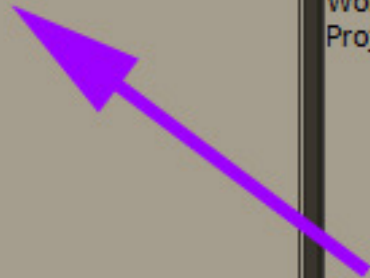
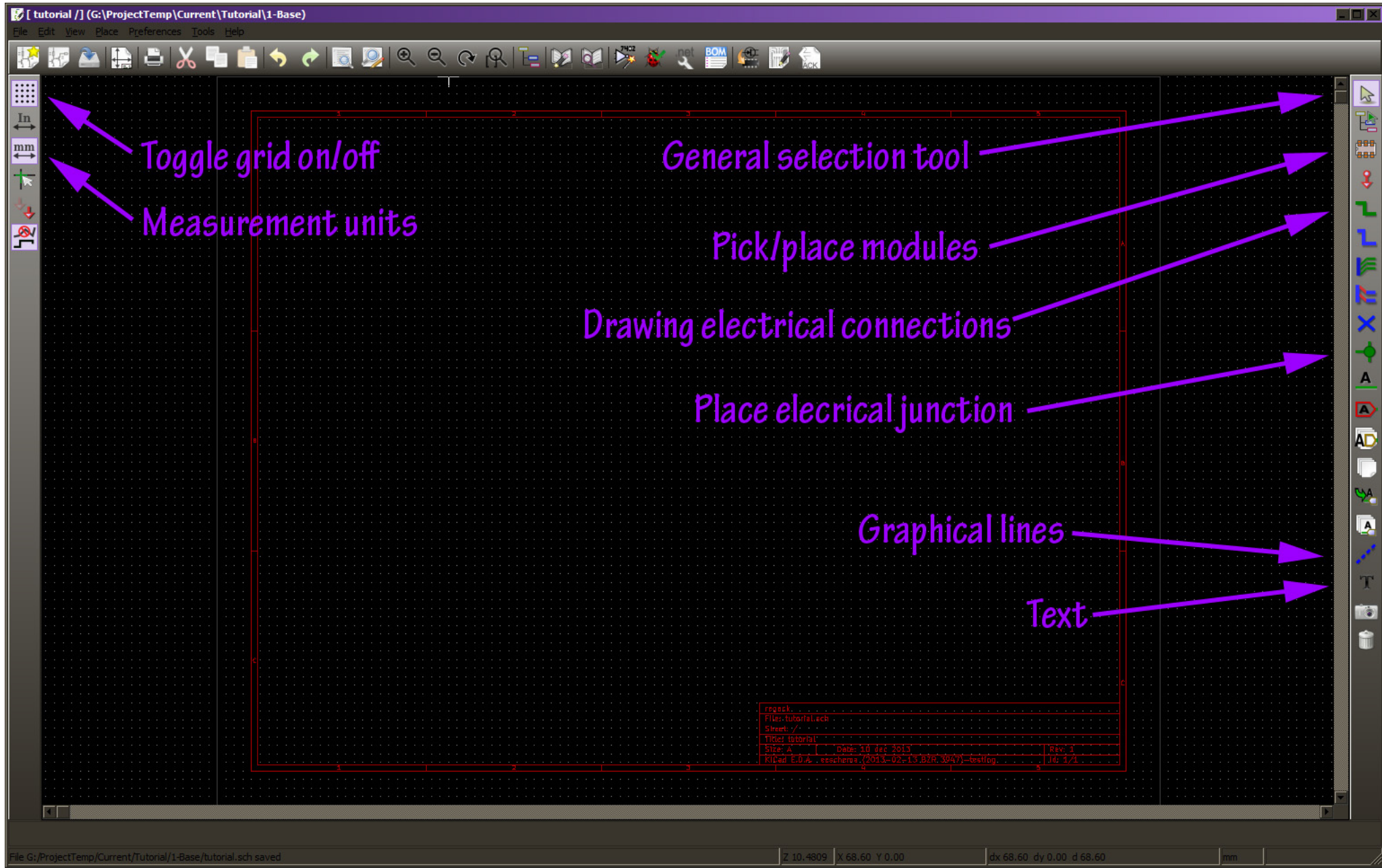
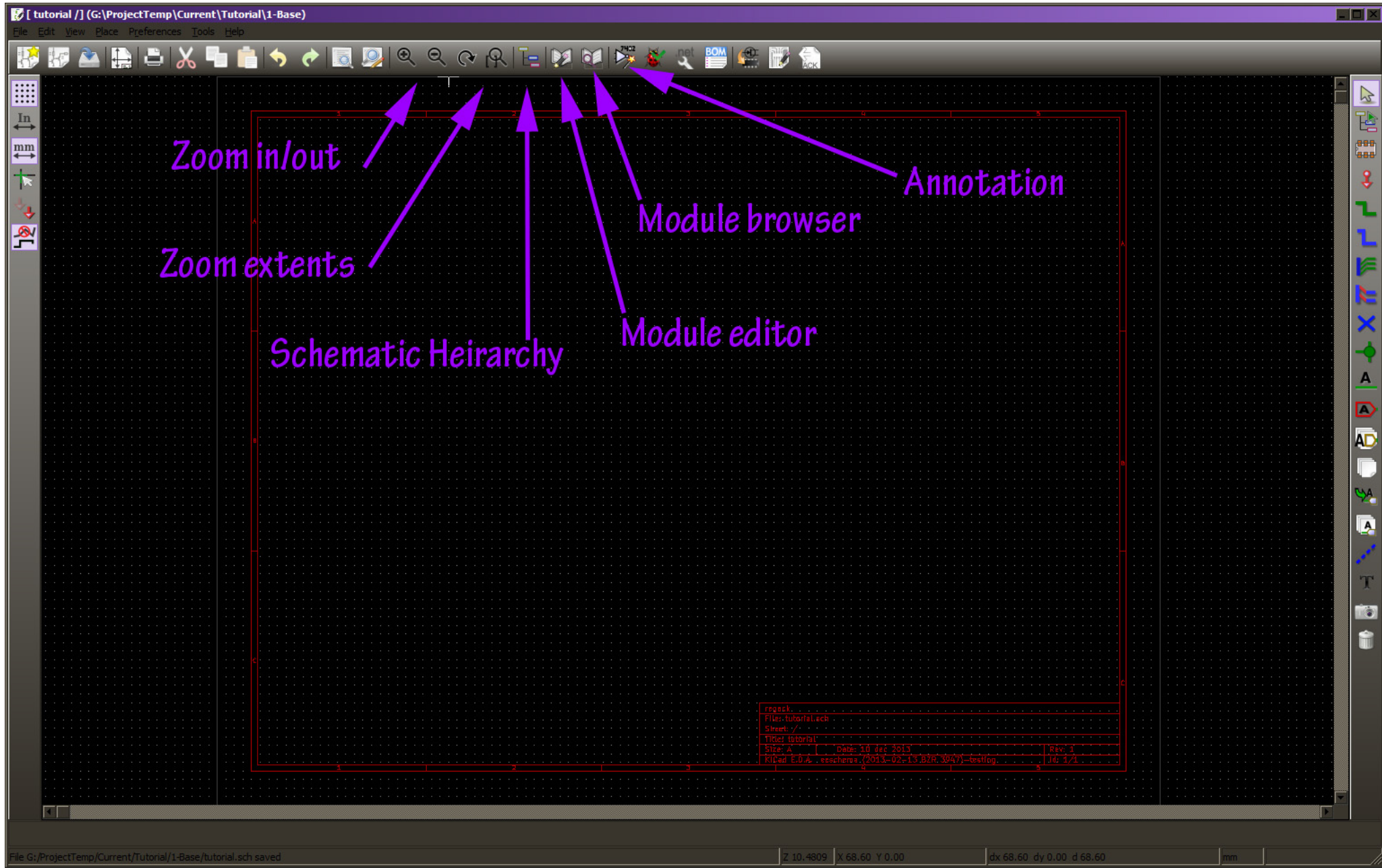


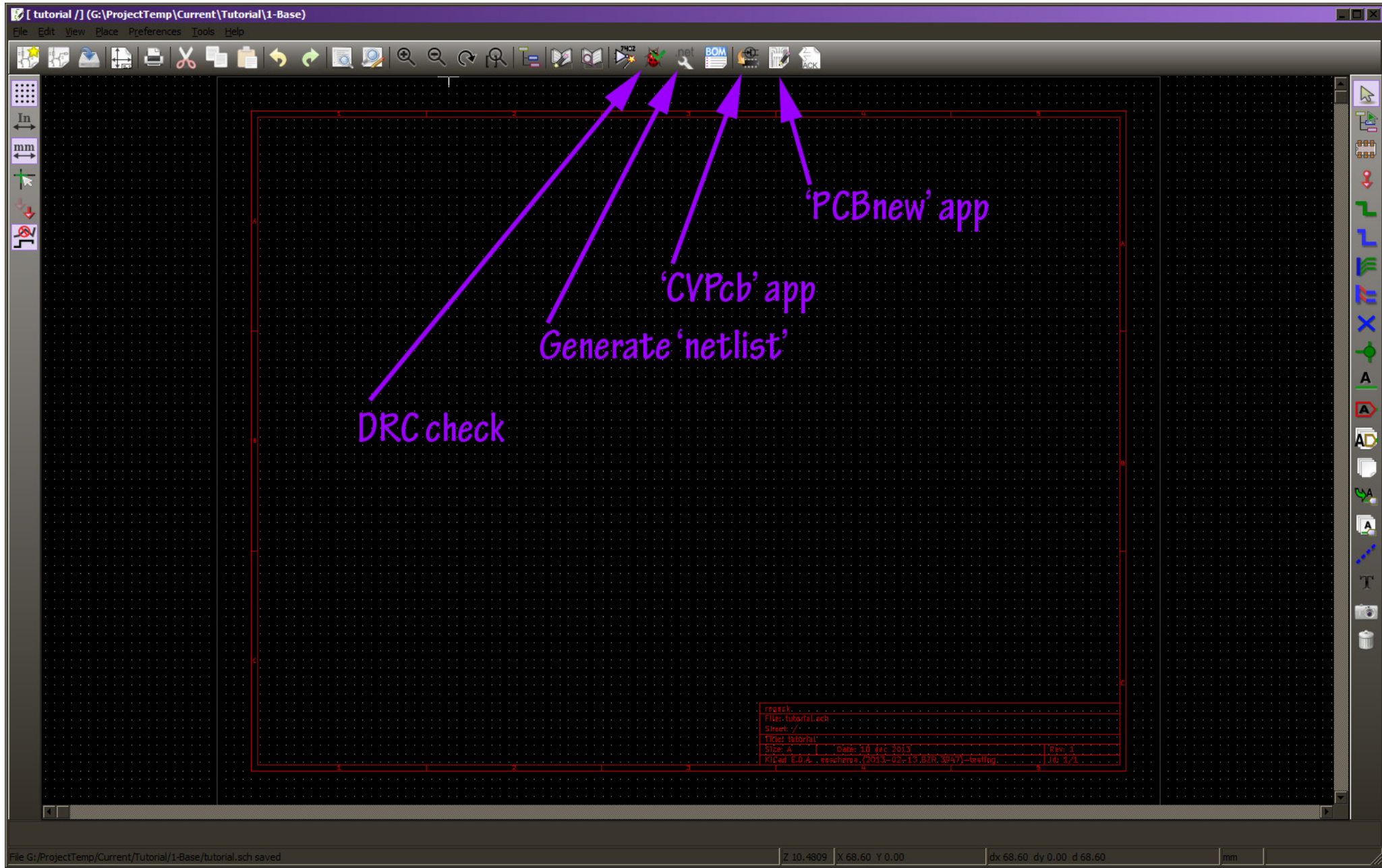
Working dir: G:\ProjectTemp\Current\Tutorial\1-Base
Project: tutorial.pro

