

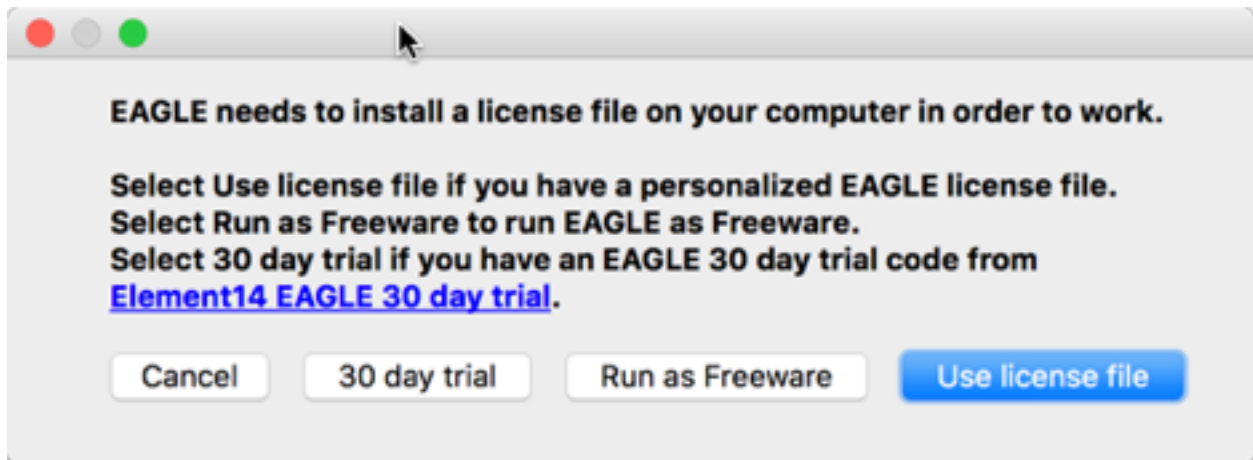
Eagle Set Up and Board

Malcolm Knapp

9/19/16

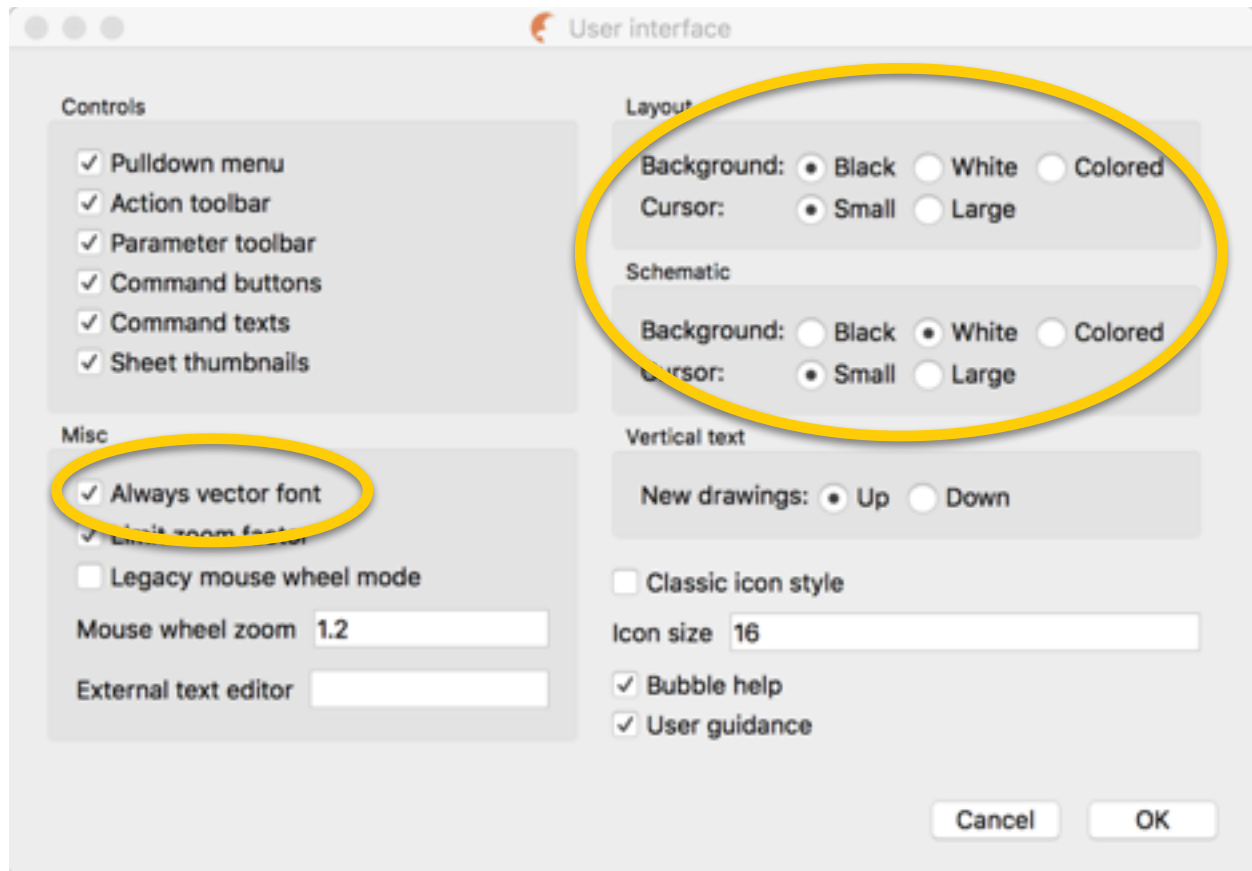
Eagle Set Up

1. Create three folders in the location where you want to store your projects. This provides a simple way to know where all Eagle related files are.
 - 1.1. Eagle Projects
 - 1.2. Eagle Scripts
 - 1.3. Eagle Libraries
2. Move 1-Generics.lbr file from Eagle folder in your Github repository and place it in the Eagle Libraries folder
3. Navigate to <https://github.com/sparkfun/SparkFun-Eagle-Libraries> and download the libraries as a ZIP file.
4. Unzip the file and rename it "SparkFun-Eagle-Libraries".
5. Place the SparkFun-Eagle-Libraries folder in the Eagle Projects folder
6. Open Eagle
7. A Dialog box will appear asking to install a license file
8. Select Run as Freeware



9. In the Control Panel Select Options —> Directories...
10. A dialog box will open that shows the paths that Eagle looks in. Different directories are separated by a ":".
 - 10.1. Remove the \$EAGLEDIR/lib: directory from the path. This simplifies the part search when you are designing schematics. You can add always back in libraries one at a time if you need them later.
 - 10.2. Select the Libraries text box and click the Browse... button
 - 10.3. Navigate to the location Eagle Libraries and select Choose.
 - 10.4. The path to that folder should appear in that text box. Eagle will now include that path where it looks for library files.

- 10.5. Add the paths to the Eagle Scripts folder to the User Language Programs and Scripts text box in the same way
- 10.6. Add the path to the Eagle Projects folder to the Projects text box in the same way.
- 10.7. Click OK to close the dialog box
- 10.8. Contract and expand the Projects, Libraries, User Language Programs, and Scripts arrows. The folders you just added should appear in the expanded list. If they do not check to make sure the path to them is correct.
11. Open Options —> User Interface...
 - 11.1. In the Misc box, check the “Always Vector Font” box if it not already selected. This will make sure your text is printed correctly when it is fabricated.
