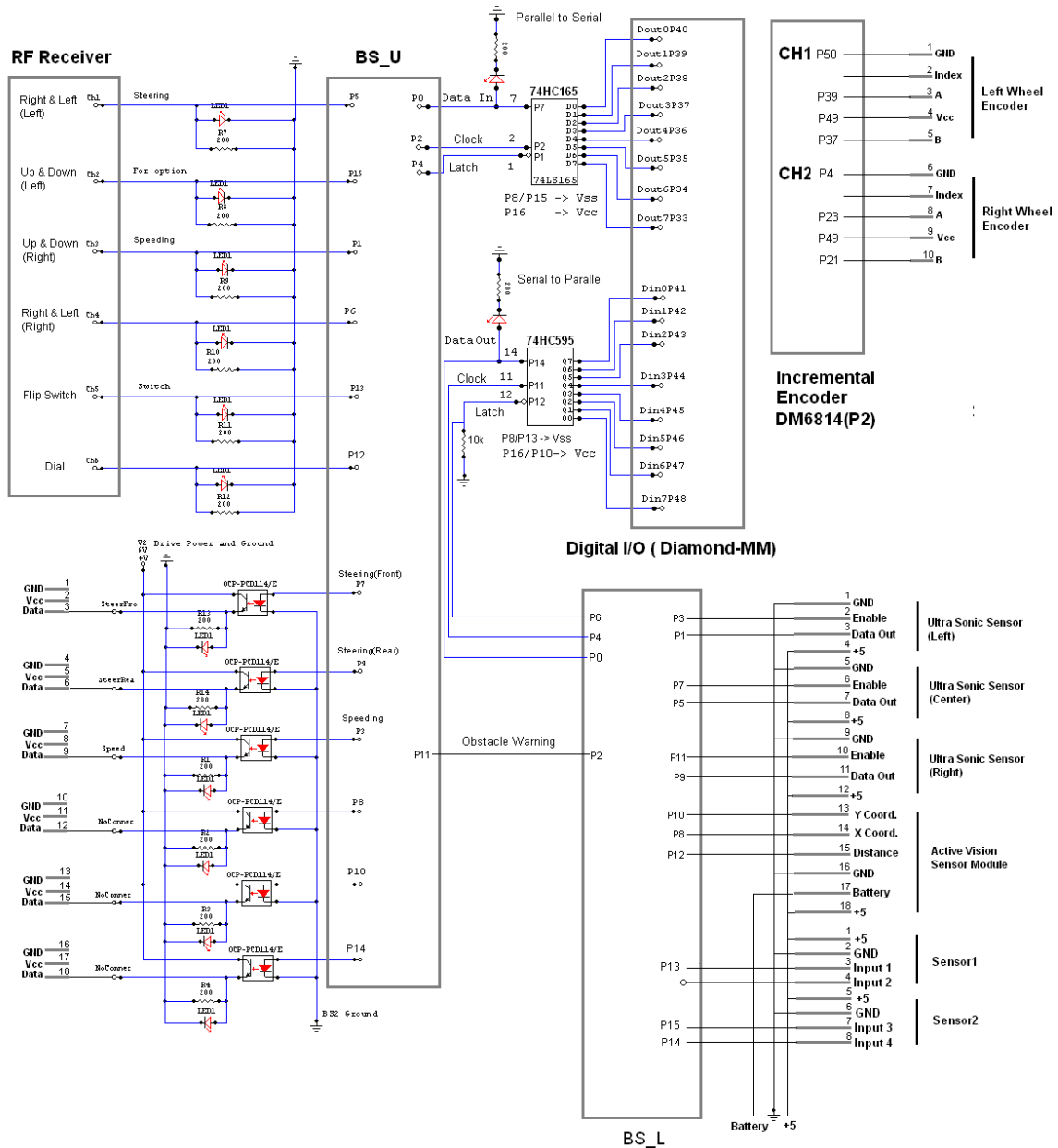


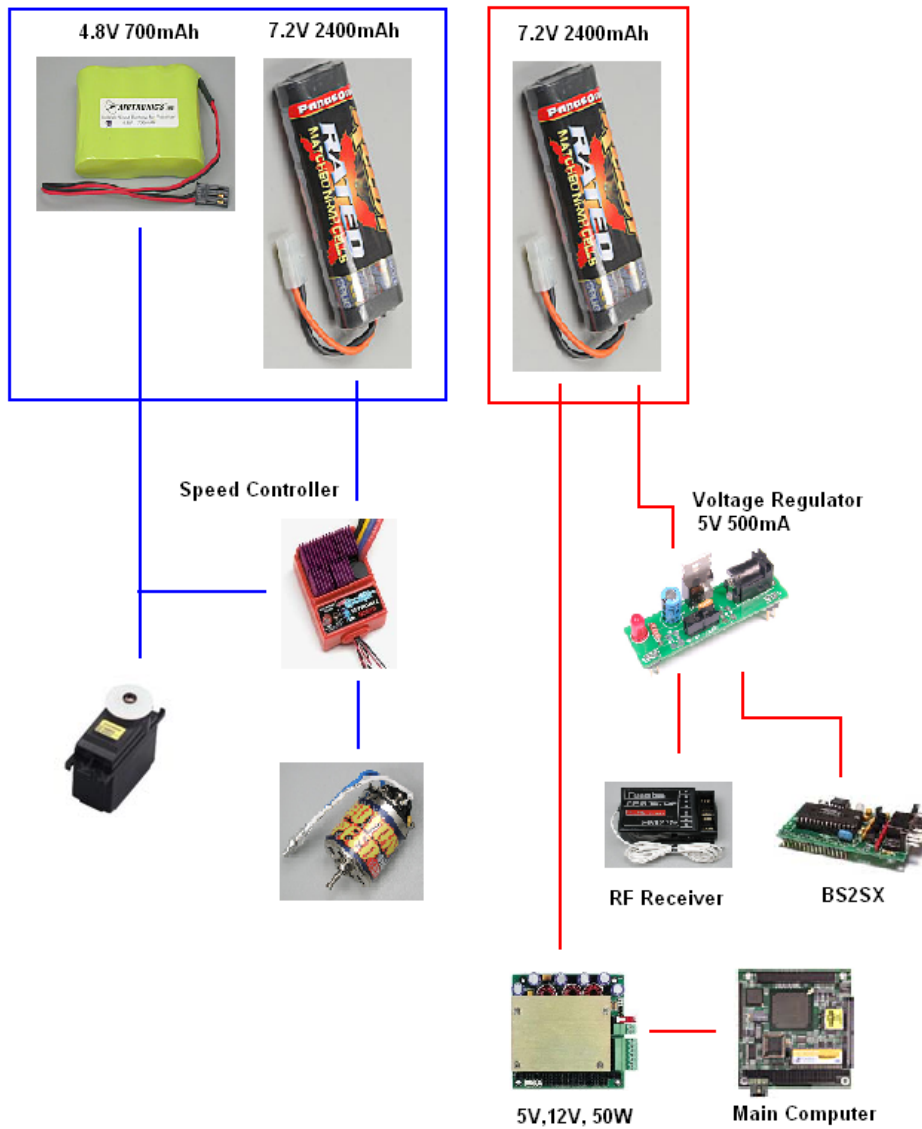
# Making PCB for SmartCar Interface Board

