Concept Session 3 - EagleCAD
Course Notes

The most widely used PCB design software for amateur constructors.

Eagle is based on three steps:

1. Define "Devices" which consist of schematic diagram symbols, physical packages

2. Capture of schematics, based on libraries of devices

3. Sending off the board file to a PCB shop for manufacture (e.g. to Eurocircuits).

If you want Eagle go to
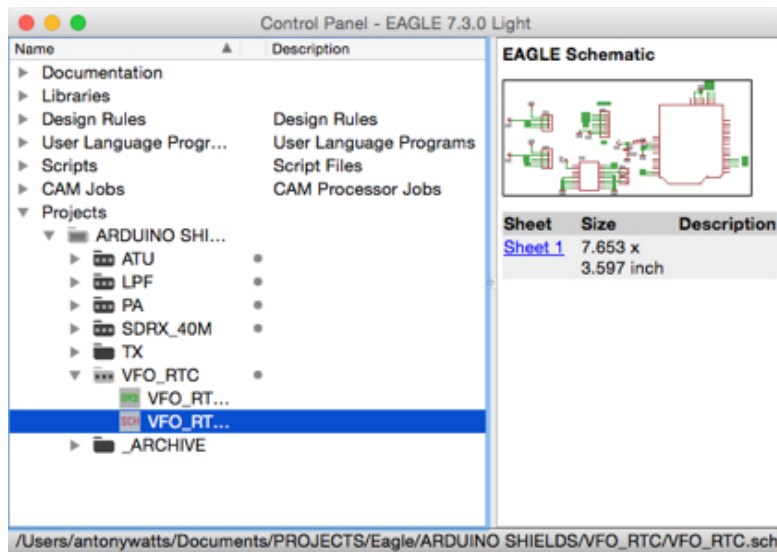
www.cadsoftusa.com/download-eagle/freeware

and I have put a three part course on my web site at

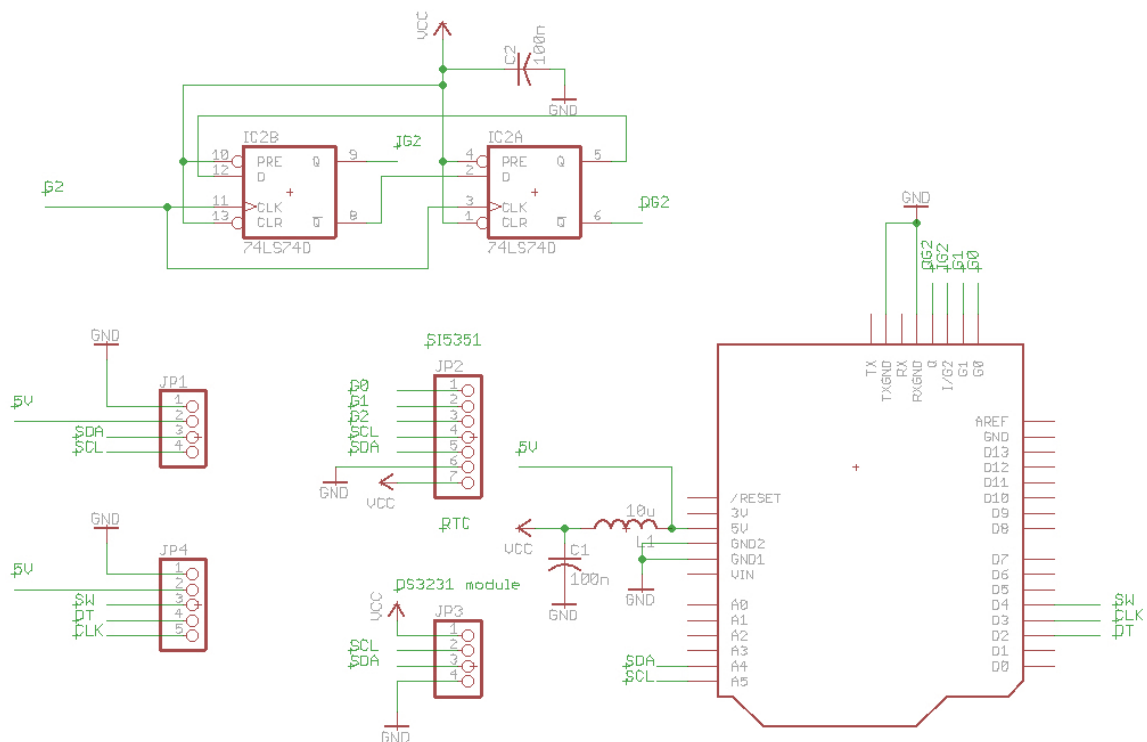www.ganymedeham.blogspot.com

See 2015 February blogs.

