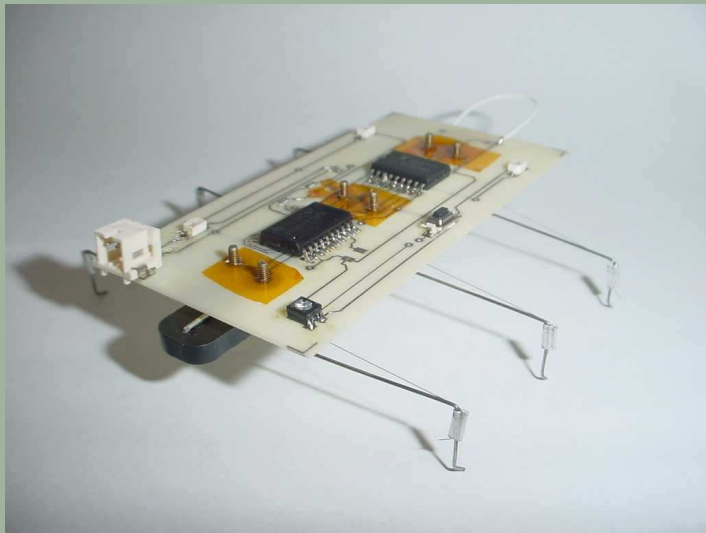


Design Review for Stiquito PCB



PCB Designer: Steve D. Tucker
Reviewers: Dr. James Conrad
Jeffery Young
Rajan Rai

Purpose of Design Reviews

- 1 Inspect the schematic and ensure the correct functionality is present in the design.
- 1 Inspect the schematic and ensure the components are of correct values and type.
- 1 Inspect the routing to see if there are any connection / crosstalk / or other problems.

Responsibilities

1 Designer Responsibilities

- Display near completed design or progress of design.
- Justify design choices and assumptions.
- Demonstrate that the design meets project goals.
- Make note of design problems and be prepared to fix them.

1 Reviewer Responsibilities

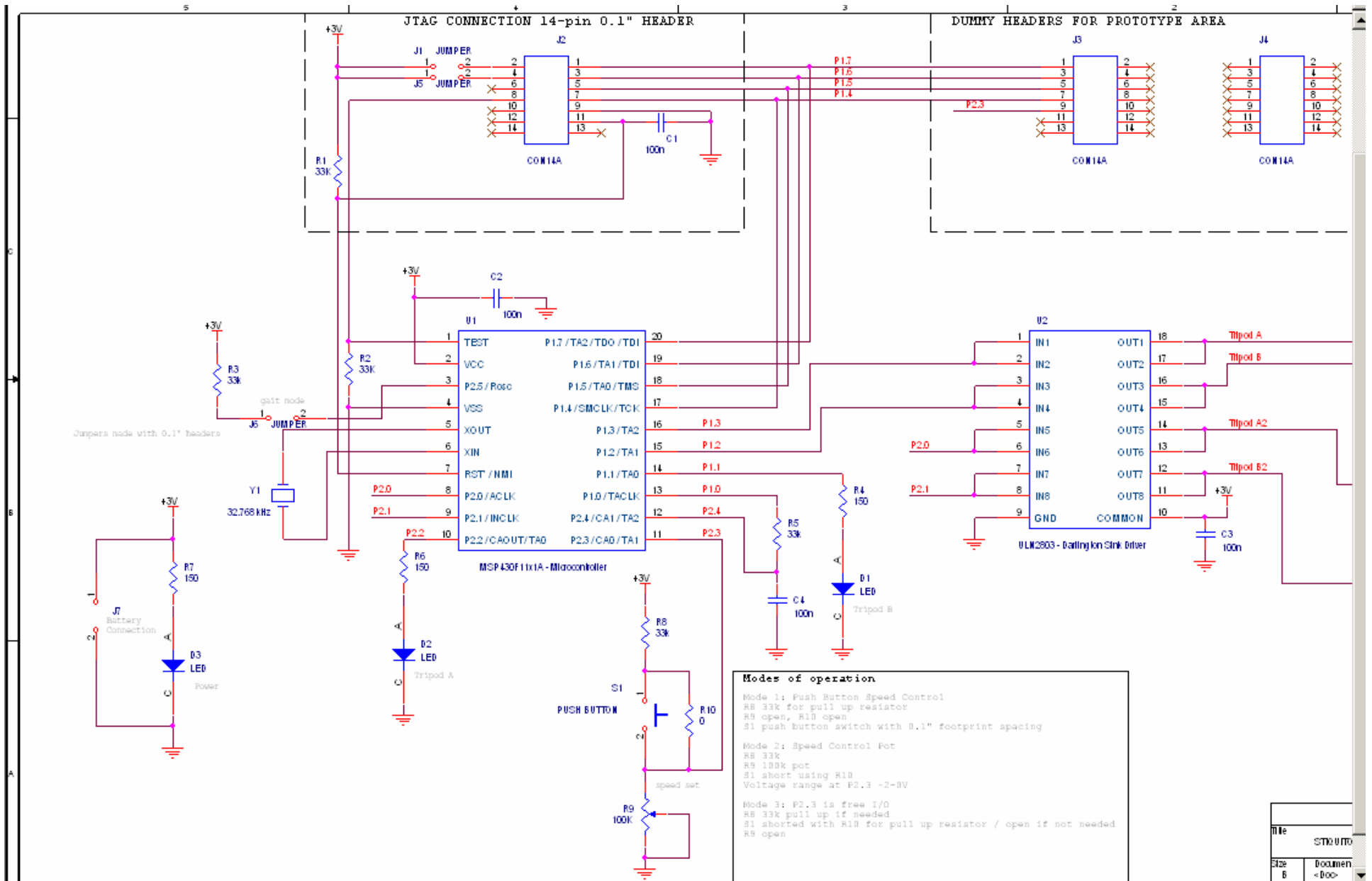
- Check the design for mistakes the could cause quality and functional problems.
- Reconsider the designers assumptions. Are they valid or need to be corrected.
- Request the forgotten features be added.
- Request that extra features be justified or remove to reduce cost.

