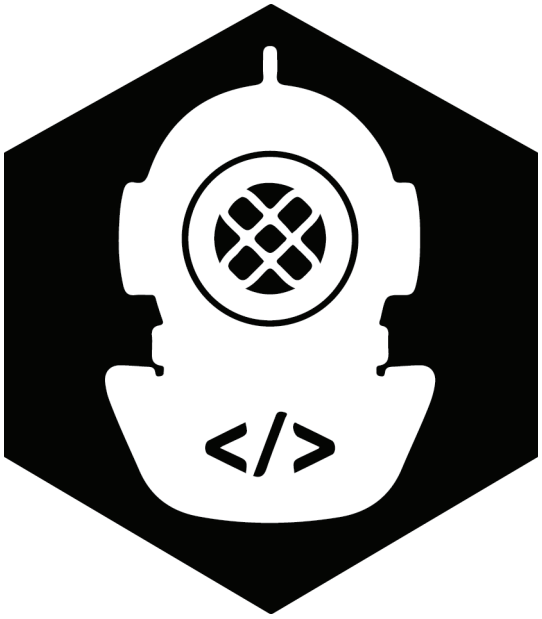
A Starter Guide to Creating Fritzing Parts



Internet of Things

Deep Dive Coding Boot Camps

Powered by CNM Ingenuity

Summer 2020 V. 1.0

(This Page is Intentionally Left Blank)

A Starter Guide to Creating Fritzing Parts

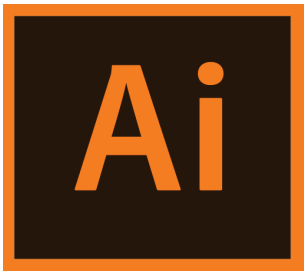
Introduction

Fritzing is a fantastic tool used to create breadboards, schematic and PCB layouts for electronic devices. The software is robust and contains many preloaded sensors, microcontrollers and parts; however from time to time the needed part is not loaded. There are many contributors who share the parts online, but more often than not there is a need for a new part to be created. This can be a complicated process and but here are a few tricks and tips to help make this process easier.

Software Needed

Fritzing is the main software that houses the elements needed to create new parts. Within Fritzing a part can be either modified or created from a template. Each part contains 3-4 different .SVG files all representing a different view. Breadboard, icon, schematic, and PCB; however icon and breadboard are often the same image. It is recommended to read through the learning section of the Fritzing website, to gain a better understanding of the tools and concepts needed to properly created or modify parts that will render correctly for future prototypes.

https://fritzing.org/learning/



Adobe Illustrator is graphic design software used to create or alter art files. It is part of a larger group of programs called the Adobe Creative Cloud to form a one stop shop for creative professionals. It is a paid service that charges monthly or annualy to gain access to the entire cloud of programs.

https://www.adobe.com/products/illustrator.html?promoid=P-GRQQLFS&mv=other

Corel Draw is also a paid creative design software that could also be use to work with the art files needed to complete Fritzing parts. Similar to Adobe, Corel offers a suite of programs many designers use to create professional projects.

https://www.coreldraw.com/en/

Inkscape is freeware that offers the basic components needed to create and alter the files incorporated to any Fritzing part. It relies on its community to contribute and donate to make the software better.

https://inkscape.org

One of the last three aforementioned software programs is required to properly create or alter any part in Fritzing. Unfortunately a simple graphic editing programing like MS paint will not work. Fritzing parts contain many layers, some hidden and a program is needed to access those embedded layers. MS paint or something similar does not have those capabilities. It is suggested to work with a freeware program like Inkscape to start and if it is decided that a higher end program is needed then compare and invest in the paid program that is right for the user.

Creating a Simple Fritzing Part

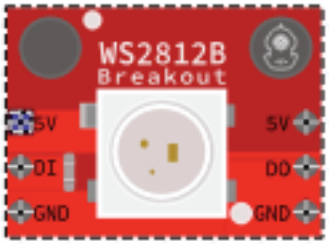
In order to properly create Fritzing part it is important to understand the different files that are used to illustrate each part. Fritzing will allow the prototype project to be viewable in Breadboard, Schematic and PCB views.

Breadboard View: The most realistic view of all the components are used and laid out in a breadboard as they would be fitted in reality.