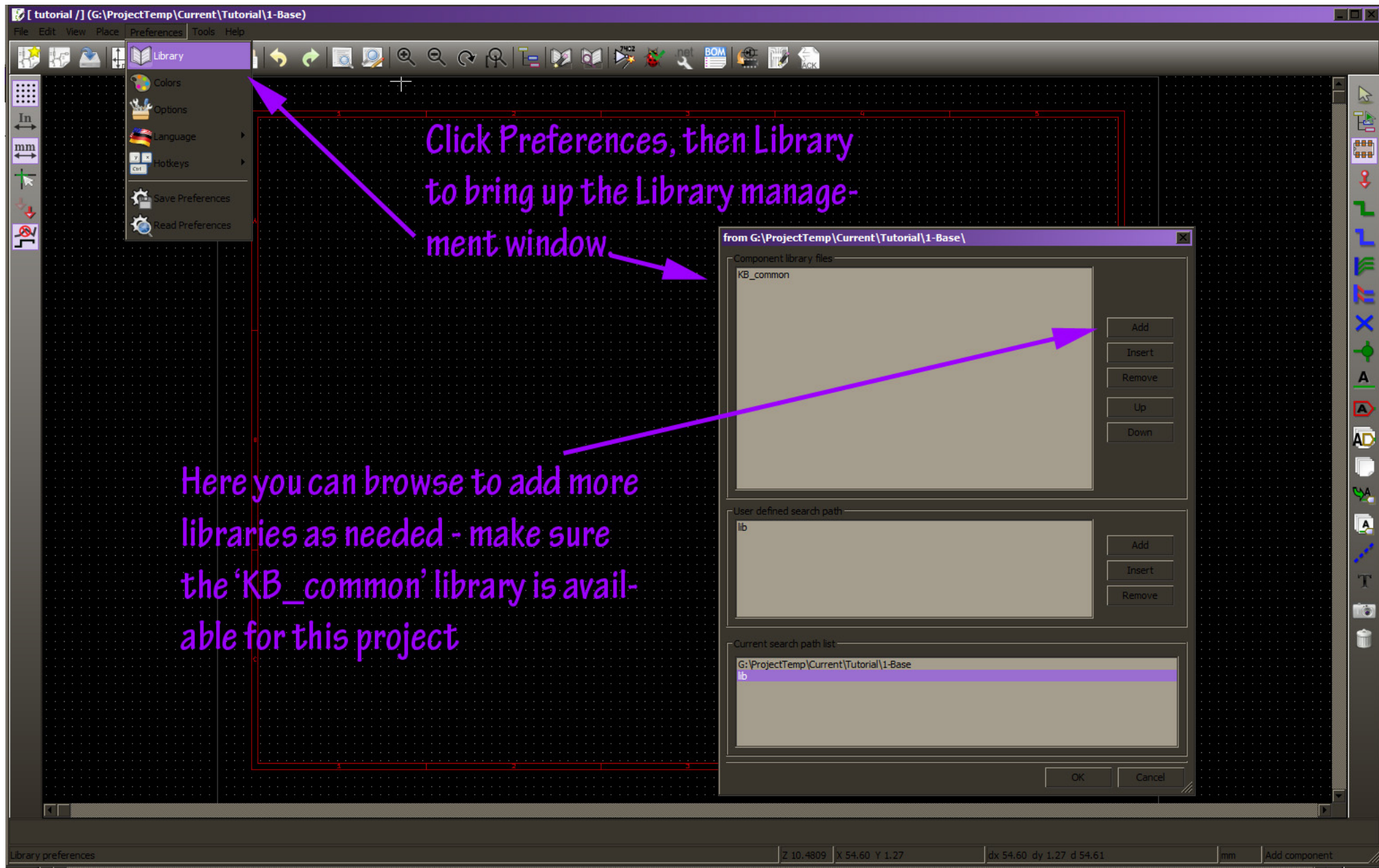


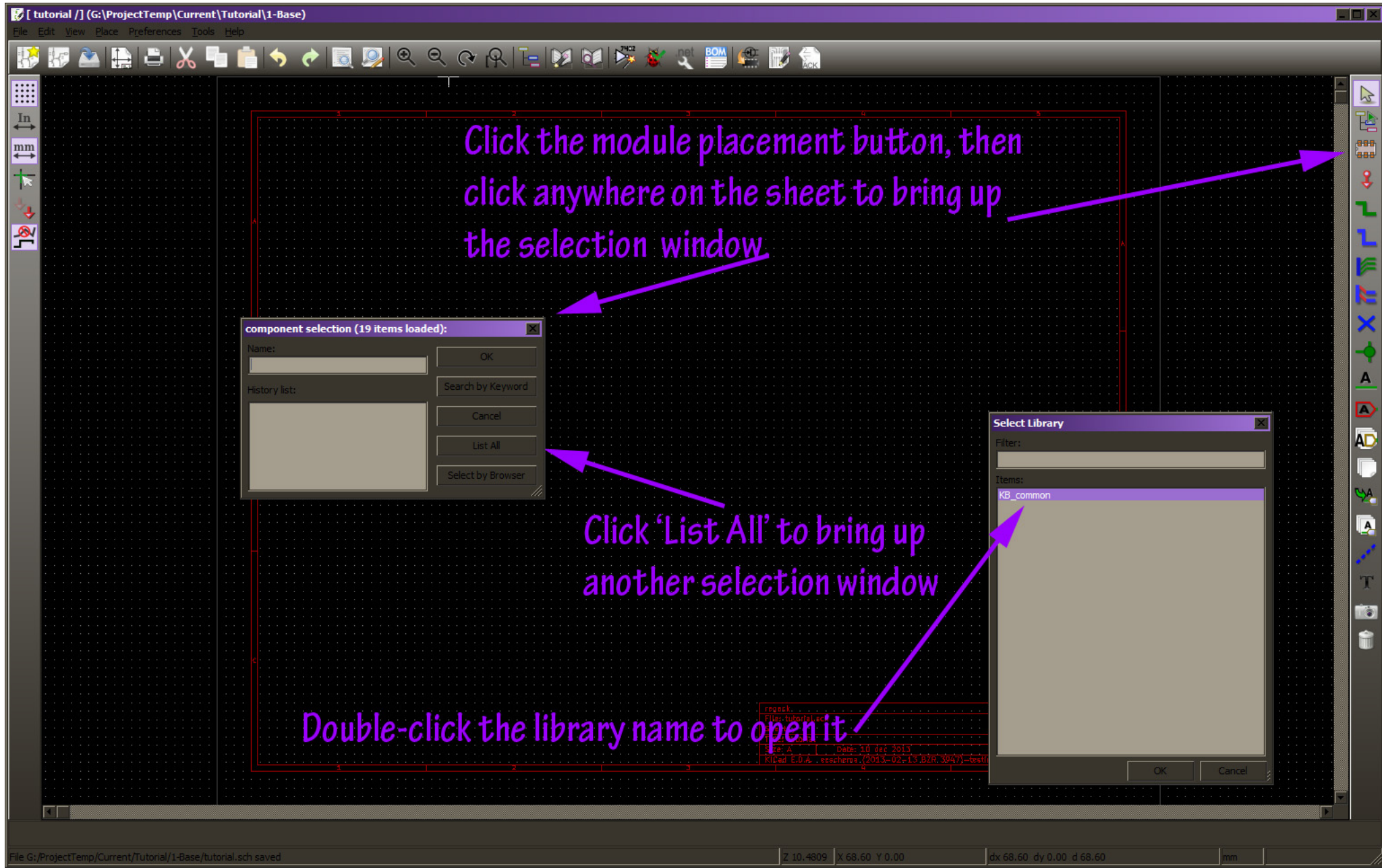
Double click to open

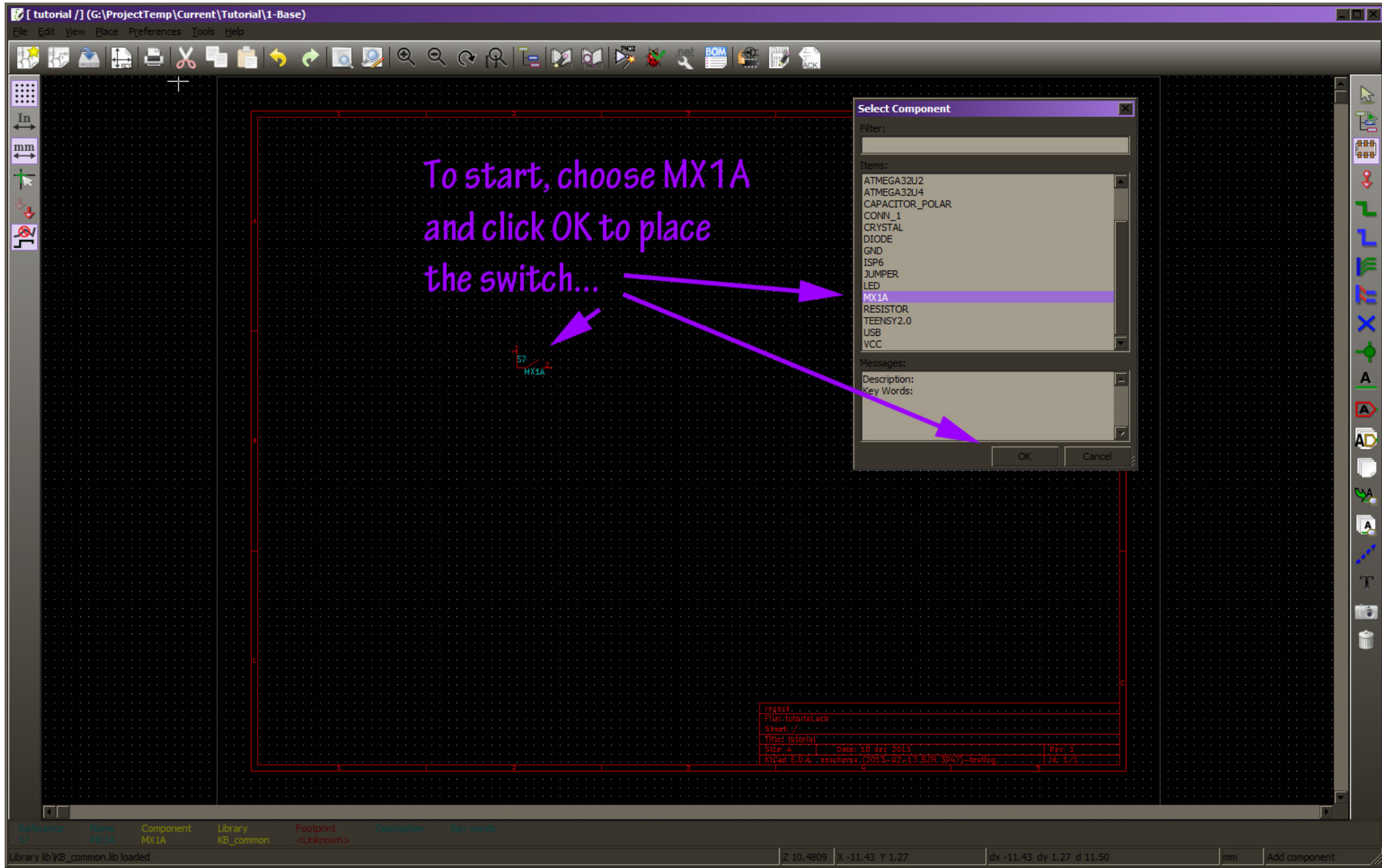


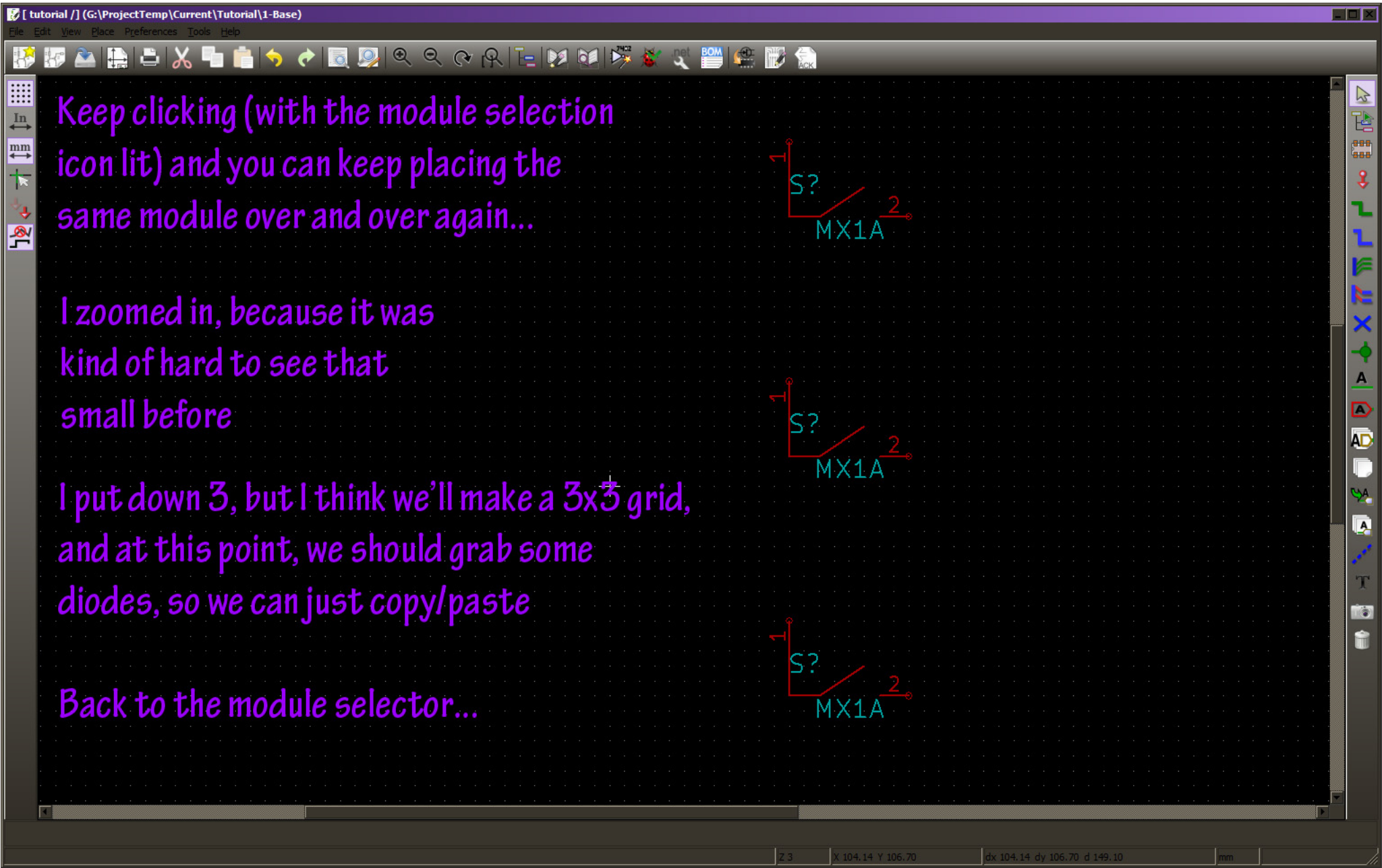










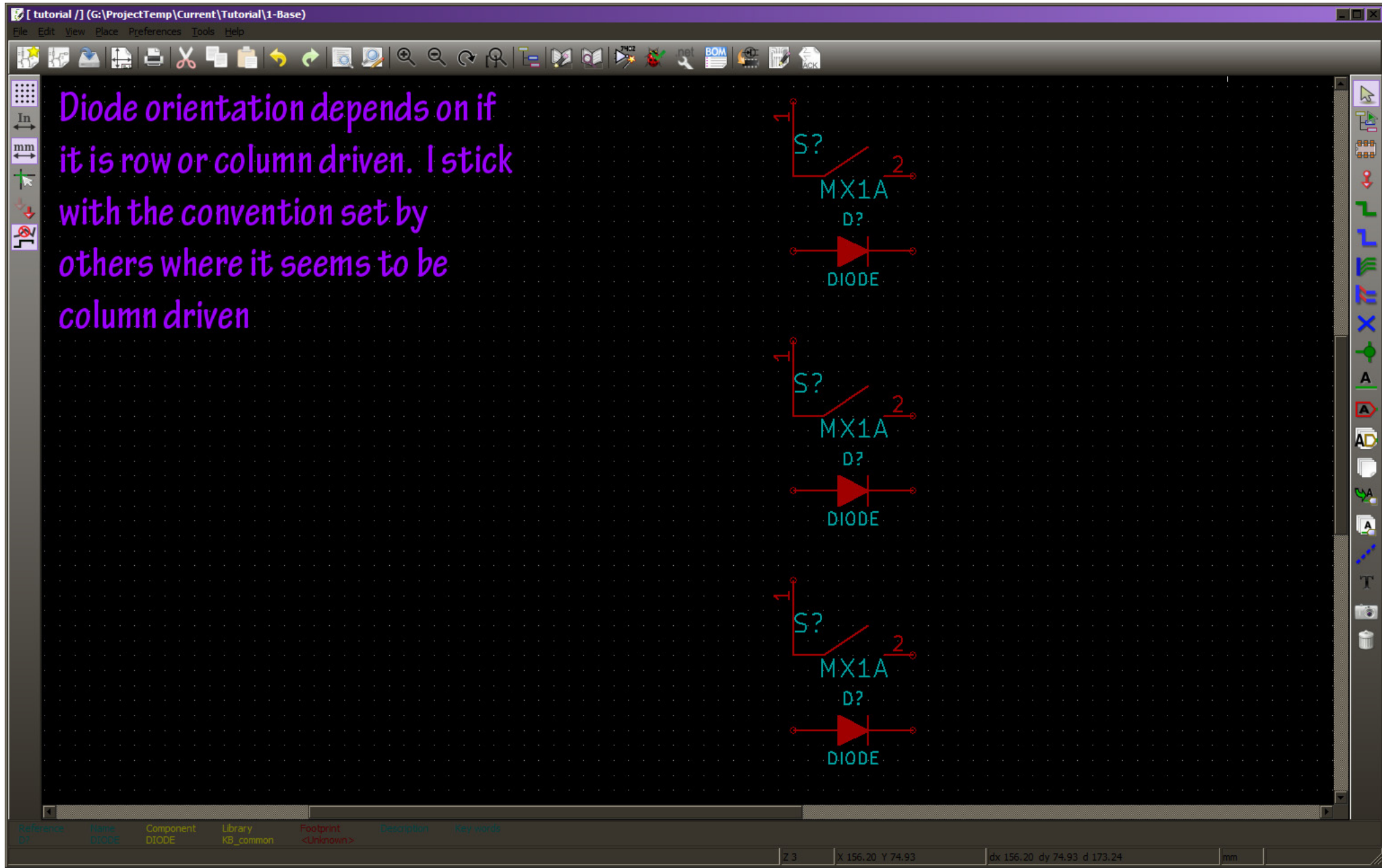


Keep clicking (with the module selection icon lit) and you can keep placing the same module over and over again...