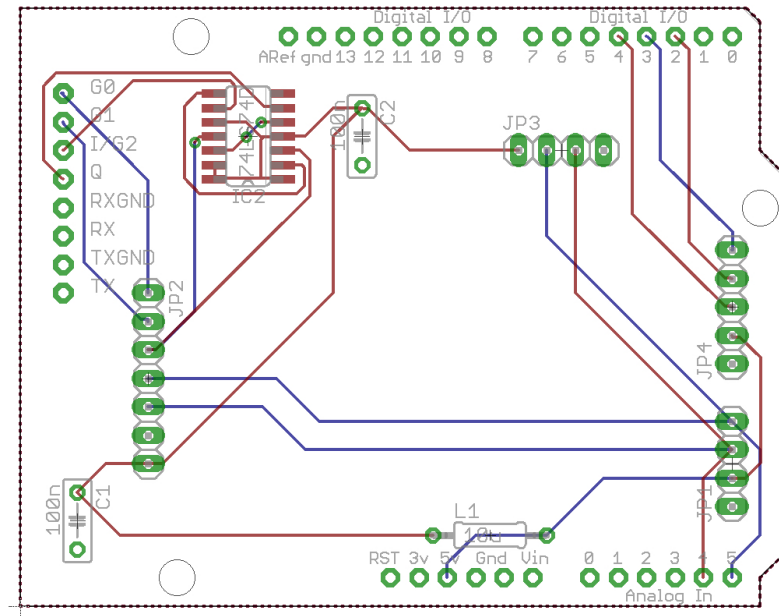## Three steps

Designing with Eagle is done in three steps

From the control panel you can chose Libraries to see those on your machine, you can open libraries and edit or create Devices and you can see your Projects

You can draw schematics using any part in a library.

e.g. the VFO_RTC_IQ board

And you can generate the PCB layout, in auto mode or manually, or both


Eagle levels

Eagle handles levels, in descending order:

1. Schematic ... leading to PCB design

2. Devices - or components (Symbols & Packages)

3. Symbols used on schematics, Packages which are used on PCbs

4. Gates which are individual building block of symbols (e.g. part of a dual op-amp)
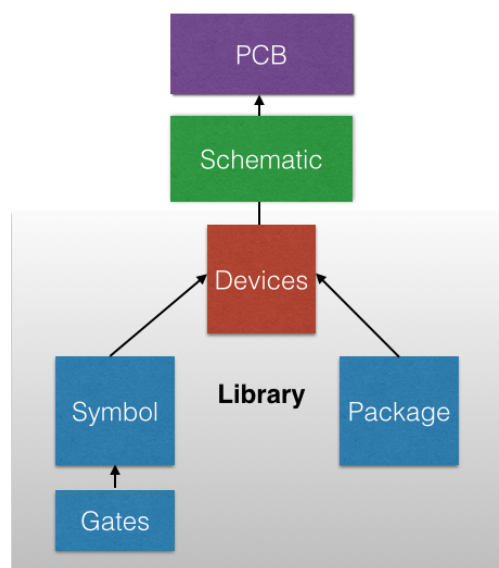
Libraries