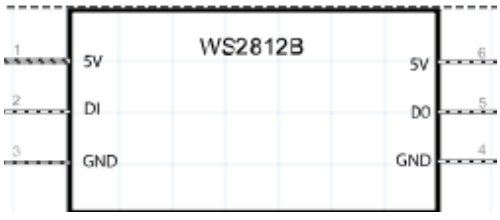
Schematic View: This view is focused on the connections made through the wiring of the components. The parts are represents as simple outlines and are arranged to best illustrate the wiring.

PCB View: Allows the user to plan how a circuit board would be printed. This view shows basic outlines of the components and indicates where the copper circuits would be soldered.

Each layout works in tandem with each other and needs to be show a accurate representation of part in size and spacing. All files are in .SVG format.



Breadboard View

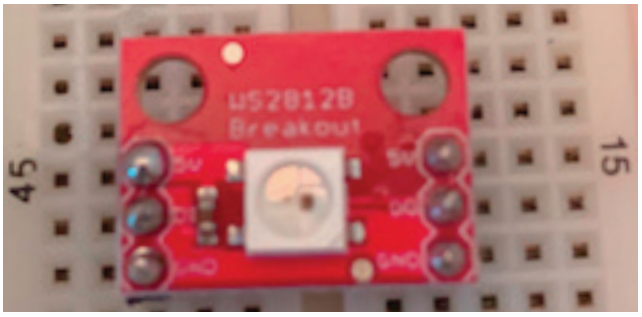


Schematic View

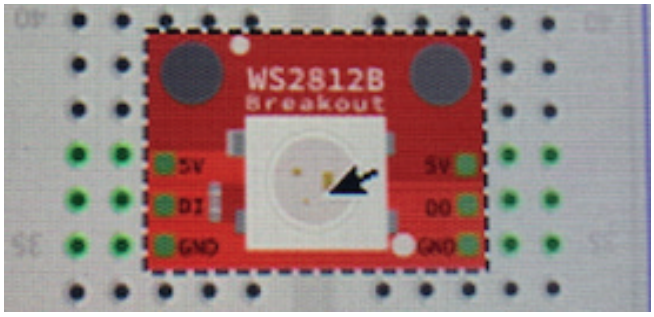
PCB View

Designing a New Fritzing Part

When planning to create a breadboard Fritzing part, use a photo of the component in which the model will be rendered after. In the graphic design software, it is good practice to also have a guide layer of a blank Fritzing breadboard image to match the size of the component in reality. This will help to model the image to the right size straight away. In the example here is a side by side comparison of a modeled component which is a bit too large compared to the real chip.