I zoomed in, because it was kind of hard to see that small before

I put down 3, but I think we'll make a 3x3 grid, and at this point, we should grab some diodes, so we can just copy/paste

Back to the module selector...



Click the interconnect tool and draw some connections

Next we'll do some basic annotation, before expanding the matrix. Once we set up the baseline, we can have the auto-annotation work for us...

The screenshot shows a PCB design software window with a dark background and a grid. Three diode components are arranged vertically. Each component is labeled 'MX1A' and 'DIODE'. The first component has a red wire connected to a terminal labeled 'S?' and another red wire connected to a terminal labeled '2'. A purple arrow points from the text 'Click the interconnect tool and draw some connections' to the interconnect tool icon on the right sidebar. The second component is identical to the first. The third component is identical to the first but has a small white crosshair symbol at the end of its second wire. The software interface includes a menu bar at the top with 'File', 'Edit', 'View', 'Place', 'Preferences', 'Tools', and 'Help'. A toolbar with various icons is located below the menu bar. A vertical toolbar on the right side contains icons for various tools, including the interconnect tool. At the bottom, there is a status bar with a table of component information and a coordinate display.

Reference	Name	Component	Library	Footprint	Description	Key words
D?	DIODE	DIODE	KB_common	<Unknown>		

Z 3 X 129.54 Y 128.27 dx 129.54 dy 128.27 d 182.30 mm Add wire

Put the cursor over 'S?' and hit 'U' on the keyboard (alternatively, you can right-click and get the same menu, but 'U' to edit the 'field reference is usually faster).

For each row, change the reference for the switch and the diode appropriately:

S1:? & D1:?
 S2:? & D2:?
 S3:? & D3:?

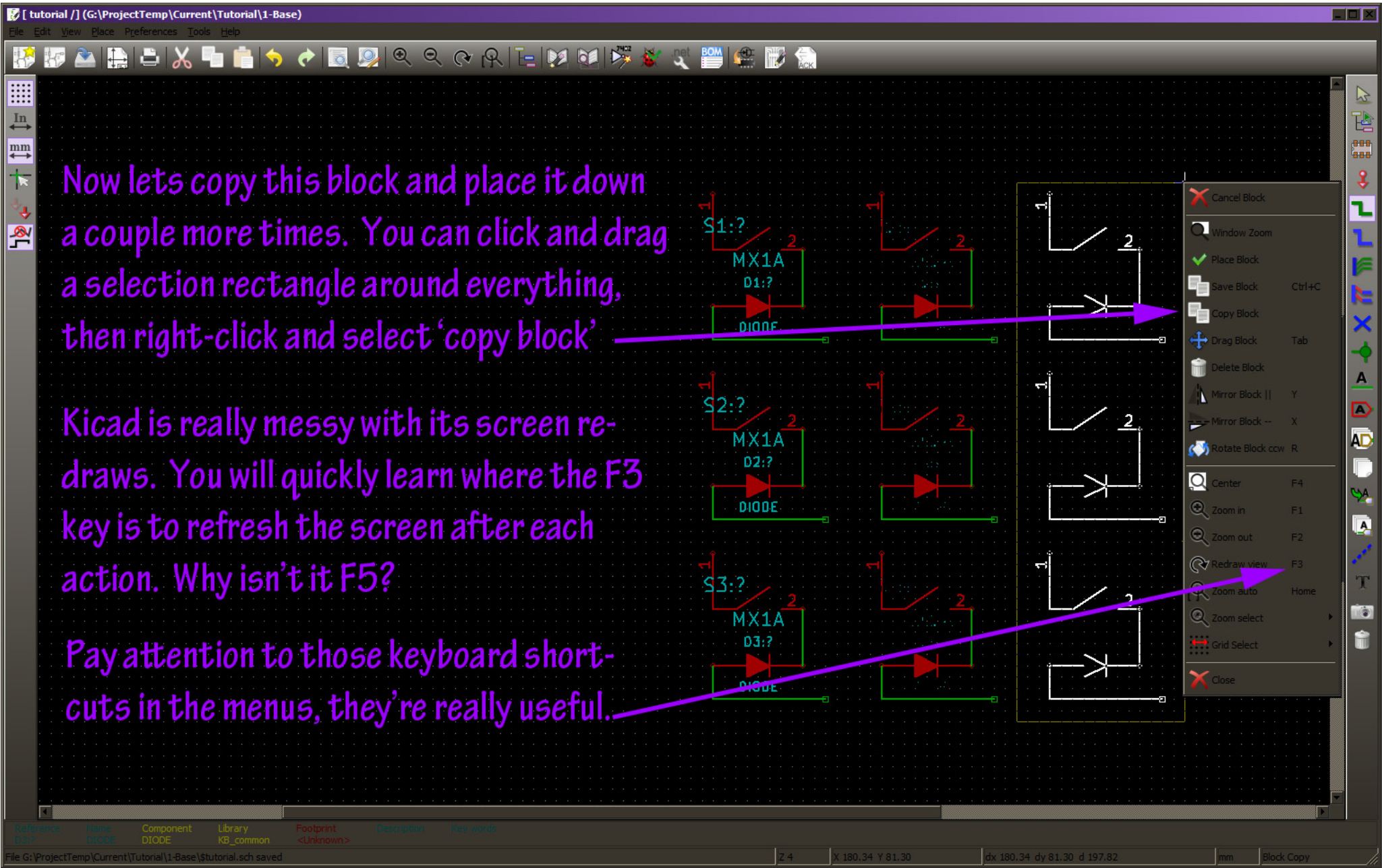
These are only partially annotated, the application can take care of the rest for us.

First, draw some more wires - double click to force the line to stop if there is no connection point for it. It will show the little square 'not-connected' icon on the end.

The screenshot shows a PCB design software window with a schematic diagram. The diagram consists of three rows of components connected by wires. Each row contains a switch (labeled S1:?, S2:?, S3:?) and a diode (labeled D1:?, D2:?, D3:?) connected to a component labeled MX1A. A dialog box titled 'Edit Reference Field' is open, showing the text 'S1:?' and 'D1:?' being edited. The dialog box has fields for 'Text' and 'Size (mm)', and options for 'Text Options', 'Text Shape', 'Horiz. Justify', and 'Vert. Justify'. A purple arrow points from the text 'S1:? & D1:?' to the dialog box. Another purple arrow points from the text 'It will show the little square 'not-connected' icon on the end.' to a small square icon on a wire in the bottom row of the schematic.

Reference	Name	Component	Library	Footprint	Description	Key words
D1:?	DIODE	DIODE	KB_common	<Unknown>		

File G:\ProjectTemp\Current\Tutorial\1-Base\Tutorial.sch saved | Z 3 | X 132.10 Y 76.20 | dx 132.10 dy 76.20 d 152.50 | mm | Add wire



Click 'Annotation' to bring up the annotation window

Here you can set your options.

These are good, we'll leave it as is, then click 'Annotation'

Check the popup, then say ok and then 'close' the annotation window, and all items will be full annotated in order...

The screenshot shows a software window titled "[tutorial /] (G:\ProjectTemp\Current\Tutorial\1-Base)". The menu bar includes File, Edit, View, Place, Preferences, Tools, and Help. The toolbar contains various icons for file operations, editing, and simulation. The main workspace displays a schematic diagram with three identical circuit blocks. Each block consists of a switch labeled S1:?, a diode labeled D1:?, and a component labeled MX1A. The components are connected in a specific configuration. A dialog box titled "Annotate Schematic" is open in the center. It has the following sections:

- Scope:
 - Use the entire schematic
 - Use the current page only
- Annotation Order:
 - Sort components by X position
 - Sort components by Y position
- Annotation Choice:
 - Use first free number in schematic
 - Start to sheet number*100 and use first free number
 - Start to sheet number*1000 and use first free number
- Dialog:
 - Automatically close this dialog
 - Silent mode

The dialog box has three buttons at the bottom: "Close", "Clear Annotation", and "Annotation". The "Annotation" button is highlighted with a red arrow. The schematic diagram shows the components being annotated with red text labels. The labels are S1:?, MX1A, D1:?, and DIODE. The components are connected in a specific configuration. The background is a dark grid.

Reference	Name	Component	Library	Footprint	Description	Key words
D1:?	DIODE	DIODE	KB_common	<Unknown>		

Z 4 X 175.26 Y 80.00 dx 175.26 dy 80.00 d 192.66 mm

[tutorial /] (G:\ProjectTemp\Current\Tutorial\1-Base)

File Edit View Place Preferences Tools Help

In mm

Yay, annotated bits... lets place some pins, and draw more wires...

Select Component

Filter:

Items:

- 3PIN
- 4PIN
- 6PIN
- ANYTHING
- ATMEGA32U2
- ATMEGA32U4
- CAPACTOR_POLAR
- CONN_1
- CRYSTAL
- DIODE
- GND
- ISP6
- JUMPER
- LED
- MX1A

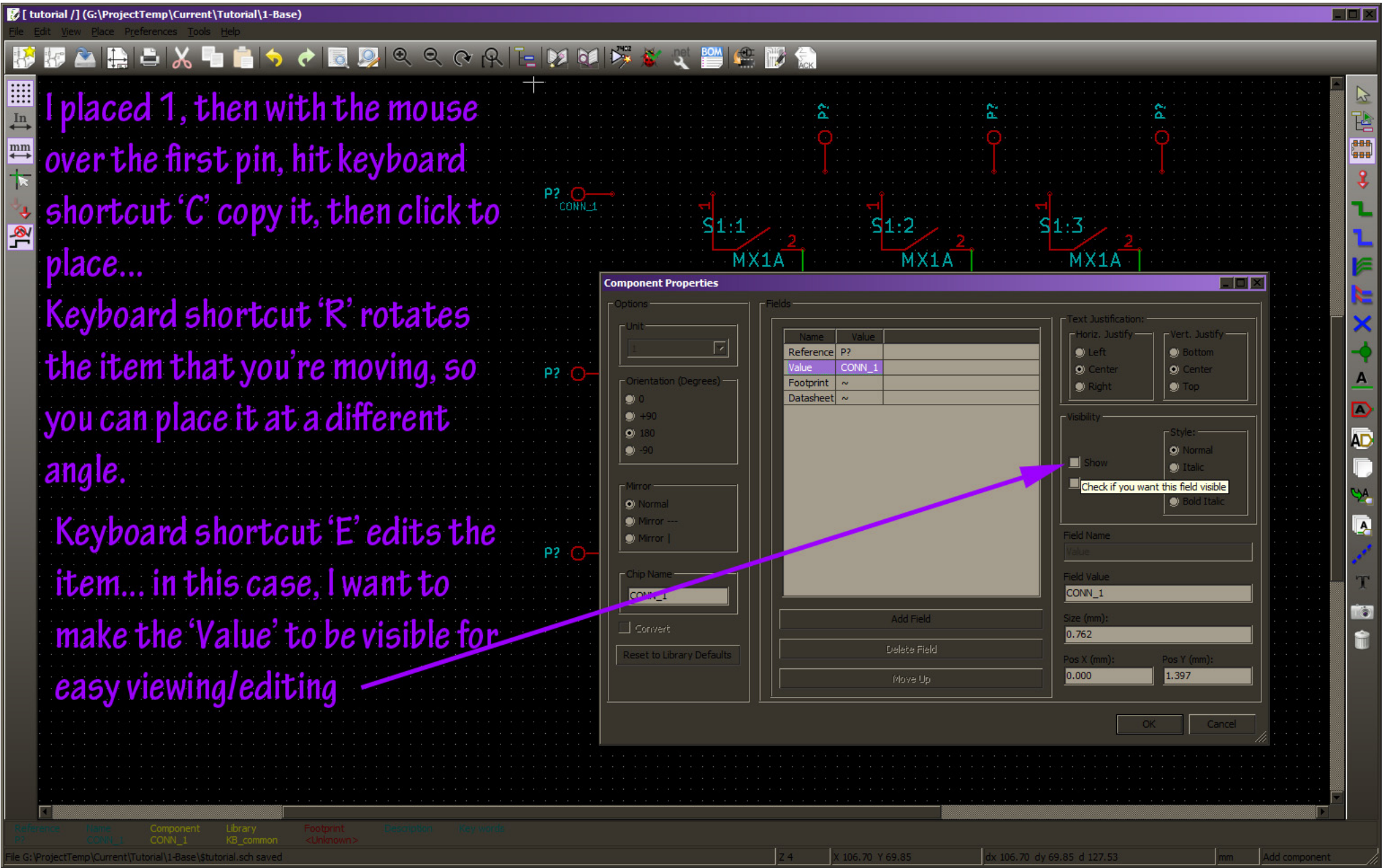
Messages:

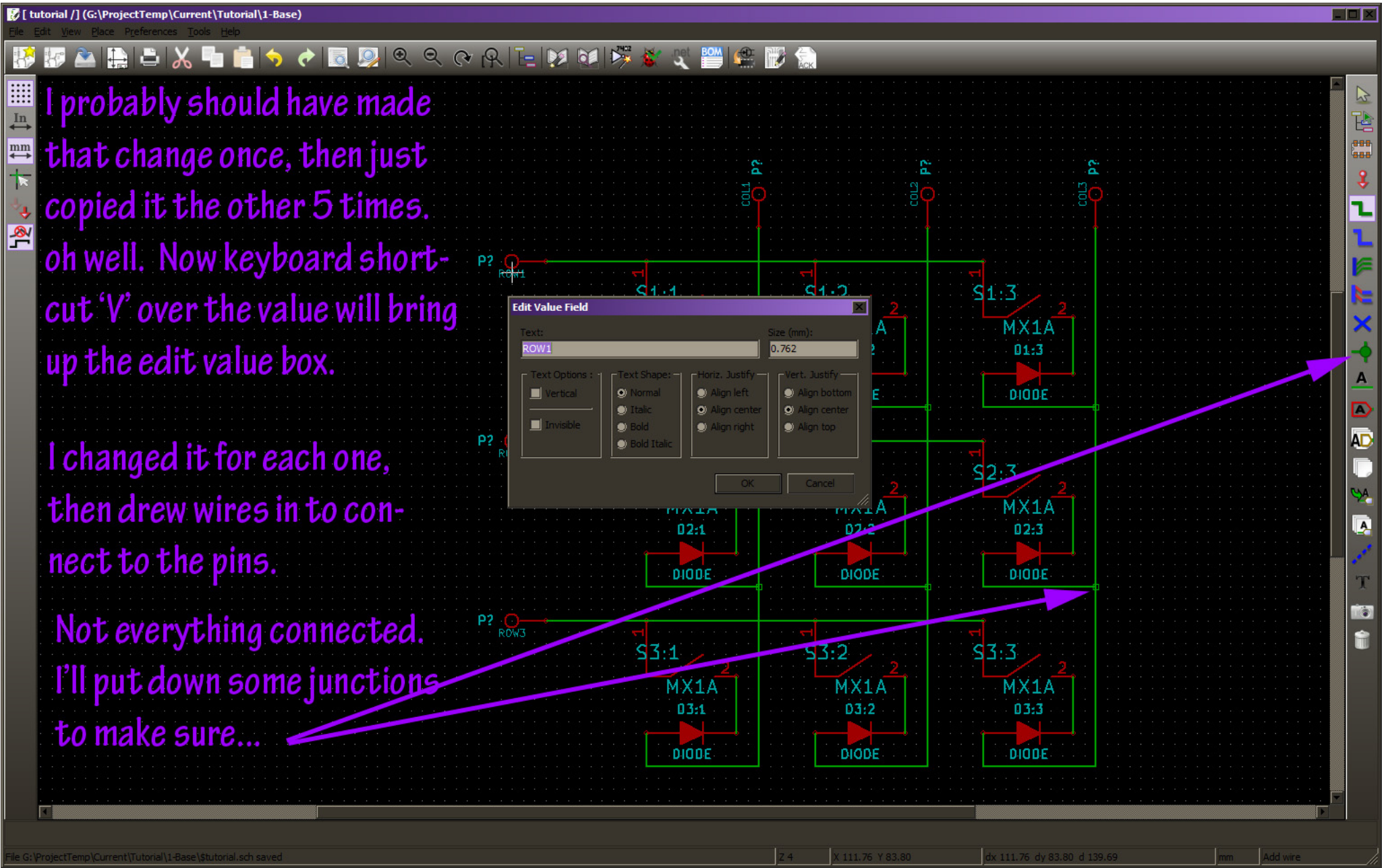
OK Cancel

Reference	Name	Component	Library	Footprint	Description	Key words
D1:1	DIODE	DIODE	KB_common	<Unknown>		

