UnPCB
PCB Reverse Engineering Tool

This program will help you to convert printed circuit boards to Eagle schematics.

Requirements

-Cadsoft Eagle 6.4.0 or newer, download at cadsoftusa.com
-UnPCB program
-Desoldering equipment
-Flatbed scanner

1. Remove all components

The first step is to remove all the components from your PCB. Before you do this it's a good idea to take some pictures from the top and bottom! For each component you need to make some documentation of it, a spreadsheet is ideal for this:
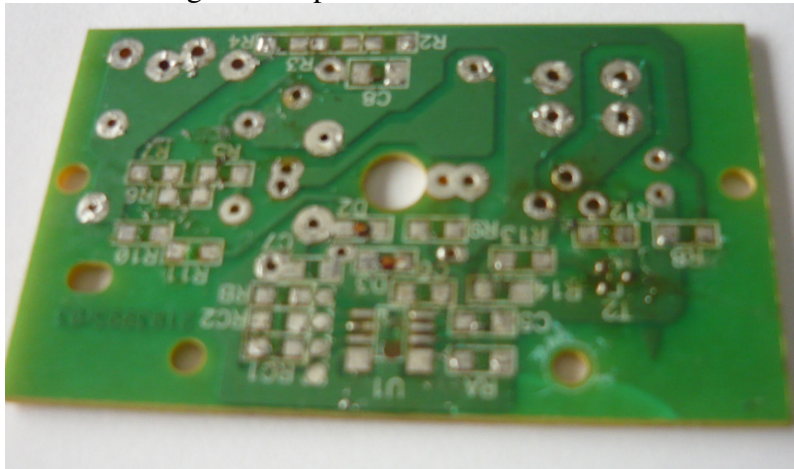
- Write down the name on the PCB (like IC1, R2, etc.),
- Search for the component's datasheet,
- Search for an Eagle library file (.lbr) that contains the right schematic symbol.

For example in this document we will make the schematic of this dimmer board:
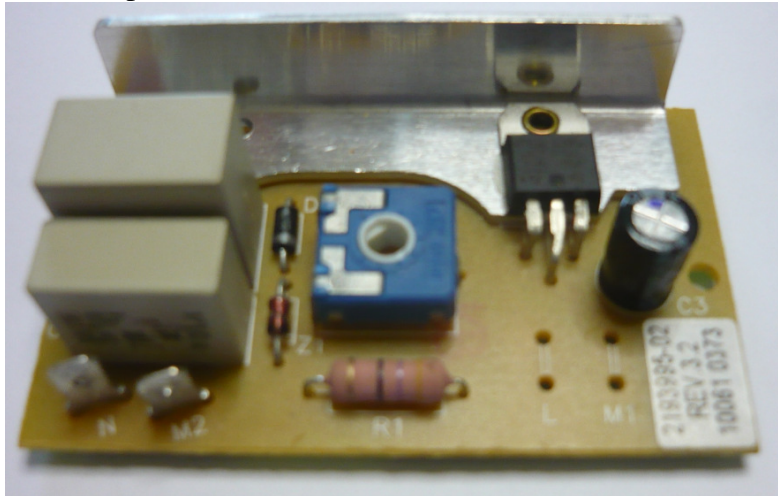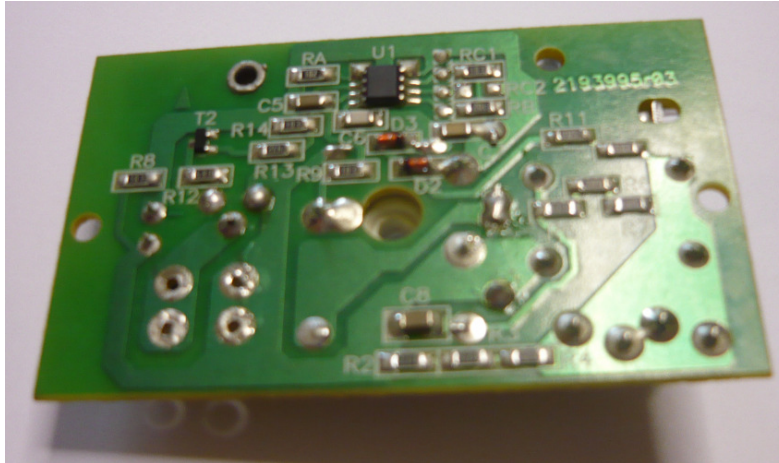
After removing the components the PCB looks like this:

The parts are all added to the components.xls file together with the corresponding schematic symbol in Eagle libraries, which looks like this:

Name	Value	Eagle component
R1	45	rlc/R-EU/R-EU_0207/10
R2	330K	rlc/R-EU/R-EU_R2010
R3	330K	rlc/R-EU/R-EU_R2011
R4	330K	rlc/R-EU/R-EU_R2012
R5	330K	rlc/R-EU/R-EU_R2013
R6	330K	rlc/R-EU/R-EU_R2014
R7	330K	rlc/R-EU/R-EU_R2015
R8	10K	rlc/R-EU/R-EU_R2016
R9	10K	rlc/R-EU/R-EU_R2017
R10	1M	rlc/R-EU/R-EU_R2018
R11	1M	rlc/R-EU/R-EU_R2019
R12	68	rlc/R-EU/R-EU_R2020
R13	4.7K	rlc/R-EU/R-EU_R2021
R14	10K	rlc/R-EU/R-EU_R2022
RA	1K	rlc/R-EU/R-EU_R2023
RC1	10K	rlc/R-EU/R-EU_R2024
RB	1K	rlc/R-EU/R-EU_R2025
C1	0.33µF	rlc/C-EU/C075-032X103
C2	0.47µF	rlc/C-EU/C075-032X104
C3	330µF	rlc/CPOL-EU/CPOL-EUE5-5
C5	100nF	rlc/C-EU/C-EUC1210
C6	1µF	rlc/C-EU/C-EUC1211

C7	100nF	rlc/C-EU/C-EUC1212
C8	10µF	rlc/C-EU/C-EUC1213
D1	1N4007	diode/1N4004
D2	?	diode/ES2D
D3	?	diode/ES2D
Z1	BZX85C 5.6V	diode/ZTK
POT	20K	pot/TRIM_EU-/TRIM_EU-PT10
T1	BTB16	triac/BT151
T2	BC847	transistor-npn/BC847/BC847ASMD
U1	MC9S08QD4	MC9S08QD4.lbr
L		Pinhead/PINHD-1x1
M1		Pinhead/PINHD-1x1
M2		Pinhead/PINHD-1x1
N		Pinhead/PINHD-1x1

Notice that the microcontroller MC9S08QD4 is not in Eagle's standard libraries so for this component a custom library was made. If you don't know how to create custom components for Eagle please read library_tutorial.pdf.

2. Start a new unPCB project

After removing all the components you must make a scan (using a flatbed scanner) of the top and bottom. Then you can start the unPCB.exe program and add the top / bottom images:
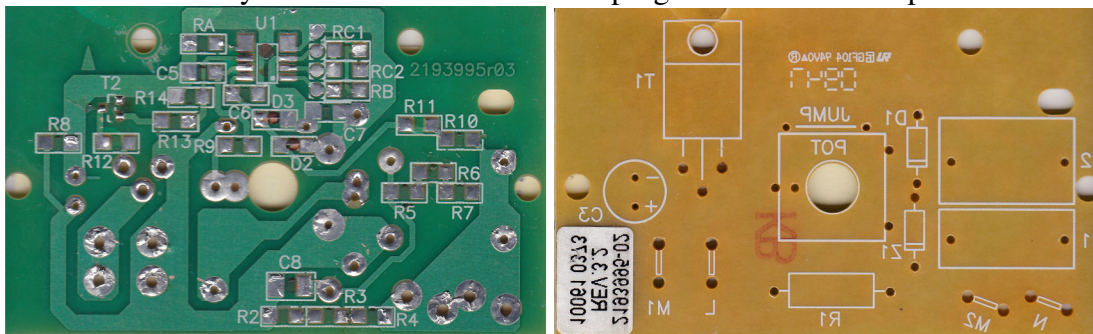

Make sure the images have the same size / DPI and when you move top over bottom image the holes must align. So it's possible you need to mirror the top image (like in this example top is mirrored horizontally).
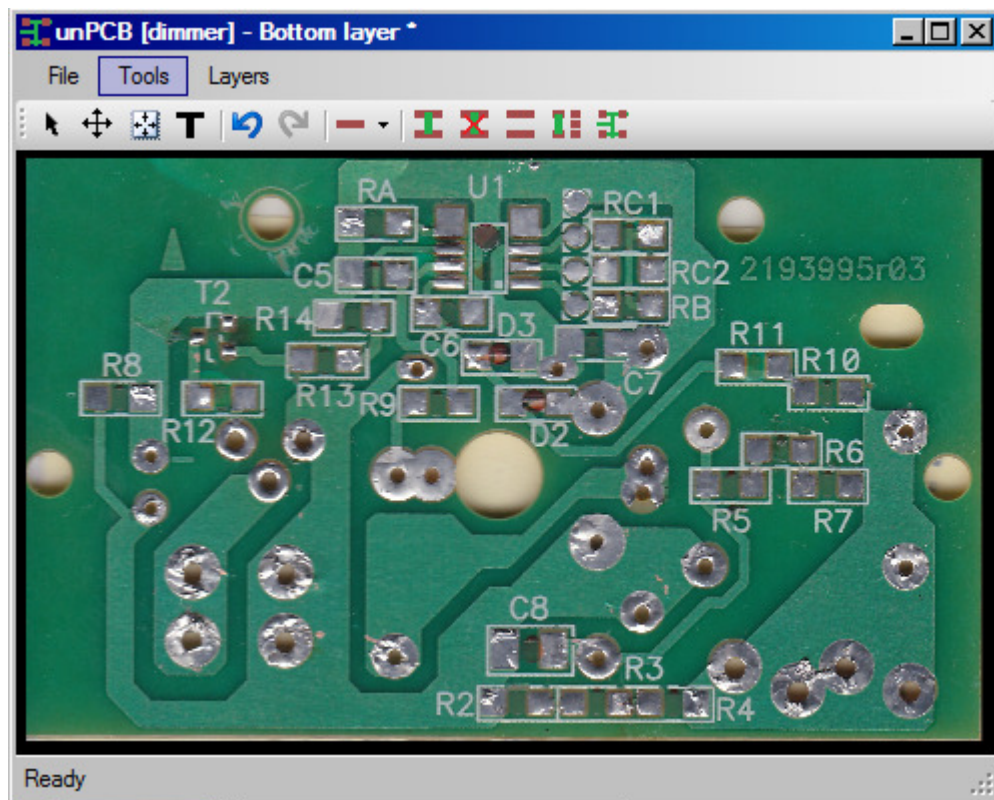
So now open the unPCB program and select the top/bottom images. For the top image you can check 'flip image horizontally'. This will make the text readable again without misaligning the holes / components.
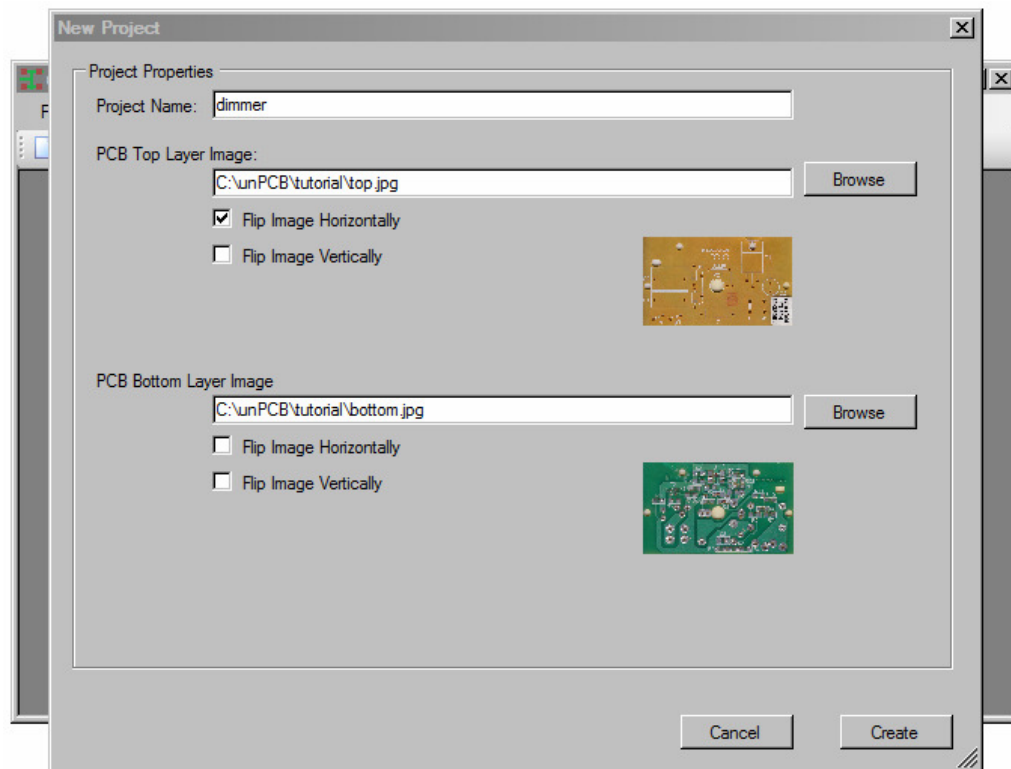
Click the create button to start the new project.