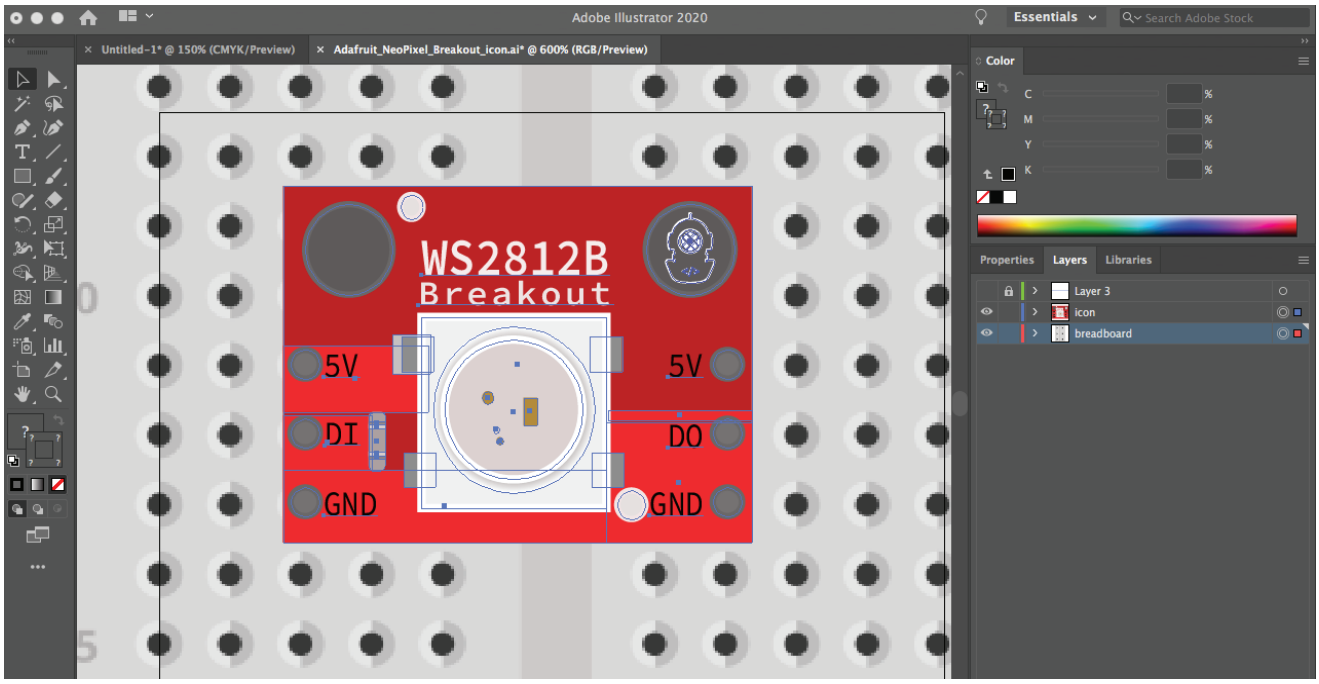


Adafruit WS2812B Neopixel



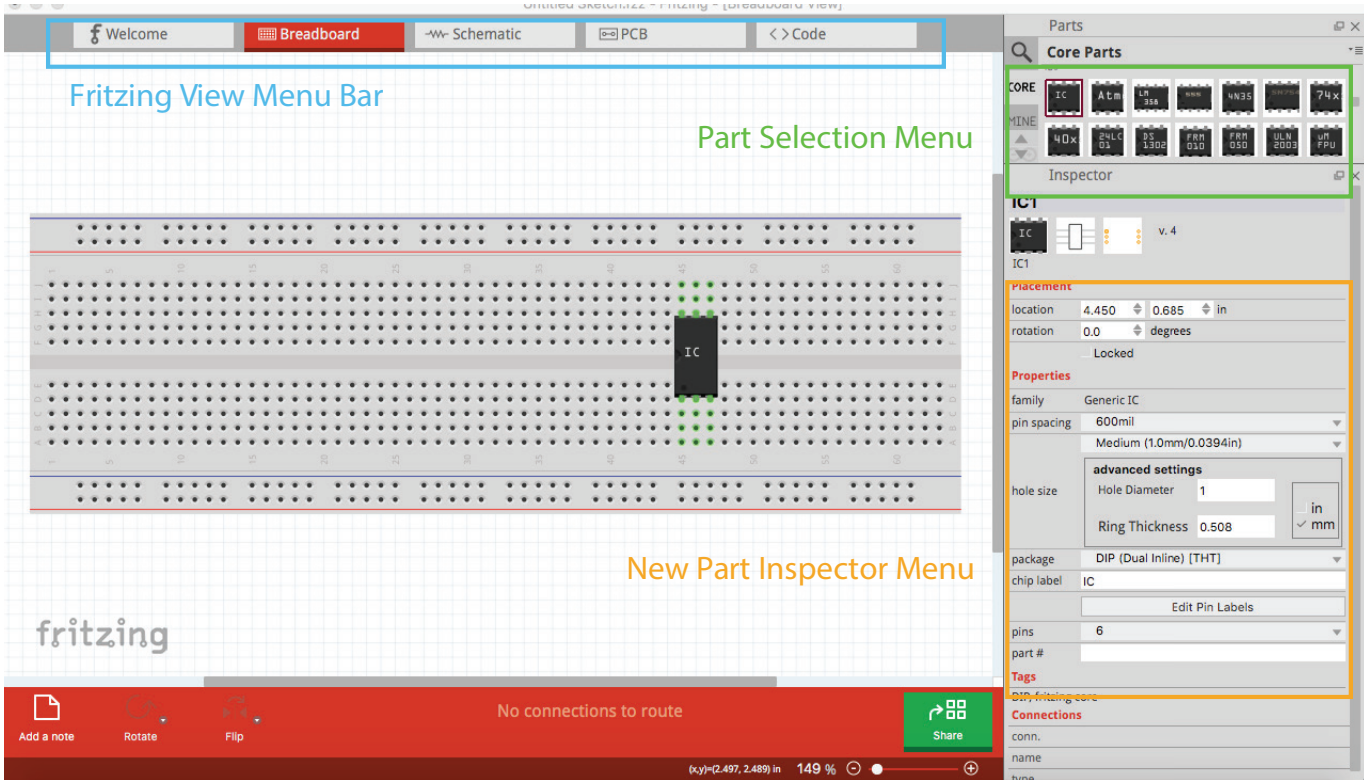
Improperly Sized Rendered Component

Comparing the two images above the Fritzing chip is one row too big. It is these small details that will cause huge issues if used in a prototype project. Therefore it is important to understand the need to model to scale. However, due to the fact that when capturing the Fritzing breadboard will import into the designer program large, it will be necessary scale the image down to a workable size. Once the render is complete, hide the breadboard layer and any guide layers, scale the art board around the component, and export the file as a non responsive .SVG file. Keep the working graphic file open as sizing adjustments may be needed after loading the file into Fritzing.



