To understand EPOXY, we will make a sinple exemple.
Note : Passing from a program to another (this file and EPOXY) may cancel pending constructions in EPOXY. It is better to print this file…

In this small tutorial, you learn how to :
- Place a component
- Repeat last action
- Move yourself in the program
- Move and turn objects
- Place electric links
- Edit a component
- Export scheme
- Make an electrical check
- Generate the net
- Create roads, using the net
- Bevel
- Manage short-circuits
- Modify objects

At first, start EPOXY, it can be useful :-) Check that the active window is the right window, which content is white. This window will contain the scheme. The other window (salmon color) will contain the PCB
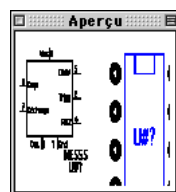
Implant a component

Below, it's the library component :

This window is normaly in the hight right corner of your screen. You can open several windows via the Window menu, in order to have different open library in same time. Open the local menu in this window and choose "Active".
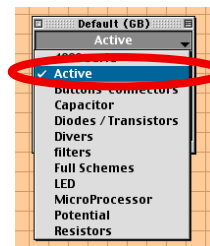

Next go down in the open list and click on the NE555. The small preview window is open :

Double click on the NE555, in the library window (not in preview window). The yellow frame says you "Click on the scheme to place the component."

Otherwise, click in the white window and begin again.
The mouse cursor is ▯ when it is over the window. Click in. The yellow frame say you "Place now the component in the copper.". So, click in the left window to place the component. Your component is displayed in the 2 windows :
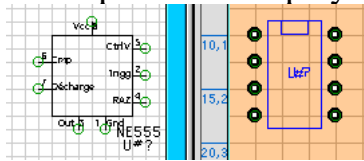
Now we will place the other component, to make an oscillator. Maybe you want to place a component in the scheme, without seeing the PCB ? For that, press the ▯ button. This button indicates to EPOXY that you won't create PCB now. Choose "Window → Full screen (bis)" in the menu. You can zoom by using the wen button.


So you can concentrate yourself on the scheme only…
In order to make an oscillator with a ne-555, you need 2 resistors and 2 capacitors. In the component library (always the window in the right high corner) open the local menu which contains the components headings and choose "resistors". Double click on R1/4W which is the 1/4 Watt resistor, and click in the white sheet near the ne555. To place a second resistor, press on Space key of your keyboard and click again in the sheet…

The SPACE key repeats last action.

Do the same to place the 2 capacitors. Choose for example,"IRC2,54mm" and place it. Press Space key to place the second. Your scheme will nearly look like this :
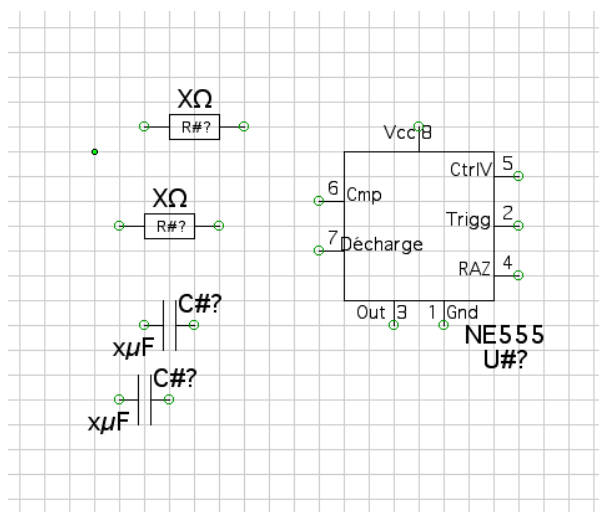
Move the objects
But you wanted the components to be standing… Well, OK. Click on a

component, choose "Object →Rotation 90°" in the menus. EPOXY waits for you to place the rotation center, the yellow window indicates "Click on the rotation center". For example click on the component extremity.
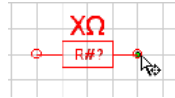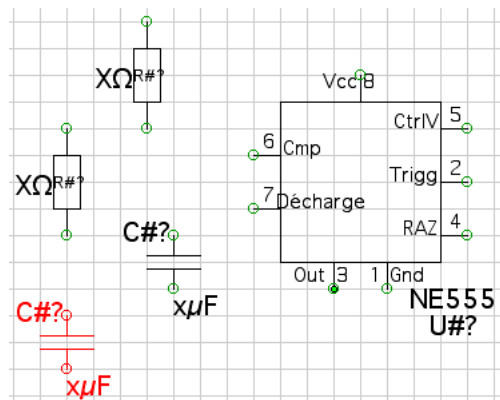

Hum, well… Is it too difficult ?
OK… In order to rotate the following component, open the "Tools Palette" in the "Window" menu. Click on a component, and press one of those 2 buttons.


