# How-To #2: Create a PCB using Fritzing and OSH Park

## Background:

This How-To provides instructions on how to create a printed circuit board (PCB) using the Fritzing (http://www.fritzing.org/) open-source design tool and then prepare that design for fabrication by OSH Park (http://www.oshpark.com/), which provides an economical fabrication service well suited for small volumes.

Because the production files Fritzing produces are not a perfect match to the requirements of OSH Park, this document provides step-by-step instructions for modifying the Fritzing files and then uploading the results to OSH Park for production. The instructions assume the circuit-board designer has an account with OSH Park; accounts are free and easy to set up on the website.

| | |
|---|---|
| **I** <br> Important | This How-To does not include information about using the Fritzing software to design a circuit board. The Fritzing website provides excellent tutorials for this purpose. Keep in mind that circuit-board design is both an art and a science. Fritzing will help with the science, but the art is mastered only with practice. |

Fritzing is unusual in that it allows a designer to begin with an interactive pictorial of a bread-board, jumper wires, components, and an Arduino™ single-board computer. The user can enter her bread-board experiment, then edit the schematic diagram generated by Fritzing, and finish by laying out the resulting circuit board. Some users prefer to go directly to the circuit-board layout and bypass the bread-board and schematic views.

The Infrared Navigation circuit board used by the Rolling Robot project is the example used here. Figure HT2-1 shows the three views of this circuit as seen inside the Fritzing software.
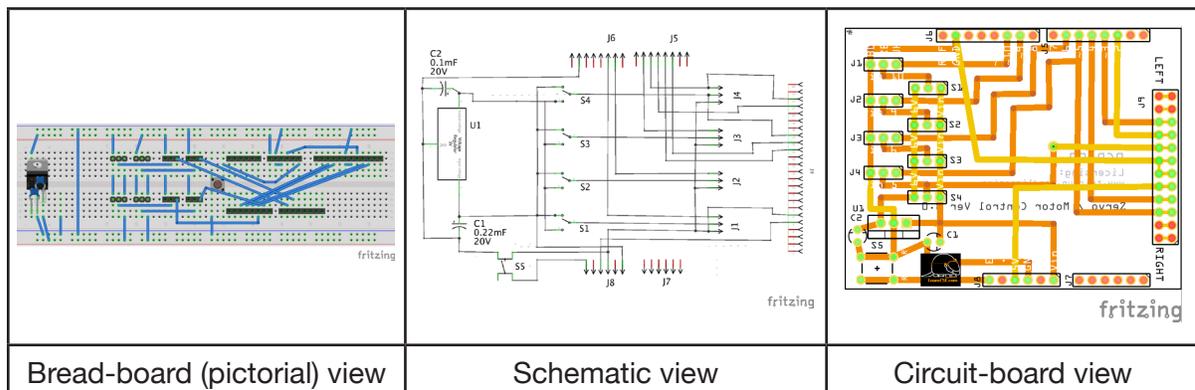


| Bread-board (pictorial) view | Schematic view | Circuit-board view |

*Figure HT2-1. Three views of IR navigation circuit-board*

The bread-board view in HT2-1, is a pictorial view — a picture or drawing of how the parts are physically connected.

The schematic view is an economical representation of the way the electricity flows.

The actual Fritzing design file for this circuit board can be found on LearnCSE.com.

## Procedure:

1. Design the circuit board using Fritzing. (Instructions and tutorials for Fritzing are found on the organization's website, http://www.fritzing.org.)

2. Create a set of Gerber files by exporting for production. The menu path is:

    File -> Export -> for Production -> Extended Gerber (RS-274X…)

A set of nine files will be created. For the IR Navigation board, the files are:

```
IRNavigationVerPt6DoubleSidedGroundFill_contour.gm1
IRNavigationVerPt6DoubleSidedGroundFill_copperBottom.gbl
IRNavigationVerPt6DoubleSidedGroundFill_copperTop.gtl
IRNavigationVerPt6DoubleSidedGroundFill_maskBottom.gbs
IRNavigationVerPt6DoubleSidedGroundFill_maskTop.gts
IRNavigationVerPt6DoubleSidedGroundFill_silkBottom.gbo
IRNavigationVerPt6DoubleSidedGroundFill_silkTop.gto
IRNavigationVerPt6DoubleSidedGroundFill_drill.txt
IRNavigationVerPt6DoubleSidedGroundFill_pnp.txt
```

This listing was taken from a Macintosh computer. The listing on a Windows computer will be similar, but the .txt file suffix will be missing from the _drill file.

3. Look for the file that ends with _drill. On a Macintosh it will also have the suffix .txt, indicating it is a text file. Change this suffix to .xln. Thus, in this example,

    IRNavigationVerPt6DoubleSidedGroundFill_drill.txt

becomes

    IRNavigationVerPt6DoubleSidedGroundFill_drill.xln

For a Macintosh computer, simply editing the file name is sufficient.

On Windows computers, the .txt suffix exists but is hidden from the user. (The file would be named IRNavigationVerPt6DoubleSidedGroundFill_drill.) Windows computer users then will open the file with a good text editor, such as Notepad++. Then save the file as a *.* type and add the file extension .xln to the name. This will result in two files ending in _drill. Delete the one without the .xln suffix.

4. Locate the file that specifies the top silk screen. It will end with _silkTop, followed by the suffix .gto. Make a copy of this file but change the suffix to .gko. This will result in two files for the top silk screen, one with the suffix .gto and the other with the suffix .gko.

5.  <u>OSH Park</u> will not use all the files created by the Fritzing software. Zip the following Fritzing files into a single file:

    ```
    boardname.GTL
    ```

    ```
    boardname.GBL
    ```

    ```
    boardname.GTS
    ```

    ```
    boardname.GBS
    ```

    ```
    boardname.GTO
    ```

    ```
    boardname.GBO
    ```

    ```
    boardname.GKO
    ```

    ```
    boardname.G2L - only if a four-layer board
    ```

    ```
    boardname.G3L - only if a four-layer board
    ```

    ```
    boardname.XLN
    ```

    In this example the files to be zipped are:

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_copperTop.gtl
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_copperBottom.gbl
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_maskTop.gts
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_maskBottom.gbs
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_silkTop.gto
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_silkBottom.gbo
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_silkTop.gko
    ```

    ```
    IRNavigationVerPt6DoubleSidedGroundFill_drill.xln
    ```

The zip file can be given any valid name. In the example, the name is:

```
IRNavigationVerPt6DoubleSidedGroundFill.zip
```

6.  Go to <u>http://www.oshpark.com/</u>, click the [Get Started Now] button, and follow the instructions.