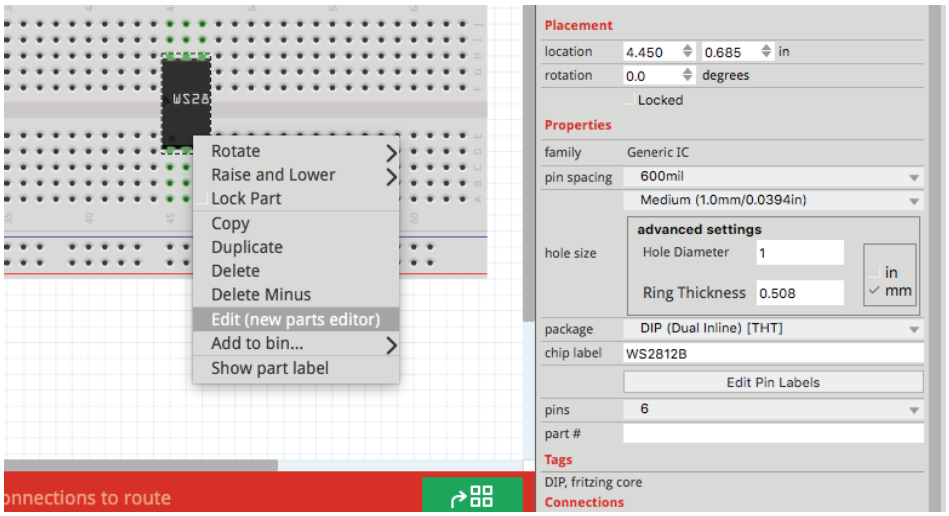
Screenshot of Working Adobe Illustrator File

Once the part image is rendered in as workable .SVG file, open a new Fritzing project. In the Breadboard View use the Part Selection Menu to chose the type of component that is going to be created. The Adafruit WS2812B is a IC chip. Drag that part onto the breadboard to start to alter the template part. In the New Part Inspector Menu basic part data can be changed. A new name can be given to the part, along with editing the name of the pins to correspond with the pin labels on the chip model. This information is also able to be changed later if needed; however be sure to select the proper pin distance and number of pins before moving on.