You'll now see 2 layer windows open with the top and bottom images loaded:
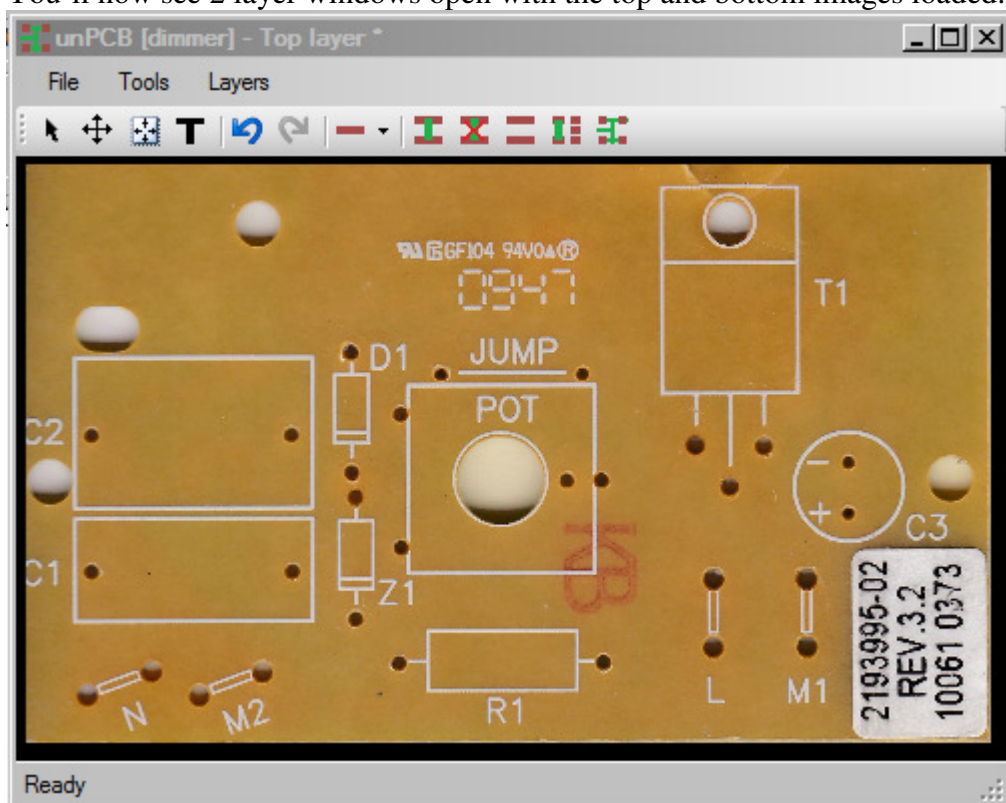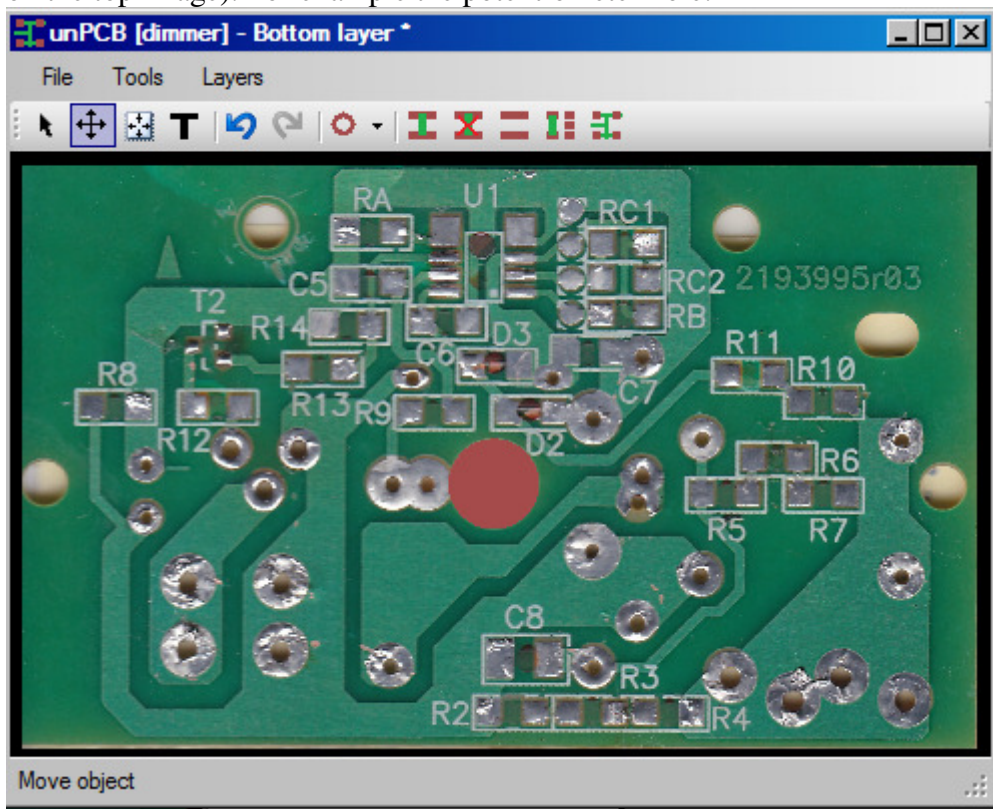
You can use the middle mouse button to move across the drawing by holding it down and moving the mouse. You can zoom in/out by using the mouse scroll button.

First thing you need to do is align the top and bottom because the holes may not be 100% at the same location. To do this draw a through hole pad or via on a hole, pad or via. You can draw this by clicking the draw toolbar button and select through hole pad:
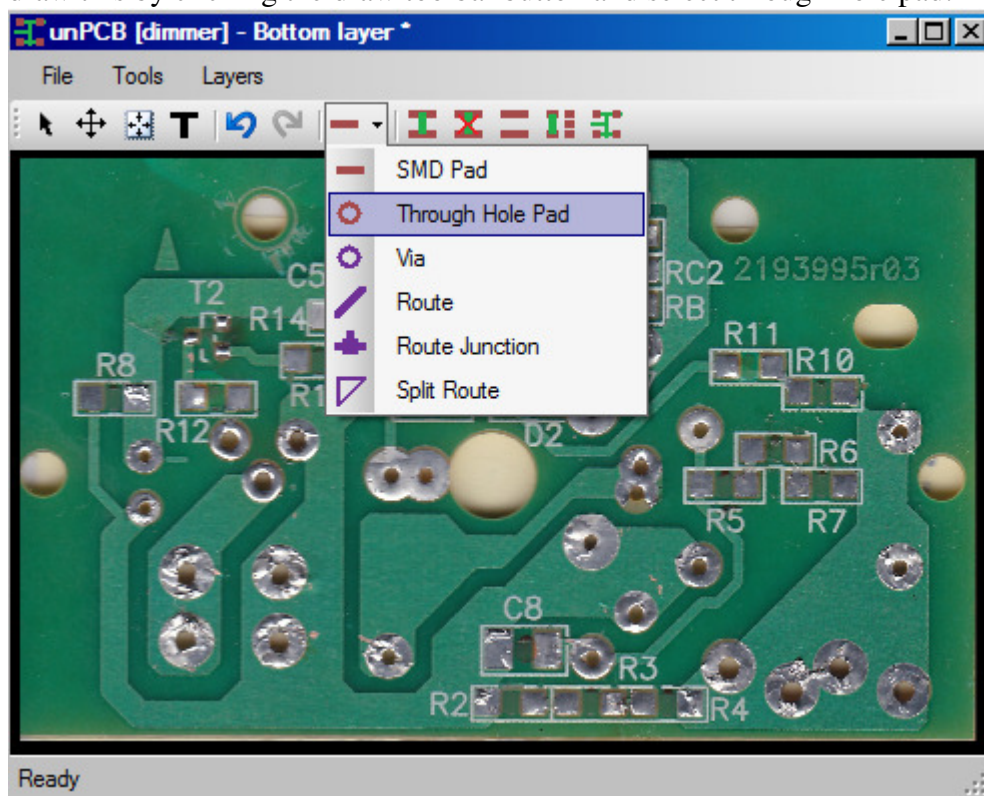
Now draw the pad somewhere on a hole (doesn't matter where as long as the hole is visible on the top image). For example the potentiometer hole:
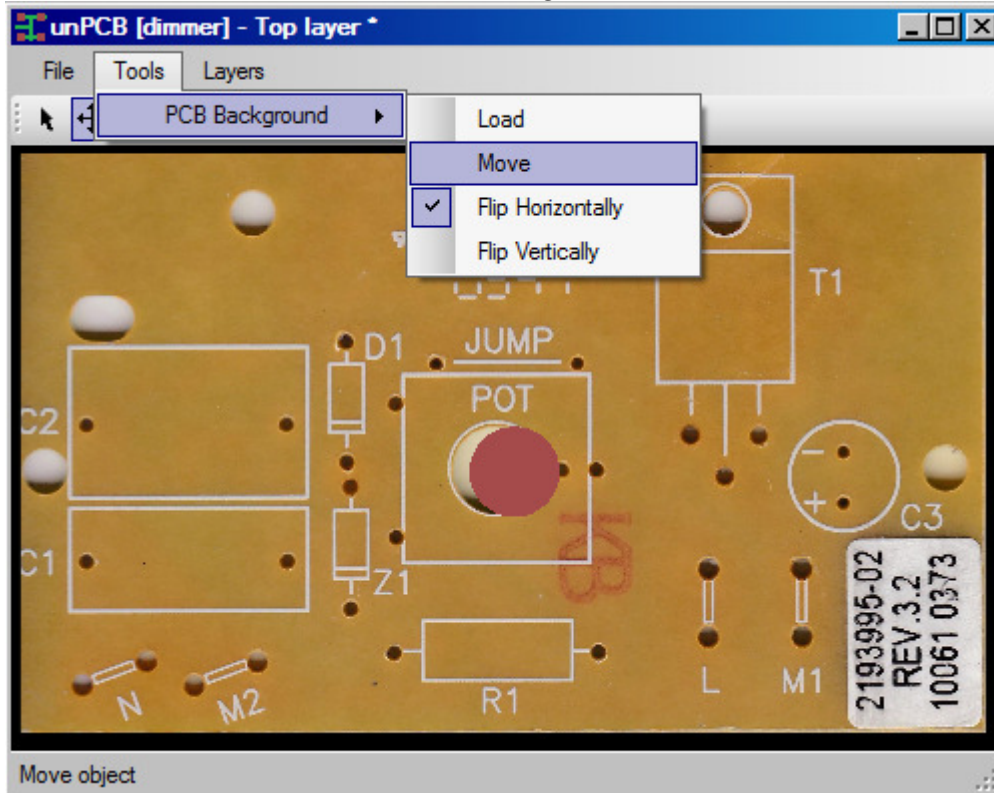