Eagle when installed provides a lot of libraries from component manufacturers, often they are confusing as they are listed under makers part numbers. There are some useful generic libraries (e.g. for R, C & L). You can find many more libraries on the web or you can make your own, like this one:
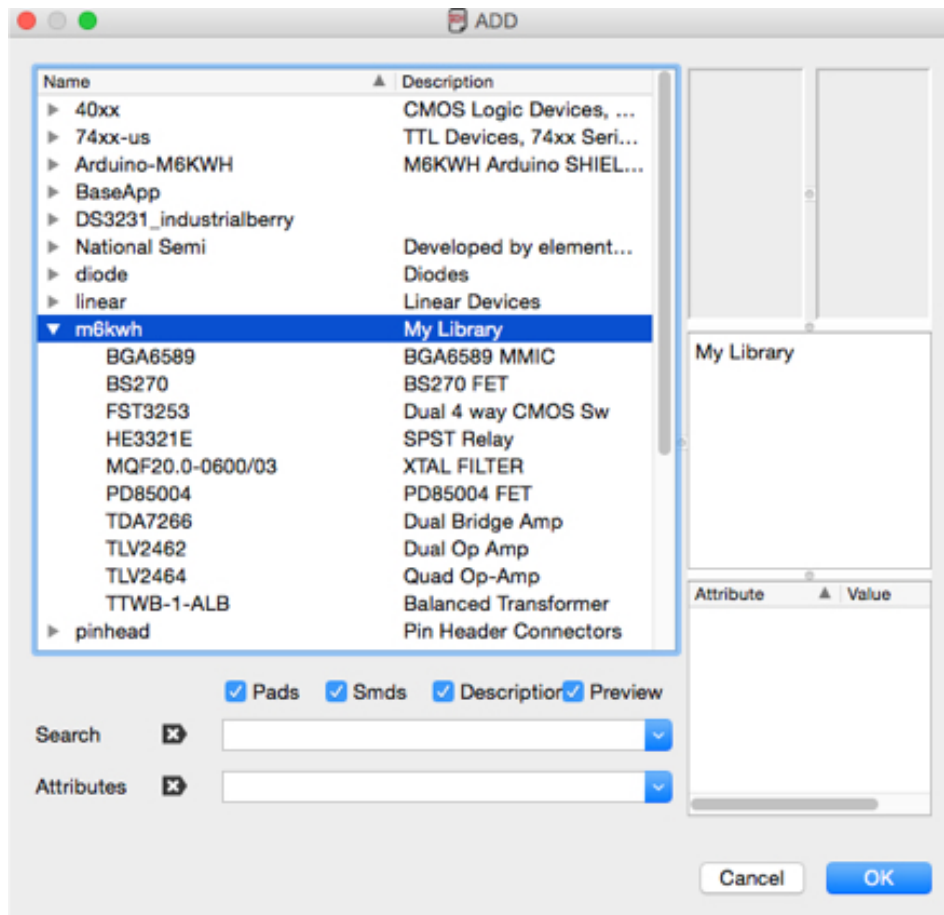
Which is my library for parts I used in the Concept program. I find it is often easier to make a new device yourself then spend time searching the web for a library that contains it...
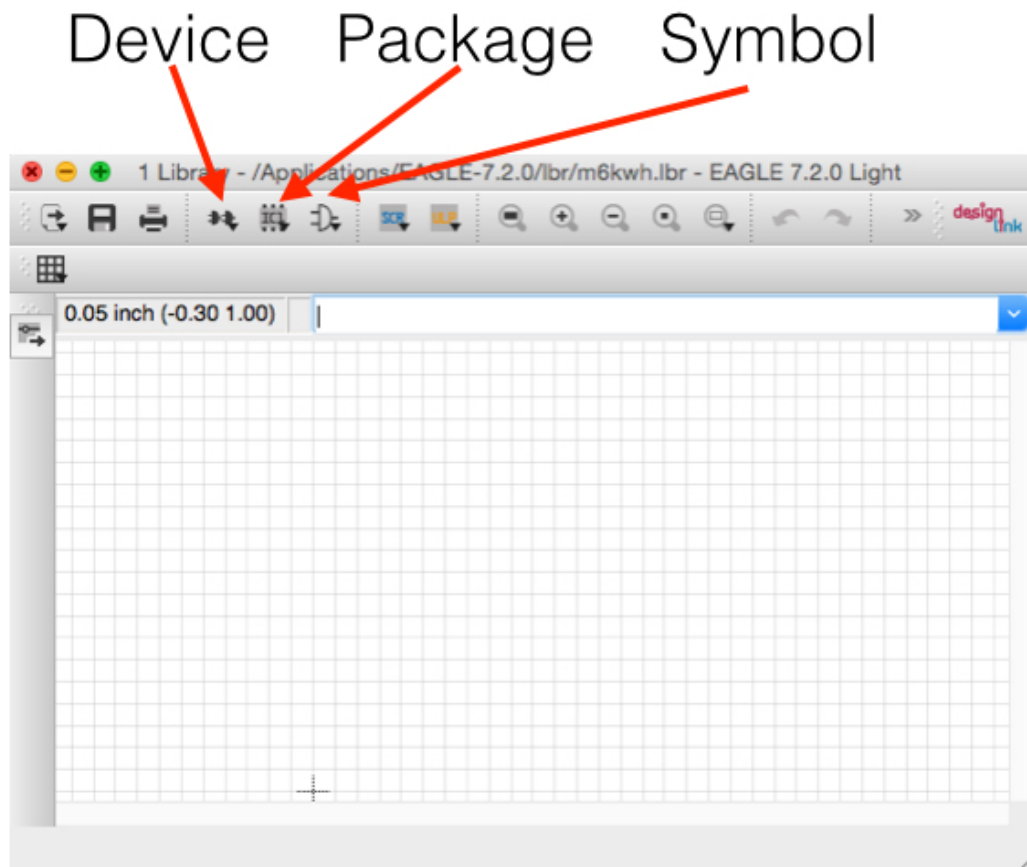
Make a new device