Is it easier for you ? Yes, but the first method is important because sometimes, we want to rotate a whole group around a precise point. When you have finished this, your scheme nearly looks like this :

Now, you must place the potentials levels : A power**,** a GND, and an ouput for the oscillator. In the "potentials" heading of the library take a GND, a power

"Ref Tension" and a "Label". Select the label and then press on to return it horizontally.
*Other way : You can also choose "Object → Horizontal Miror".Then click to define the symmetry axis (for a horizontal symmetry).*
Your scheme nearly looks like this :
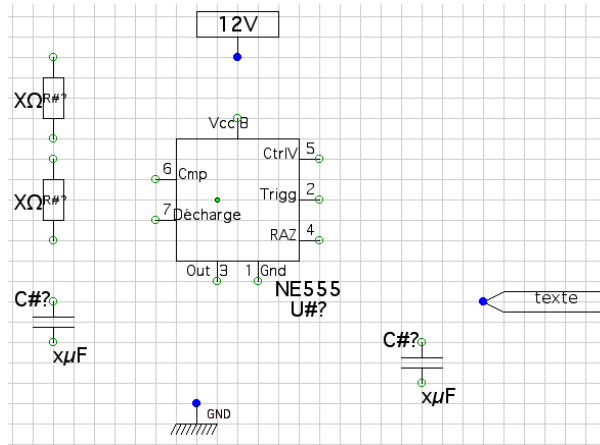
Place the components approximately like on this sheme…

Place wires

Now we will draw the wires. Check that the SuperGrid is disabled. Then, press the button . Note that your cursor is white and the yellow frame says "Wire : Enter the first point." When you make the first click, the cursor become , and EPOXY is ready to enter other points. Click on the point in this order :
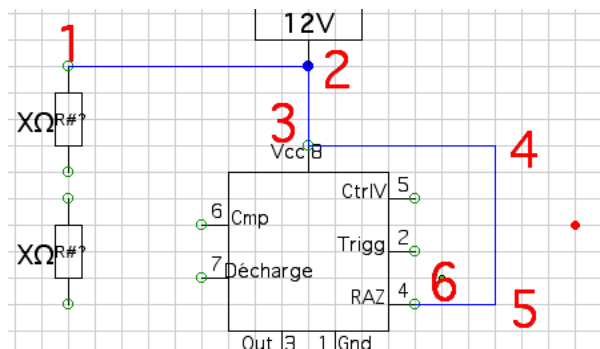
It's not useful to click again on the same point. Press Space bar to make a new wire. Press escape to finish a wire, and your mouse cursor is . To delete a wire (or other object), click on when your cursor is and press the delete key
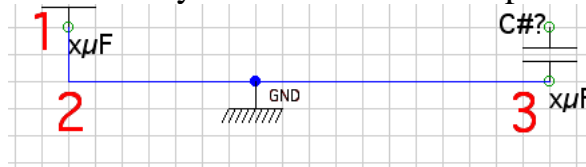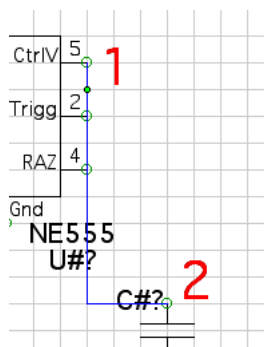
Don't hesitate to adjust the objects position to avoid useless wires. The more wires there are, the more time EPOXY will require to calculate…
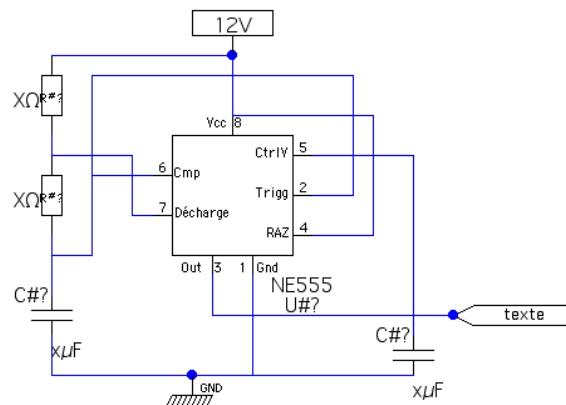I will tell you a guile : To make this wire directly :
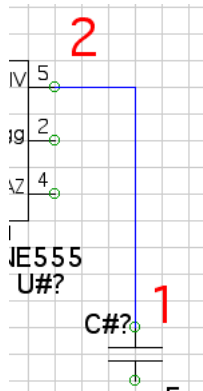
Light the Shift Lock key ⬆ then enter the two points. The wire is automatically orthogonal. EPOXY makes fisrt the longer segment. If you invert the way, so EPOXY does :