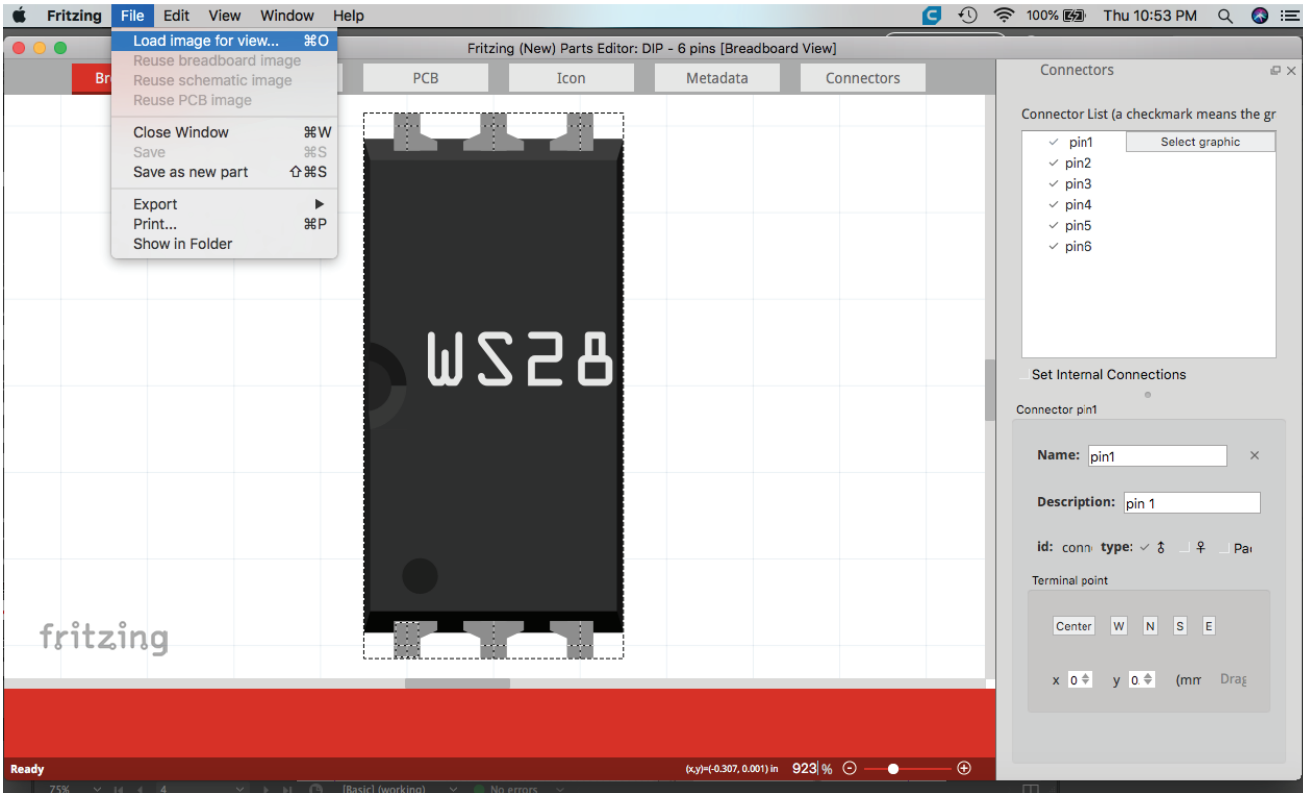
Breadboard View of a New Project in Fritzing

After selecting the proper basic setting right click on the template part and select New Parts Editor.

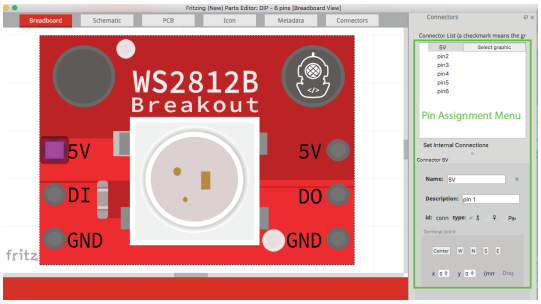


New Part Editor Selection

In the New Parts Editor, use the FILE option to load the .SVG file that was created.
FILE => LOAD IMAGE FOR VIEW