By Seung kook Jun

## 1. Circuit Diagram



## 2. Power Supply Diagram



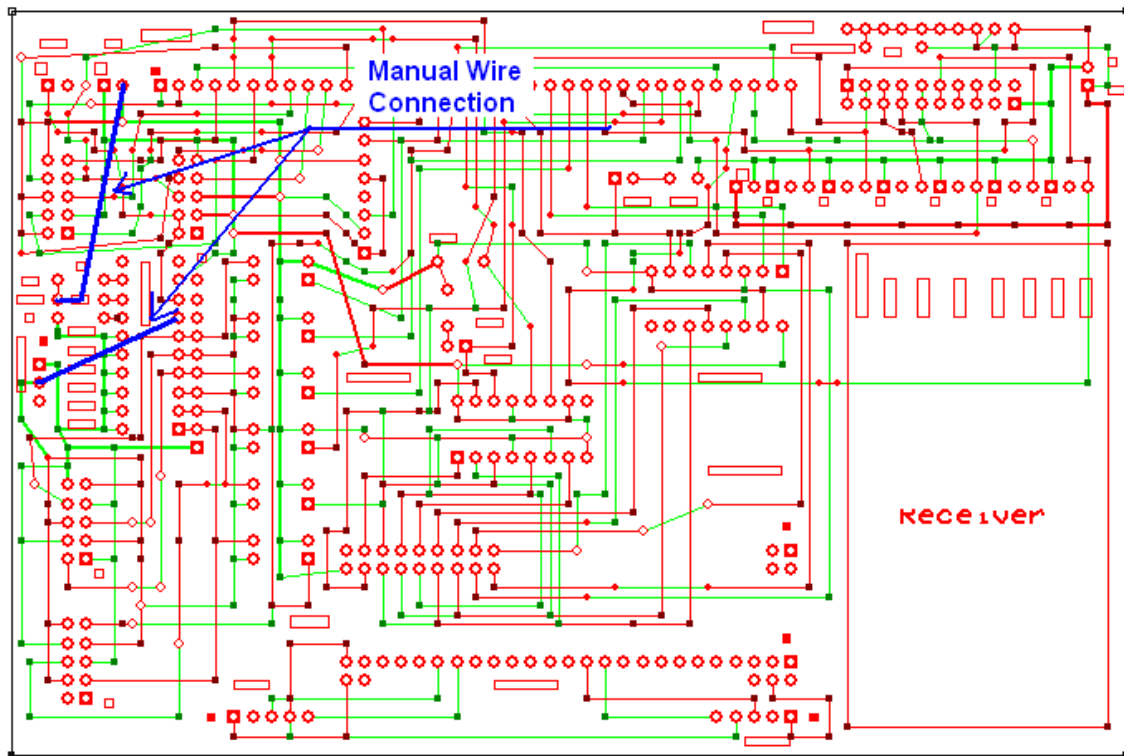
## 3. Working environment on ExpressPCB

Size: 4 in × 6 in  
Grid: 0.1 in : Snap:0.025in  
Hole: 0.075" round with 0.040" hole  
0.075" square pad with 0.040" hole ( for Vcc, Pin 0 or 1)  
0.056" round pad with 0.029" hole (for connectoin)  
Line: 0.01 trace and 0.02 trace

#### 4. List of Components

No	Description	Specification	Q'ty
1	RF receiver	2.6 in × 1.4 in	1
2	74HC595	Serial to Parallel	1
3	74HC165	Parallel to Serial	1
4	LED Array	10 port	2
5	Resistor Array	220Ω, 9port	2
6	Resistor	10kΩ	1
7	Resistor	220Ω	2
8	Resistor	1kΩ	4
9	LED	Red, Blue	2
10	OCP-PCD114/E	Photo isolator	4

#### 5. PCB Board



## 5. Appendix

- <http://www.expresspcb.com/ExpressPCBHtml/Tips.htm>

### Placing Components

Generally, it is best to place parts only on the top side of the board.

When placing components, make sure that the snap-to grid is turned on. Usually, a value of 0.025" for the snap grid is best for this job.

First place all the components that need to be in specific locations. This includes connectors, switches, LEDs, mounting holes, heat sinks or any other item that mounts to an external location.

Give careful thought when placing component to minimize trace lengths. Put parts next to each other that connect to each other. Doing a good job here will make laying the traces much easier.

Arrange ICs in only one or two orientations: up and down, or, right and left. Align each IC so that pin one is in the same place for each orientation, usually on the top or left sides.

Position polarized parts (i.e. diodes, and electrolytic caps) with the positive leads all having the same orientation. Also use a square pad to mark the positive leads of these components.

You will save a lot of time by leaving generous space between ICs for traces. Frequently the beginner runs out of room when routing traces. Leave 0.350" - 0.500" between ICs, for large ICs allow even more.

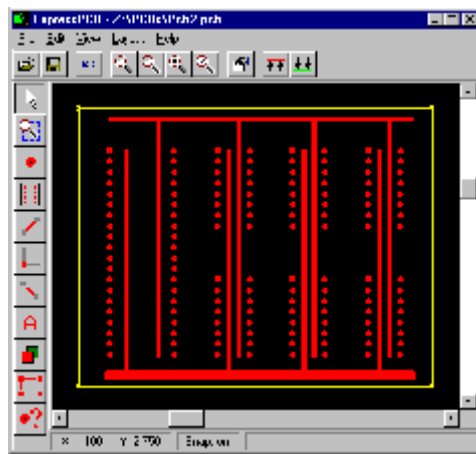
Parts not found in the component library can be made by placing a series of individual pads and then grouping them together. Place one pad for each lead of the component. It is very important to measure the pin spacing and pin diameters as accurately as possible. Typically, dial or digital calipers are used for this job.