Click and hold down the left mouse button, then move the mouse to make the pad bigger/smaller so it fits the hole. You can use the *move* toolbar button to make the center align perfectly. Now go to the top image, you will see that the pad is not perfectly aligned with the potentiometer hole on the top image.
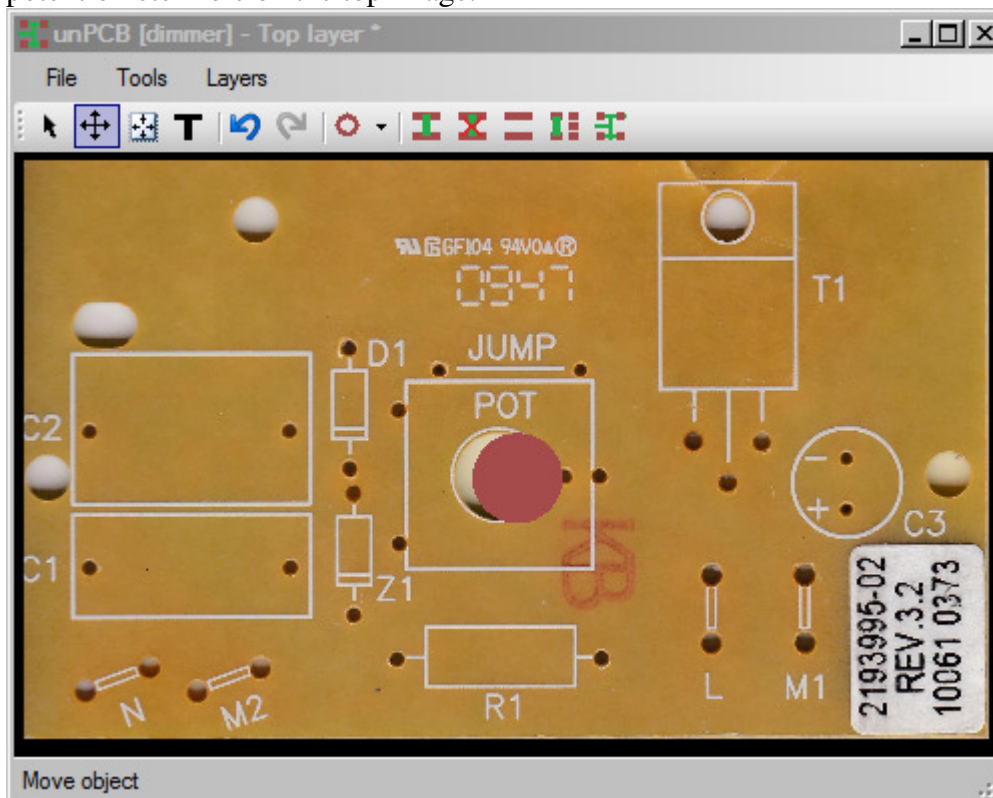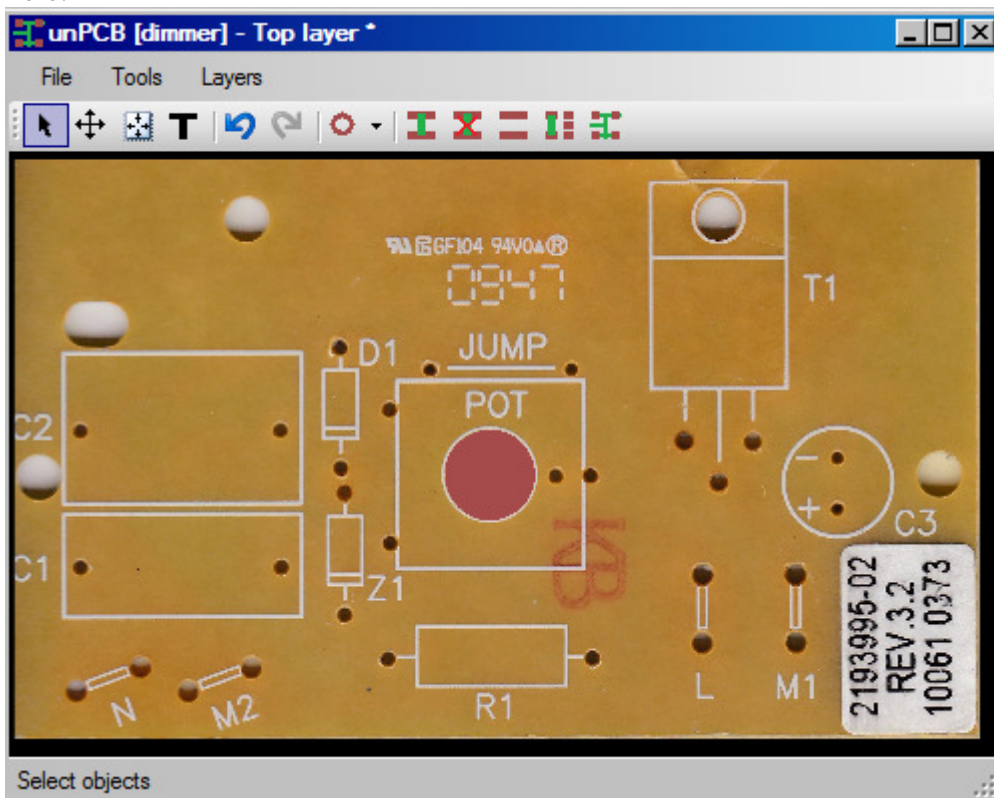
Click the *tools* menu and select *PCB background -> move*.

You can now move the background without moving the pad, move the background by holding down the left mouse button and dragging the image. Make it align with the potentiometer hole.
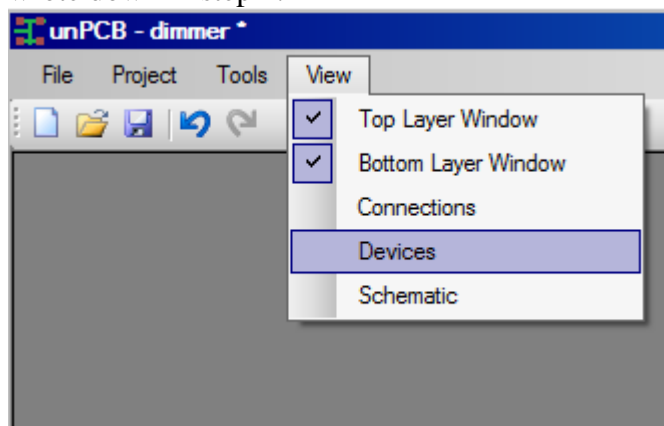