File G:\ProjectTemp\Current\Tutorial\1-Base\tutorial.sch saved

Z 4 X 130.80 Y 72.40 dx 130.80 dy 72.40 d 149.50 mm

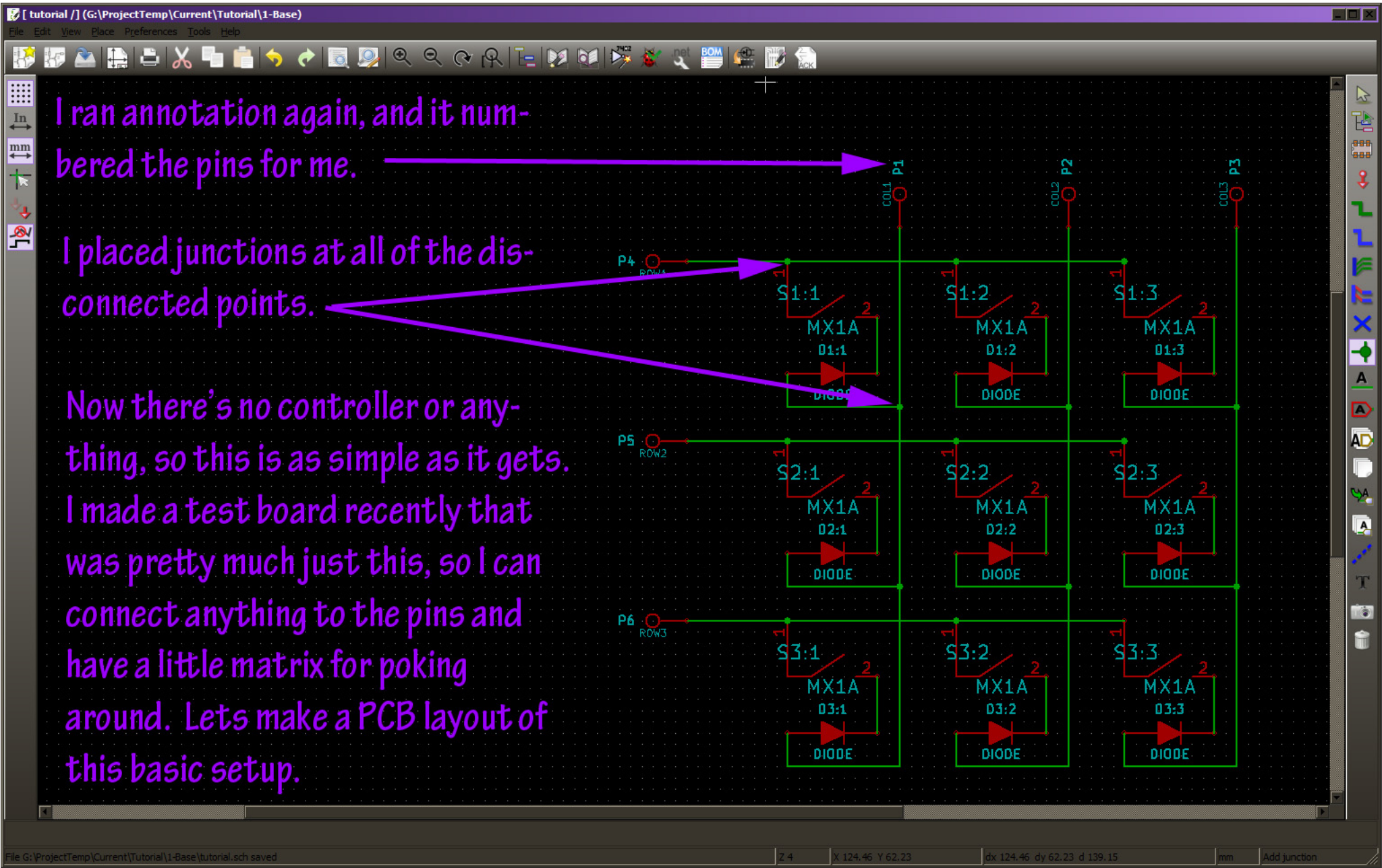


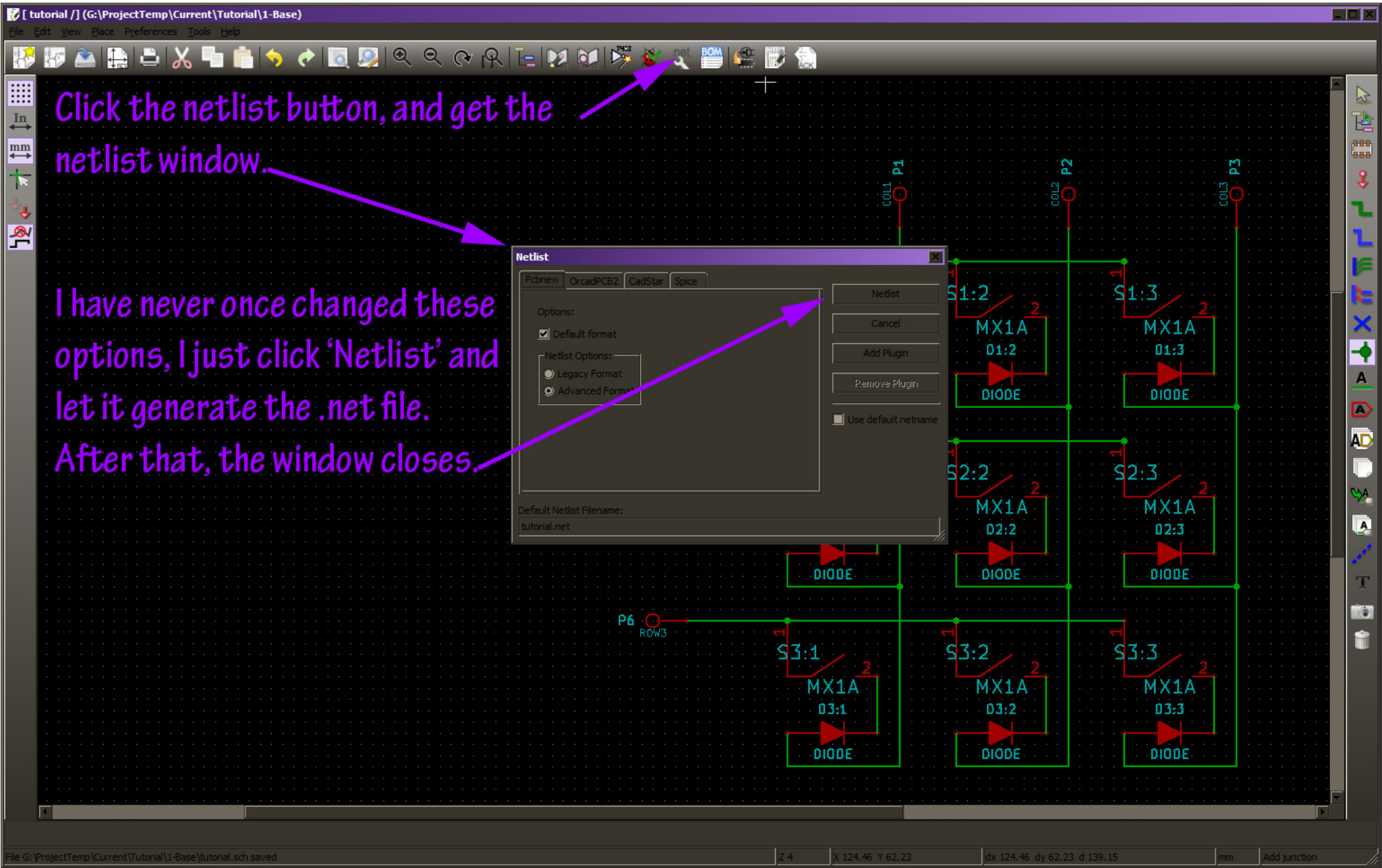


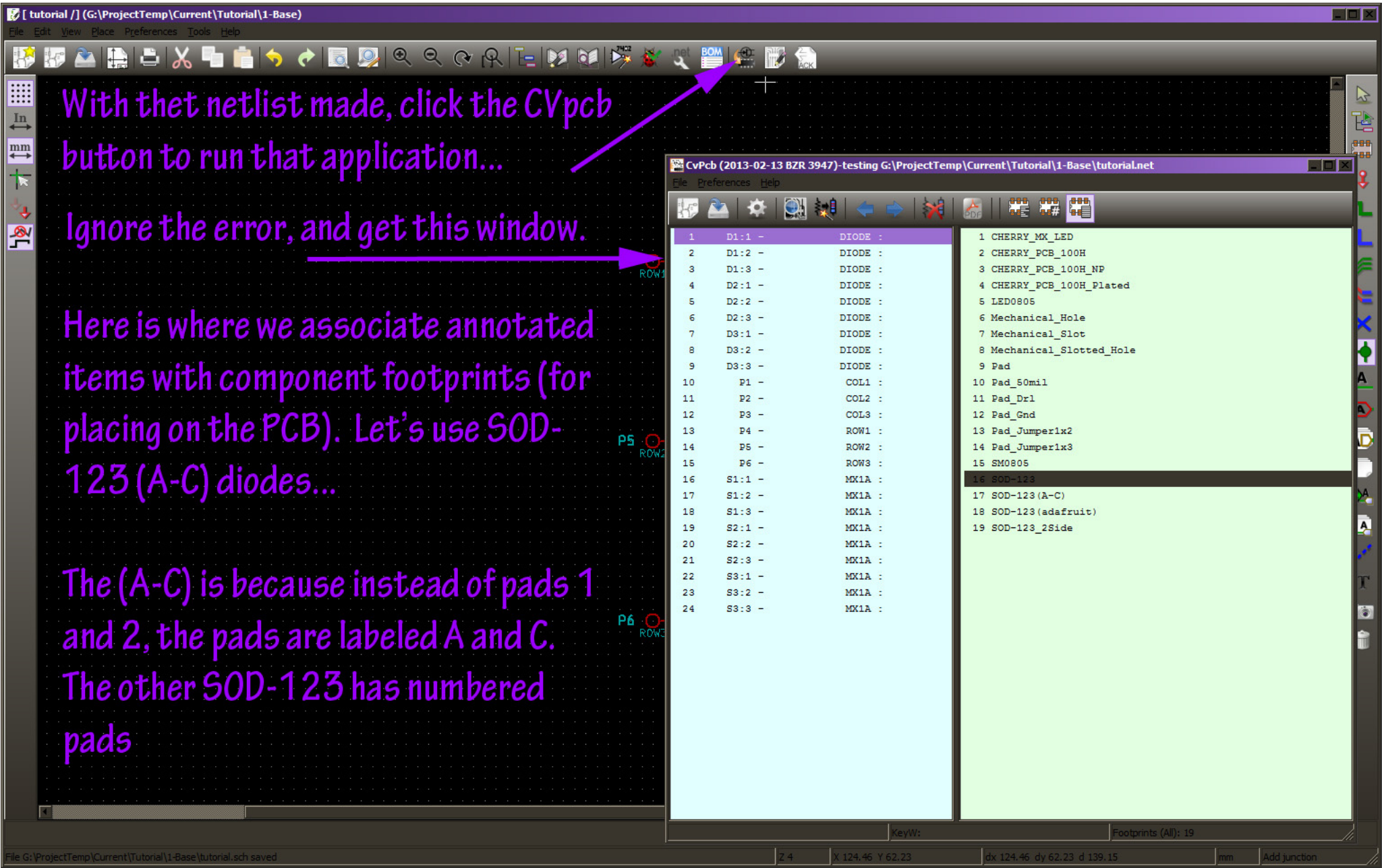
I probably should have made that change once, then just copied it the other 5 times. oh well. Now keyboard short-cut 'V' over the value will bring up the edit value box.

I changed it for each one, then drew wires in to connect to the pins.

Not everything connected. I'll put down some junctions to make sure...