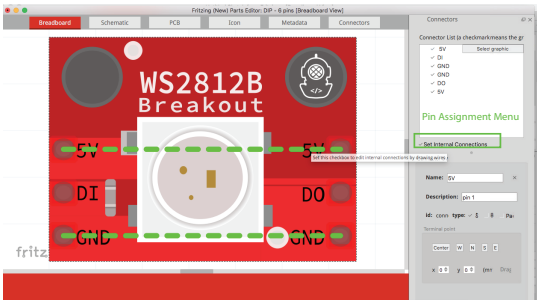
New Parts Editor in Fritzing



New Parts Editor Pin Assignment

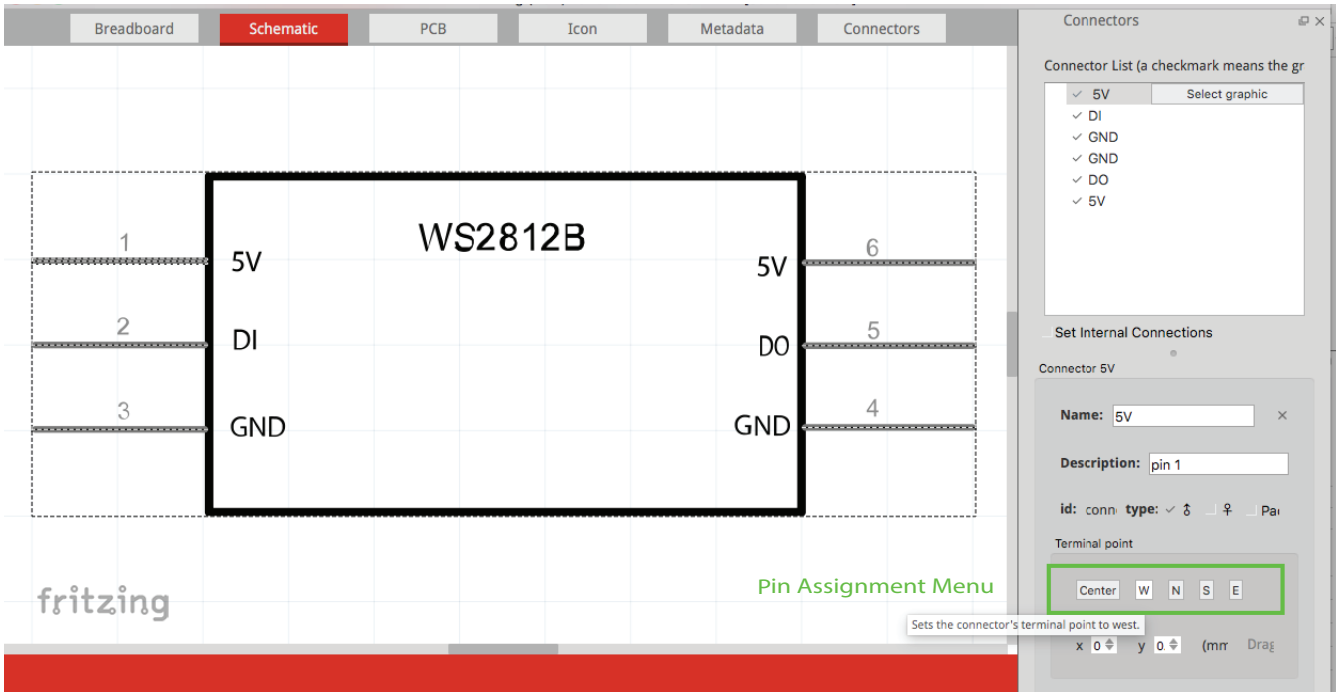
Once the New Parts Editor loads the .SVG file that was created, the pins needs to be labeled and assigned in association to the image. After naming a pin, click away for the change to take effect. Re-click the newly named pin and press the SELECT GRAPHIC button. From there any section of the image can be clicked to assign the pin.

Because this component has two sets of connections that are the same, internal points of connection need to be made. In order to do this check the SET INTERNAL CONNECTION box and click and drag the two connections that are the same. This will ensure that when the part is in use both pins will become active when one pin is activated.