Not very beautiful, isn't it ? Now you can finish all the wires. An oscillator with ne-555, you may learn by heart :-).
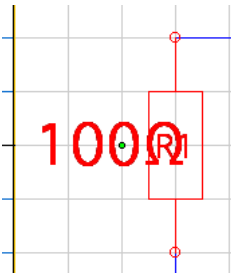
Edit the components

The scheme is not finished. You need to edit the components to enter the good values and references. Choose "Document → Number the components". The half of the work is did. But EPOXY can't guess the component value. Click on a component (a resistor for example) and choose "Object→Property". The window appear :
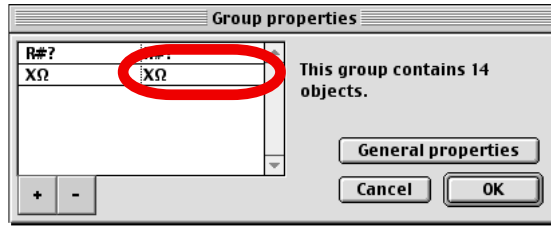
Click in the square at right, below. Enter your value, for example 100Ω. Click OK


If text is over other object, you can move only it if you maintain the alt key pressed. Edit all the components with the same way. Edit also the 12V, replace with 5V and the label, name it "Output"
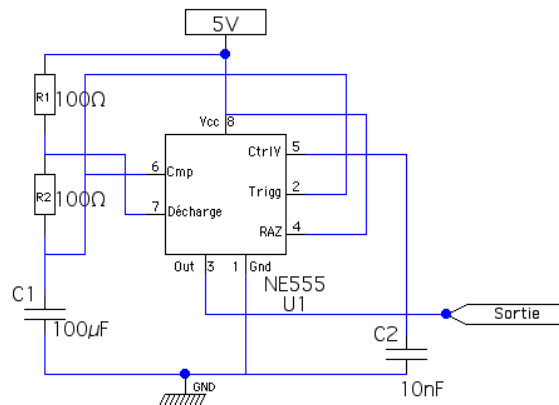
Presto ! Your scheme is almost finished. At the end of the tutorial, you will add the edge and a title block.

Note : If you envisage to use EPOXY to make reports, this paragraph will satisfy you. Here is the method to insert in a document a nice scheme like above. It is necessary to reduce the document size to its minimum. Be sure that your work is in the left high corner of the window. If it is not the case, select all and move all in the right high corner. Then, "Document → Size of the document" and enter the size of your document, the smaller possible, reading on the rule.
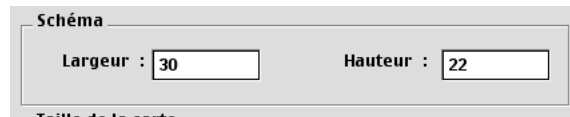
Then choose "File → Export". Click on "clipboard" and check the box "Schematic" and uncheck the box "PCB". Press on Export. Here is your nice draw in the clipboard, ready to be pasted in an Appleworks document or another software…

With the practice it will go faster, you'll see.You'll also learn the shortcuts and the tricks not to do the same work two times. Refer to the manual for the tricks.