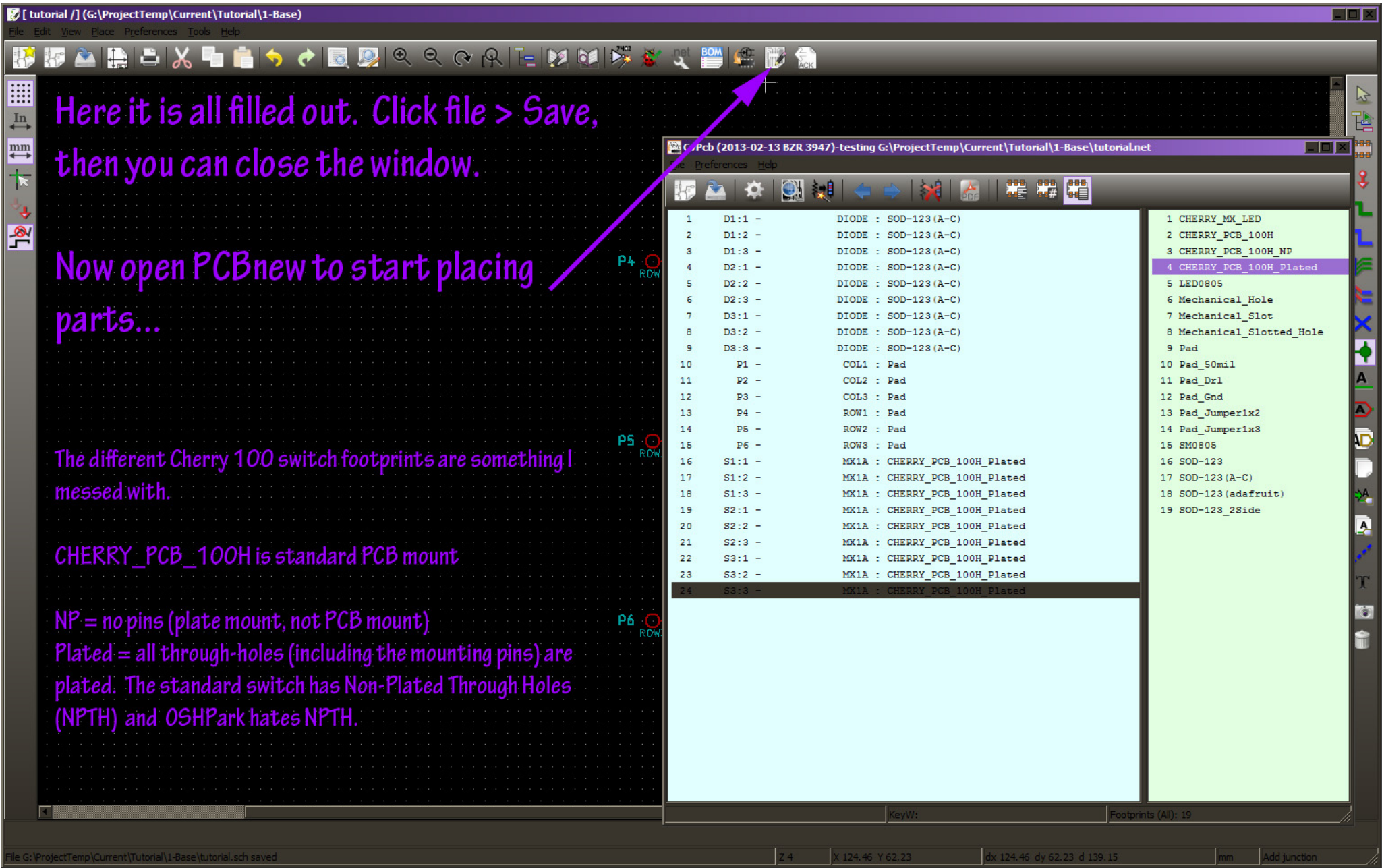


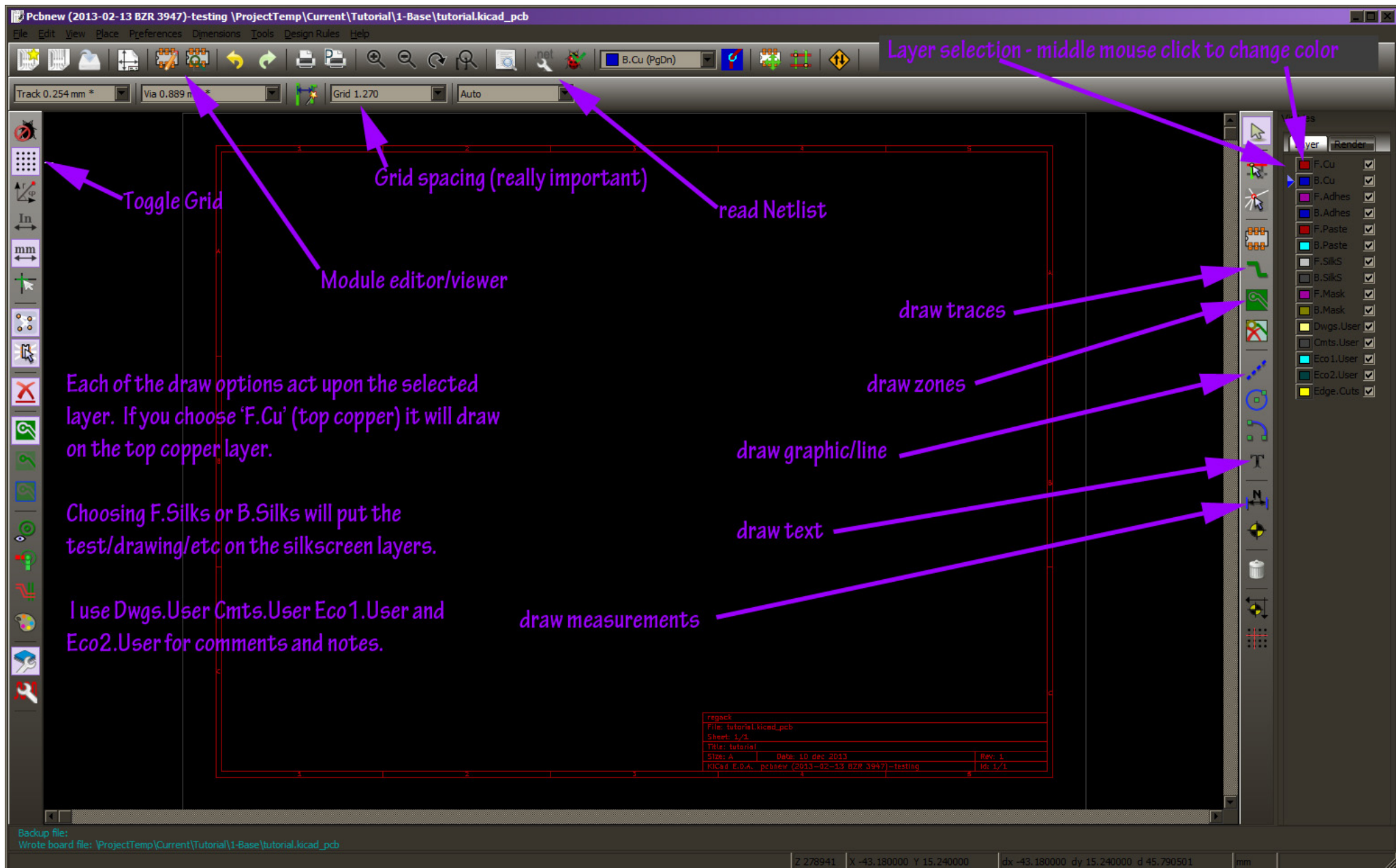
With that netlist made, click the CVpcb button to run that application...

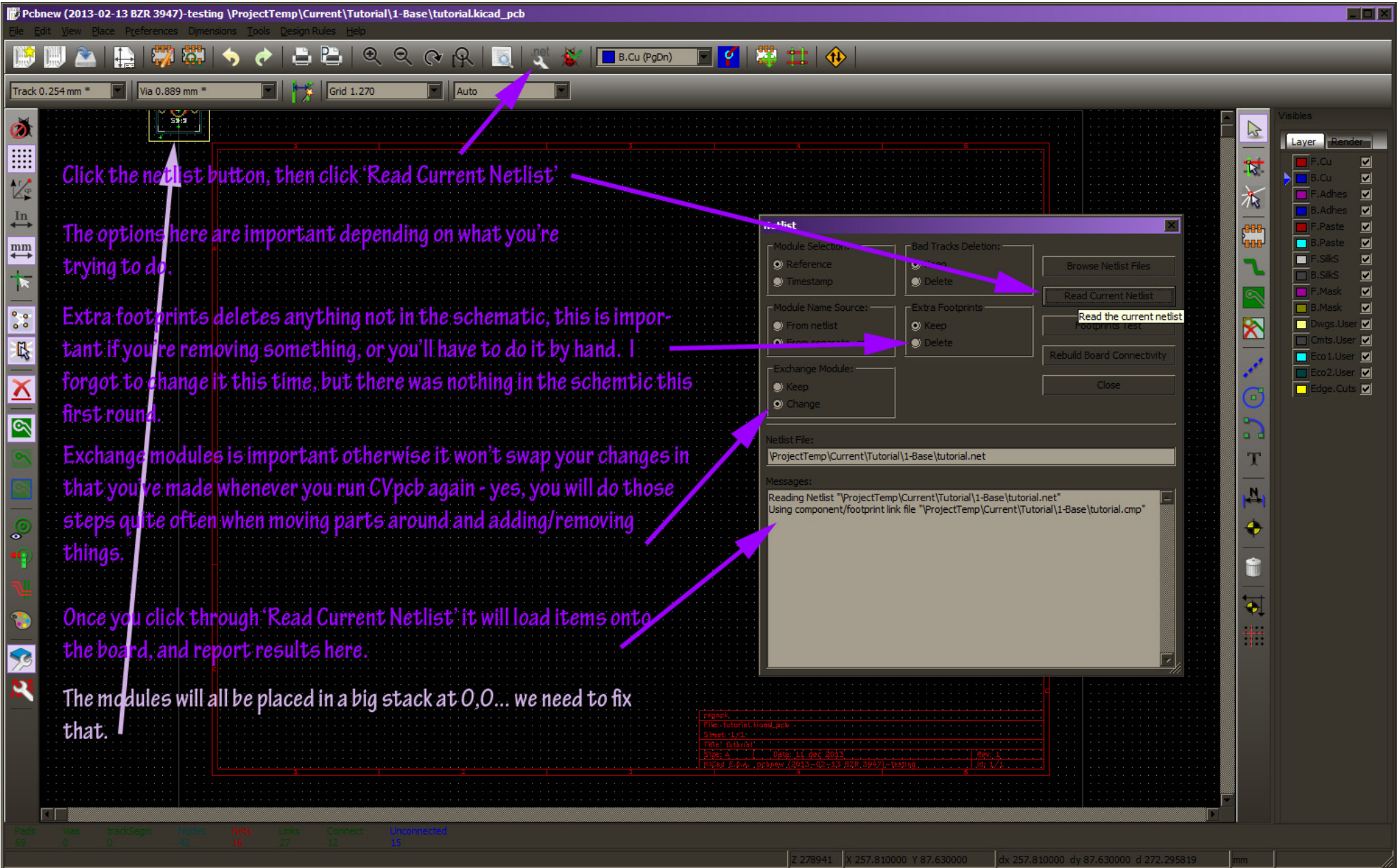
Ignore the error, and get this window.

Here is where we associate annotated items with component footprints (for placing on the PCB). Let's use SOD-123 (A-C) diodes...

The (A-C) is because instead of pads 1 and 2, the pads are labeled A and C. The other SOD-123 has numbered pads







Click the netlist button, then click 'Read Current Netlist'

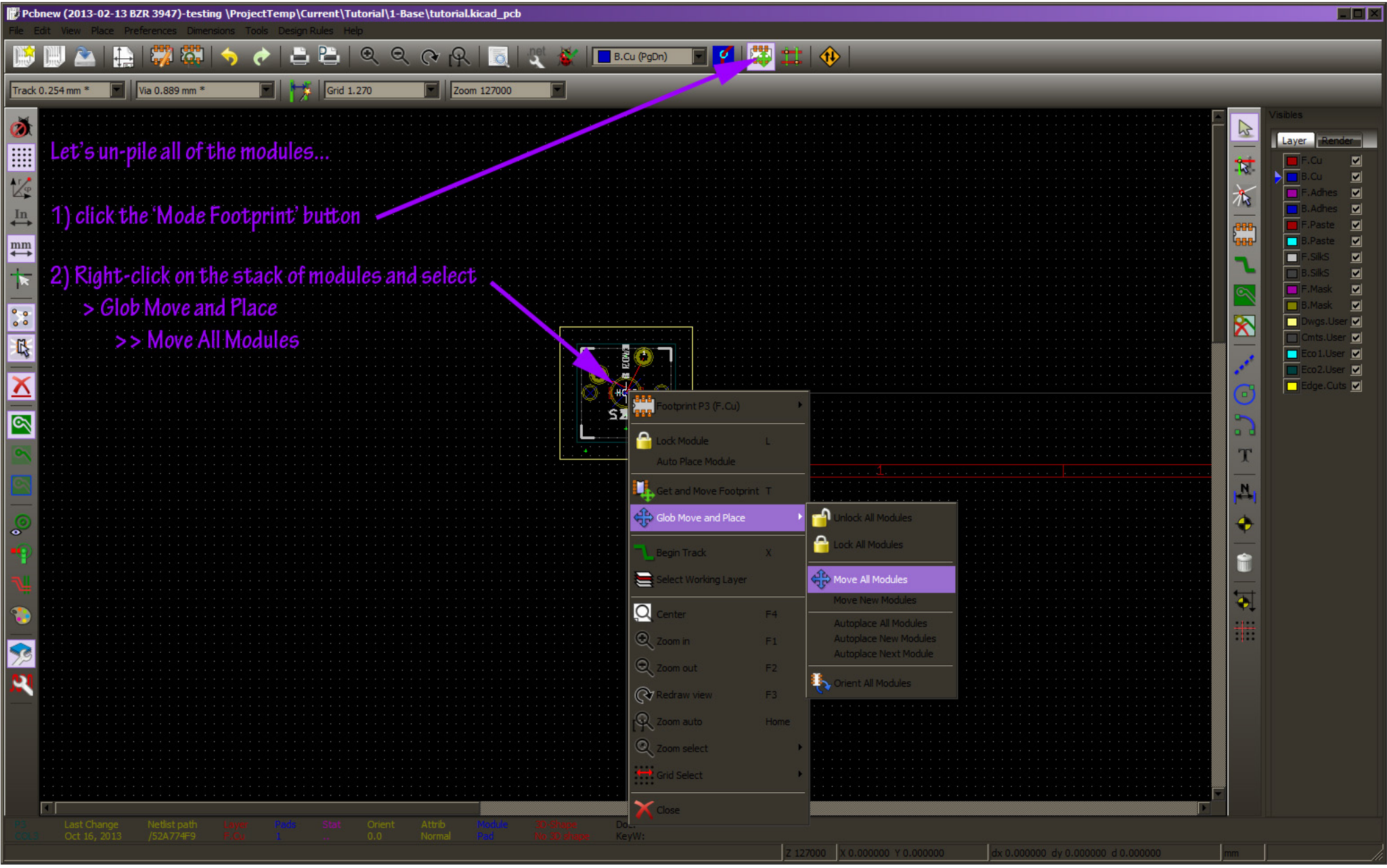
The options here are important depending on what you're trying to do.