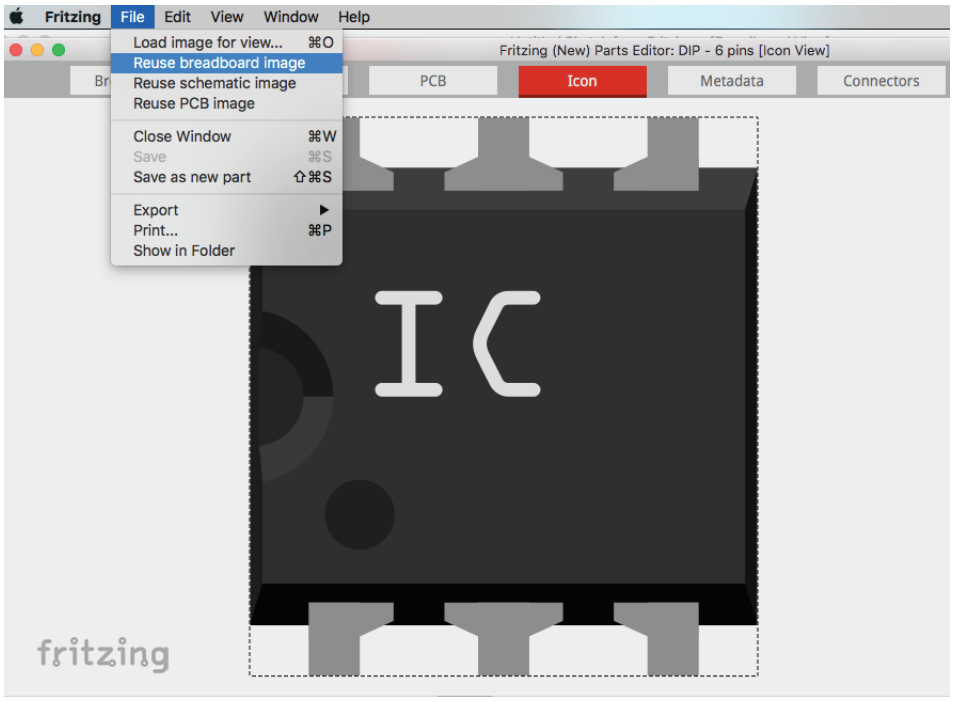
New Parts Editor Internal Connection Assignment

Moving on to the Schematic View. Set the terminal point for each pin. The pins on the left (1,2 & 3) have west terminal points, the pins on the right (4,5 & 6) have east terminal points.



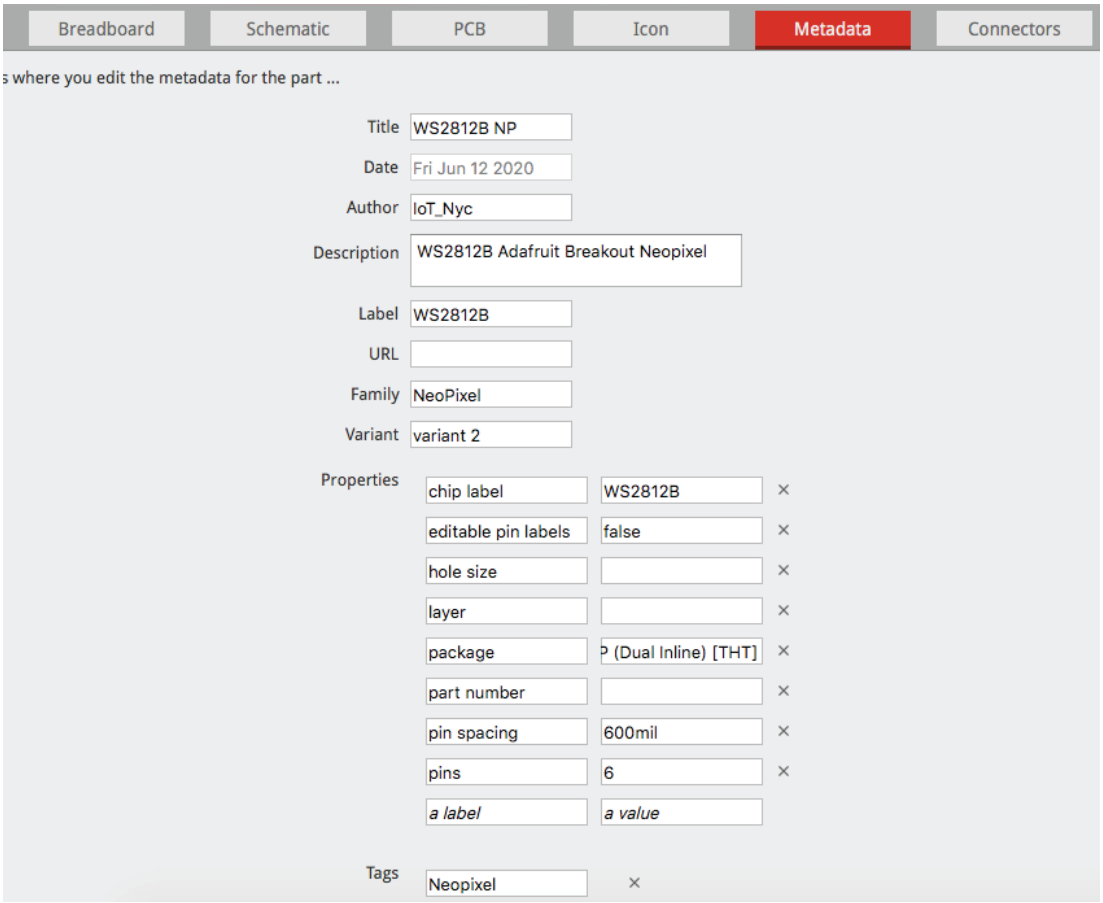
New Part Editor Schematic View Terminal Point Selector