Now, you will draw roads on PCB.
At first, you've to place the components in PCB. To do this, you need the scheme and PCB . Do "Window → Juxtapose the windows".

Place the components

All your components, except from the first must be packed down in the left high corner, because you placed the components in the scheme without worry about PCB…
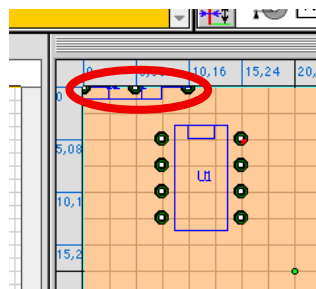

In the scheme, click on a component. Notice that it becomes red in PCB too. Click in the PCB window and move the selected component. Go back to the scheme and click on another component. Then, in the PCB move again the selected component and do so until the 5th. Your components are now disposed in the sheet.
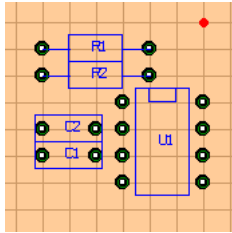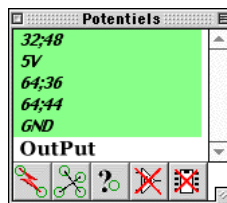
Electrical check

But, we don't know which pin goes where. In the Window menu, choose Potential Palette. The following window is opened :
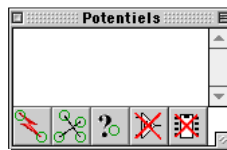
Be sure that the two right buttons are disabled. Click on . The window now contains text lines :


Each line represents an electrical potential (a voltage level) generated by the scheme. The roads not drawn are represented by green lines. The "output" is considered as cabled because it is not connectedat physical component
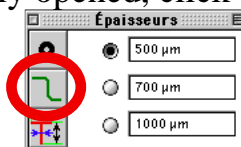
Press now this button . Your components are now linked in the PCB by white lines, the net indicating the roads to draw…

Draw roads

You have to draw roads. In the main palette, choose a copper layer (green) :
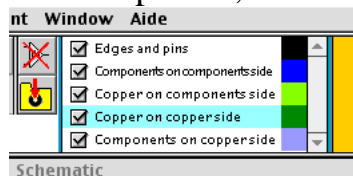
In the Thickness palette, normally opened, click on the middle button.

Choose a road thickness… For this work, 0.02 inches (0,5 mm) are fully sufficient !

You can of course modify the content of the fields.

Then in the main palette, click on the ⌐ button.The yellow frame indicates : "Road / Line : Enter the first point" and the cursor of the mouse become white when it is over the PCB ⬉ . Draw the roads like you drew the wires in the scheme. You have to connect the points linked by white lines. When you have maid some, you can press again ⚡ and possibly ✂ two times waiting for I modify the procedure ;-).

At the end, all the roads must be cabled, and the yellow frame must indicate "You've got finish your card !", when you click on ⚡

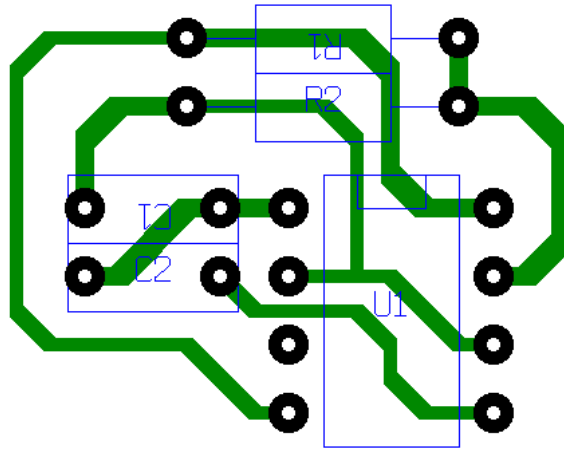Here is your PCB, now you know how to draw roads on a card with EPOXY
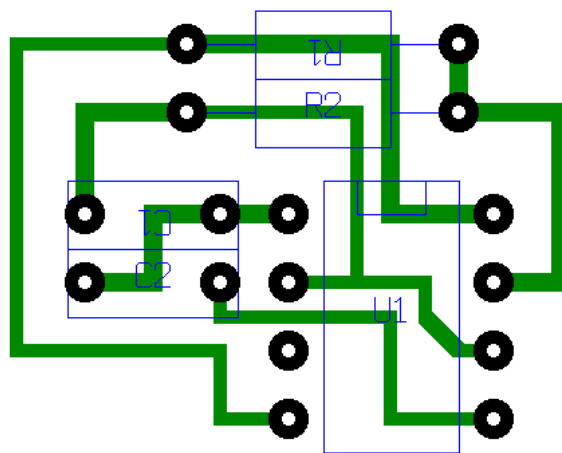
note : You can export the PCB as you do for the scheme.
Isn't it nice my PCB ? OK , let's bevel.