 - 11.2. In the Layout and Schematic set the Background to the color you like. The setting I use are Layout background is black and Schematic background is white



Project Work Flow

1. Create blank library and schematic
2. Copy in generic parts to library
3. Update IC blanks with component signal names and footprints
4. Schematic Capture

5. Design for Test
6. Clear the ERC
7. Board Layout
8. Clear the DRC
9. Generate gerbers using CAM

Creating a New Project

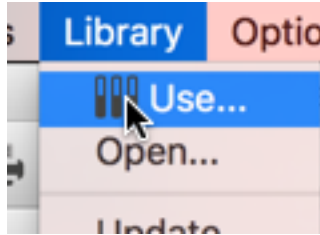
1. In the Control Panel expand the Projects arrow and then expand the Eagle Projects Arrow.
2. Right click on the Eagle Projects Folder and select New Project. An empty project folder will appear.
3. Rename that folder to the Project Name
4. Right click on that new project folder and Select New Schematic
5. A new blank Schematic will open
6. Save the Schematic with the project name
7. Right click on that new project folder and Select New Library
8. A new blank Library will open
9. Save the Library with the project name

Part Management

Eagle gives you a very large number of default libraries that have thousands of parts in them. However, the search functionality is not very good so it is virtually impossible to find and parts in these libraries. This means that they are basically unusable. To work around this issue I use a Generic parts library and a project library. The Generic library contains all the parts you will always you like Resistors, Capacitors, rail symbols GND symbols etc as well as blanks fro IC. These blanks speed up the process of creating the symbols and footprints for the specific ICs in your design. The project library contains all the parts need for that specific project. The method for creating and populating a project library are shown below. This structure makes it very easy to find your parts and prevents your parts libraries from becoming unwieldy as you do more designs. NOTE: The “1-...” in front of Generic guarantees that this library will be at the top of the list when you are adding components.