After placing all the components, print out a copy of the layout. Place each component on top of the layout. Check to insure that you have allowed enough space for every part to rest without touching each other.

### Placing Power and Ground Traces

After the components are placed, the next step is to lay the power and ground traces. It is essential when working with ICs to have solid power and ground lines, using wide traces that connect to common rails for each supply. It is very important to avoid snaking or daisy chaining the power lines from part-to-part.

One common configuration is shown below. The bottom layer of the PC board is used for both the power and ground traces. A power rail is run along the front edge of the board and a ground rail along the rear edge. From these rails attach traces that run in between the ICs. The ground rail should be very wide, perhaps 0.100", and all the other supply lines are 0.050". When using this configuration, the remainder of the bottom layer is then reserved for the vertical signal traces.



## Placing Signal Traces

When placing traces, it is always a good practice to make them as short and direct as possible.

Use vias (also called feed-through holes) to move signals from one layer to the other. A via is a pad with a plated-through hole.

Generally, the best strategy is to layout a board with vertical traces on one side and horizontal traces on the other. Add via where needed to connect a horizontal trace to a vertical trace on the opposite side.

A good trace width for low current digital and analog signals is 0.010".

Traces that carry significant current should be wider than signal traces. The table below gives rough guidelines of how

wide to make a trace for a given amount of current.

<b>0.010"</b>	<b>0.3 Amps</b>
<b>0.015"</b>	<b>0.4 Amps</b>
<b>0.020"</b>	<b>0.7 Amps</b>
<b>0.025"</b>	<b>1.0 Amps</b>
<b>0.050"</b>	<b>2.0 Amps</b>
<b>0.100"</b>	<b>4.0 Amps</b>
<b>0.150"</b>	<b>6.0 Amps</b>

When placing a trace, it is very important to think about the space between the trace and any adjacent traces or pads. You want to make sure that there is a minimum gap of 0.007" between items, 0.010" is better. Leaving less blank space runs the risk of a short developing in the board manufacturing process. It is also necessary to leave larger gaps when working with high voltage.

When routing traces, it is best to have the snap-to grid turned on. Setting the snap grid spacing to 0.050" often works well. Changing to a value of 0.025" can be helpful when trying to work as densely as possible. Turning off the snap feature may be necessary when connecting to parts that have unusual pin spacing.

It is a common practice to restrict the direction that traces run to horizontal, vertical, or at 45 degree angles.

When placing narrow traces, 0.015" or less, avoid sharp right angle turns. The problem here is that in the board manufacturing process, the outside corner can be etched a little more narrow. The solution is to use two 45 degree bends with a short leg in between.

It is a good idea to place text on the top layer of your board, such as a product or company name. Text on the top layer can be helpful to insure that there is no confusion as to which layer is which when the board is manufactured.

## Checking Your Work

After all the traces are placed, it is best to double check the routing of every signal to verify that nothing is missing or incorrectly wired. Do this by running through your schematic, one wire at a time. Carefully follow the path of each trace on your PC layout to verify that it is the same as on your schematic. After each trace is confirmed, mark that signal on the schematic with a yellow highlighter.

Inspect your layout, both top and bottom, to insure that the gap between every item (pad to pad, pad to trace, trace to trace) is 0.007" or greater. Use the *Pad Information* tool to determine the diameters of pads that make up a component.

Check for missing vias. ExpressPCB will automatically insert a via when changing layers as a series of traces are placed. Users often forget that vias are not automatically inserted otherwise. For example, when beginning a new trace, a via is never inserted. An easy way to check for missing vias is to first print the top layer, then print the bottom. Visually inspect each side for traces that don't connect to anything. When a missing via is found, insert one. Do this by clicking on the *Pad* in the side toolbar, select a via (*0.056" round via* is often a good choice) from the drop down listbox, and click on the layout where the via is missing.

Check for traces that cross each other. This is easily done by inspecting a printout of each layer.

Metal components such as heat sinks, crystals, switches, batteries and connectors can cause shorts if they are placed over traces on the top layer. Inspect for these shorts by placing all the metal components on a printout of the top layer. Then look for traces that run below the metal components.