com
returns many useful devices



ADS1115 - 4 channel, 16 bit a/d with
programmable gain amplifier

Program and Test

L to R: 4G router, serial to IP converter, datalogger, RS485 pressure sensor.



Deploy!

Summer 2020: two tide stations, two salinity stations, one multiparameter floating platform.

Telemeters processed data, but 8GB flash allows local storage of every instrument scan (probably for the life of the station) for reprocessing or troubleshooting - has already proven useful.

Expect to upgrade six existing data stations in the next six months.



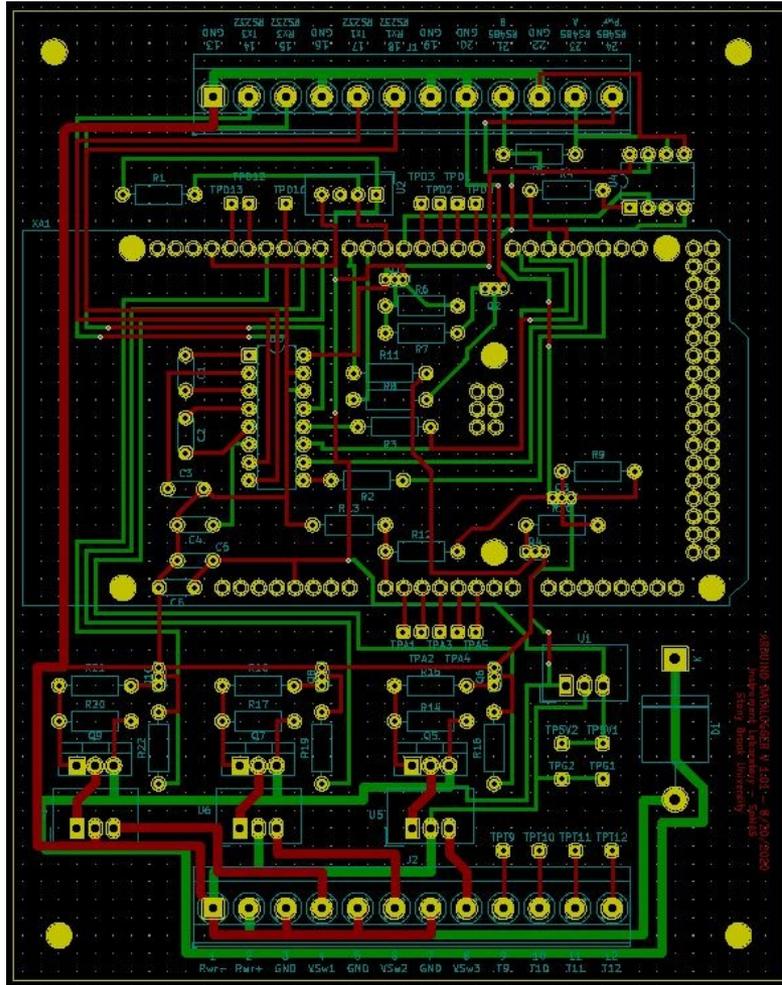
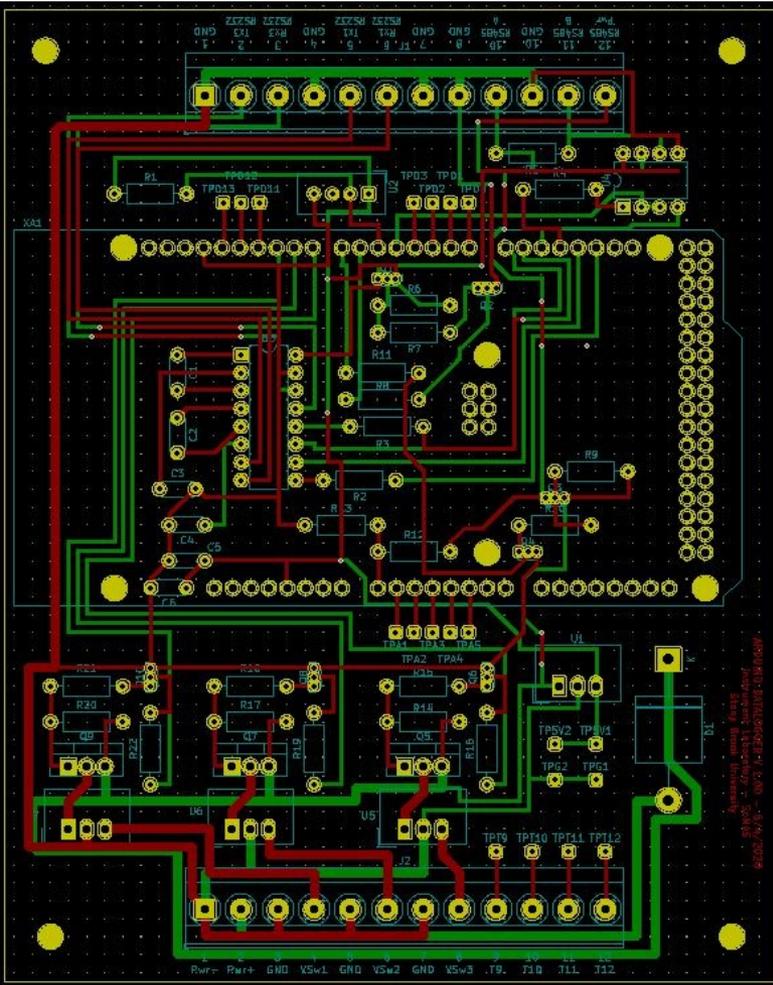
Version 1.00 worked - but there's always a punch list

Version 1.01 - Removed orphan via and trace to nowhere, moved one digital line to fix hardware conflict, renumbered terminal strip #2 to 13-24 instead of duplicated 1-12.

Next version 1.1 - more accurate Real Time Clock add 4-channel 16-bit A/D with PGA.

Version 1.00

Version 1.01



ACKNOWLEDGEMENTS

To Forrest W. Mims III - for simple systems that work.

To my parents, who always found a few dollars to invest at Halley Electronics, Radio Shack, and Pagoda Hardware - and to the longsuffering employees of Halley Electronics, Radio Shack, and Pagoda Hardware.

To teachers, mentors, students, and colleagues including Henry Harrison, David Lucyk, Bob Slavonik,



Trevor Young



Greg Smith



Alex Sneddon



Chris Crosby



Miles Litzmann

and of course my shipmates at RVTEC.

QUESTIONS? I AM NOT AFRAID.



Thank you!

Now go build stuff!

(I'm going to have some popcorn)

Thomas C. Wilson, Jr. • thomas.wilson@stonybrook.edu