Project Library

1. Open the blank project library
2. In the Control Panel expand the Library arrow and then expand the 1-Generic Library.
3. Right click on the Frame that is the right size for your project and select Copy to Library.
4. The part will copy into the to the project library.
5. For ICs and Connectors copy in the IC# blank or M# that corresponds to the number of pins you need
6. Add all the other parts that you will use or adapt. You can also look in the Sparkfun libraries for parts that you can use.
7. In the Schematic window Select Library —> Use



8. Navigate to the your project director and select the library you just created.
9. Select Open. The library will now show up in the Add Component selection window.
10. To confirm this Click the Add button. Confirm that you can see your project library in the selection window. If you can not go back to step 7 and try again.

Project Specific Parts

For all components that do not show up in the Generic library a new part must be created. For a tutorial on creating new parts in Eagle see:

<https://www.sparkfun.com/tutorials/110>

and

<https://learn.sparkfun.com/tutorials/designing-pcbs-smd-footprints>

Adapting an IC blank

1. Rename the symbol to the component part number
2. Rename the pins to the signals for that component
3. Update the footprint name to match the package type
4. Update the footprint pads to match the package.

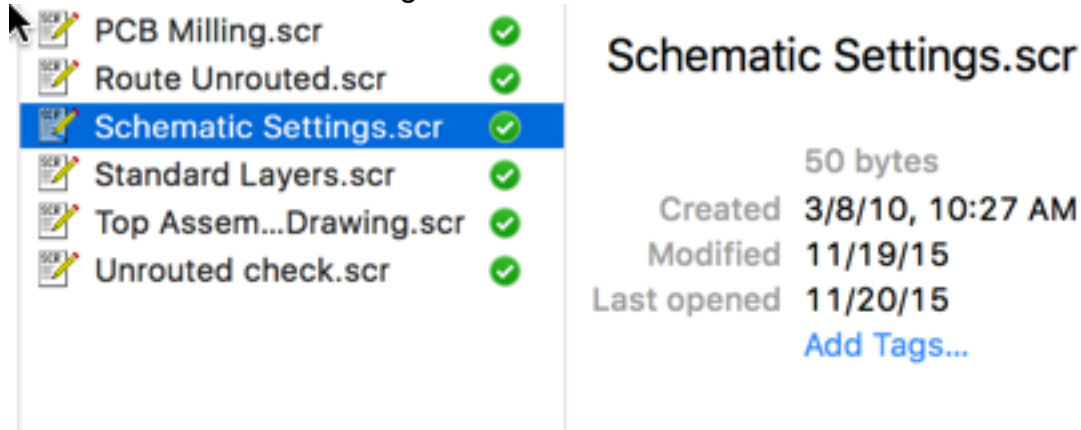
Schematic Setup

1. Click the Execute Script button

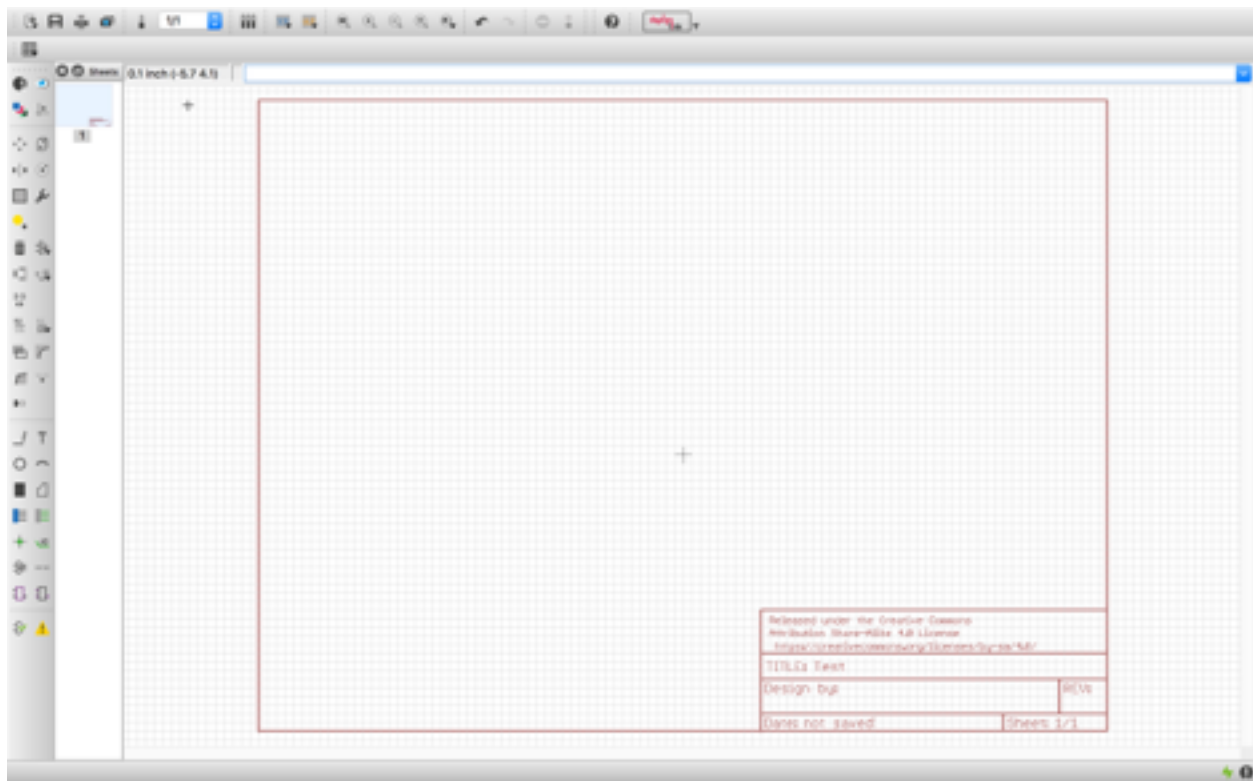


2. Navigate to Eagle Scripts folder

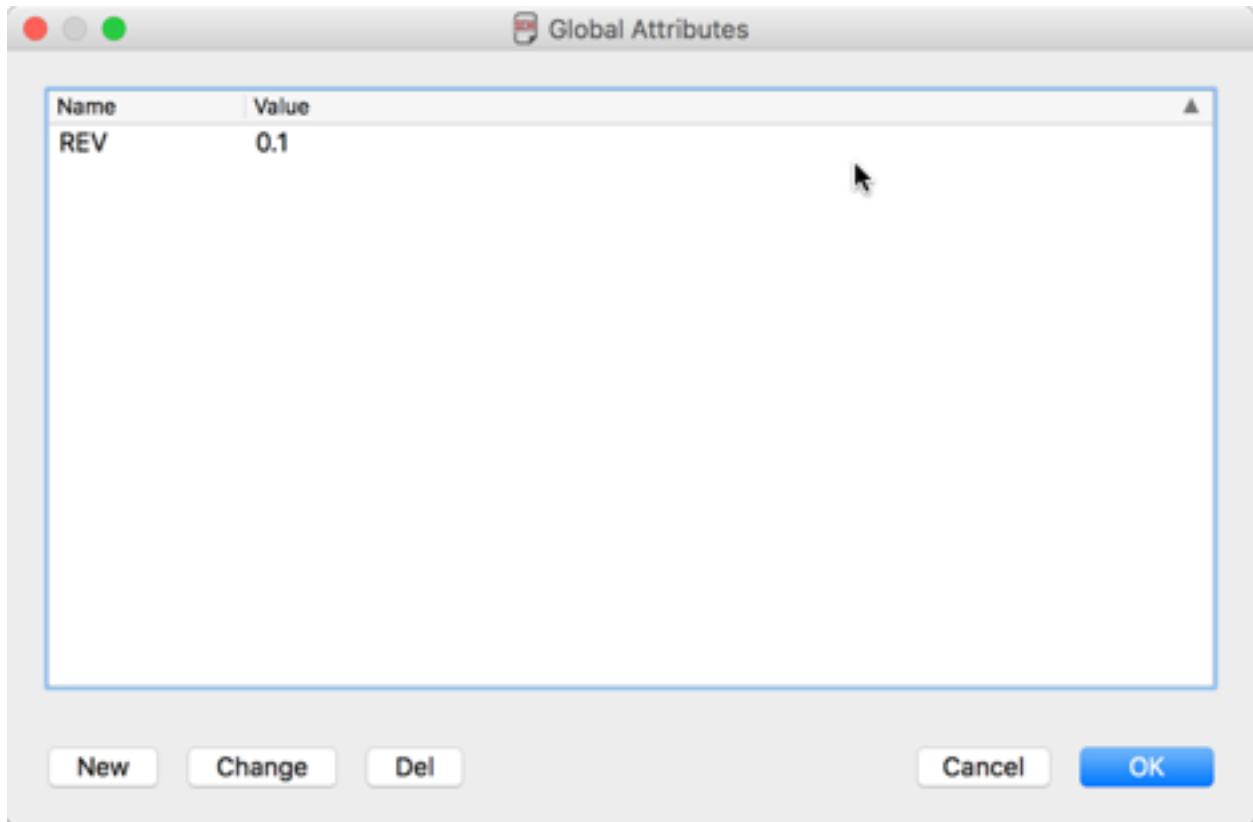
3. Select the Schematic Setting.scr and run it