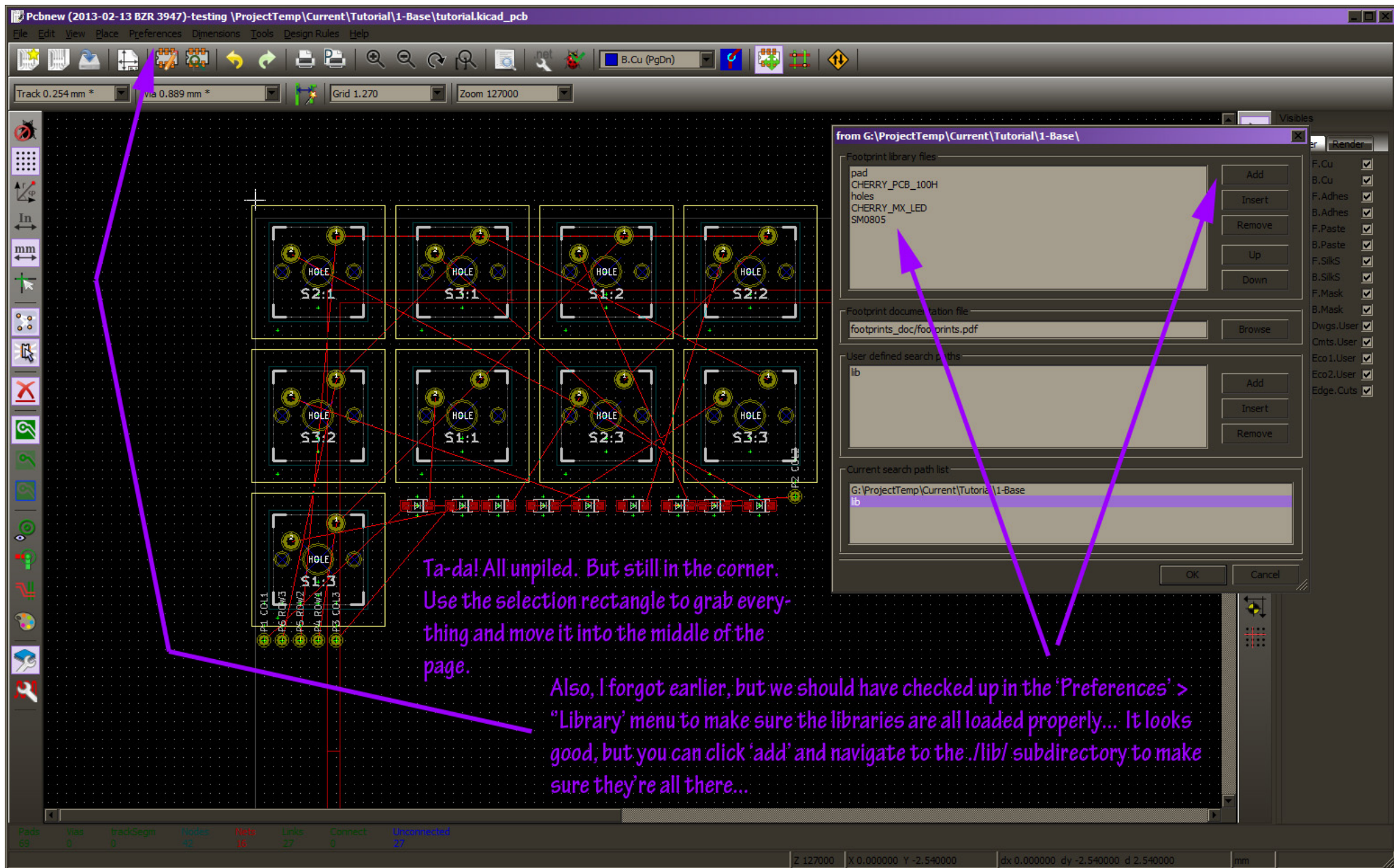
Extra footprints deletes anything not in the schematic, this is important if you're removing something, or you'll have to do it by hand. I forgot to change it this time, but there was nothing in the schematic this first round.

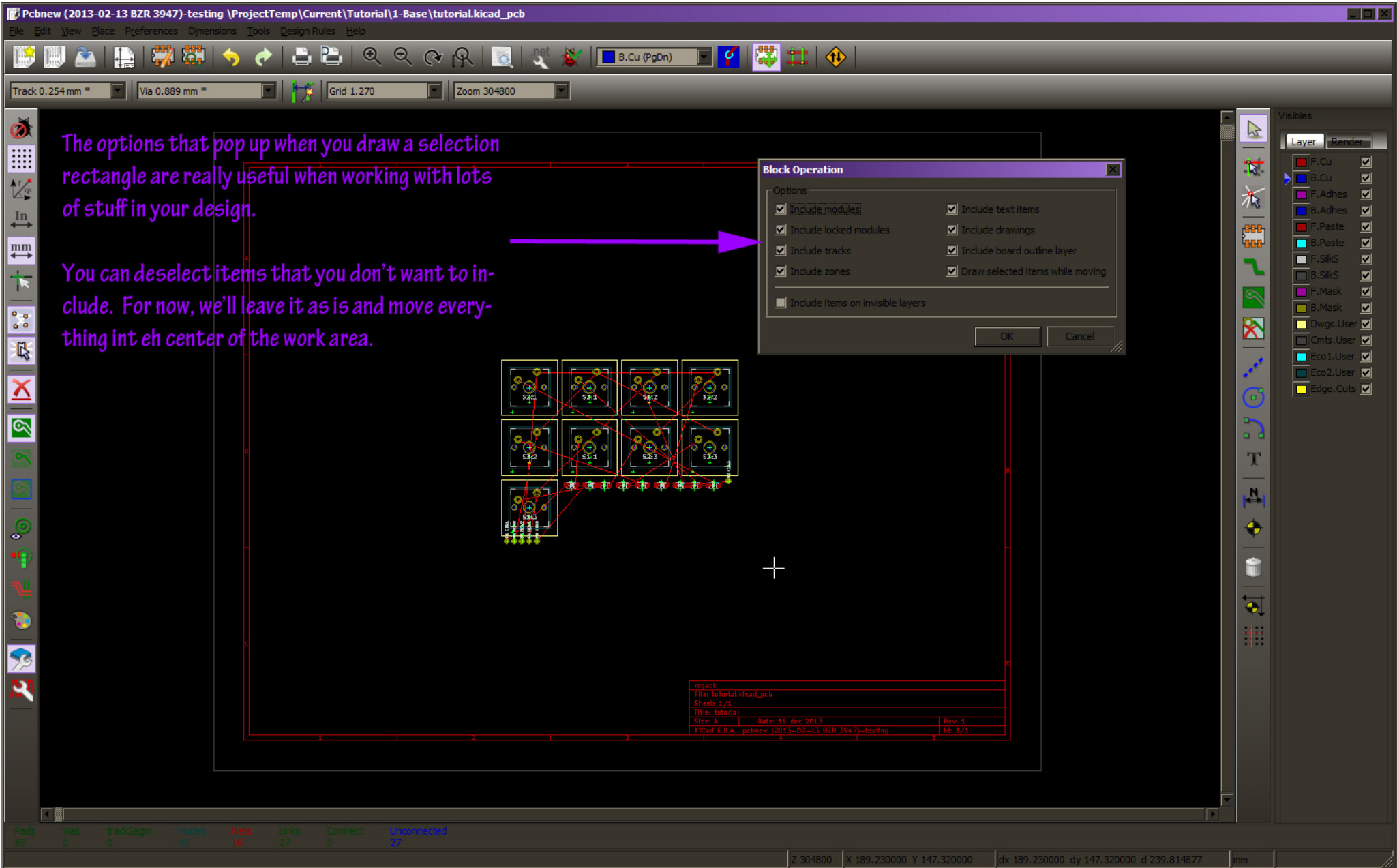
Exchange modules is important otherwise it won't swap your changes in that you've made whenever you run CVpcb again - yes, you will do those steps quite often when moving parts around and adding/removing things.

Once you click through 'Read Current Netlist' it will load items onto the board, and report results here.

The modules will all be placed in a big stack at 0,0... we need to fix that.

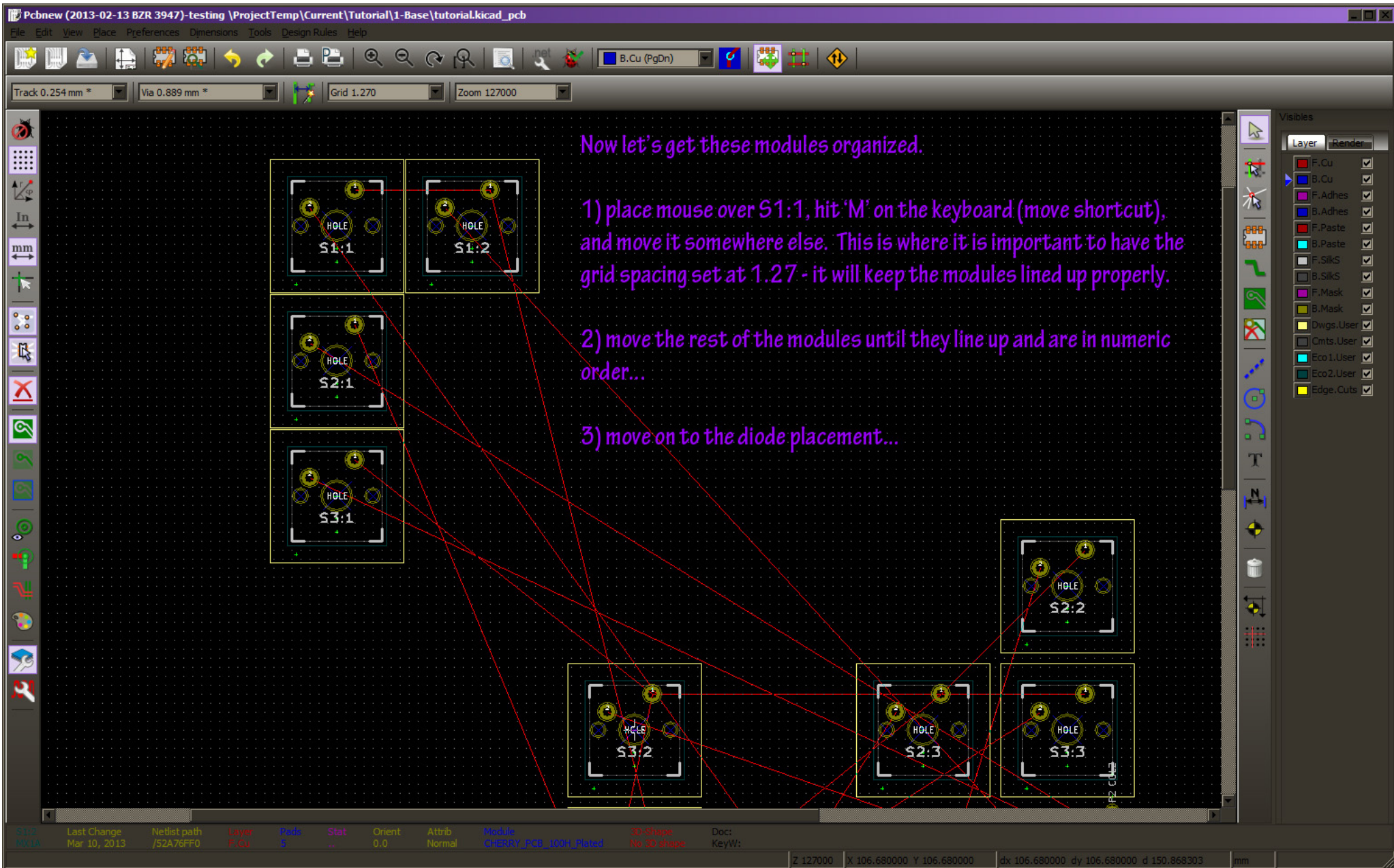


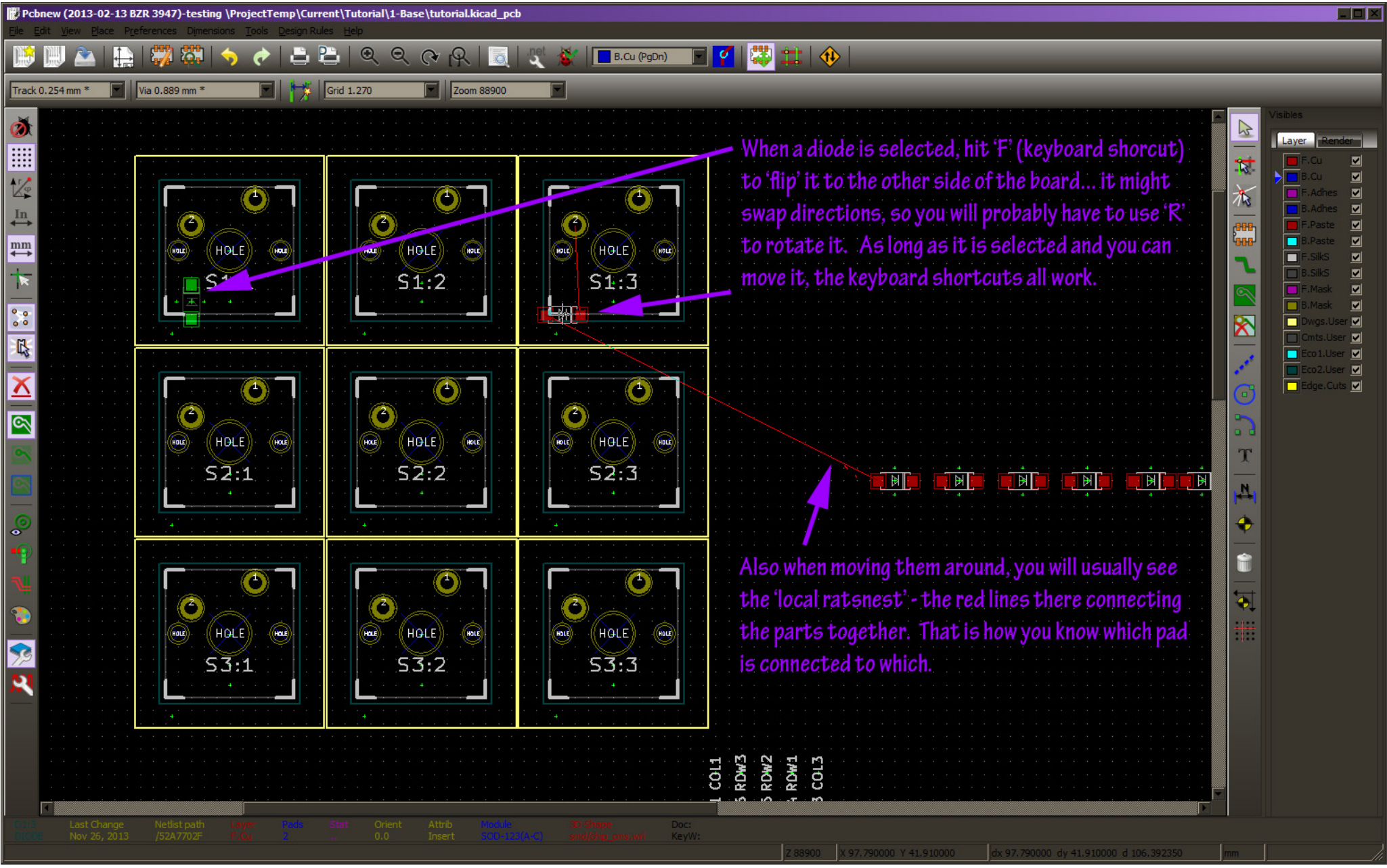


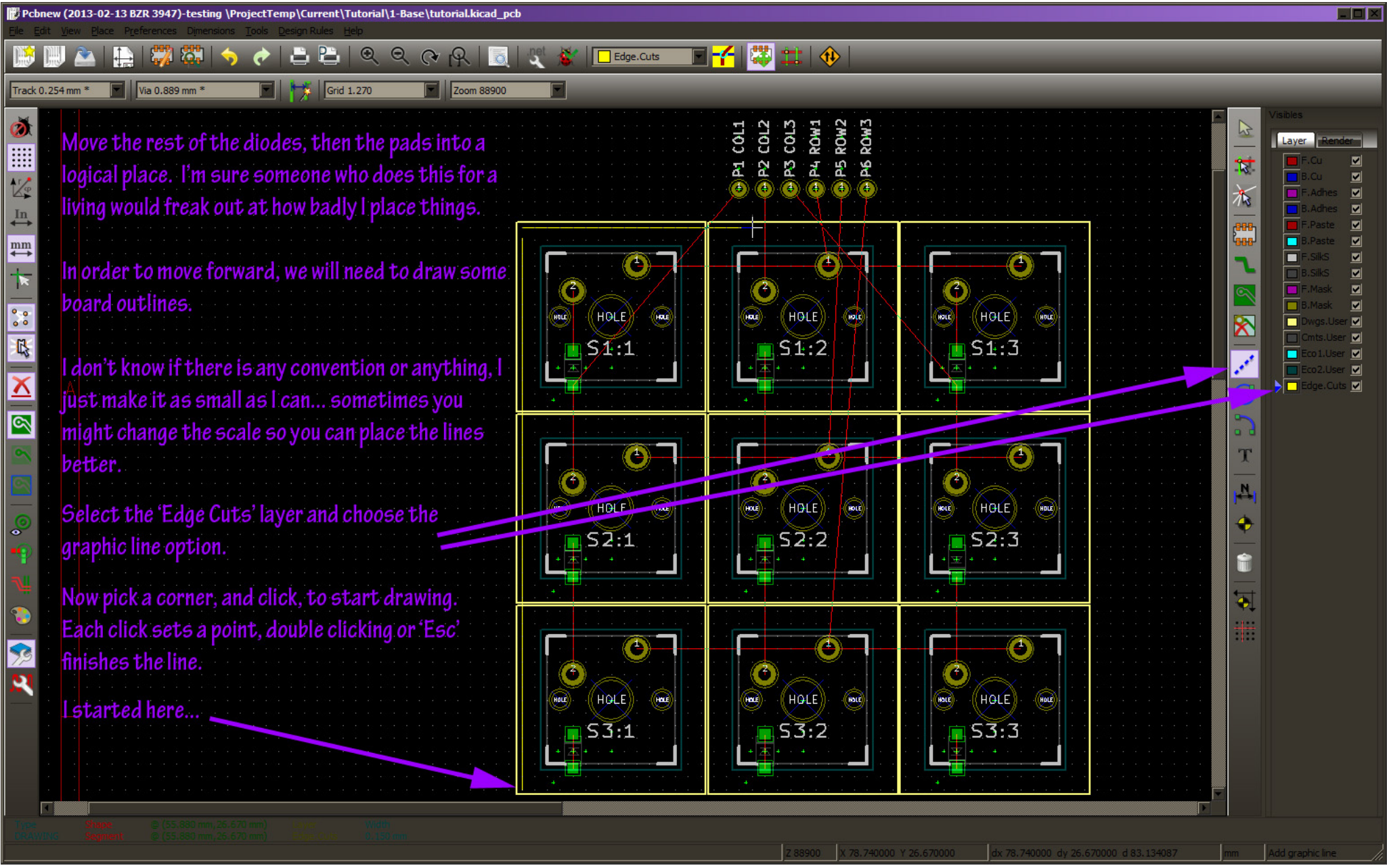


The options that pop up when you draw a selection rectangle are really useful when working with lots of stuff in your design.

You can deselect items that you don't want to include. For now, we'll leave it as is and move everything into the center of the work area.







Move the rest of the diodes, then the pads into a logical place. I'm sure someone who does this for a living would freak out at how badly I place things.

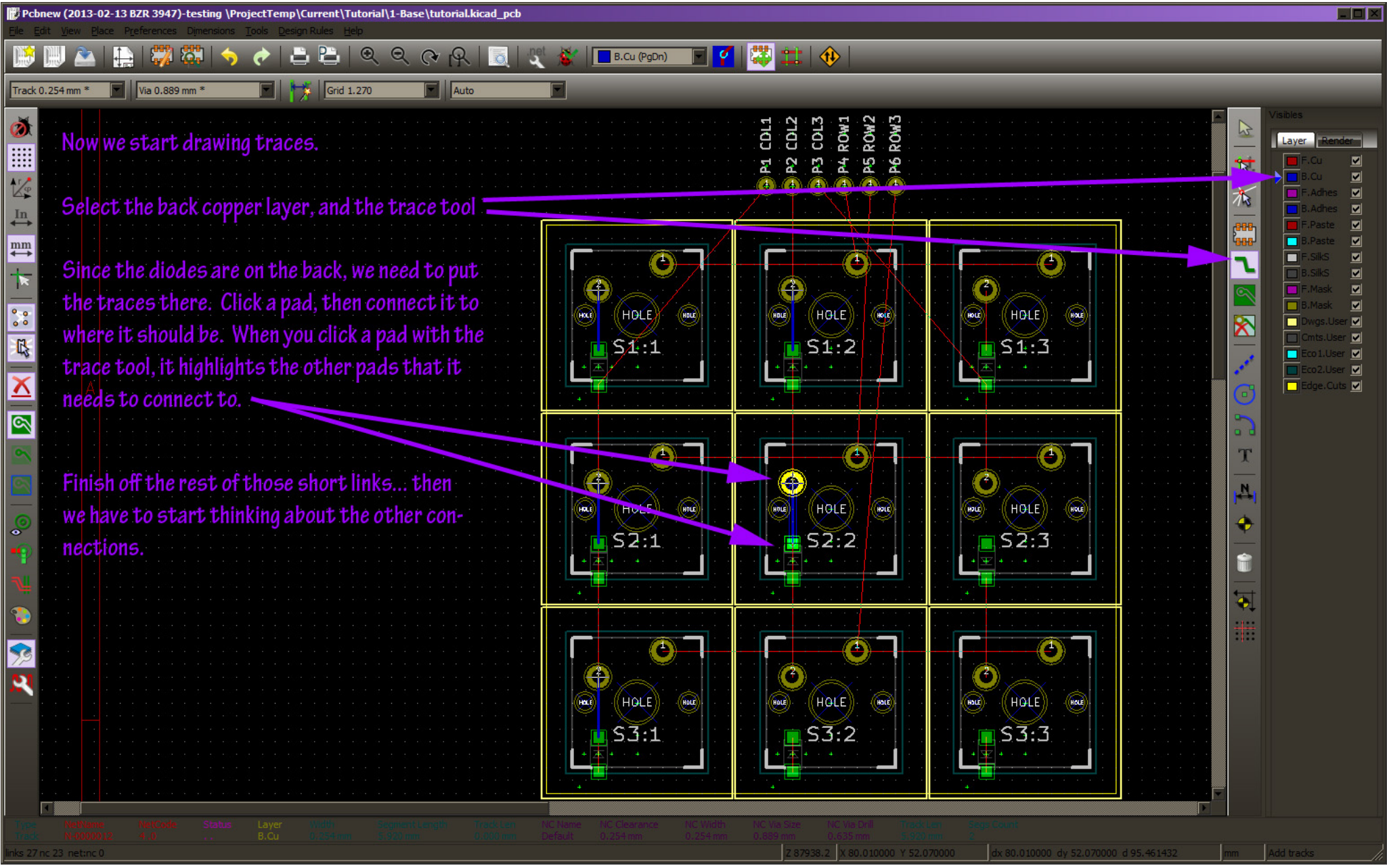
In order to move forward, we will need to draw some board outlines.

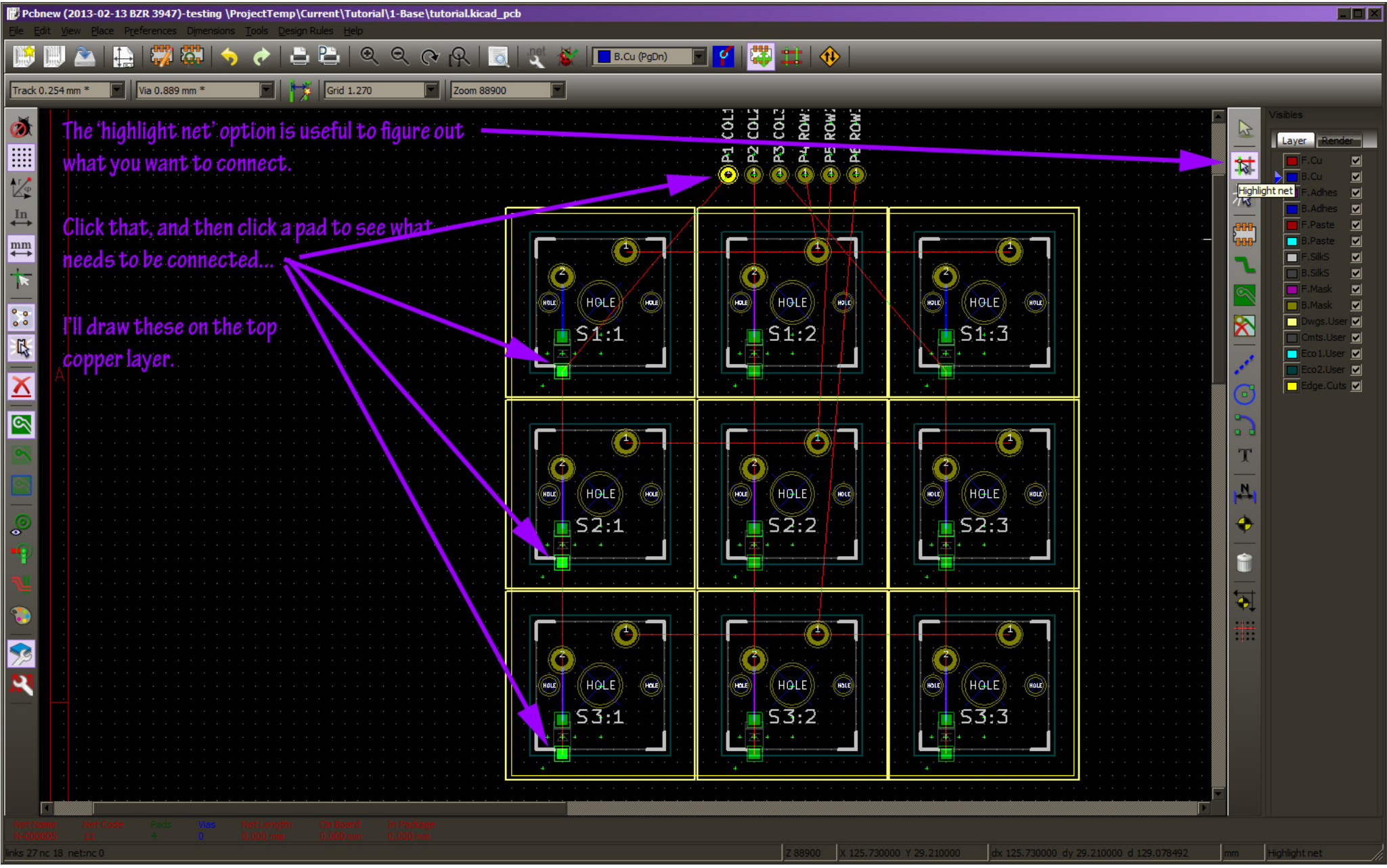
I don't know if there is any convention or anything, I just make it as small as I can... sometimes you might change the scale so you can place the lines better.

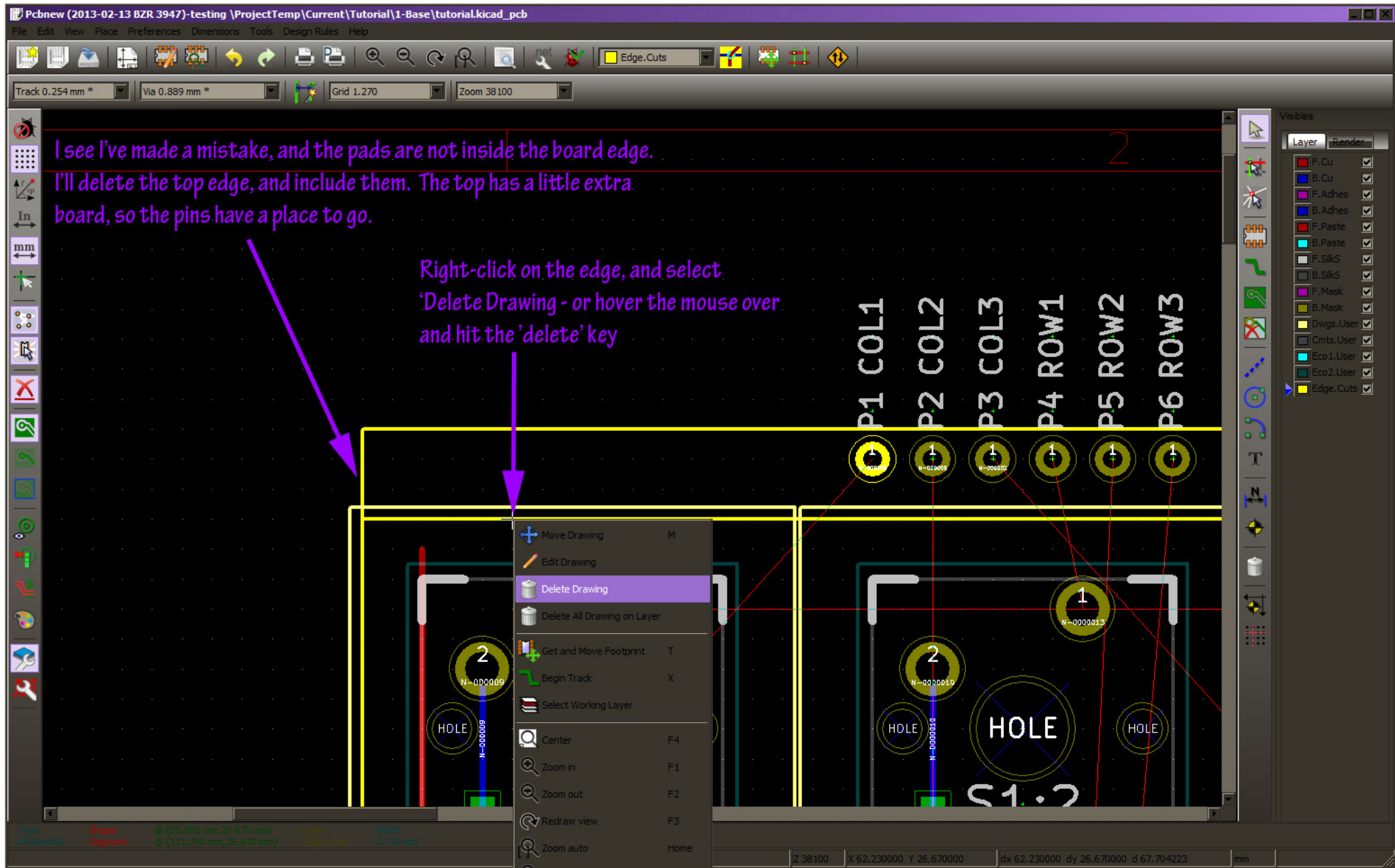
Select the 'Edge Cuts' layer and choose the graphic line option.