You can now delete the pad in the potentiometer hole. Use the *select* tool (=first button) in the toolbar, select the pad and right click it, click *delete*.
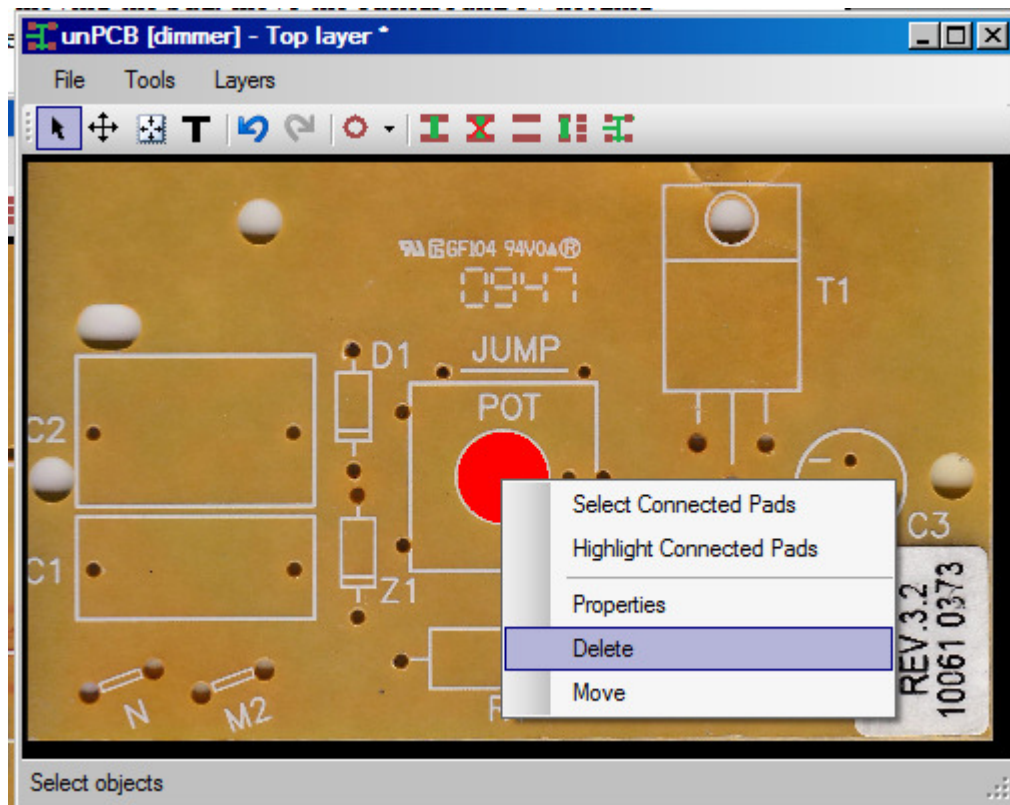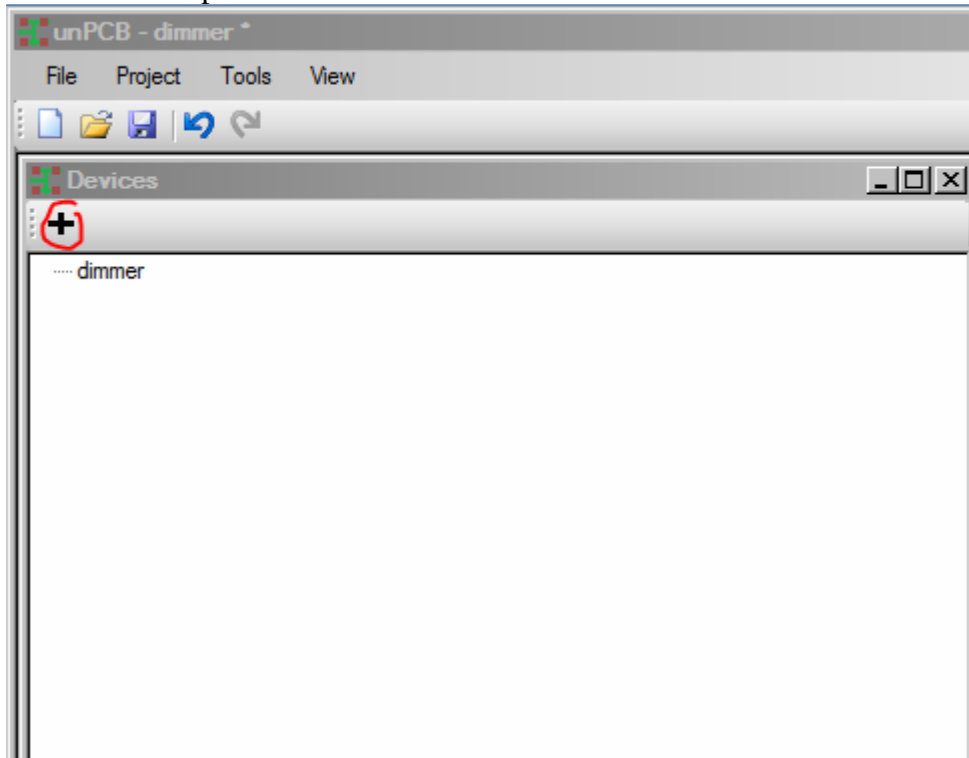
3. Add all devices

Now you must open the devices window via *view->devices*, you can now add all devices you wrote down in step 1.
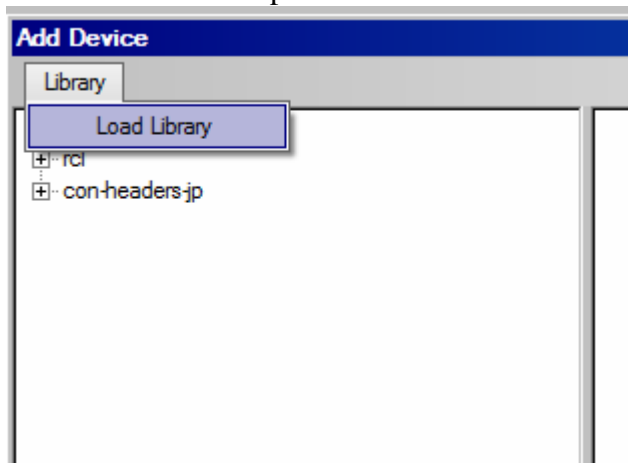
New window opens:

Click the + icon to add a new device, the first time you'll need to load your Eagle libraries by clicking the *library->load library* menu:

You can select multiple .lbr files at once or add them one by one as soon as you need them.

Now you can select the device you wish to add, give it a name and optional value.
In this example we need the rcl, diode, pot, triac, transistor-npn, MC9S08QD4 libraries.

Notice: If you get an error saying that the lbr format is not valid XML format you are trying to open an old version of lbr file. To solve this problem you must open the lbr in a newer version of Eagle and just save it. Then it will be saved as XML format and you will be able to open it.

Now you can add the first device from the list, R1. From the *rcl* library select *R-EU_* and click R-EU_0207/10 give it the name R1 and value 45.
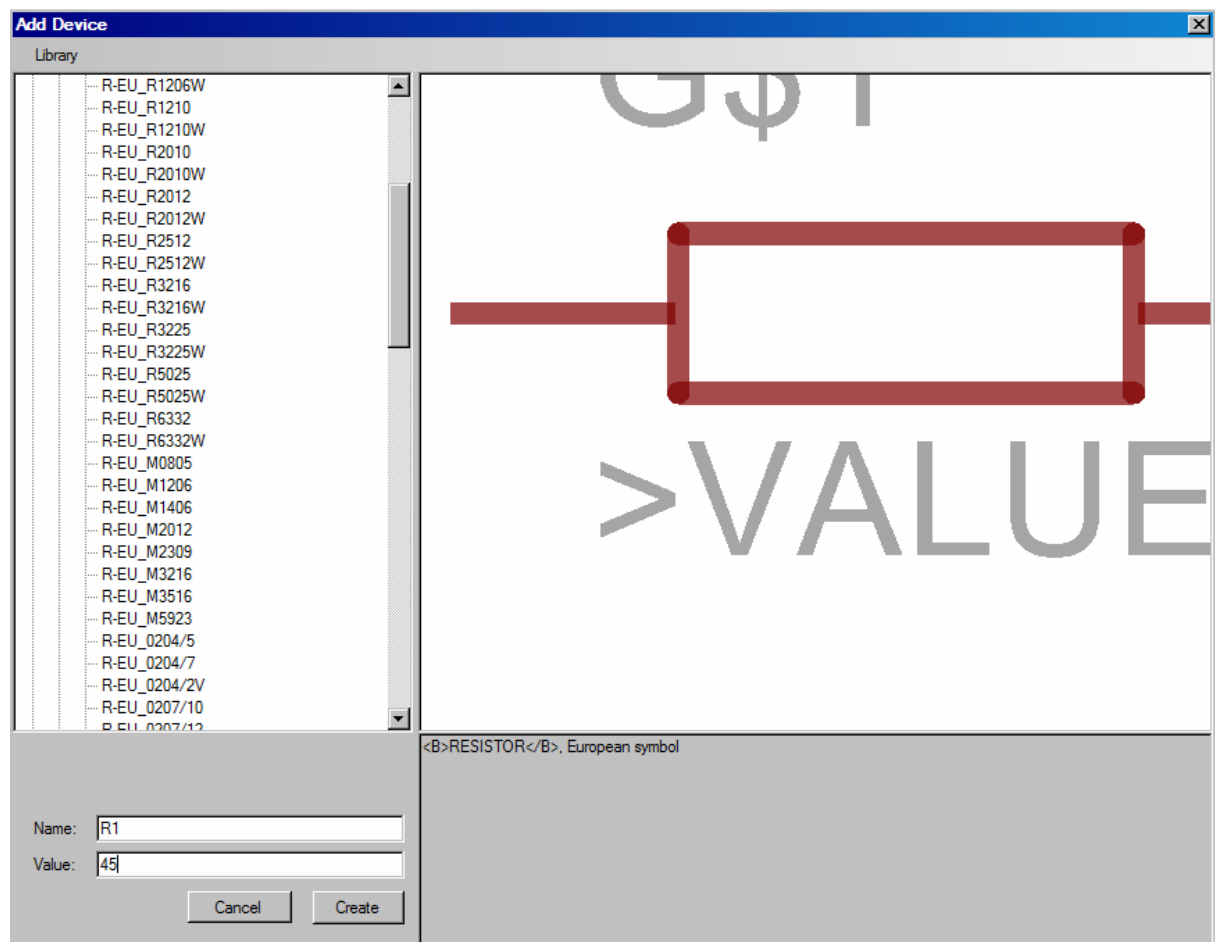
Click the *Create* button. Now from the *devices* window you can right click the R1 and then select *Place a pad for each device pin*.
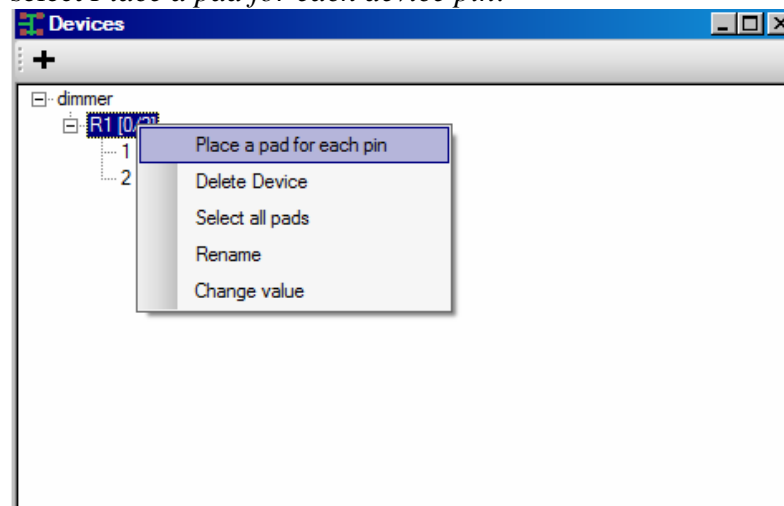