First create a new library for yourself, in which you will keep your devices
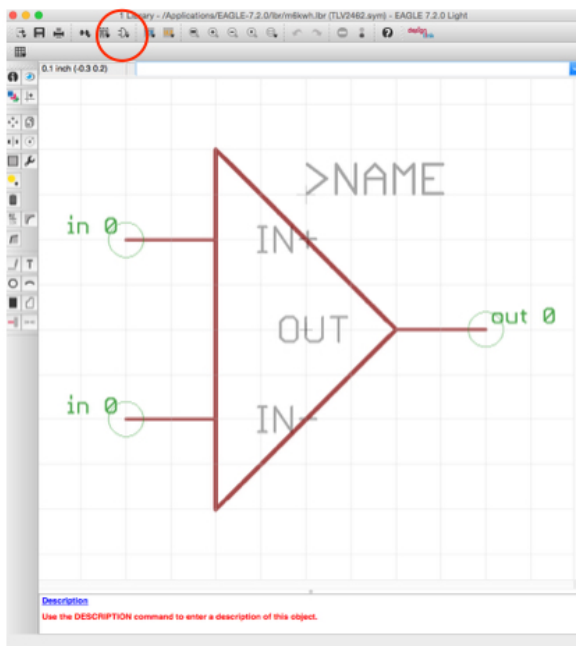
```
Control Panel,  File > New > Library
```

Name it and enter a description
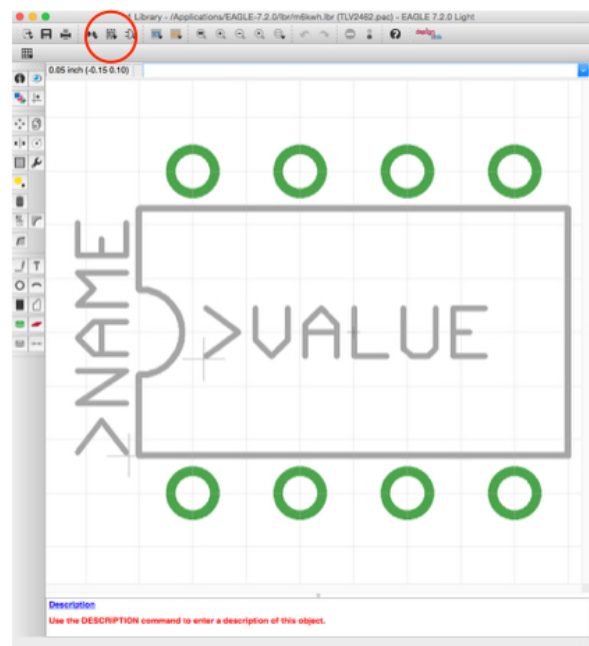
```
File > Save As ...
```

With the library open you will have this window:

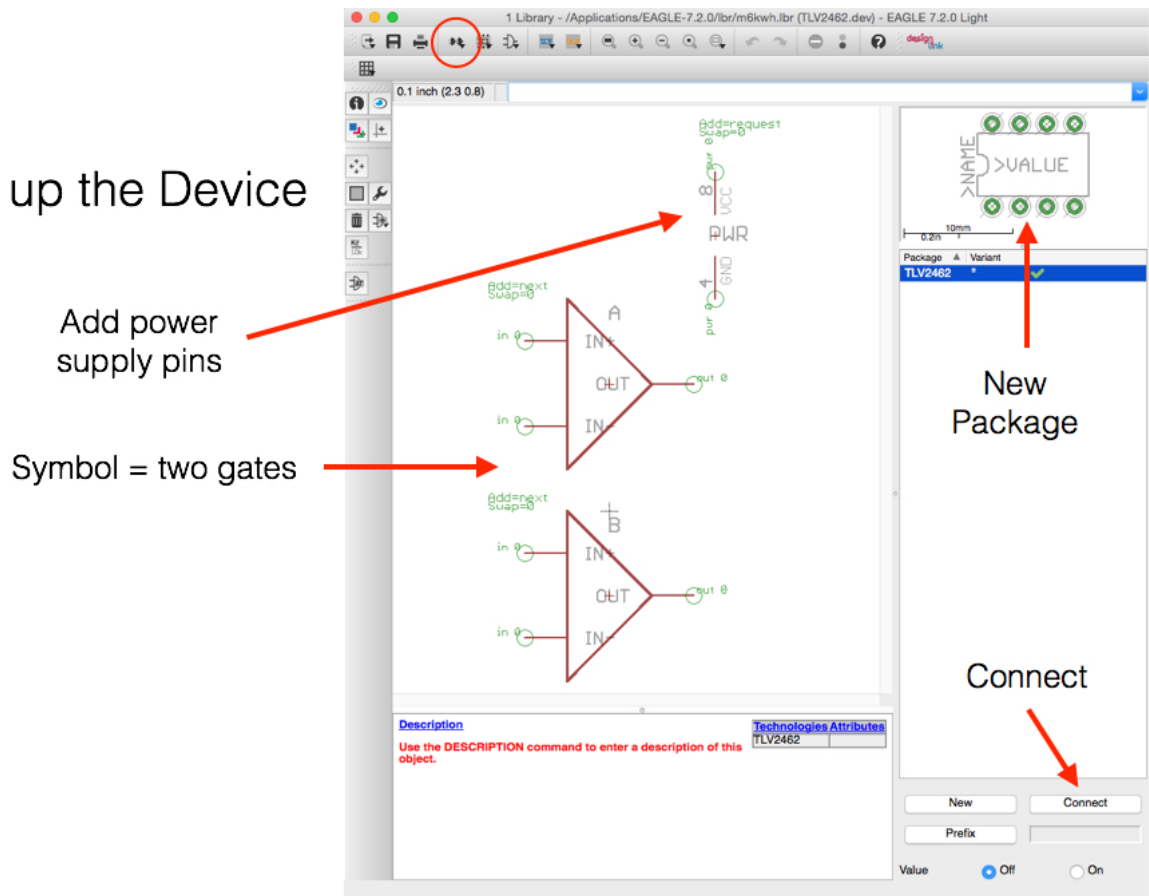Now draw a Gate and a Package - this is the TLV 2462 Dual op-amp
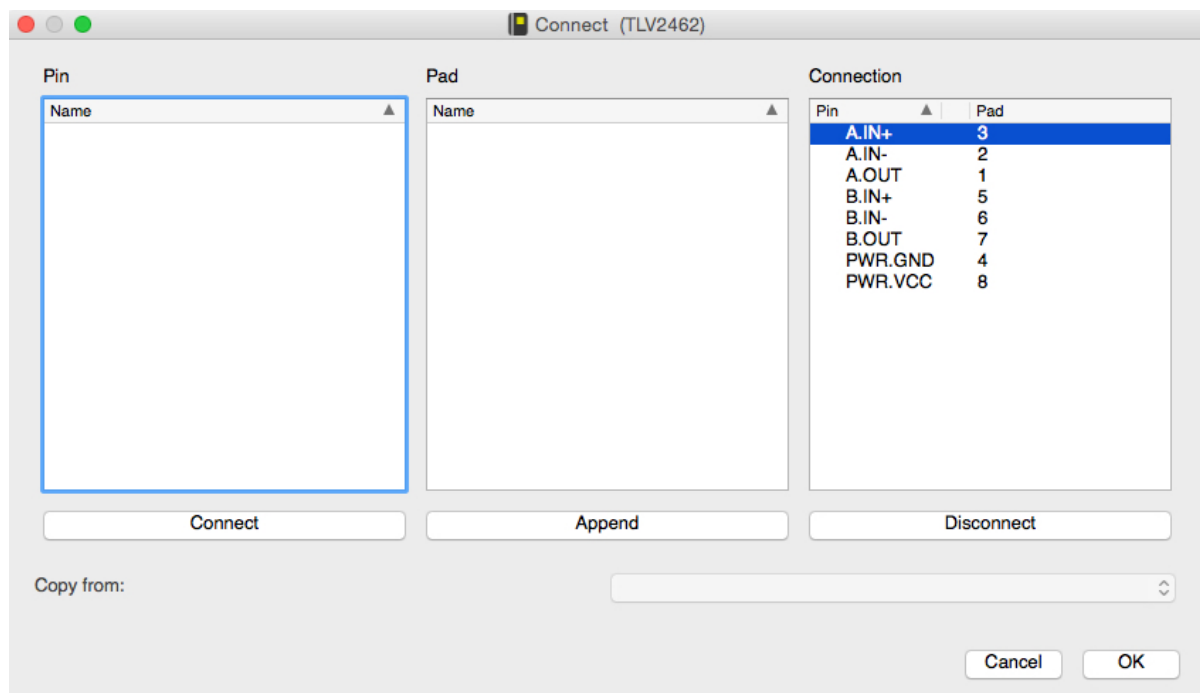

Draw a
Gate

Draw the
Package

Next build the Device from the Gates and the Package

Make up the Device

Add power supply pins

Symbol = two gates
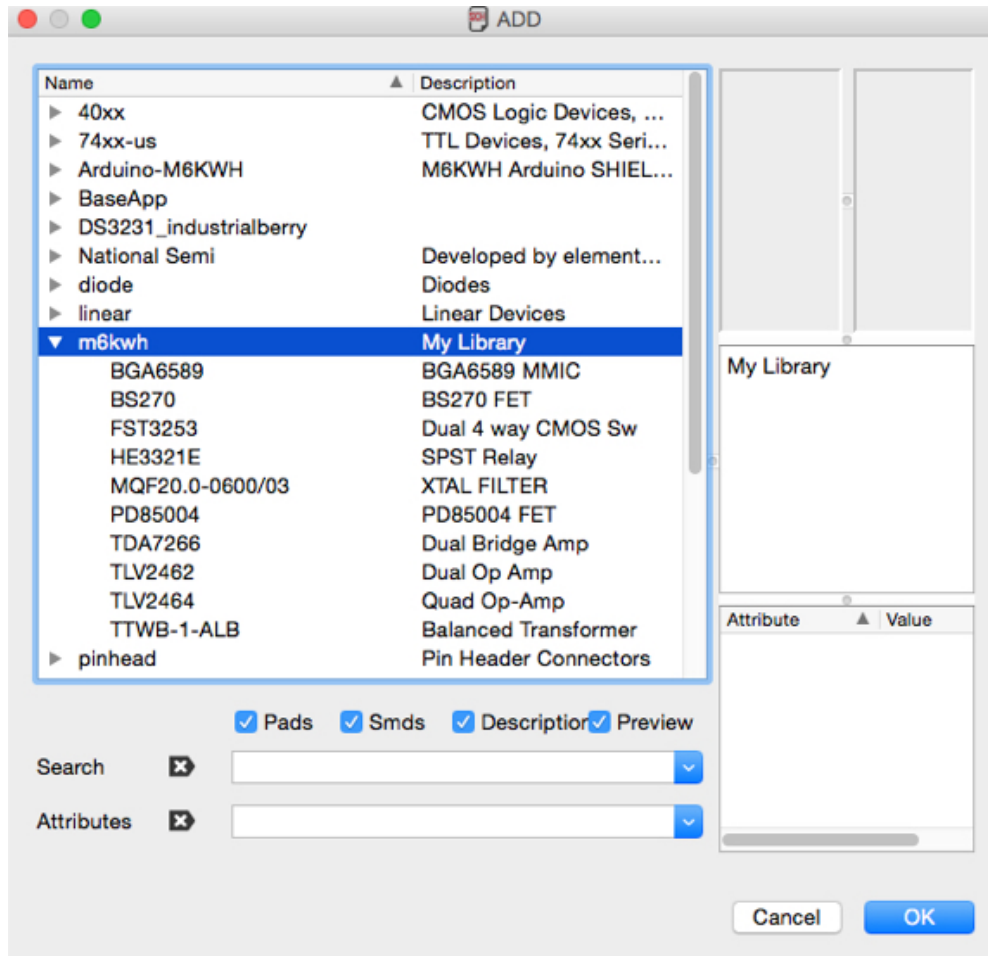
New Package

Connect

Note that I have included a new item the power supply, this does not show on the schematic, but allows automatic routing of VCC and GND to the device.