Design Requirements - Schematic

- 1 Current Stiquito design needed correction and features added to the schematic.
 - Add JTAG interface.
 - JTAG power option selected with jumpers using 2-pin, 0.1” header on PCB.
 - Add Prototype Area.
 - Route extra I/O to the Prototype Area.
 - Correction added for A/D conversion.
 - Speed control input wired for different options.
 - Thru-hole area added for crystal to test another clock mode of the microcontroller.
 - Add decoupling to IC’s to reduce voltage sag on power rails.
 - Schematic Symbols corrected to reflect true wiring and pin outs.
 - Microcontroller I/O reordered to include new features.
 - **Make everything as cheap as possible. Remove expensive parts like connectors and switches.**

Design Assumptions - Schematic

- 1 Values of R5 and C4 did not matter for now just needed place holder and footprints for layout.
- 1 Three modes were designed for the input P2.3.
 - Mode 1: Push button speed control.
 - Mode 2: Speed control pot
 - Mode 3. P2.3 is a free I/O routed to prototype area.
- 1 Headers used to represent prototype area.
- 1 Crystal added to pin 5 and 6 of Micro-controller to add footprint for testing external clock mode.
- 1 Symbol for the Leg Assembly is just a “black box” representation.



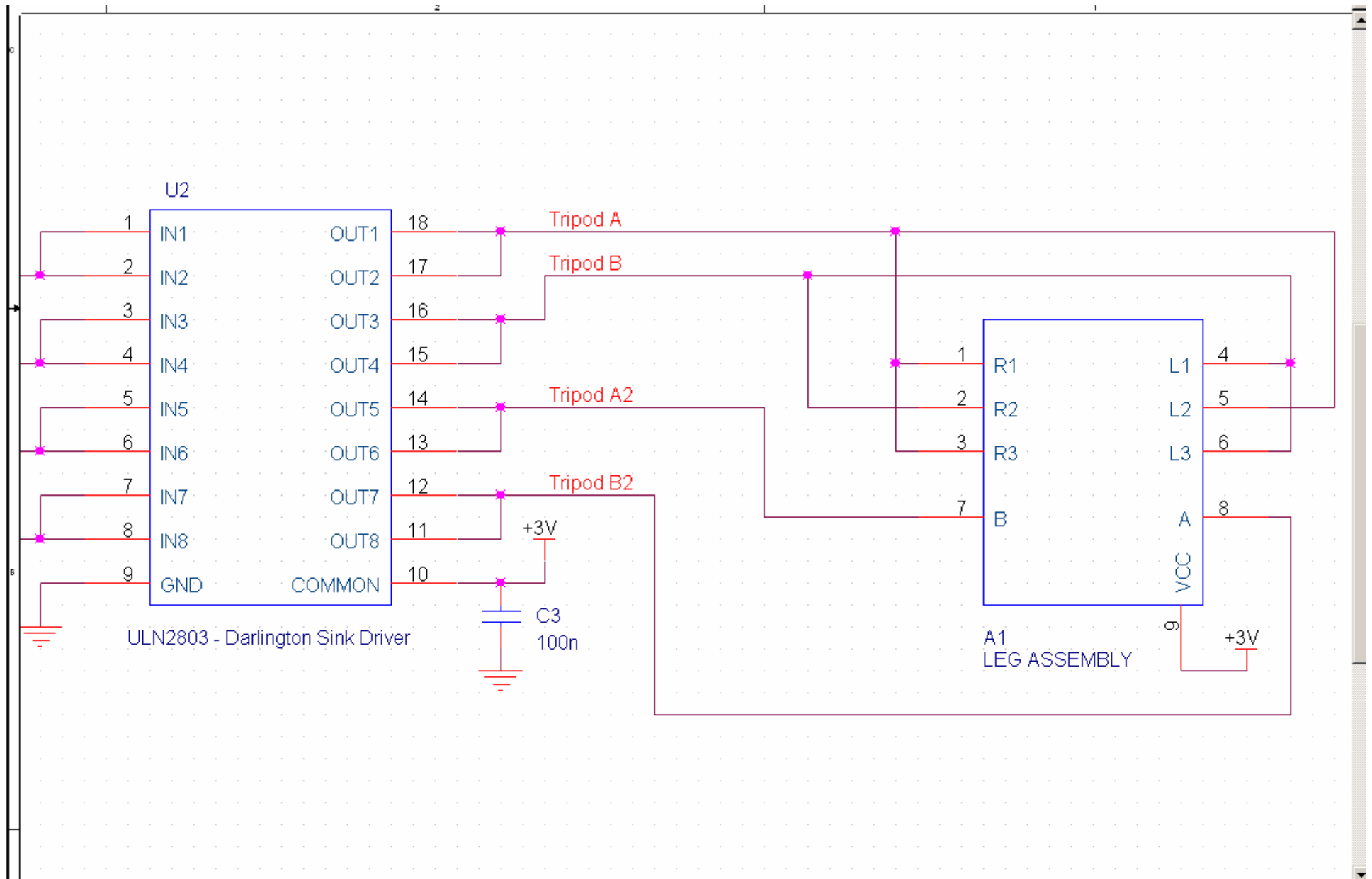
Modes of operation

Mode 1: Push Button Speed Control
 R8 33k for pull up resistor
 R9 open, R10 open
 S1 push button switch with 0.1" footprint spacing

Mode 2: Speed Control Pot
 R8 33k
 R9 100k pot
 S1 short using R10
 Voltage range at P2.3 ~2-8V

Mode 3: P2.3 is free I/O
 R8 33k pull up if needed
 S1 shorted with R10 for pull up resistor / open if not needed
 R9 open

Micro-controller and JTAG



Sink Driver and Leg Assembly

Design Requirements - Layout

1 Board Info

- 2-layer with solder mask and one side population.
- Silk Screen maybe possible but label in metal.
- All holes plated.
- Size: approx. 1 x 3 inches (smaller if possible)
- Material: FRU-62 (cheapest).
- 2 IC's (Microcontroller and Sink Driver), 3 LED's, 1 Trim Pot

1 Current Stiquito PCB have problems affecting quality and also needed the features added to the schematic.

- Footprint for Stiquito Leg Assembly needed better fit and better pads.
- Limit to 3 drills.
- R and C are 0402 size. Other parts need the footprint checked.
- Heavy traces need to ULN2803 Sink Driver and to Spine Connection of the Leg Assembly.
- Power soldered in without connector.