Now you go to the top layer window and you left click on the first pin of the resistor to place a through hole pad, then click the next pin. You can hold down the left mouse button to adjust the size while placing the pad. If you use the right mouse button you can swap between through hole and SMD pads.
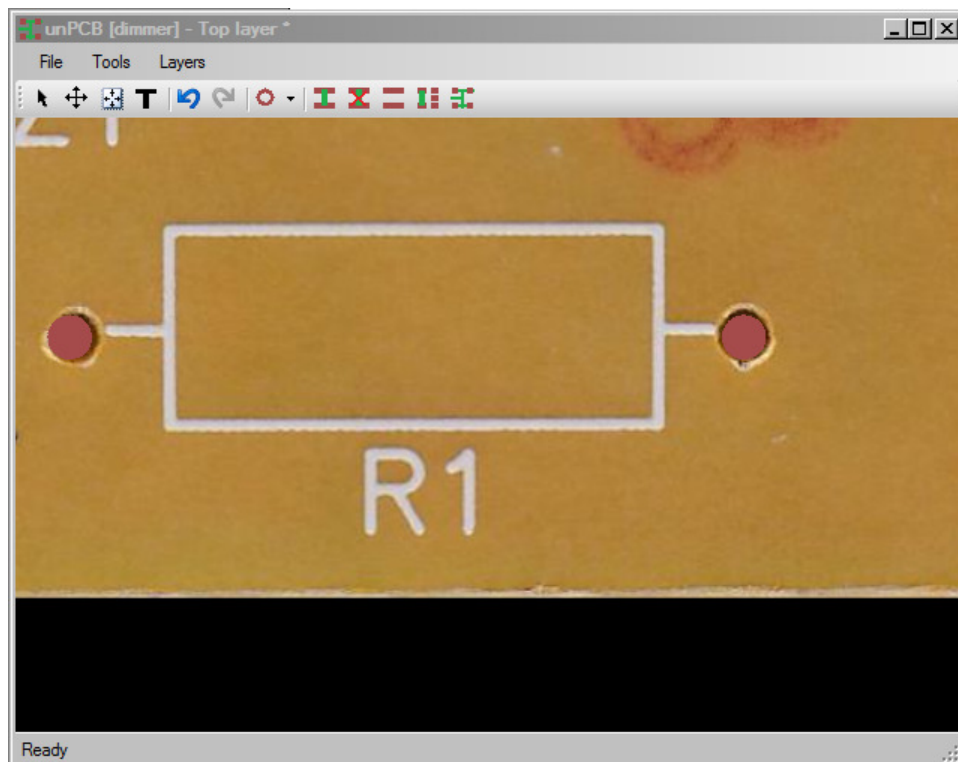
Now you can place the other devices in the same way.

It's a good idea to open the schematic window as well! Everytime you add a new device you can move it to what you think is the best location in the schematic. You can open the schematic window through the *view* menu on the main window.
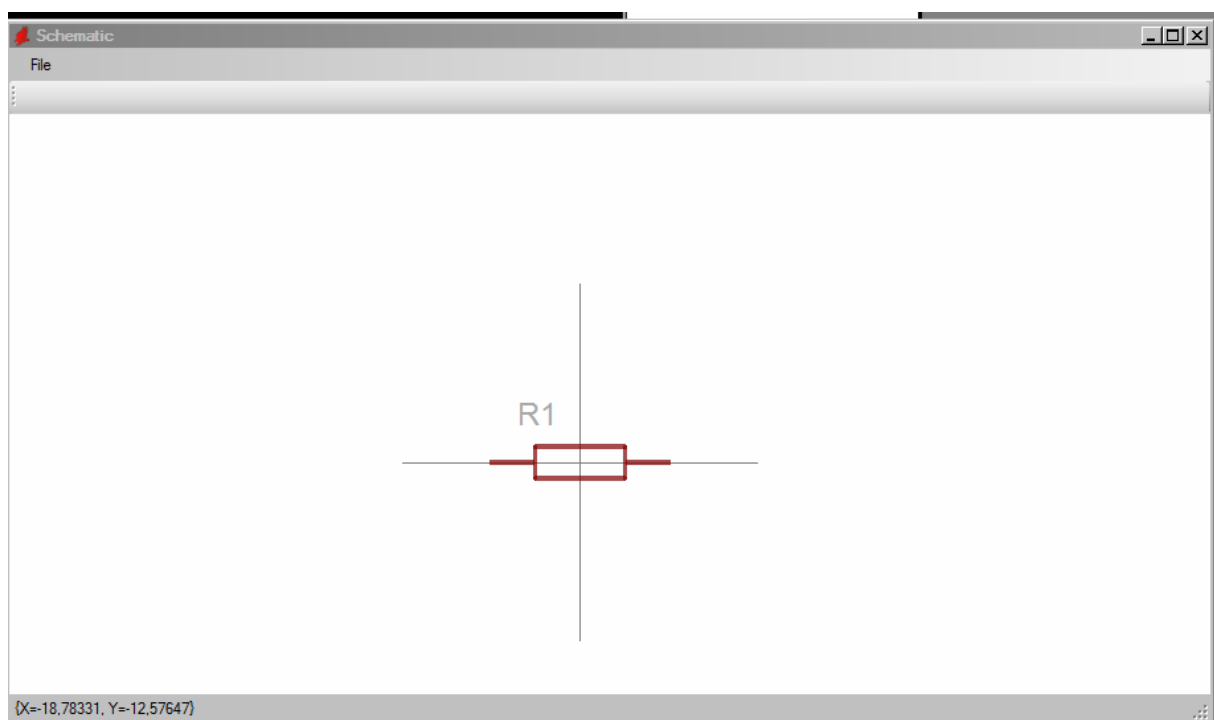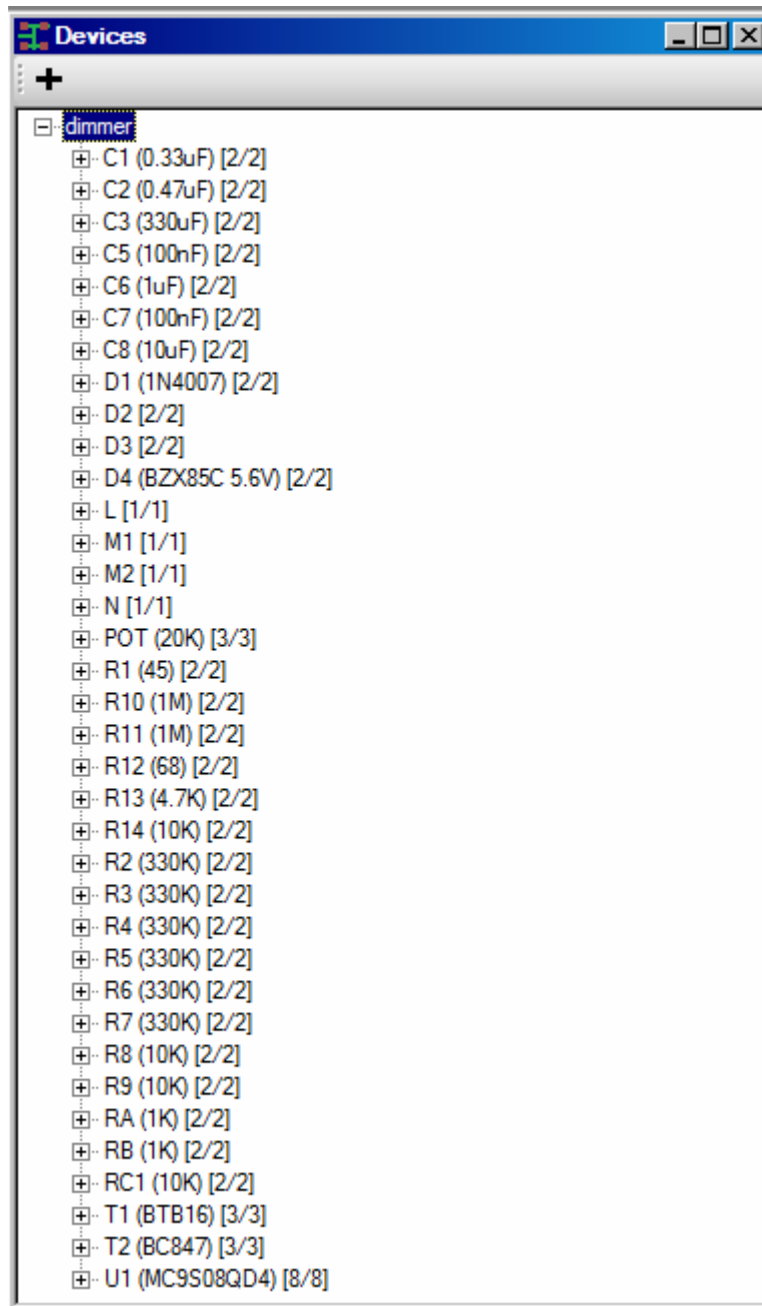