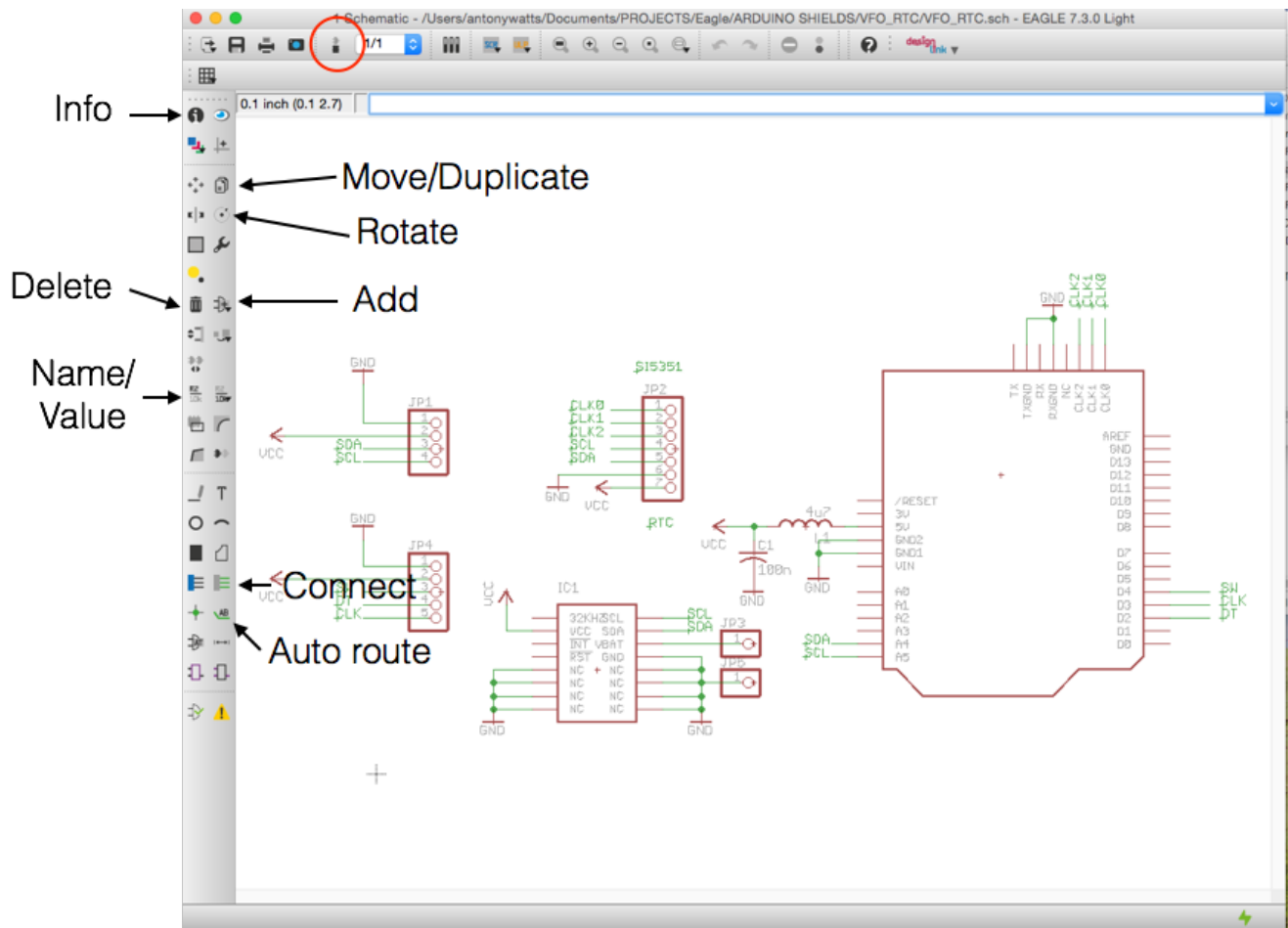
Finally connect up the pins and save the pa

Pin	Pad	Connection	
Name	Name	Pin ▲	Pad
		A.IN+	3
		A.IN-	2
		A.OUT	1
		B.IN+	5
		B.IN-	6
		B.OUT	7
		PWR.GND	4
		PWR.VCC	8

Connect | Append | Disconnect

Copy from:

Cancel   OK

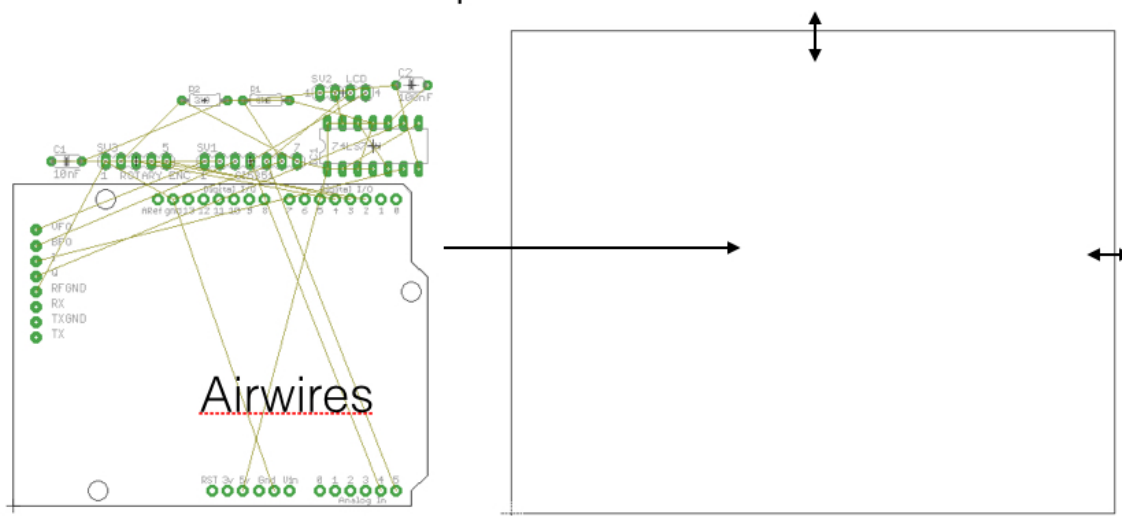Your part is now in your library and can be used in PCB designs

Schematic capture

With the schematic editor open you can add parts, place and connect them

Make the PCB, chose the PCB symbol and this will be the result

- Define Board size
- Move parts to Board area

Airwires

All the components used are in the free space and must be mover one-by one to the PCB area. In the case of an Arduino board, delete the board outline rectangle as the board outline is defined by the Arduino Device itself.

Then move the board and the components over. and auto wire, then correct and improve manually to get your board. Put a polygon round the outside an name it GND both top and bottom, to create ground planes.
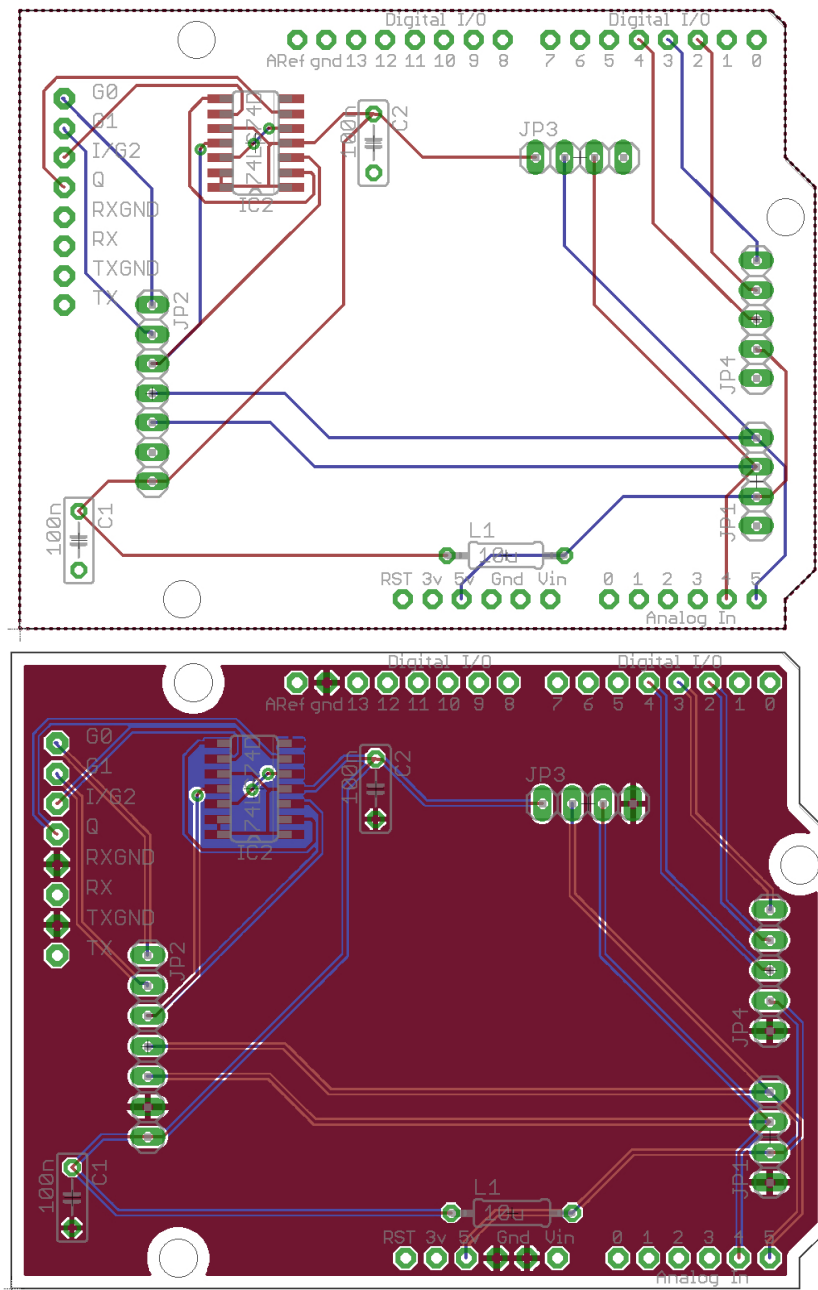
If you have installed Eagle there is a very simple example on the USB stick you copied to your computer called "invamp" you can load this and practice doing the PCB layout.