Bevel

Select the roads you want to bevel and do "Object →Bevel".
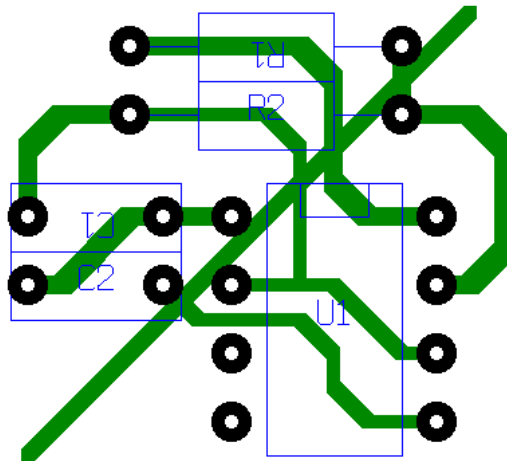*To make a multiple selection, you can use the Shift key.You can also do "Edit → Special selection → Select the work layer".*

Here is the rooting after having bevelled…that's better, don't you think ?
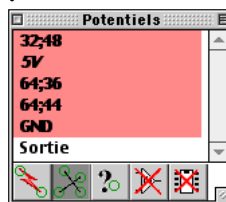
Short-circuits

A little game : Draw a road crosswise as on the picture :

The following window must appear :

If it is not the case, contact-me immediatly :-) . The potential palette must show many red lines as on this picture :

Of course, if on my picture all the roads are in short-circuits, in is not necessarily

the case for you…

Delete this horrible road. Phew, all become again normal !

Now you will add a component in the scheme, and take into account the modification in the PCB.

Objects'Additions and modifications

Click in the scheme or use ⊡ . In the library choose a connector (4 points, it is one superfluous !). Add also the 5V, the label "output" and a GND. You can select and duplicate existing objects
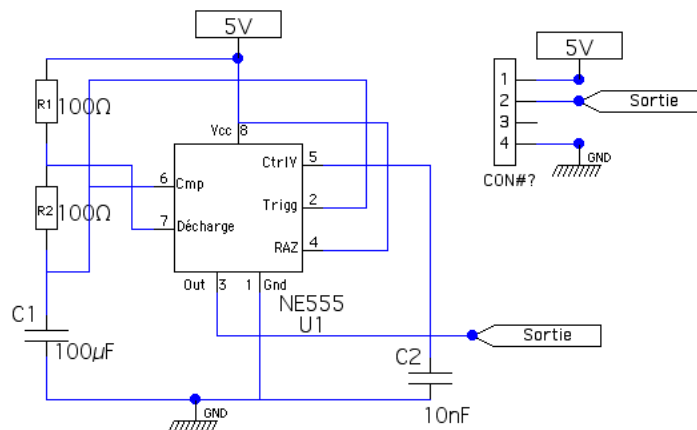

Connect the connector as on the scheme above, edit the potentials.…

Now, as the scheme becomes elaborate it is time to present it correctly. Use the tools ⊡ , △ and T to make frames and place texts. For the text, think to enter the text in the palette before clicking in the sheet… You can also use the library title-block, "Diverse" heading.

You can distort a wire, a edge, or a road (in the PCB) when you select the object. To do this, maintain the Option key pressed, click on a corner or a segment and move it. Try with a corner and with a segment to see the difference.
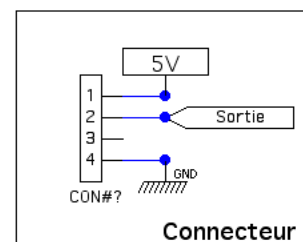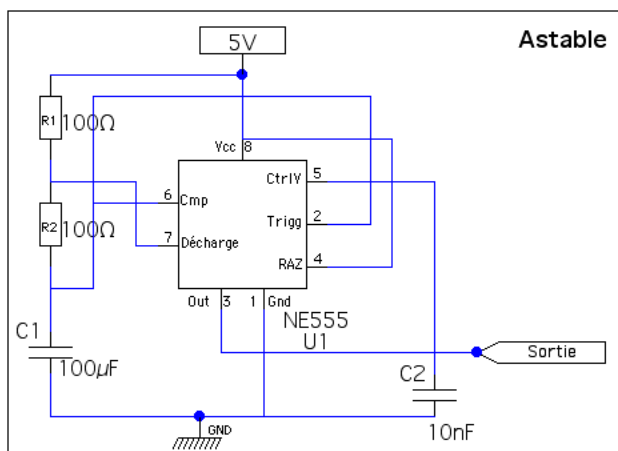
Return in the PCB, place your components, be sure that the buttons ✗ ✗ are

disabled if you enabled them to work in the scheme, press ![icon] and draw the missing roads. Be careful, as the black layer is present on every layer, copper layers included, an object must not make a short-circuit (I say this in particular for the tittle-block) Vous can also use the light green layer (Copper on Component side).
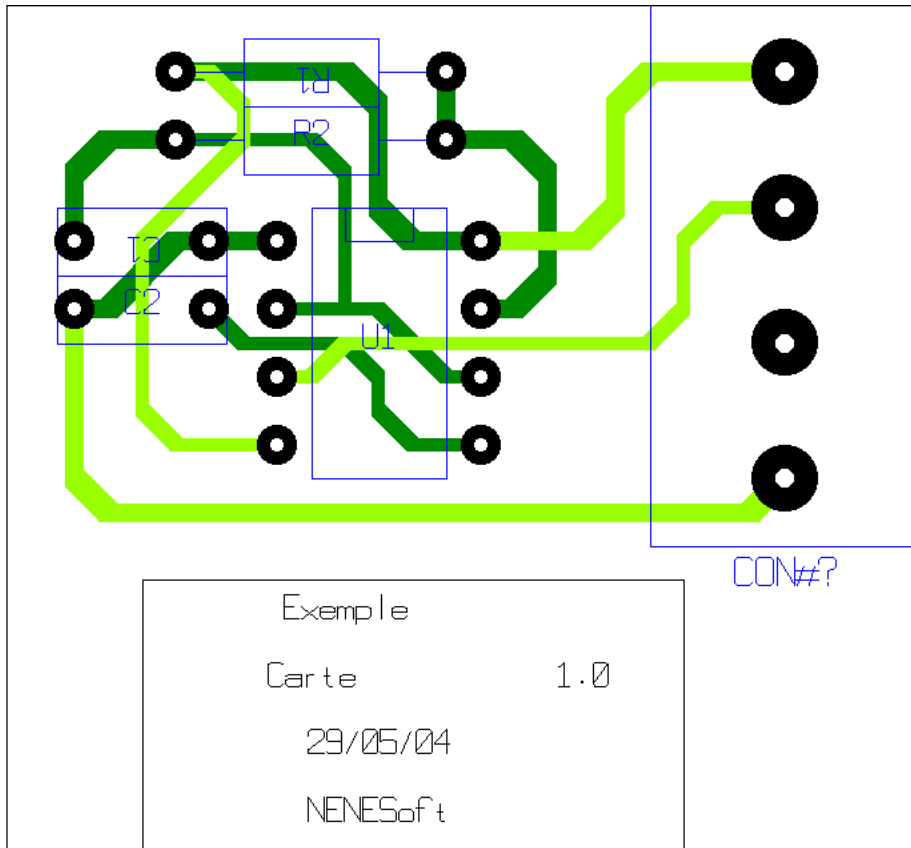
Here is your card, finished !
I let you discover the printing, refer to the manual for more details…