Just click the R1 to move it to a new location, by default all devices are placed on the origin.
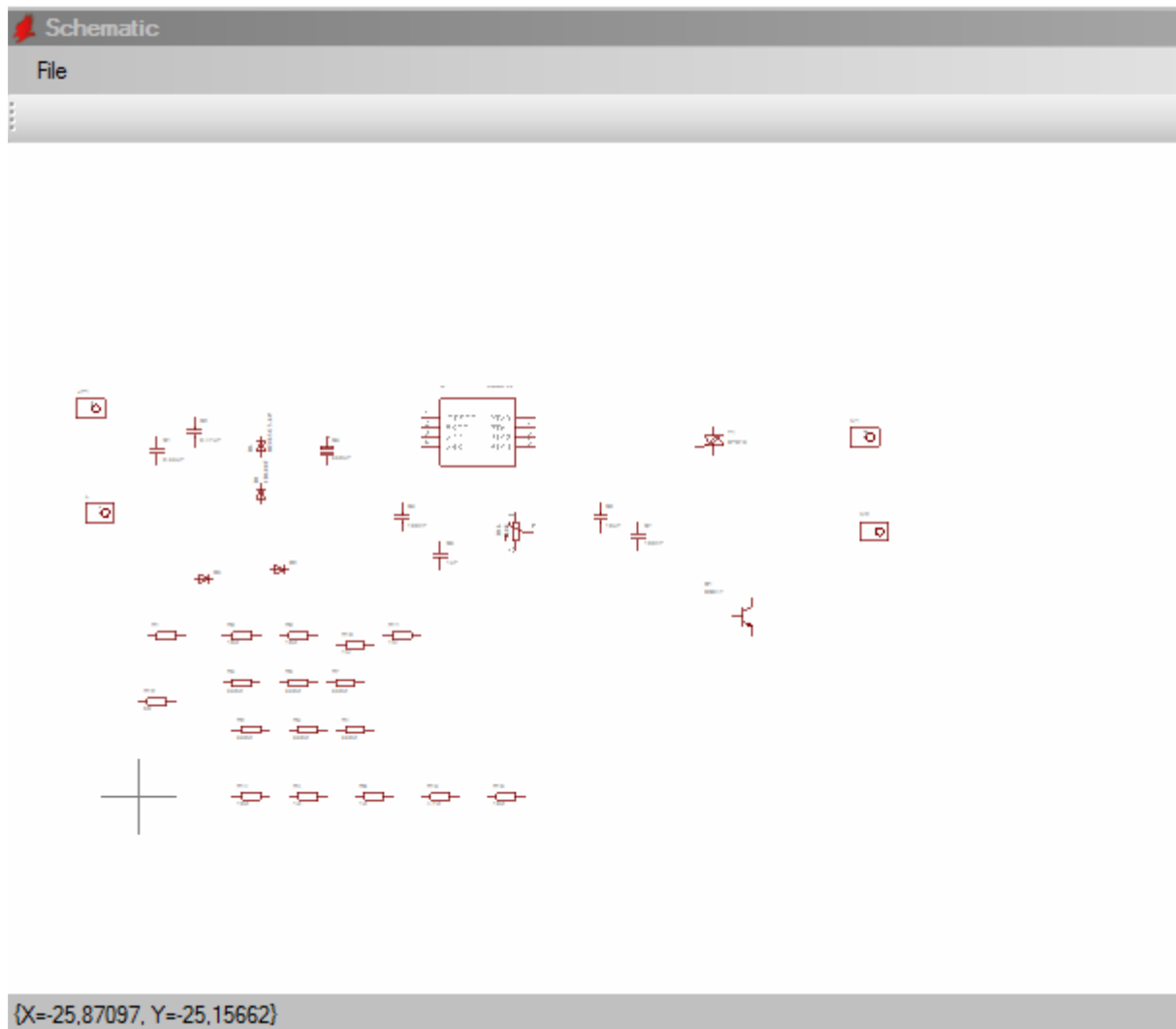
If all devices are placed you can add vias or continue with the next step. Vias will make it easier and faster to connect pads with each other as you only need to look on one layer. Devices window:
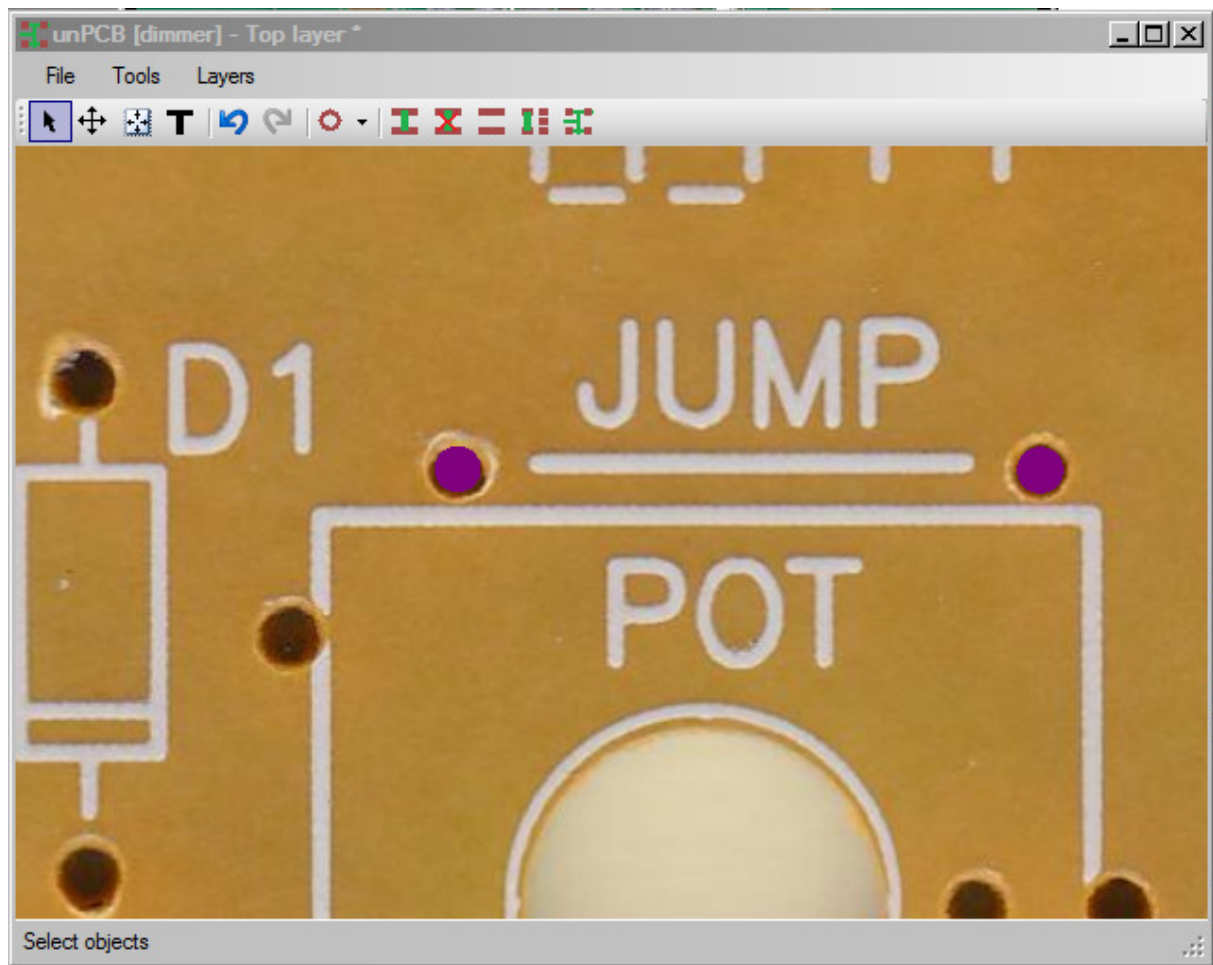

Top and bottom layer:

The schematic looks now like this:
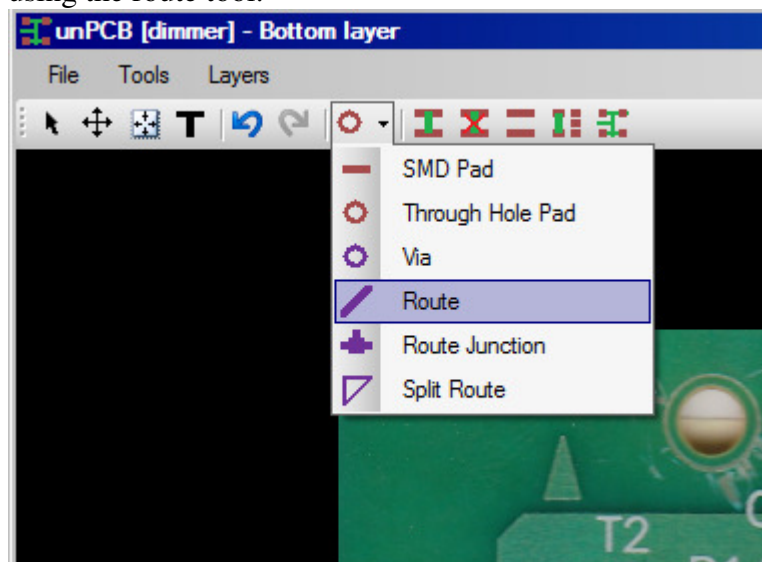
{X=-25,87097, Y=-25,15662}

Our example is a single layer board so it doesn't have any real vias that switch routing between layers but we can use the jumper wire to explain it. Place a via on the jumper wire pins.
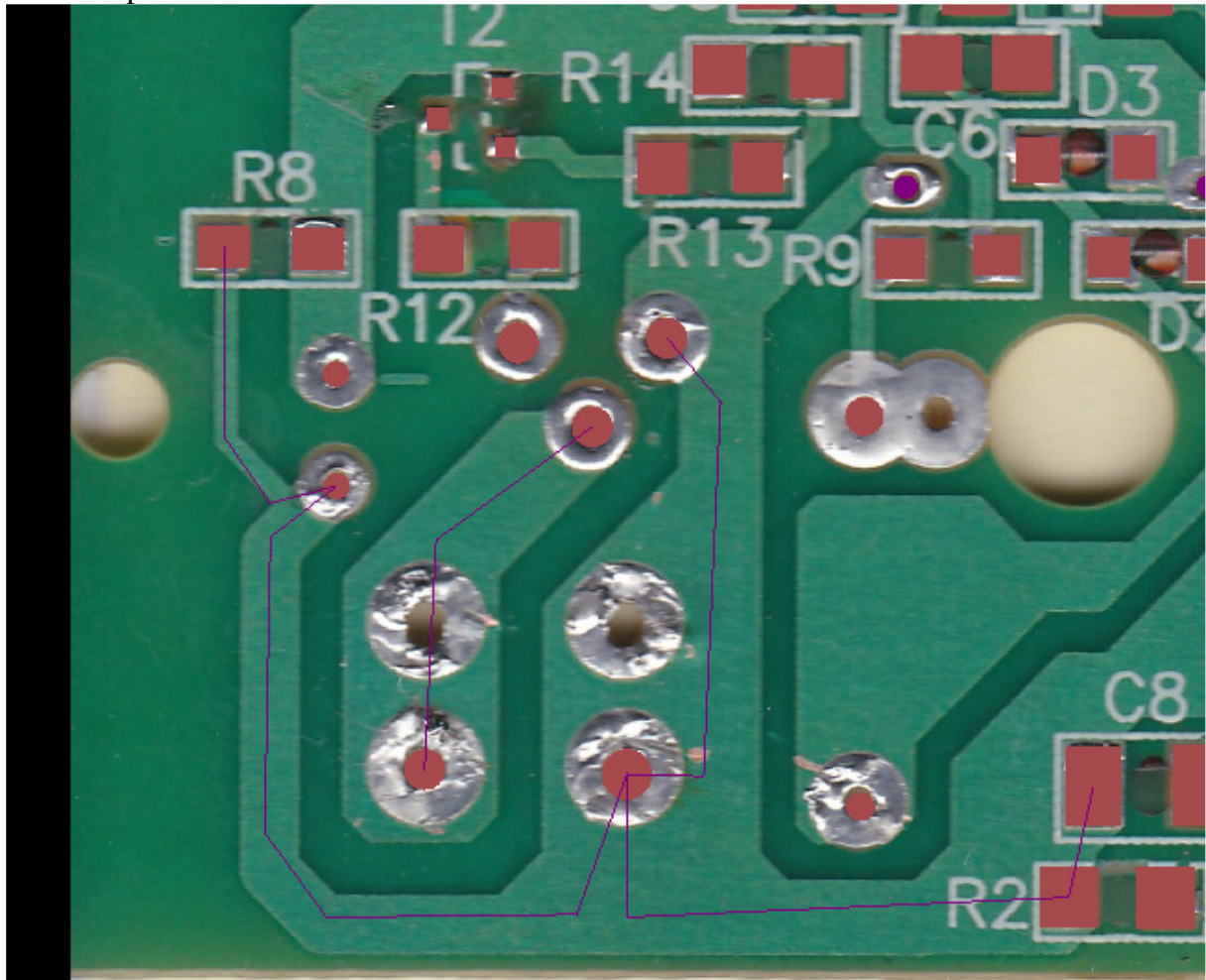
4. Connect pads

Now you can start connecting as many pads and vias by hand as possible. You can do this by using the route tool.