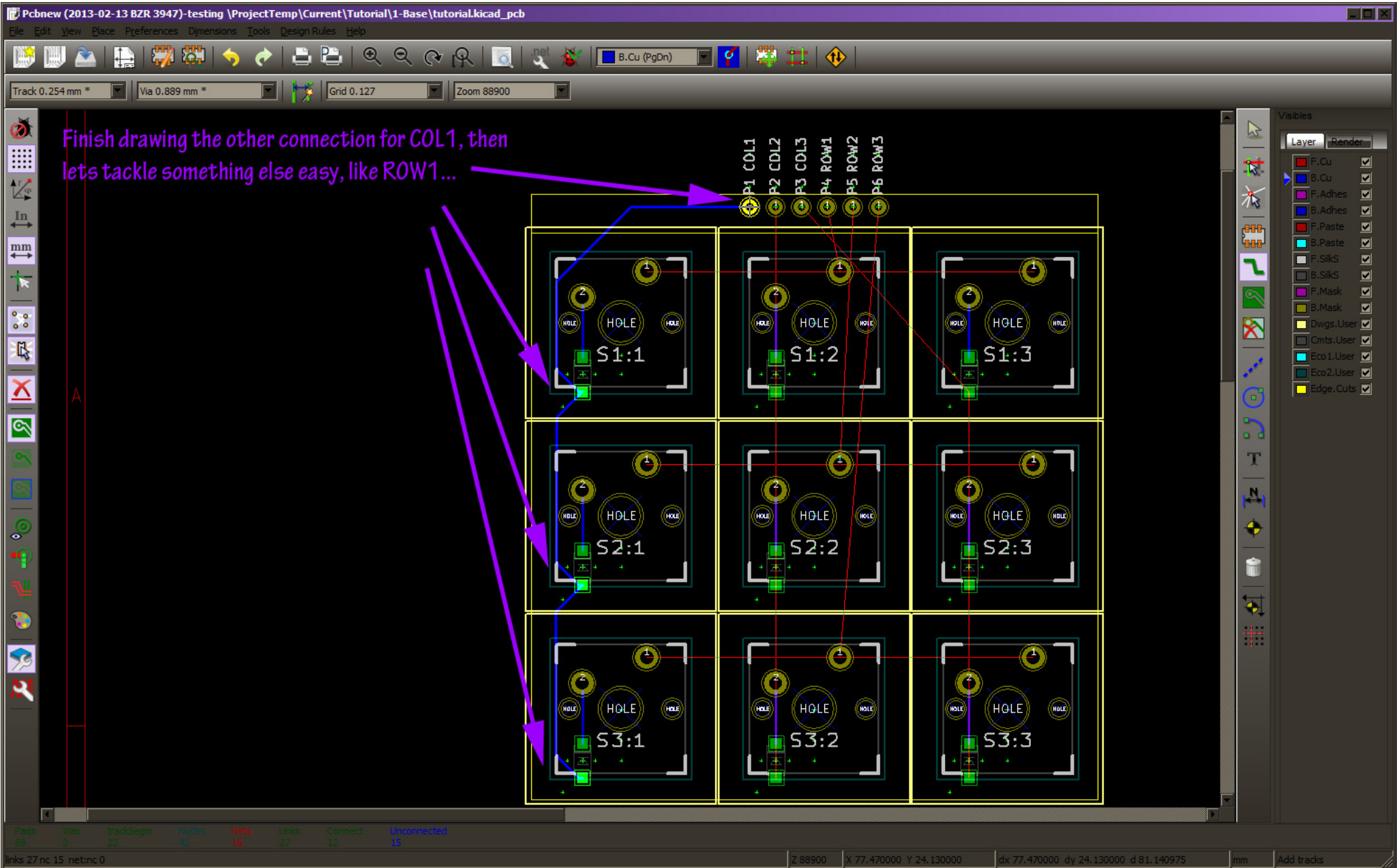
Now pick a corner, and click, to start drawing. Each click sets a point, double clicking or 'Esc' finishes the line.

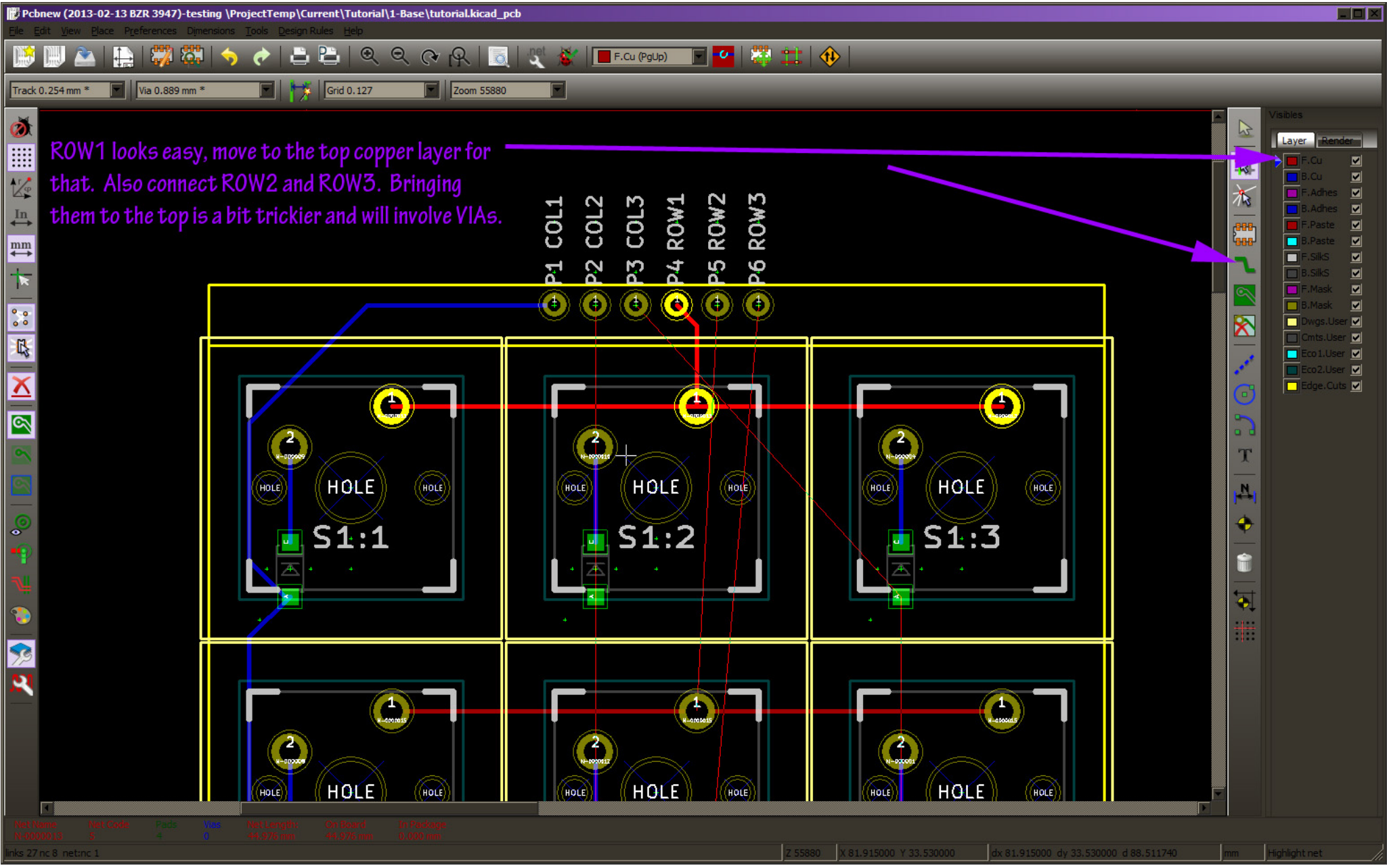
I started here...

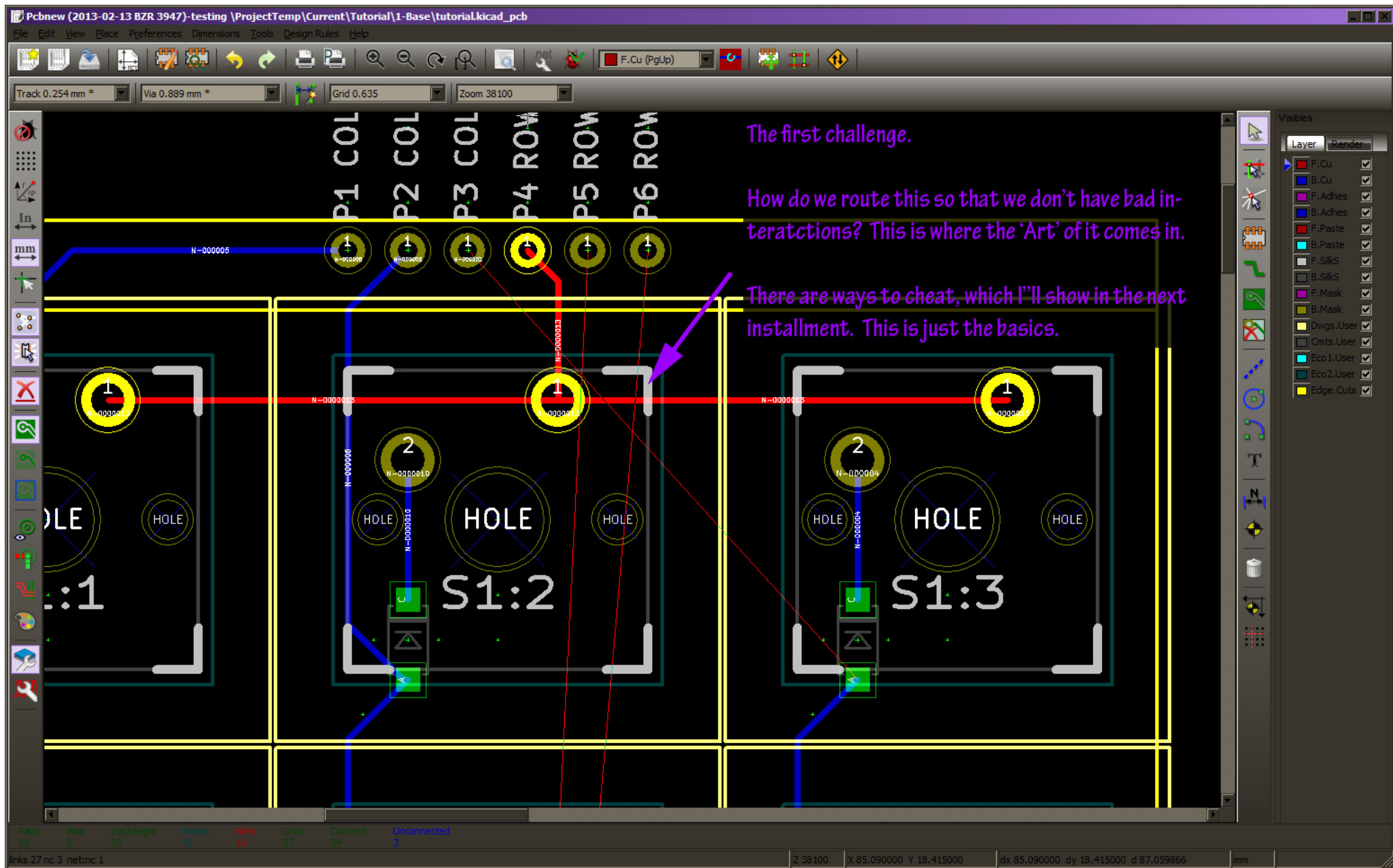








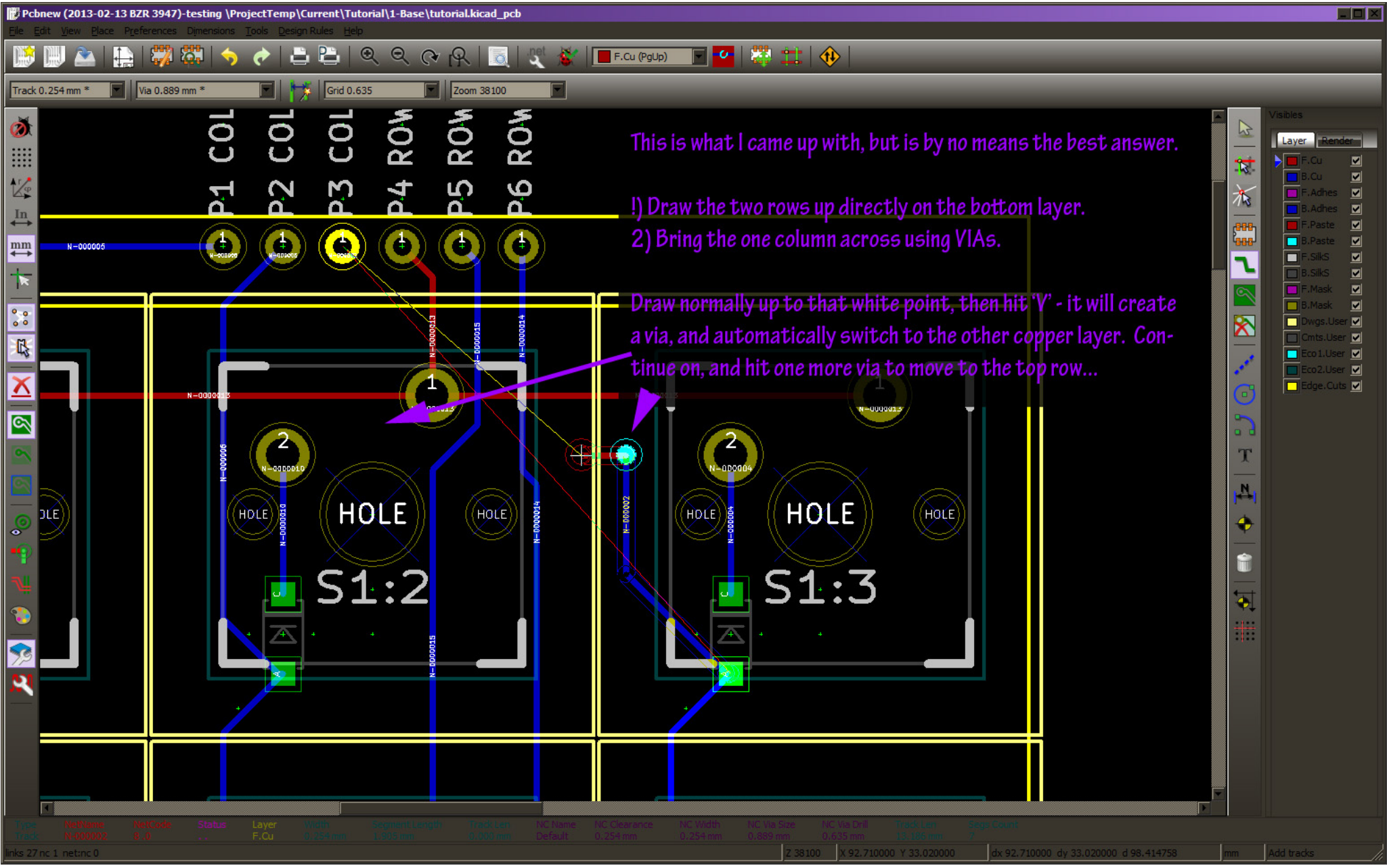




The first challenge.

How do we route this so that we don't have bad interactions? This is where the 'Art' of it comes in.

There are ways to cheat, which I'll show in the next installment. This is just the basics.



This is what I came up with, but is by no means the best answer.

- 1) Draw the two rows up directly on the bottom layer.
- 2) Bring the one column across using VIAs.

Draw normally up to that white point, then hit 'V' - it will create a via, and automatically switch to the other copper layer. Continue on, and hit one more via to move to the top row...