Switching to the Icon View reload the .SVG file that was used in the Breadboard View.
FILE => REUSE BREADBOARD IMAGE



New Part Editor Icon View

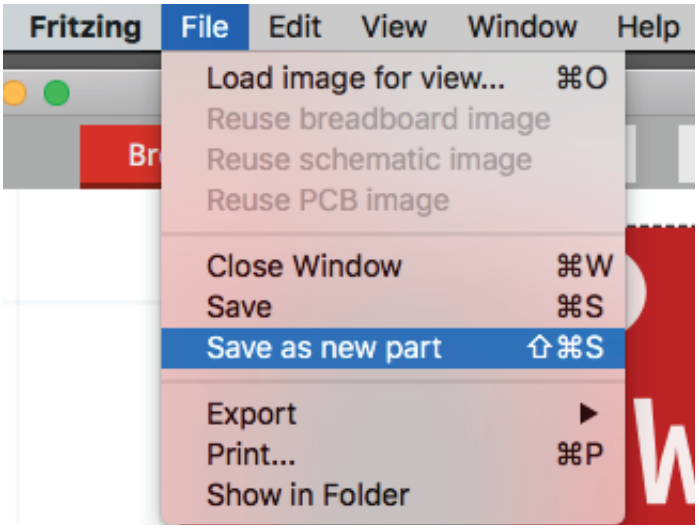
The Metadata View is where the index data for the new part will be edited. The more information that can be inputed the better the new component will be cataloged.



New Part Editor Metadata View

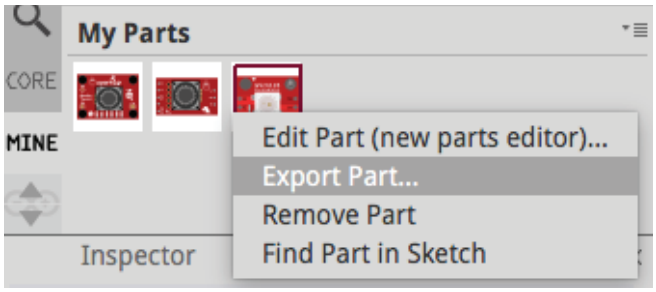
When the metadata information is inputed, and all aspects of the part are completed, save the component as a new part.

FILE => SAVE AS NEW PART

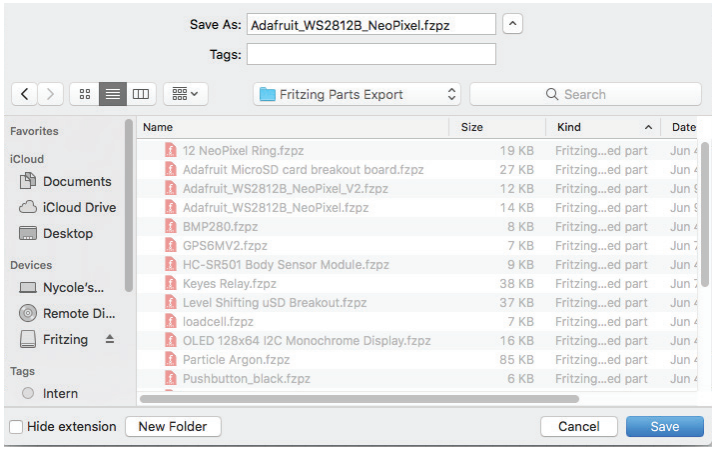


New Part Editor File Menu Option

Once the new part is saved, it will appear in the MY PARTS bin. From there the part can be exported to be shared by right clicking on the part and selecting EXPORT PART. Choose the location in which you would like the save the file. Be sure that the file saves with the .FZPZ extension. This type of file will include all embedded files necessary for the file to be shared.



Part Selection Menu My Parts



Part Selection Menu My Parts