Design Assumptions – Layout - 1

1 Leg Assembly Footprint

- Uses bolt size and nut size #0-80.
- Bolt tread dia. is 60 mils
- Nut outer dia. is 156 mils
- Padstack made with 160 mils pad and 75 mils hole this accounts for 5 mil plating. Final hole size 65 mils. May be a tight fit.
- Checked some on-line info. (expressPCB) Starting with ½ oz. copper and plating to 1 ¼ oz. the plating is ~1.7 mils thick. This gives a finished hole approximately 71.6 mils.

1 Leg Assembly Pad Spacing

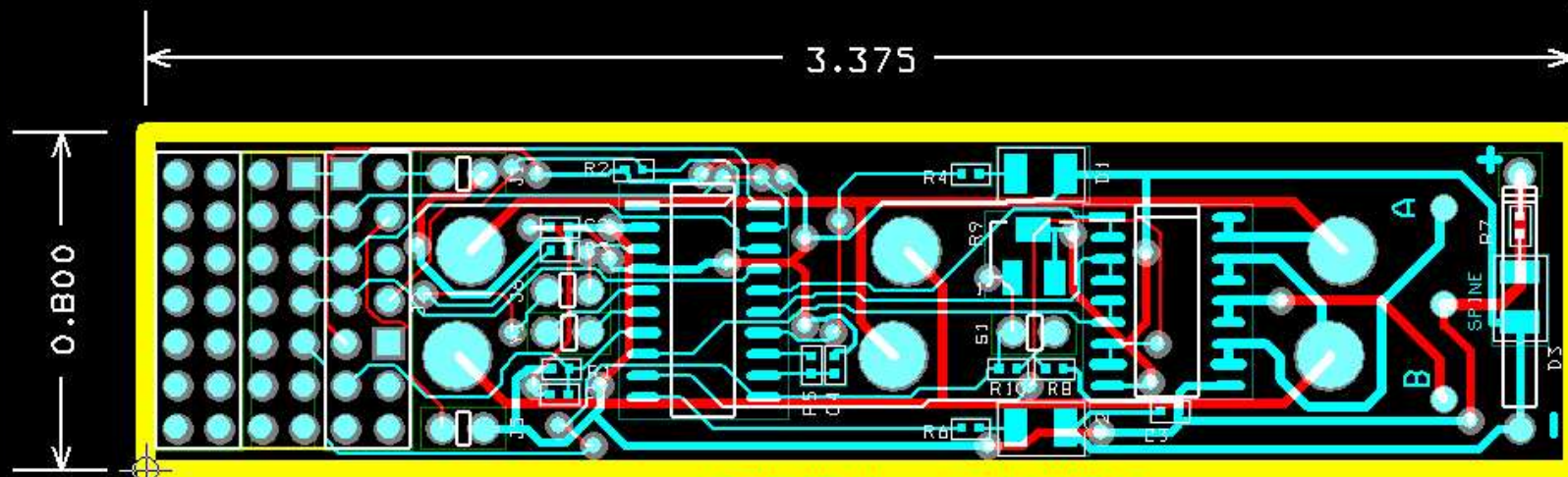
- 3 x 2 array with 1031.25 mils spacing in the x direction and 250 mils in the y direction. The rows are offset by 31.25 mils.
- Connections for the VCC Spine and two 2nd degree of freedom pads included with footprint for ease of later placement.
- Used Cadence Layout's Pad Array Generator to create footprint once spacing was calculated.
- Footprint could not automatically be placed using netlist. Instead a 9-pin header was placed by the program then replaced later.

Design Assumptions – Layout - 2

- 1 Main current consumption is in the nitinol wires and sink driver.
 - Assumed that with 2 way gait that the current would pulse up to 0.5 amps.
 - Routed power to Vcc spine connection of leg assembly at 30 mils.
 - Routed power and ground to sink driver at 25 mils.
 - Continued to rout power and ground at 25 mils where possible but used a minimum of 15 mils.
 - Signal trace minimum 8 mils.

- 1 Trace current chart from expressPCB. 1 ¼ oz. copper.

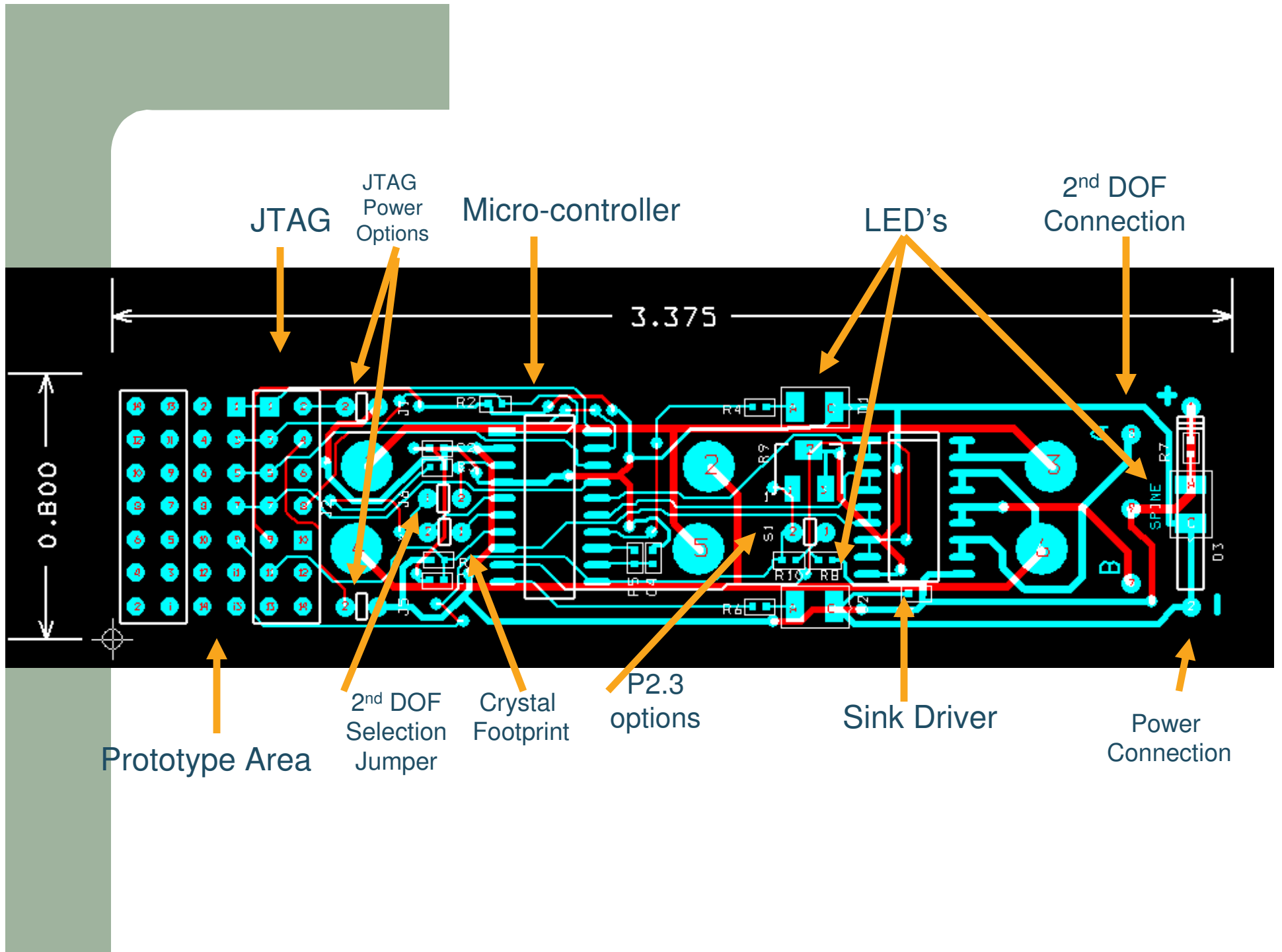
mils	Amperes
10	0.3
15	0.4
20	0.7
25	1.0
50	2.0
100	4.0
150	6.0

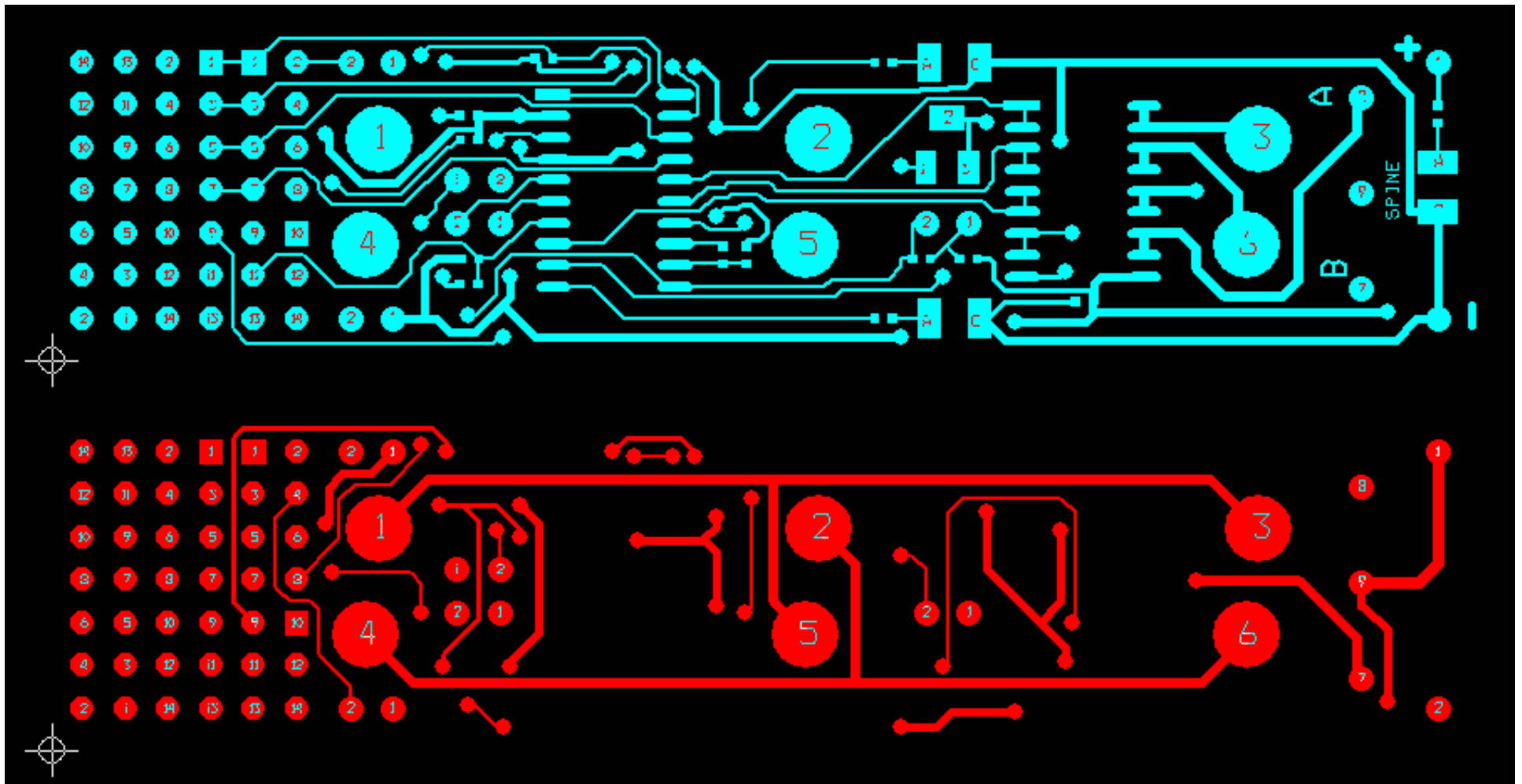


DRILL CHART

SYM	DIAM	TOL	QTY	NOTE
⊠	0.020		32	
+	0.038		57	
◇	0.075		6	
TOTAL			95	

Layout View showing top, bottom, outline and silk screen layer.





Top and bottom layer displayed individually.

Supporting Documentation

- 1 MSP430F1101A Data Sheet
- 1 ULN2803/04 Data Sheet (Toshiba)
- 1 Application Report – “Economic Measurement Techniques With the Comparator_A Module”
- 1 Data Sheets for components with footprint information.
- 1 MSP430-H1121-B Header Board Data Sheet for information on the JTAG interface