Now you click a pad or via and you follow the route on the PCB until you reach another pad the route is connected to. Once you click that other pad they both will be connected automatically.
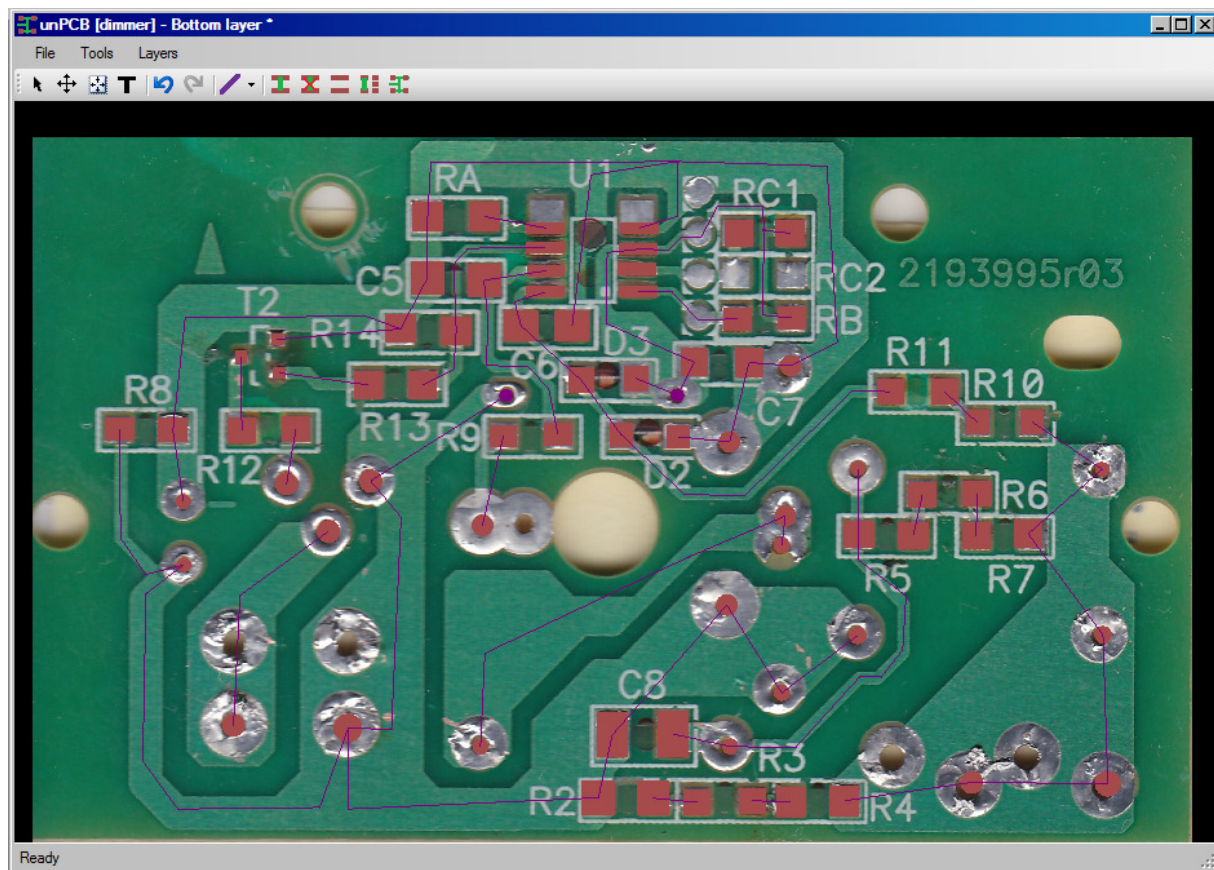
For example:

The purple lines (routes) connect the pads, you can draw multiple routes from the same pad. You can double left click to terminate with an SMD route junction or right double click to terminate the route with a via.

Top layer now looks like this:
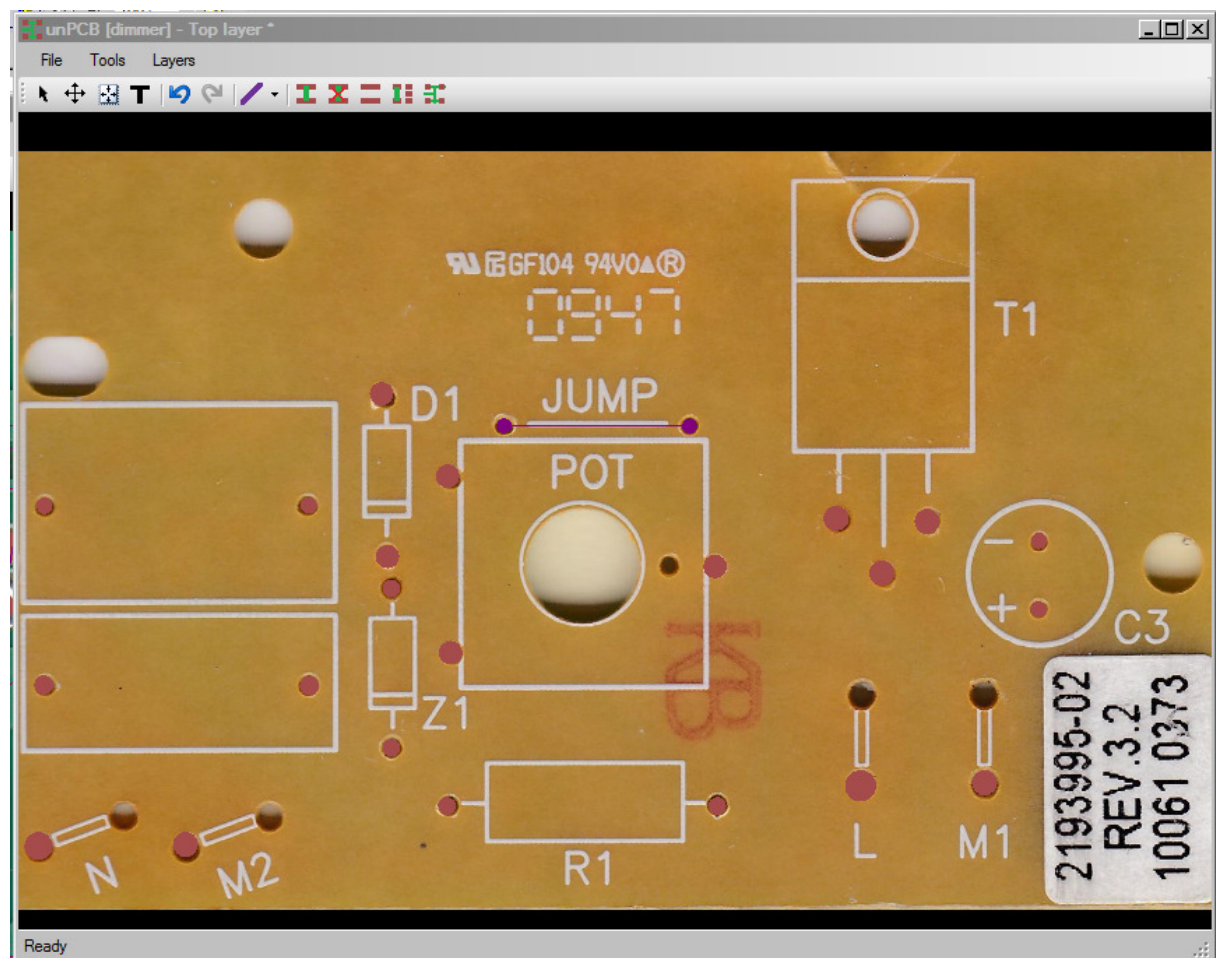
Bottom layer:

It's also possible to manually connect pads using the toolbar buttons. You can do this by using the select tool and clicking multiple pads while holding down the control key. Then you click one of the toolbar buttons:

![connect]	Connects the selected pads to each other
![disconnect]	Disconnects the selected pads, meaning the connection state goes to "uknown"
![not connected]	The selected pads are NOT connected
![connected only]	The selected pads are connected to each other but NOT connected to any other pads. The automatic connection tester tool will not ask to test the selected pads anymore.
![tester]	Opens the connection tester / manual router tool. This will ask you to check if certain pads are connected to each other.

All connections are saved in a connection matrix. You can open this table by clicking *view->connections* at the main window.

	R1.1	R1.2	via_8	via_9	R2.1	R2.2
R1.1	1	?	?	?	?	?
R1.2	?	1	?	?	?	?
via_8	?	?	1	1	1	?
via_9	?	?	1	1	1	?
R2.1	?	?	1	1	1	?
R2.2	?	?	?	?	?	1
R3.1	?	?	?	?	?	1
R3.2	?	?	?	?	?	?
R4.1	?	?	?	?	?	?
R4.2	?	?	?	?	?	?
R5.1	1	?	?	?	?	?
R5.2	?	?	?	?	?	?
R6.1	?	?	?	?	?	?
R6.2	?	?	?	?	?	?
R7.1	?	?	?	?	?	?
R7.2	?	?	?	?	?	?
R8.1	?	?	1	1	1	?
R8.2	?	?	?	?	?	?
R9.1	?	?	?	?	?	?
R9.2	?	?	?	?	?	?

Here you can see that R1 pin 1 is connected to R1 pin 1 and also to R5 pin 1. The connection state between R1 pin 1 and R1 pin 2, via_8, via_9,… is a question mark, meaning the state is unknown. The automatic connection tester tool will ask you to check for the connection between these pins (using Ohm meter). If you see a 0 in the table this means these pins/pads/vias are NOT connected (= open connection measured with Ohm meter).

5. Run the automatic connection tester tool

This tool will help you to check any unknown connections between pads. These connections could be made with routes in the middle layers of the PCB that are not visible. With a single layer or double layer board you could skip this step and immediately extract the schematic. If you chose to do this you must be sure you connected all the pads that seem to be connected using routes!