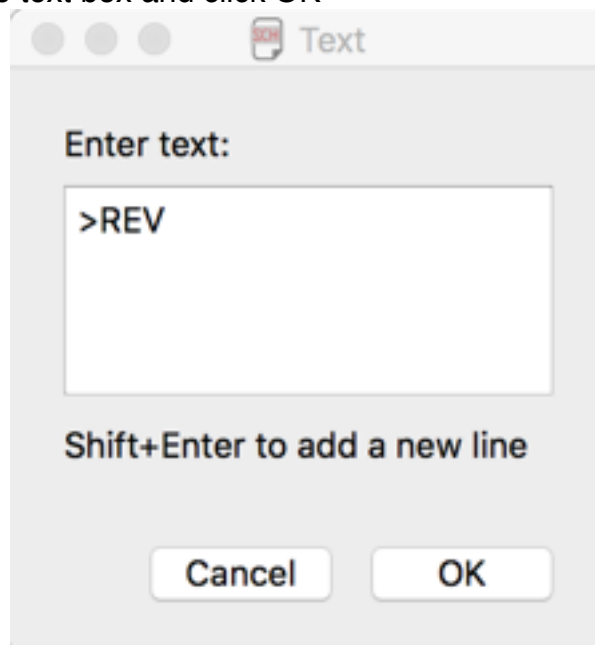
4. The grid will appear and the user interface will be set up for schematic capture. You can change the setting by editing the Schematic Setting.scr file.
5. Click Add Part and find the frame you put in the project library
6. Click Ok. Place the Frame that appears so the origin if the schematic is in the middle.



7. Select Edit —> Global attributes...
8. A dialog box will appear. Select New
9. Another dialog box will appear
10. Enter "REV" in the Name text box and "0.1" in the Value text box



11. Click OK and then OK again.
12. Click the Text button
13. Enter ">REV" in the text box and click OK



14. A "0.1" will appear attached to the cursor. The ">" acts as a marker to look for the same text in the Global attributes. If it is found then the text is replaced by whatever

Value is set to. This is a good way to make changes if the text appears in multiple places.



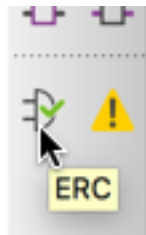
Schematic Capture

For a tutorial on creating schematics in Eagle see:

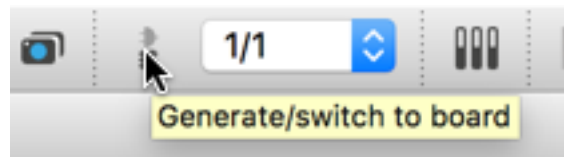
<https://learn.sparkfun.com/tutorials/using-eagle-schematic>

Schematic Closure

1. Click the ERC button to run the Electric rules check



2. Fix any errors that are reported and rerun the ERC until you get no errors.
3. Click on the Switch/Generate Board button

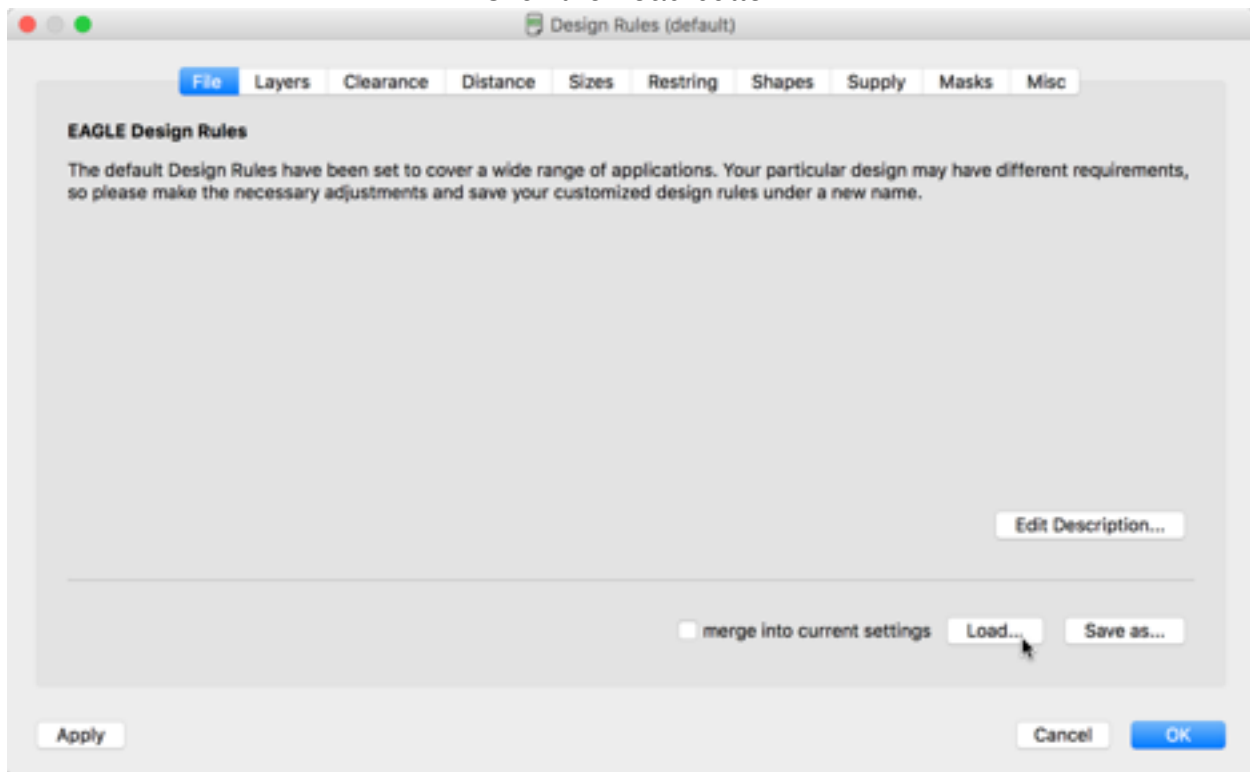


4. The Layout window will open

Layout Setup

1. Click the Execute Script button
2. Navigate to Eagle Scripts folder
3. Select the Layout Setting.scr and run it
4. The grid will appear and units will be set to mils and the user interface will be set up for layout. You can change the setting by editing the Layout Setting.scr file.
5. Select Edit —> Design Rules...

Click the Load button



6. Navigate to where EA OthermillDRC_30_deg_v2.dru is and click open.
7. The design rules for using the 30° v-bit will be loaded into the design rules.

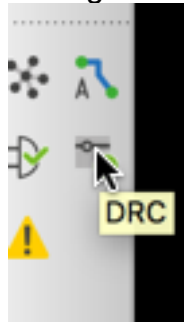
Board Layout

For a tutorial on creating layouts in Eagle see:

<https://learn.sparkfun.com/tutorials/using-eagle-board-layout>

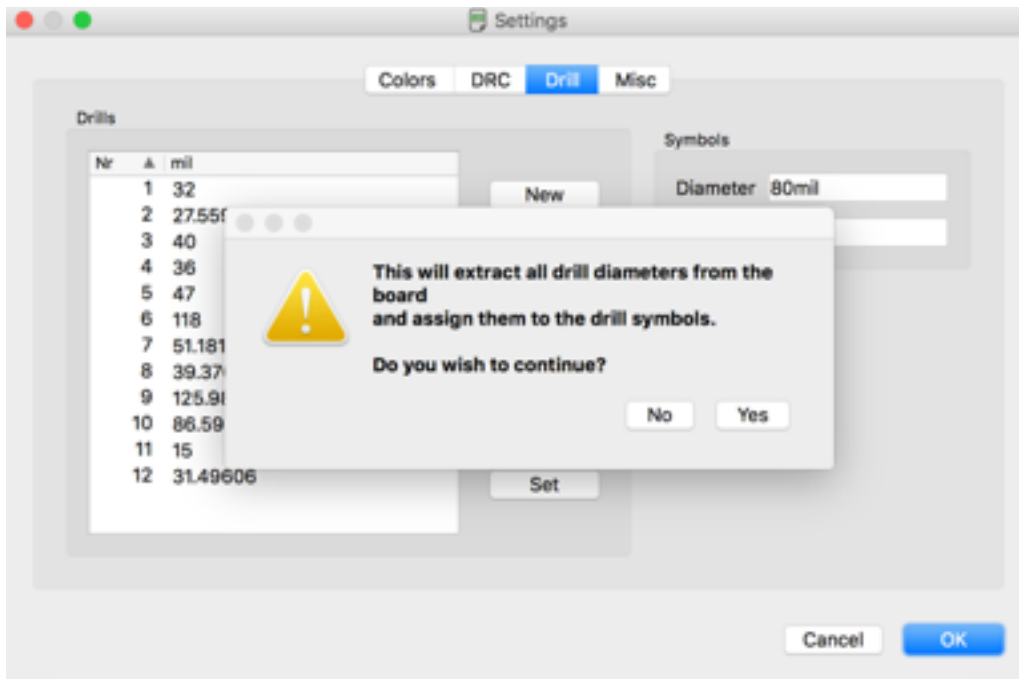
Layout Closure

8. Click ok the DRC button. The DRC dialog box will appear

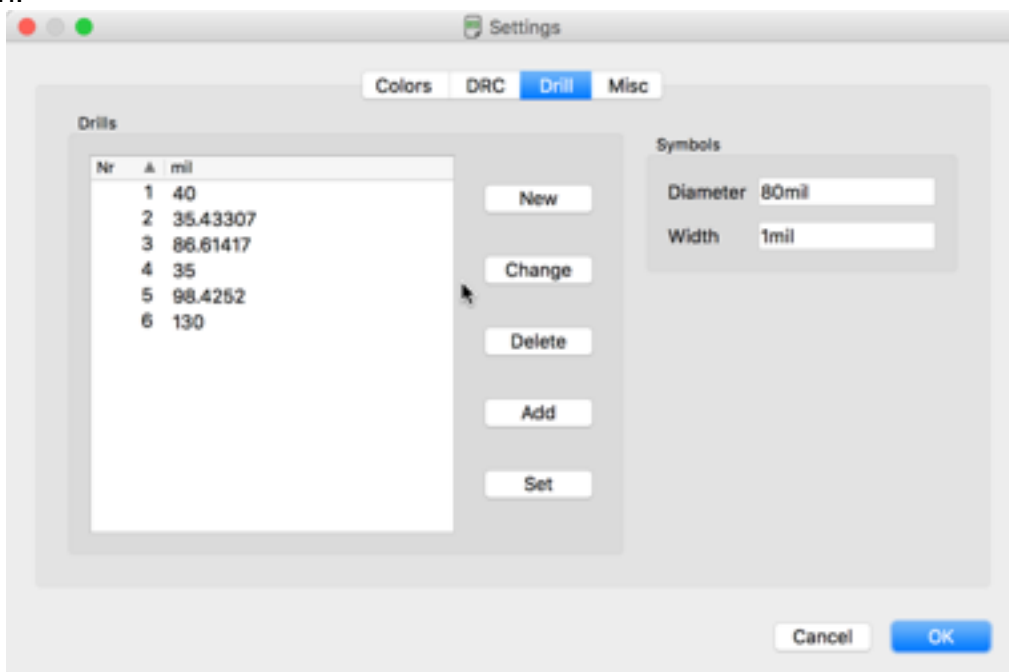


9. Click the Check button
10. The board will be checked for design rule violations. Many, such as Stop Mask errors, can be ignore. Mark those as Approved.

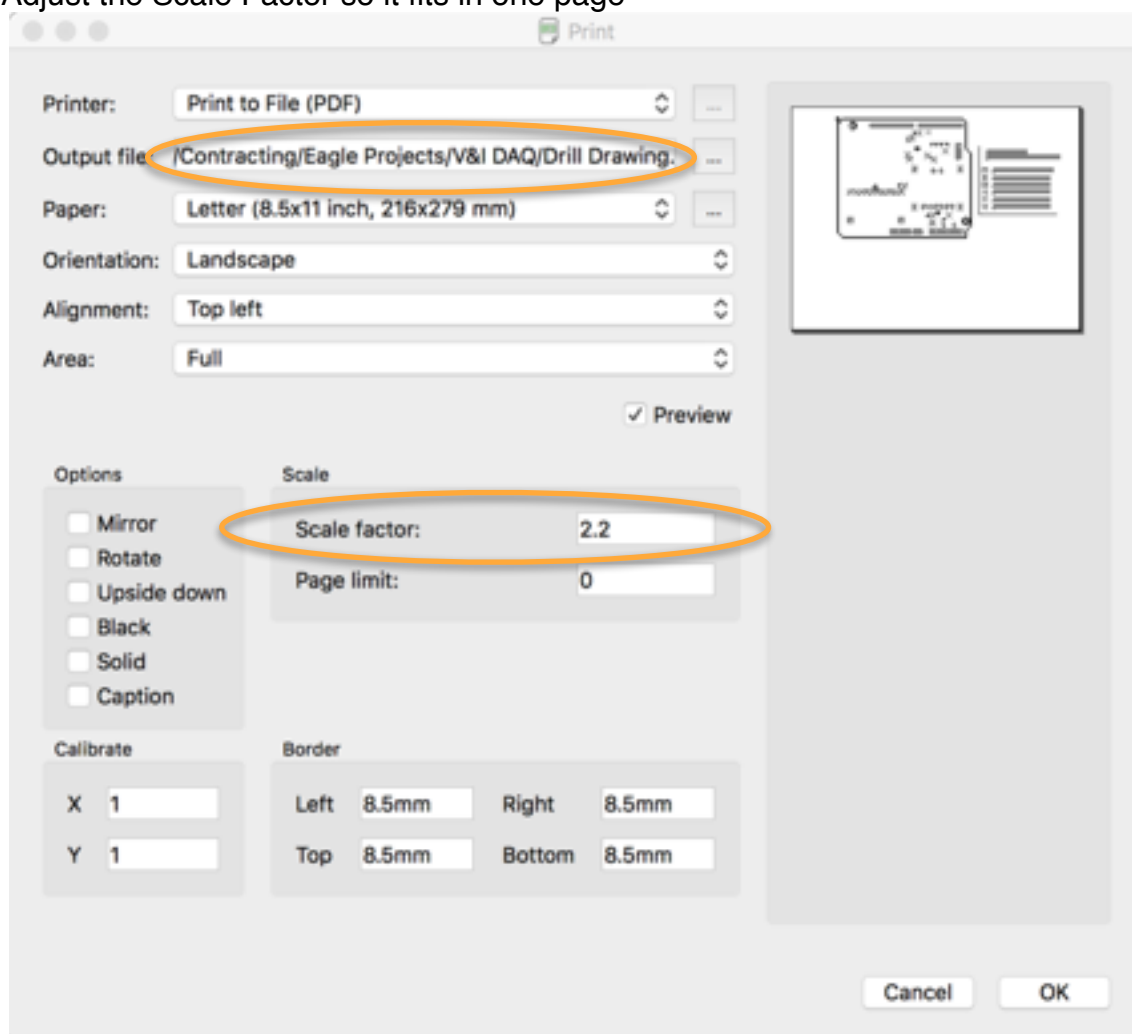
11. For Clearance, Drill, Overlap errors move the components or traces to make the error disappear.
12. Cycle through steps 8-11 until all errors are either fixed or marked as Approved.
13. Click on Options—>Set... and select the Drill tab
14. A dialog box will appear. Click the Set button. A dialog box will appear.



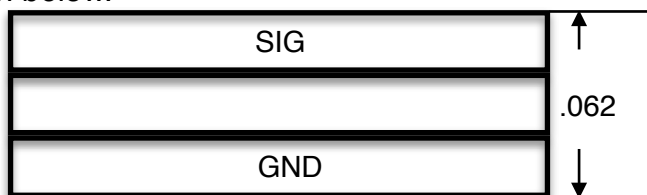
15. Click Yes and the Drills list will update for this design. If you see drills that are close in size you can match the sizes if you want and the go through steps 12 and 13 again.



16. Select the Execute ULP button and run drill-legend.ulp. The drill tables and drill locations will appear.
17. Run the Drill Drawing.scr. This hides all the layers except for the drill drawing layer and the board out line.
18. Move the drill table so that it looks clean
19. Select Print
20. Name the file and use the extension .pdf
21. Adjust the Scale Factor so it fits in one page



22. Run the Standard Layers to show all the trace layers and hide the drill drawing
23. Draw the Layer Stack Up by hand. The layer stack in a drawing of the board layers, what each layer is (signal, GND plane, power plane), and how thick the board is. An example is shown below.



24. Click the ULP button



25. Navigate to the Eagle Scripts folder and select fab-notes.ulp. A standard set of fab notes will appear. Move and modify them these as needed.

Export CAM

For a tutorial on exporting CAM in Eagle see:

<https://learn.sparkfun.com/tutorials/using-eagle-board-layout/generating-gerbers>

NOTE use The Othermill's CAM file instead of Sparkfun