So click the manual router tool:

A new window opens and 2 pads are highlighted in yellow.
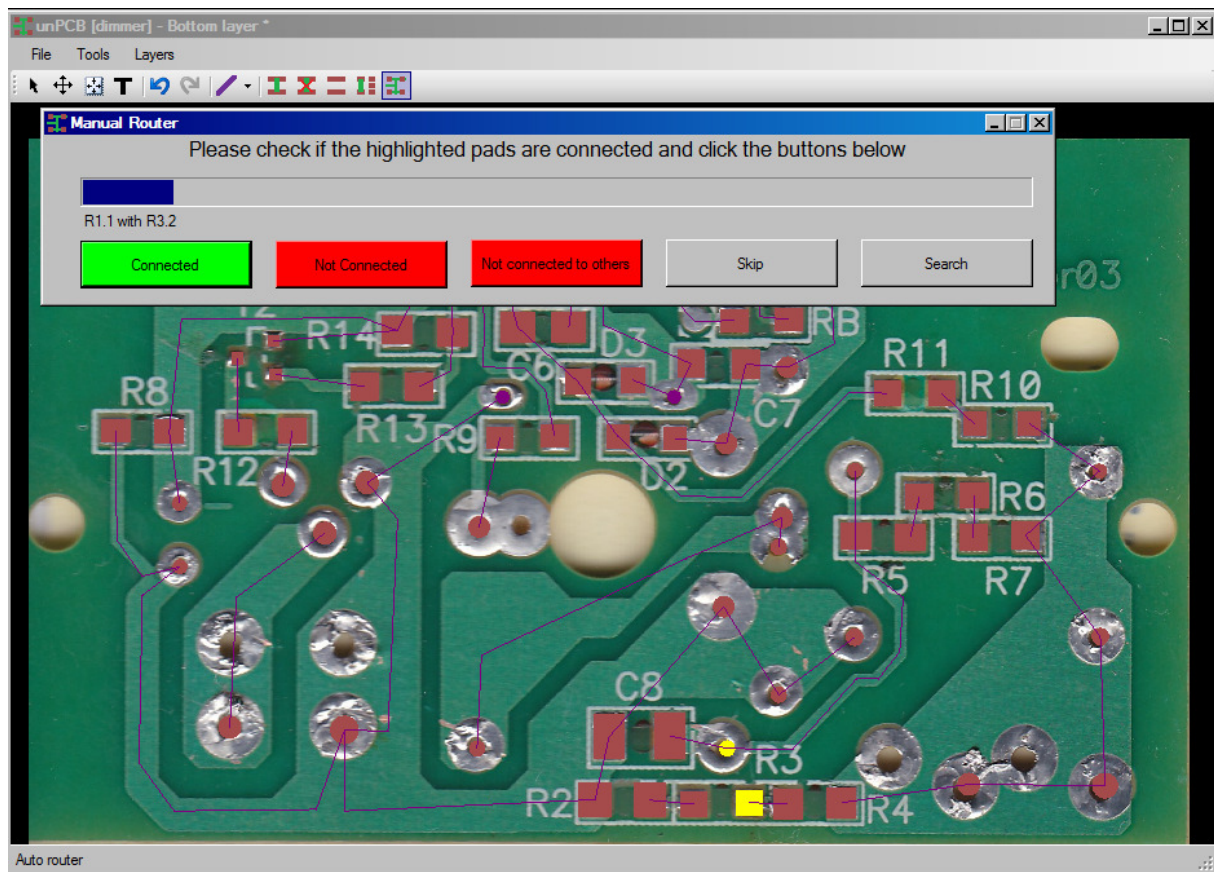
Now you take an Ohm meter and you test if the yellow marked pads are connected, if this is not the case you click the *Not Connected* button. If you measure 0Ohm you must click the green *connected* button. The tool will then ask you to test R1.1 with the next unknown pad. If you are 100% sure R1.1 is not connected to anything else than what it's already connected to at this moment you can click the red *not connected to others* button (this speed things up!). This will then finish pad R1.1 and move to R1.2.
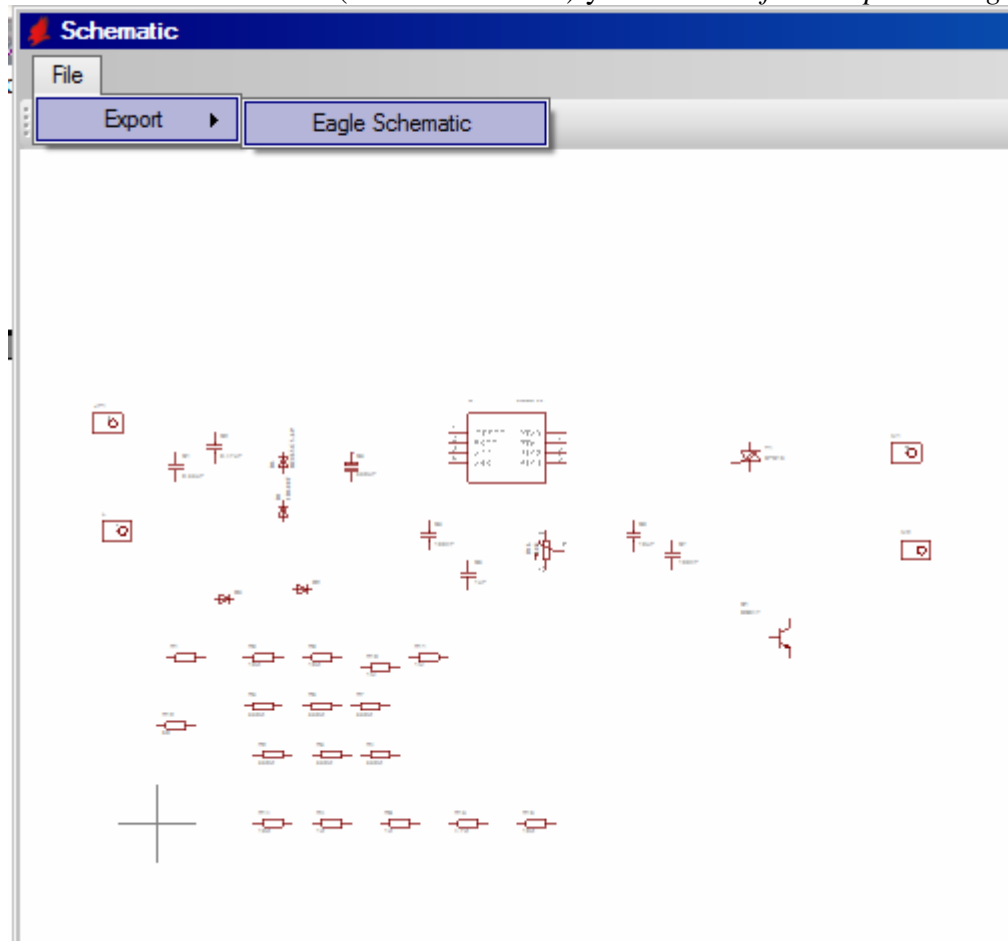
After a lot of testing you will see this box:
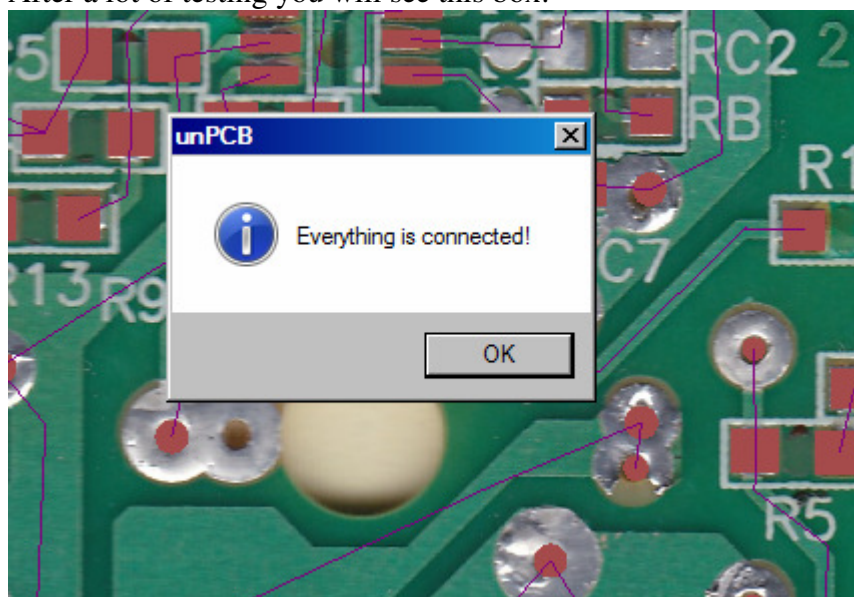
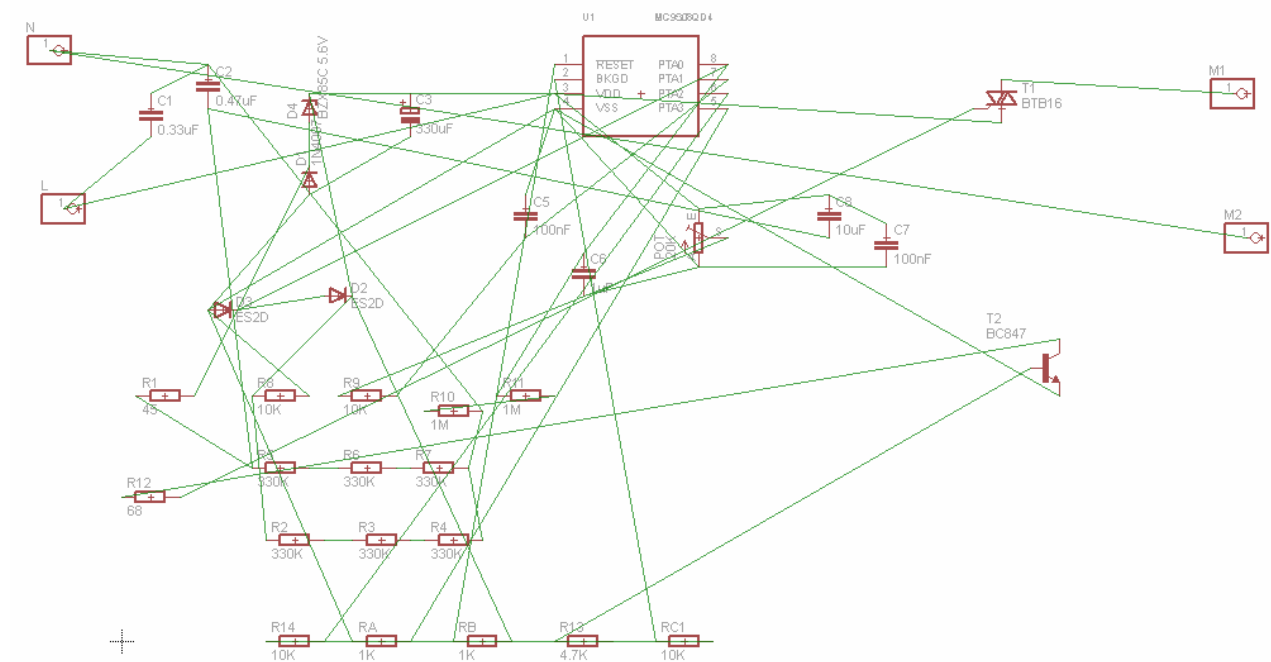Now all connections are tested and you can extract the schematic

6. Extract the schematic

In the schematic window *(view->schematic)* you can click *file->export->Eagle schematic*.

Choose a file name, save and open the schematic in Eagle. You will now need to move all the components so the schematic becomes more readable. Best thing to do is to add GND and VCC symbols to all components connected to it.

Schematic in Eagle looks like this:

After clean up we get: