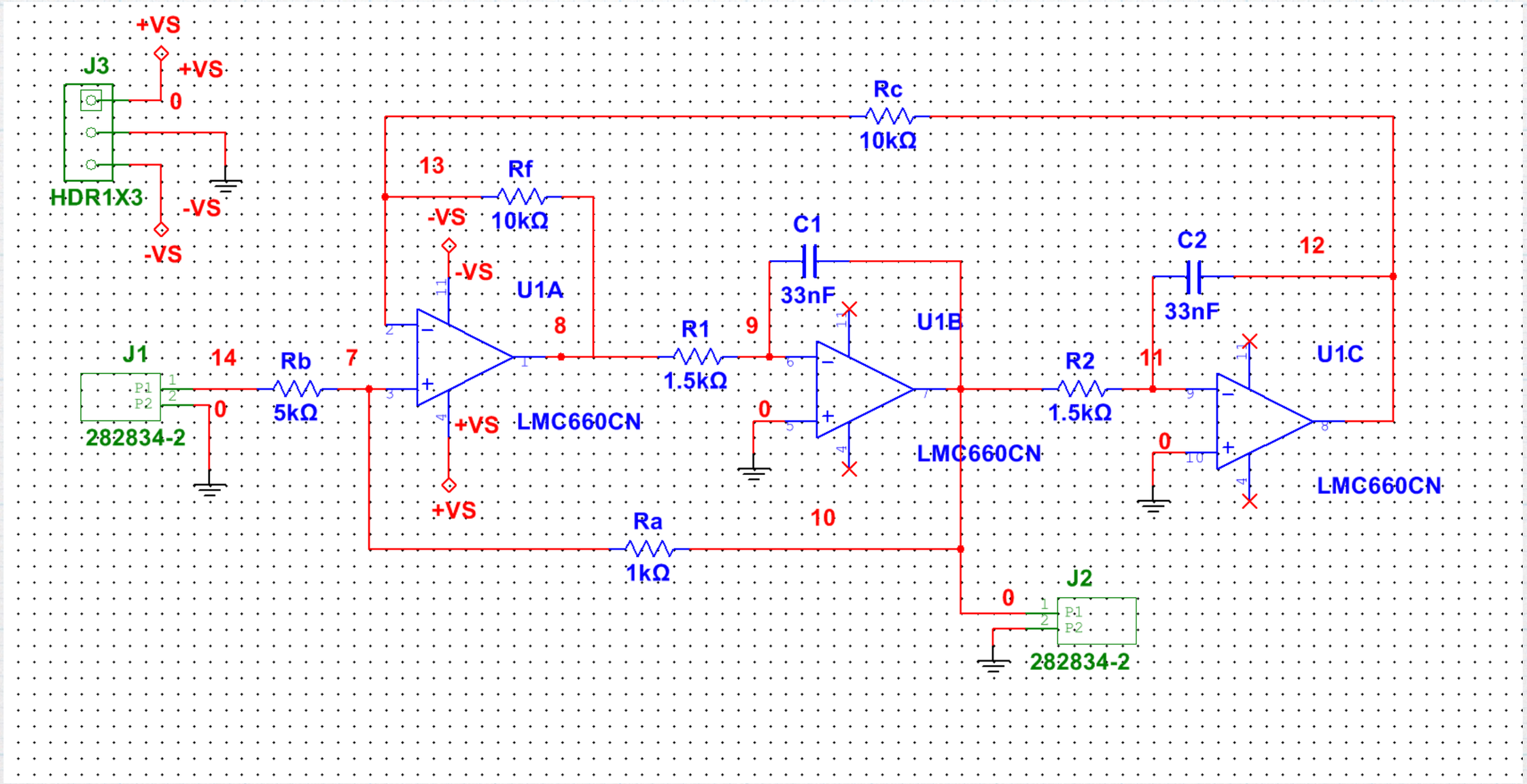
PCB layout tutorial – MultiSim/Ultiboard

The basic steps in designing a PCB

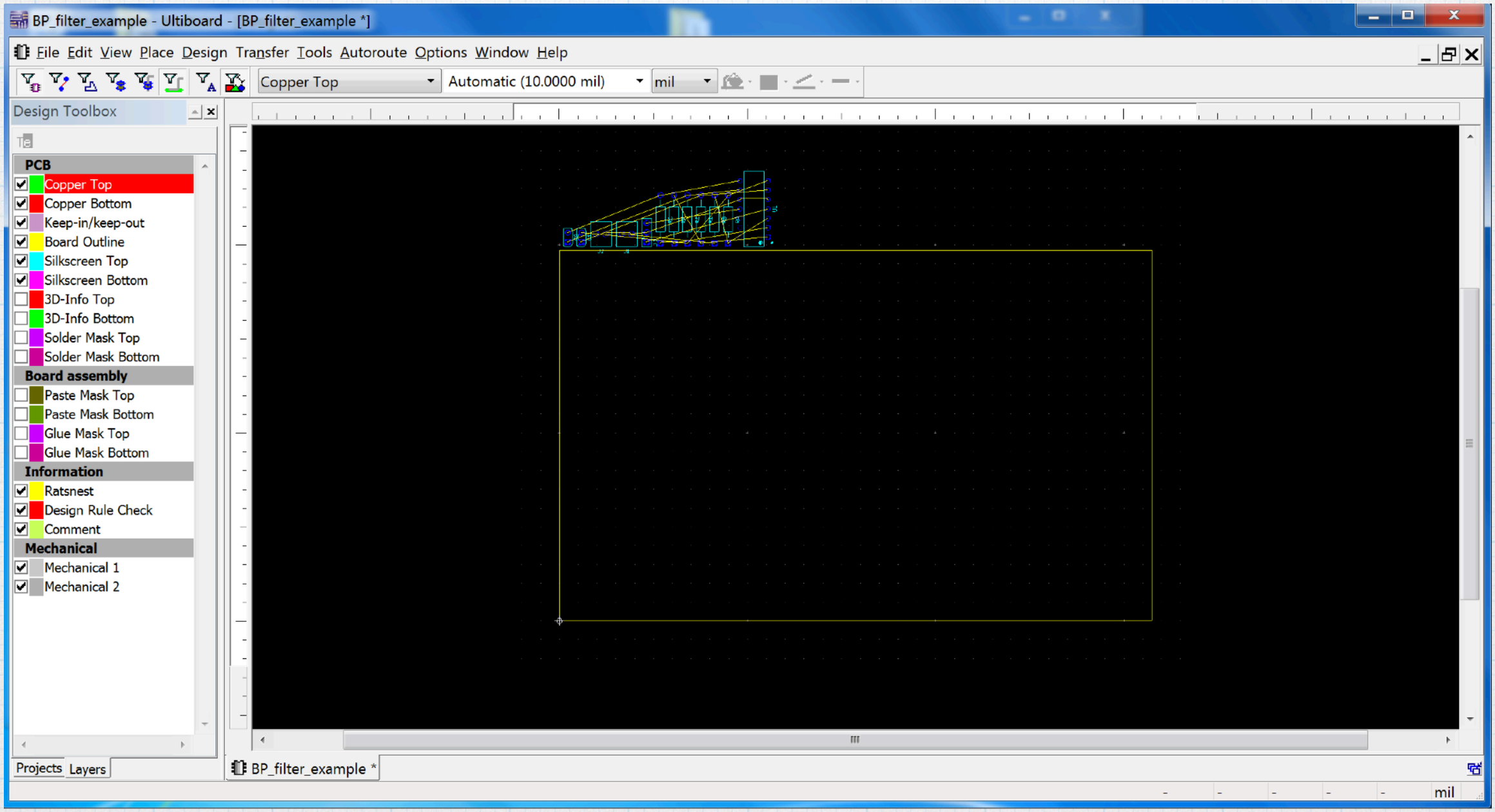
- Paper design and prototype of the basic circuit.
- Identify the parts — and the footprints — that will be used.
- Make a circuit schematic, with the correct parts identified.
- Transfer the schematic to the board layout view.
- Arrange the parts on the board.
- Make the interconnects on the board (autoroute, manual, or combo).
- Tweak the details.
- Design-rule check.
- Generate Gerber files.

In part one of our tale, we created a schematic for an active band-pass filter, as shown below. Now we are ready for the actual layout step.

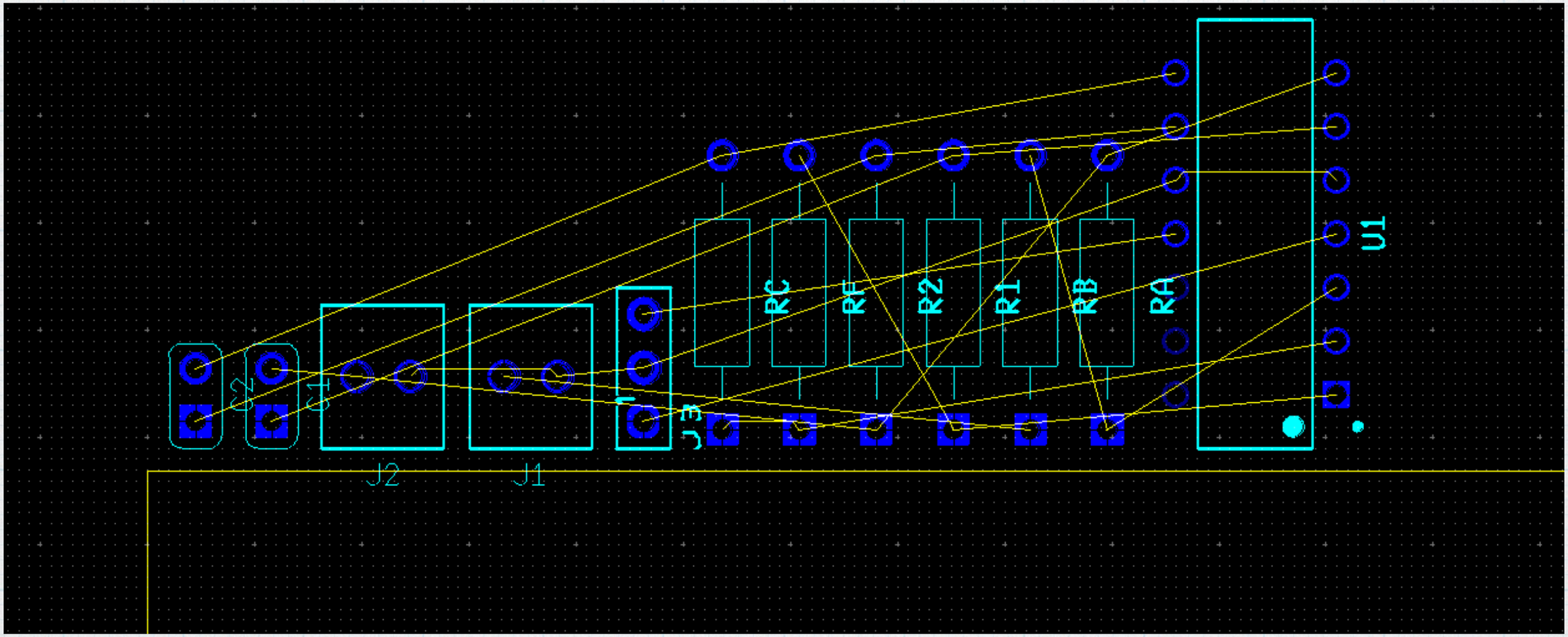


We start where part 1 left off, with the schematic open in Multi-Sim.

To begin the transfer, use the menu item: “Transfer to Ultiboard -> transfer to Ultiboard 14.0” from the “Transfer” menu. A new file is created (an Ultiboard file of type .ewnet), the Ultiboard program is launched, and a window showing a Spice-type net list specifying the connections in the circuit appears. Click OK to import the net list. We are now in the board layout view.

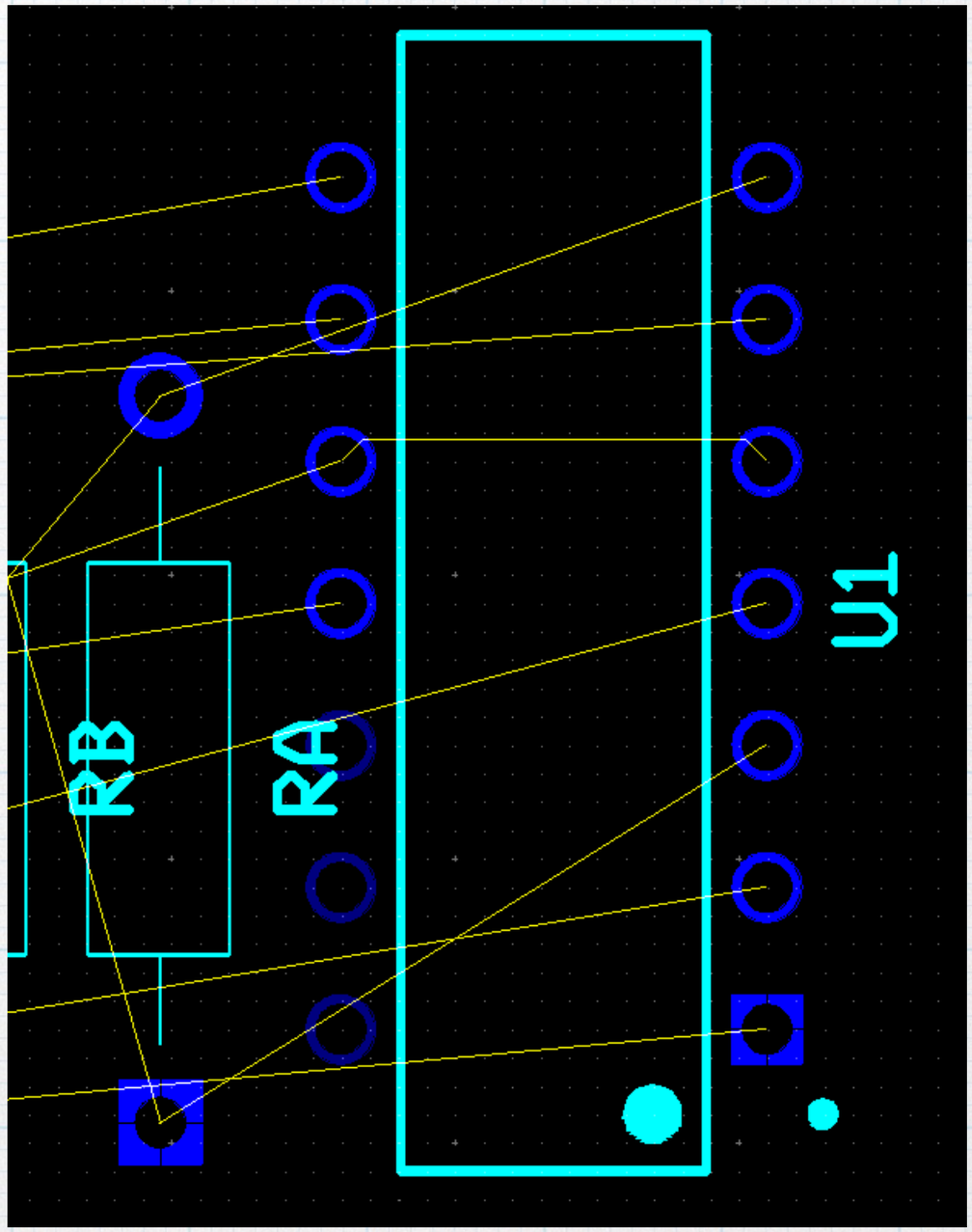


We see an outline of the board, footprints of the various components, and the “ratsnest” wires indicating the connections. Zooming in a bit (use the scroll wheel), we recognize the parts: the quad op-amp chip (U1), six resistors, two capacitors, two two-pin connectors, and the three-pin header whose footprint we will use for a three-pin connector.

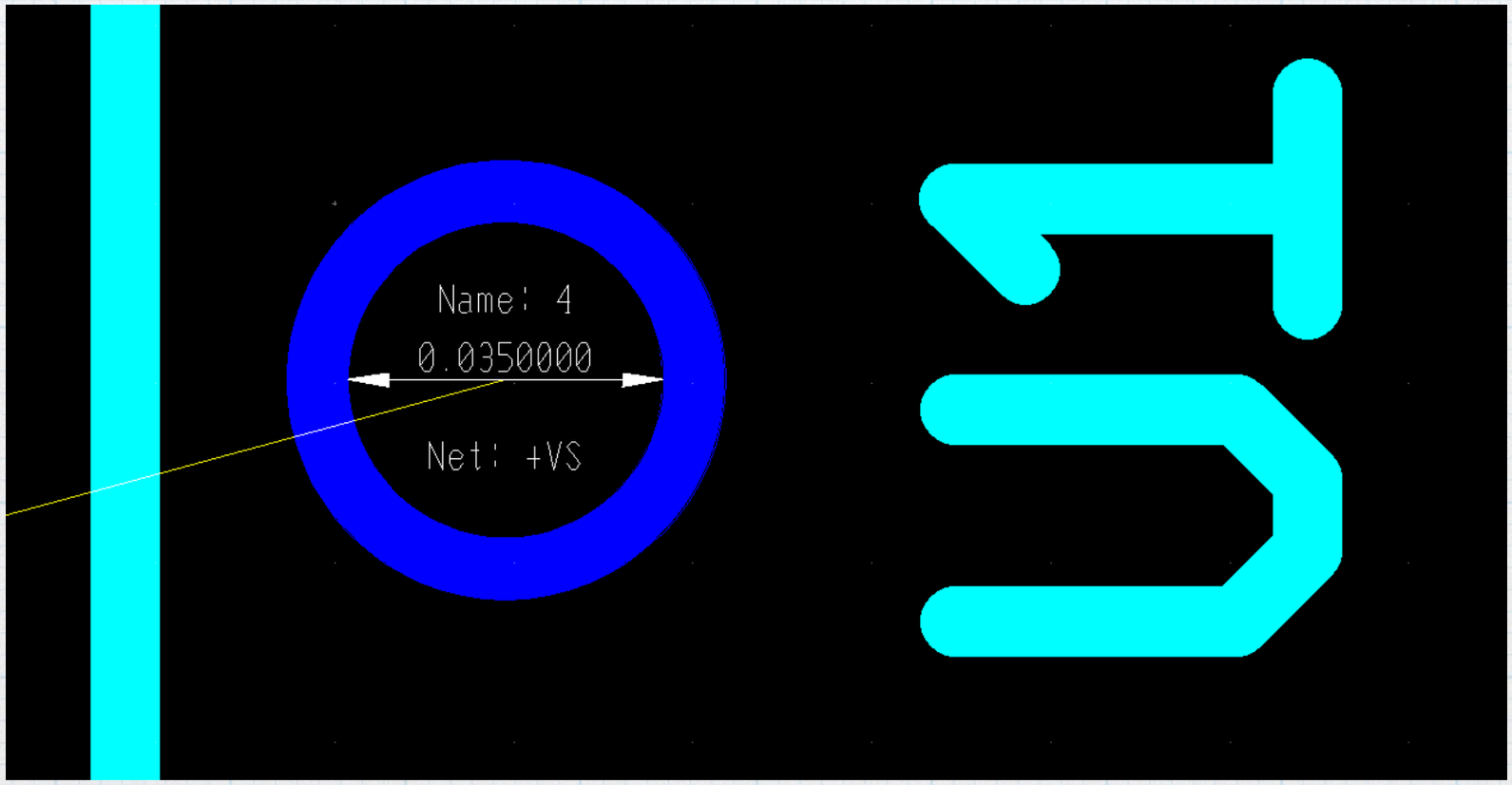


The default size of the board is 6.3 inch x 3.9 inches (160 mm x 100 mm). Obviously, this is much too big for our little circuit. We will change this later.

The components are indicated by their footprints. In this default veiw, we see the component outline (in light blue), the label (also light blue), and the locations for the holes that will be drilled for the through-hole connections (dark blue). The dark blue represents the metal contact region, and the holes will be centered within. The outline and the labels shown will be printed on the board itself. Of course, the holes will be quite obvious in the board. The ratsnest lines will replaced with copper traces on the board — we will do that shortly.



However, there is considerably more information about the layout than what we see here. Some of it is invisible at the moment — we will be able to make it visible when needed. Some info is simply too small to see at a zoomed out scale. For instance, if we zoom in (use the mouse scroll wheel and window scroll bars) on pin 4 of the op amp, we find some information about it — pin number, inner ring diameter, and the node (net) connection. (This is the positive power supply connection.) Some of this info may be useful as we proceed.

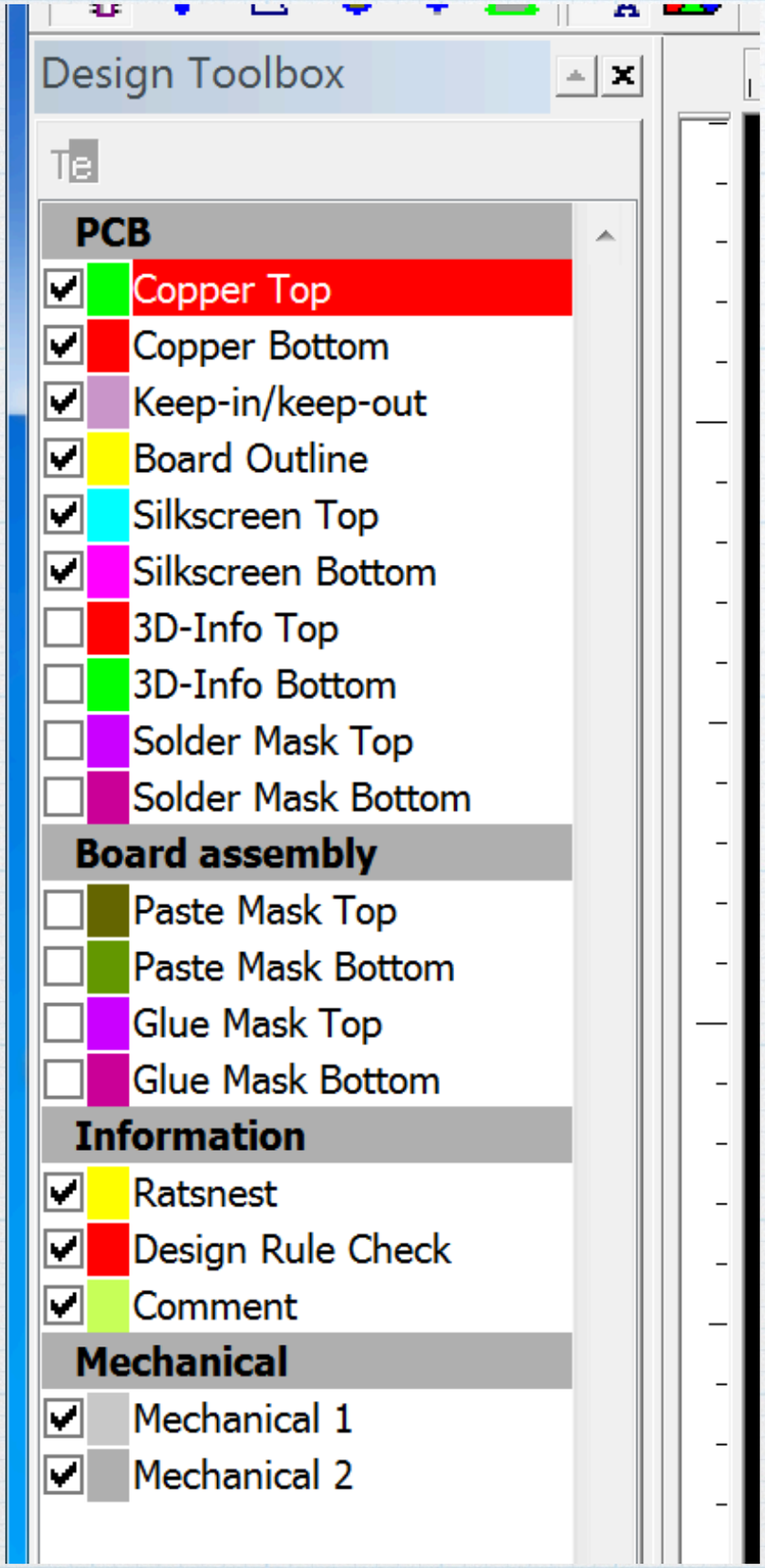


As we discussed earlier and as you saw in the PCB videos (You watched those, right? right?!), there are a number of layers involved in defining the complete board layout. For simple two-layer boards — which almost everyone in EE 333 will be doing — the main layers we need to consider are:

- Copper Top – traces on top of the board
- Copper Bottom – traces on the bottom of the board
- Solder Mask Top – contact openings in the topside solder mask layer
- Solder Mask Bottom – openings in the bottom side solder mask
- Silkscreen Top – labels, lettering, and artwork on the topside
- Silkscreen Bottom – same thing but on the bottom
- Board Outline – outer perimeter of the board
- Ratsnest – the initial tangle of interconnects that are converted to traces

Board with more trace layers (4, 6, 8, etc) will have correspondingly more layers.

The layers above are the essential definition of the board that will be sent to the board manufacturer. The ratsnest is not included, but there will be an extra layer for the drill holes that will added to the above list.



The design toolbox, on the left side of the UB window, lists all these layers (and more). The toolbox allows us to select which layers are visible and which layer is currently “active”.

Layers with check marks are visible. The active layer is highlighted in red. The active layer indicates where editing actions take place.

Change the active layer to one of the unused layers (Paste Mask Top or something) by double clicking. Then turn off and on some of the layers by clicking on the check marks and observe the changes in layout window. (Note that it takes 3 clicks to cycle through the check marks.) Also, turn on and off the solder mask layers to see those areas.

(Note that you can’t uncheck the current active layer.)

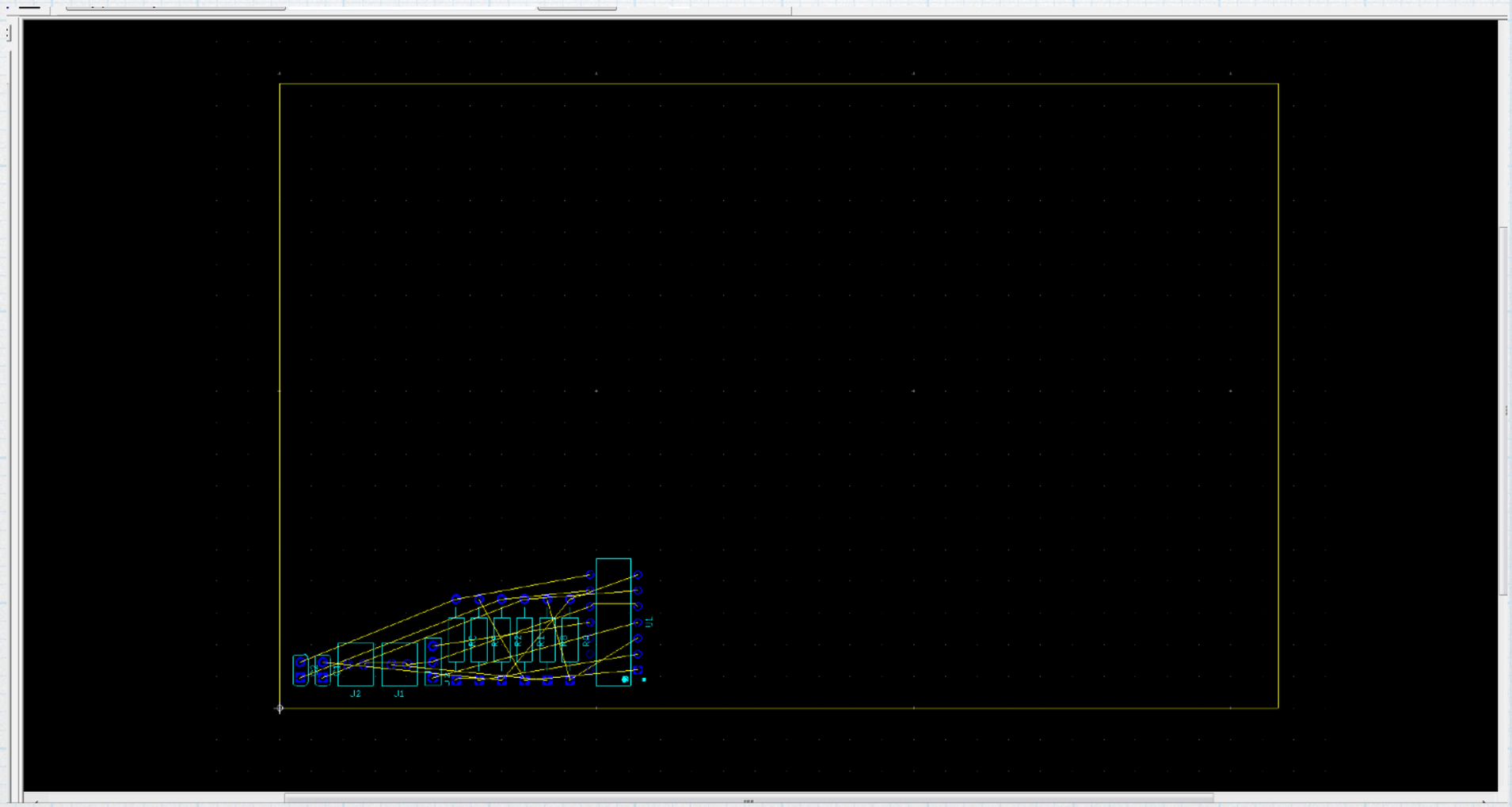
Because there are so many different items in the drawing window, selecting specific items can be tricky. To facilitate manipulation of the various parts of the drawing, there is a cursor selection toolbar in the upper left corner of the program window. (Activate it by choosing the "Select" toolbar from the "View" menu.) This toolbar allows you to enable or disable the various items that you can select with the mouse.

The icons represent (from left to right),

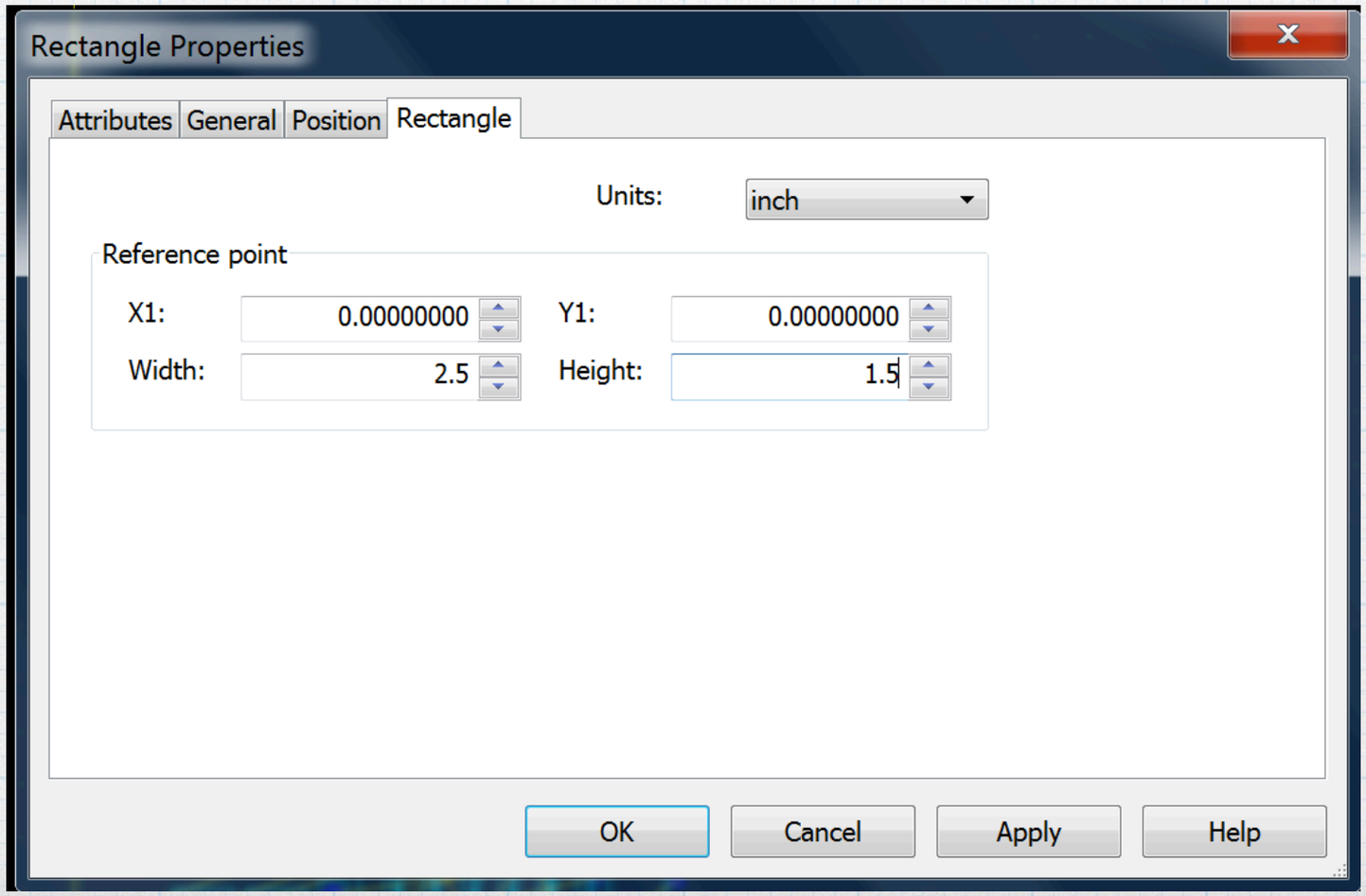
- enable / disable selecting parts
- enable / disable selecting traces
- enable / disable selecting copper areas
- enable / disable selecting vias
- enable / disable selecting THT (through hole) pads
- enable / disable selecting SMD (surface mount) pads
- enable / disable selecting attributes (enabled above)

As we move and edit the various components on the board, we will use the design toolbox and the select menu to facilitate our actions.

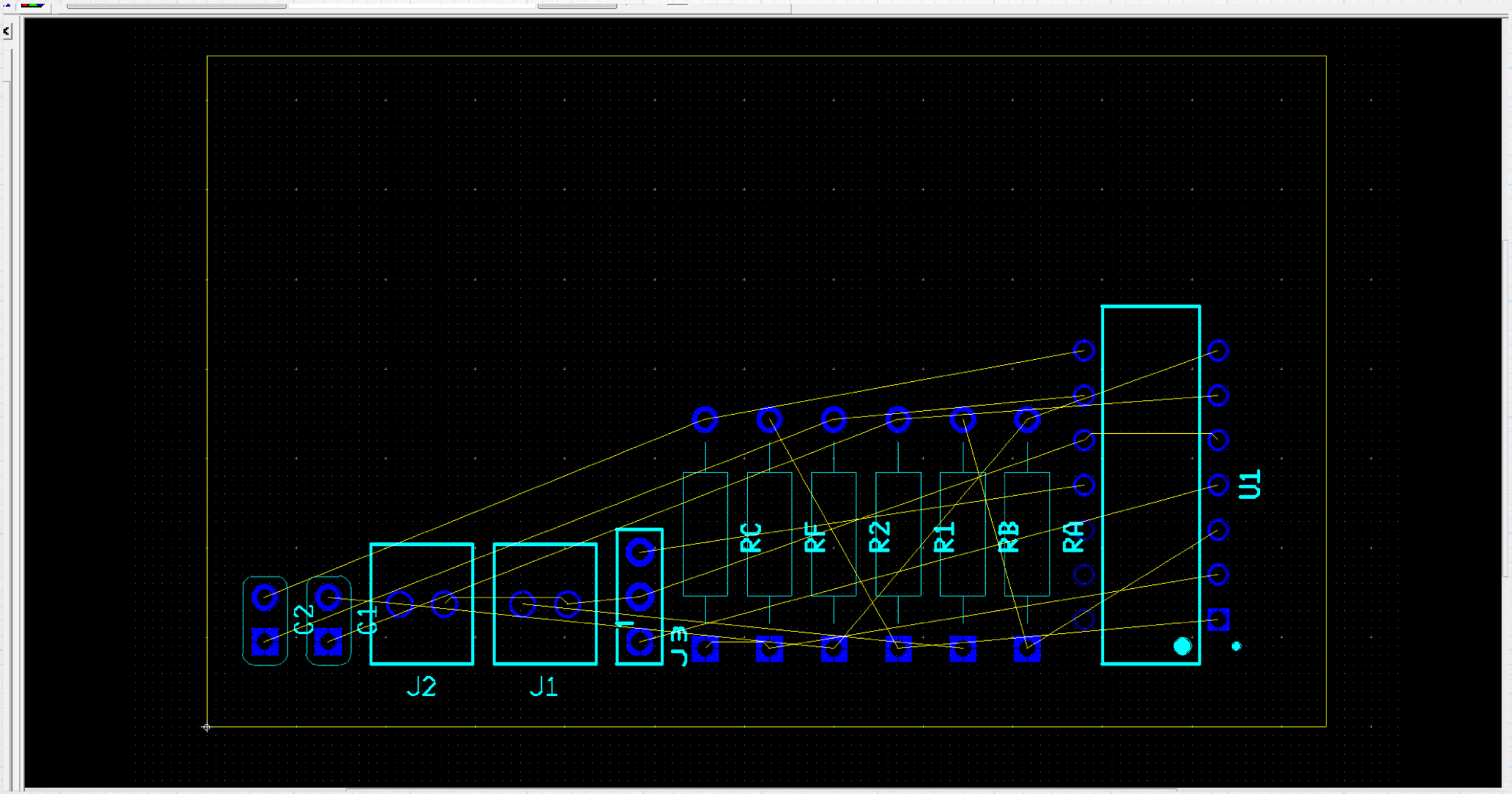
To move the components onto the board, enable the “selecting parts” item in the select toolbar. Click and drag a rectangle around the parts to select them all. Then click to select and drag the collection onto the board. The origin for the board is in the lower left corner, so you might want to move everything into that region, but location is not terribly important.
(Components can also be moved onto the board one at a time.)



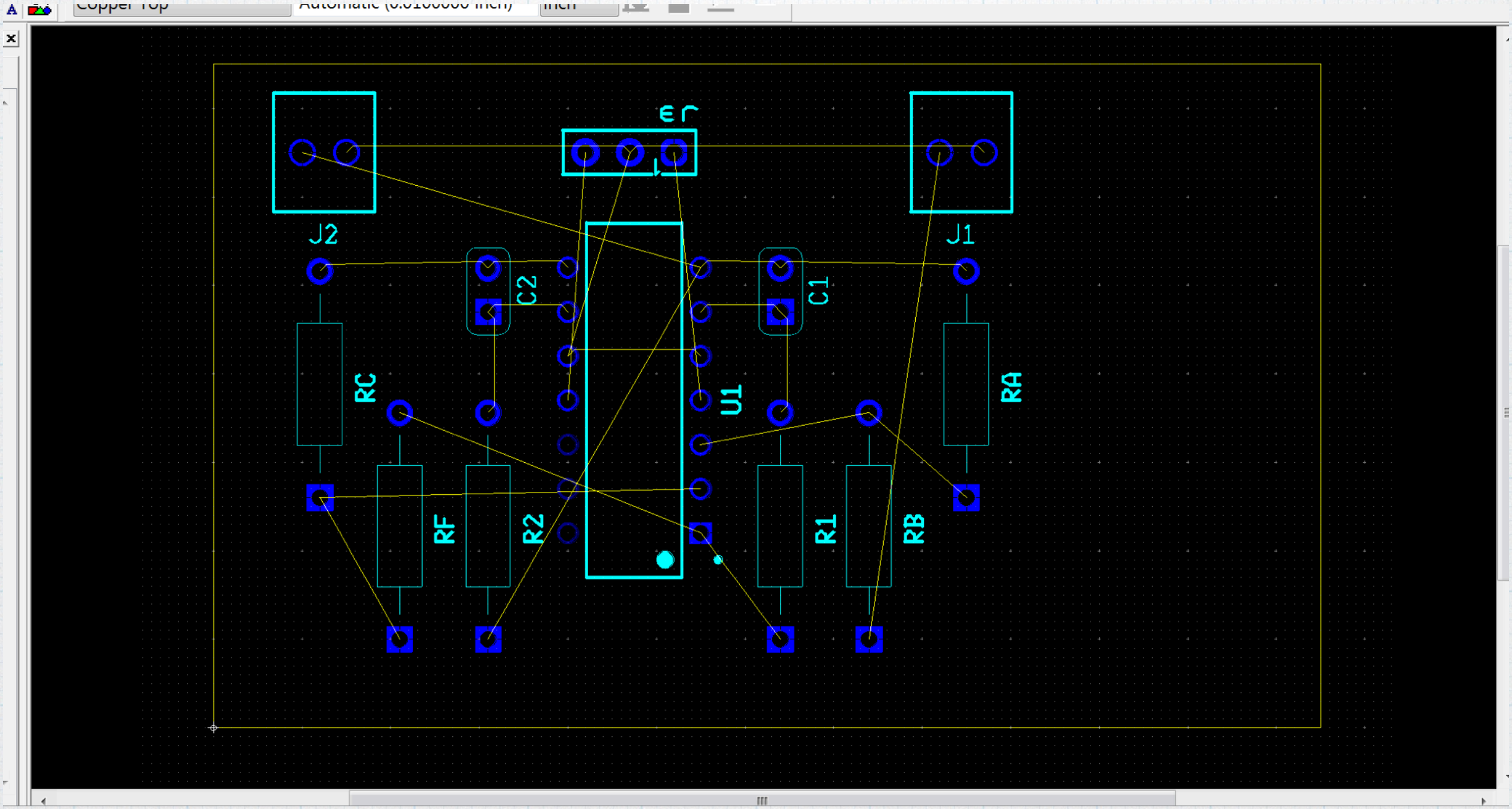
Reduce the board size. Disable “selecting parts” and enable “selecting other objects”. Double-click the Board Outline layer to make it active. Then click on the yellow board outline to select it. Next, click on the upper-right corner and drag the box to make it smaller — something a bit bigger than the area taken up by the components. We can make a final adjustment again later. Or another way to re-size the board outline: double-click on the board outline to bring up the “Rectangle Properties” dialog. Under the “Rectangle” tab, the desired new Width and Height can be entered directly. Be sure to make sure of the units. A size of 2.5″ x 1.5″ might be reasonable for now.

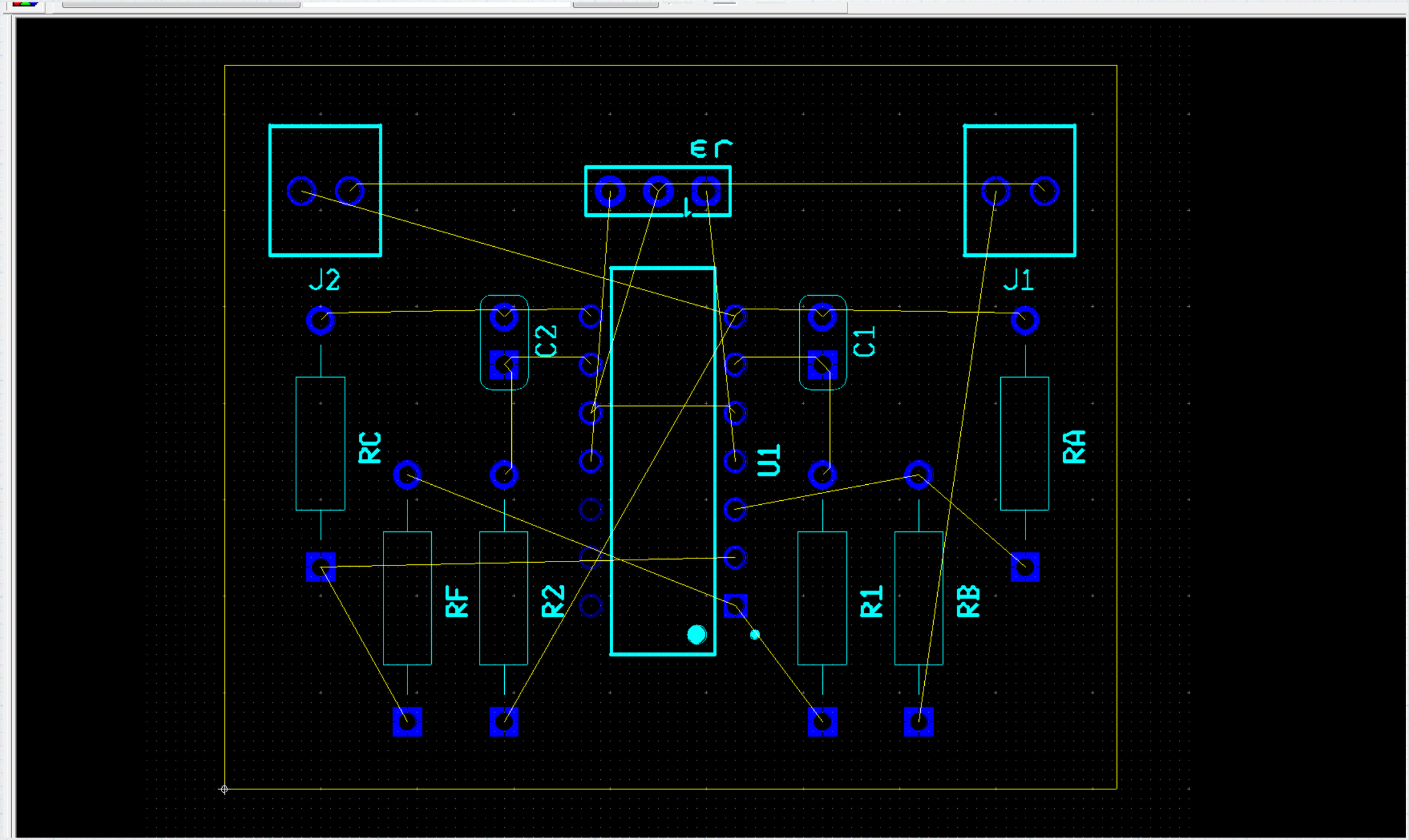


Zoom in to get a better view of the components on the board.



Next comes the (possibly tedious) process of moving the components to make the physical arrangement of the layout. Make sure to enable "selecting parts" in the select tool bar.


J2
J1
C2
C1
RC
RA
U1
RF
R2
R1
RB


εr
J2
J1
C2
C1
RC
U1
RA
RF
R2
R1
RB

Setting traces sizes

The trace widths should be chosen to minimize the resistance of the traces in order to minimize heating. The IPC (www.ipc.org) has developed an industry standard for PCB technology (http://www.ipc.org/TOC/IPC-2221.pdf), including a calculator for determining minimum trace width for different currents. The calculator can be found on many web sites, including DigiKey (https://www.digikey.com/en/resources/conversion-calculators/conversion-calculator-pcb-trace-width).

To use it, enter the expected current along with the copper layer thickness (in oz/ft^2, remember), the allowable temperature rise (usually 10°C), the ambient temperature (usually 25°C), and the trace length. (The default length is 1 inch, which is a reasonable starting since we probably don't yet know how long any trace will be.) The calculator returns the minimum trace width for layers that are external (applicable to our two-layer boards) and internal (for the extra layers in "bigger" boards) layers.

Also, we must be aware of the minimum sizes that can be manufactured. Each fab facility has its own limits.

1 oz/ft² copper, 25°C ambient, 10°C temperature rise, 1 inch length

current	external	internal
10 mA	0.02 mil	0.05 mil
100 mA	0.5 mil	1.28 mil
500 mA	4.5 mil	11.8 mil
1 A	11.8 mil	30.8 mil
5 A	109 mil	283 mil
10 A	283 mil	737 mil

	minimum trace	min. clearance
OshPark	6 mil	6 mil
Advanced Circuits	3.75 mil	4.5 mil
SEEED Studios	6 mil	6 mil
PCB Fab Express	4 mil	4 mil

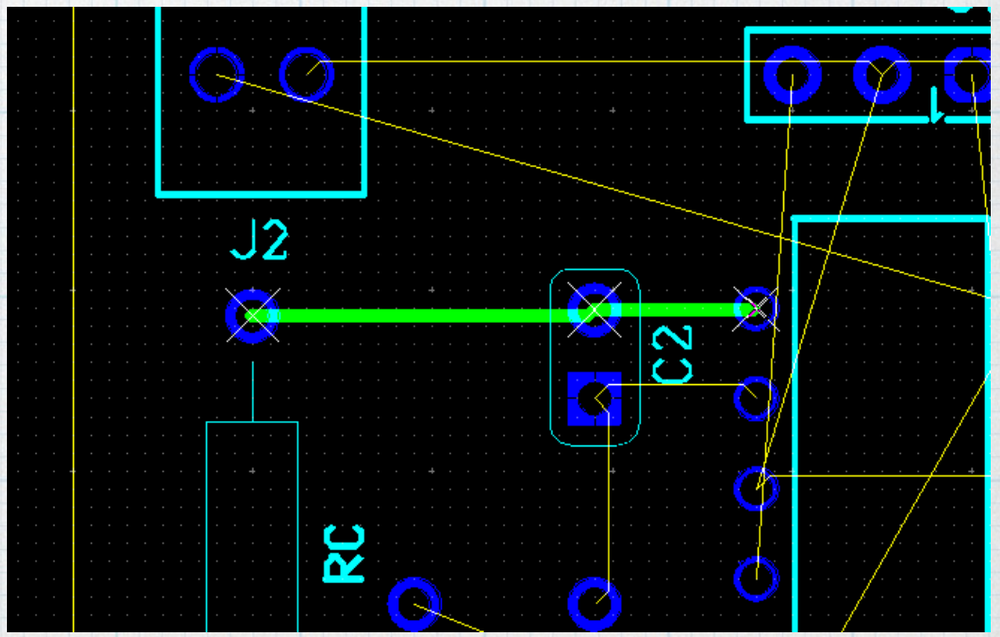
Making traces

Circuit traces can be laid out manually or using the auto-router. First, we'll try the manual approach.

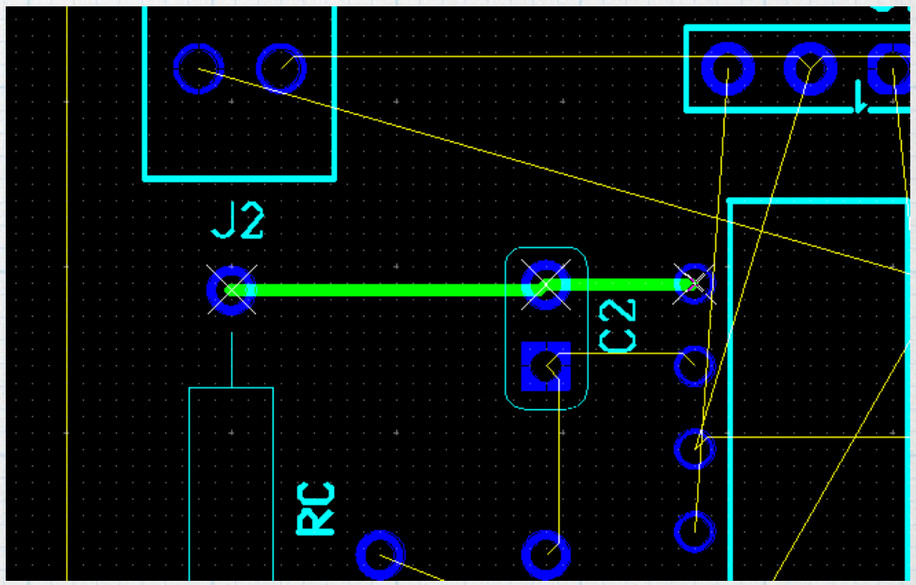
Start by setting the width. The default value is 10 mils, which would be adequate for currents of 0.85 A or less in outside layers. To provide a bit more cushion, we will bump the standard trace width to 15 mils, which will handle 1 A easily. For ground and power supply traces, we might go a bit further and use 20 or 25 mils.

In the "Draw Settings" toolbar, increase the trace width setting to 0.015 inches (15 mils).

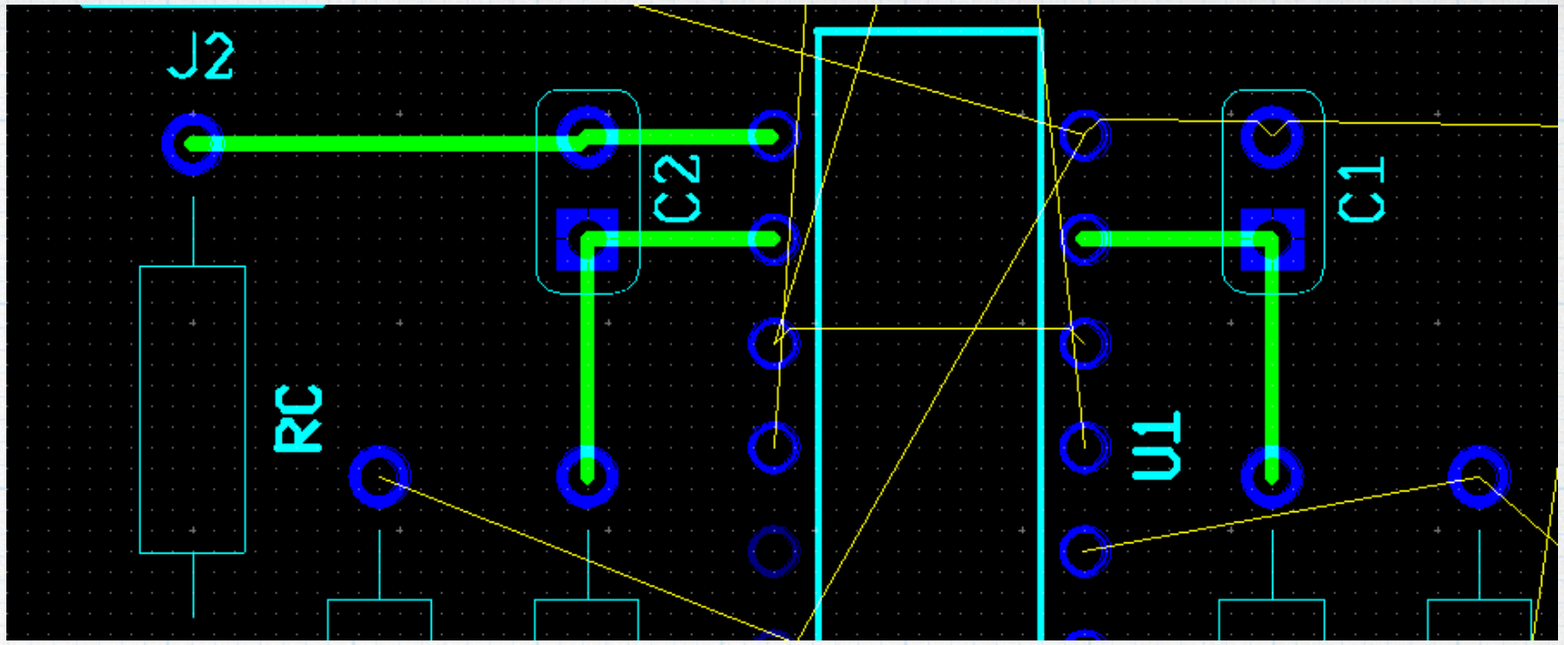
Choose "Line" from the Place menu. Move the cursor to a connection at the end of a rats-nest wire and click. All of the connections that are common to that rats-nest line will have "X" drawn over. Move the cursor to the next contact point. A trace will be drawn. Click on the contact point to complete that section of the trace.



- Choose “Line” from the “Place” menu.
- Move the cursor to a through-hole via at the end of a rats-nest wire and click on it. All of the vias that are common to that rats-nest line will be indicated with an “X”.
- Move the cursor to the next via. A trace will be drawn as you move the cursor. (The color is green because we are drawing in the top copper layer. Check the “Design Toolbox” for the color code.)
- Click on the second via to complete the first section of the trace. The rats-nest line disappears, indicating that the two vias are now connected.
- Move the cursor to the next contact along the rats-nest line, and click to complete that section.
- When all of the contacts along the rats-nest are connected, hit the “esc” key once to stop drawing that particular trace. (We are still in drawing mode and can start on another trace.) If the “esc” key is tapped a second time, the drawing mode is ended — the cursor changes back to the standard arrow. To continue drawing, we must go back to the “Place” menu and choose “Line” again.

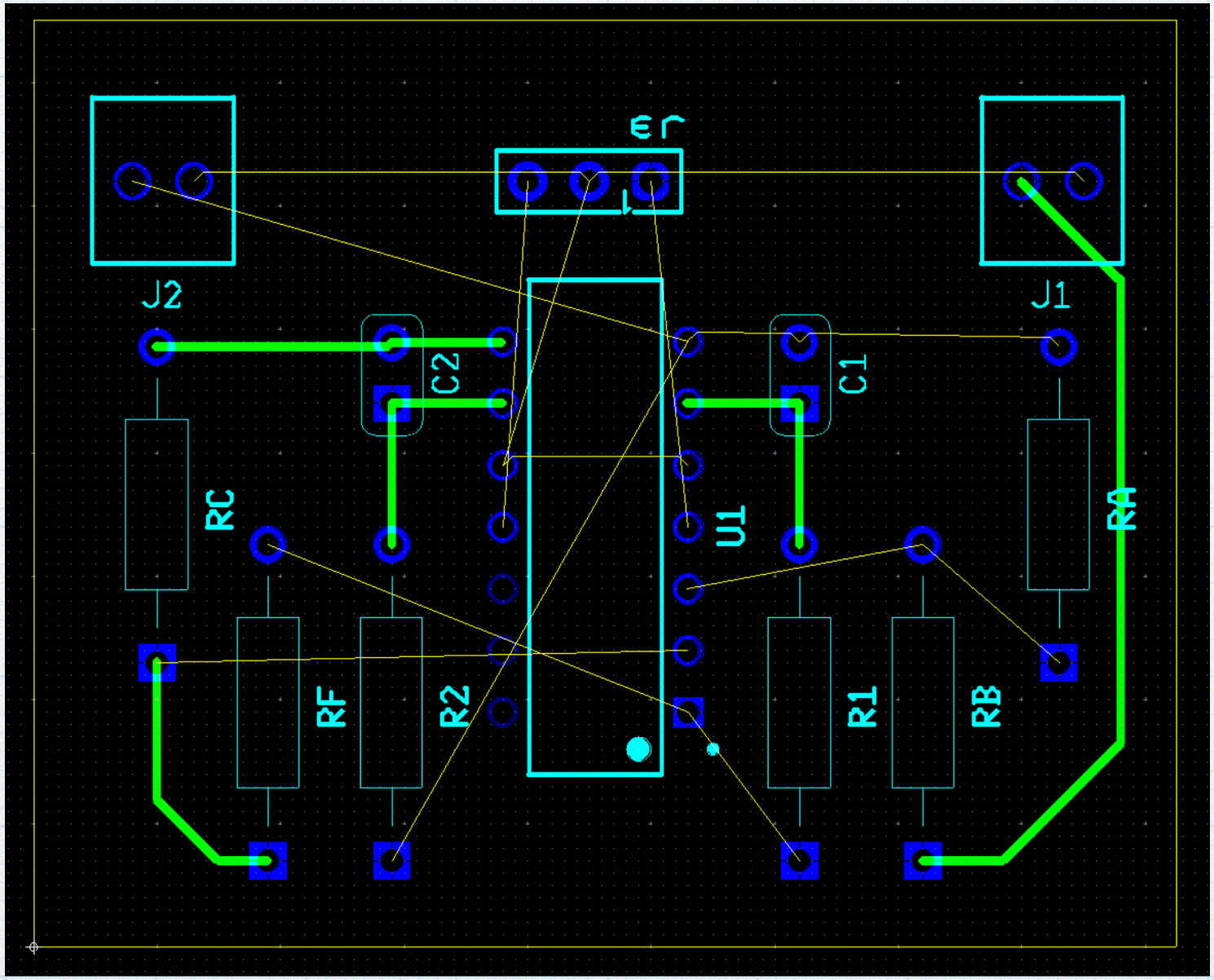


Run two or three more of the shorter traces.



When we start making traces that cannot be completed with short, straight runs, we will need to make turns. The program will try to "help" in making corners — as the cursor is deviated from a straight line, the trace will turn to follow. The program will not make 90° turns. (Right-angle turns lead to current crowding, which increases resistance at the turn.) Instead, the program will use a 45° turn. With a bit of practice, we will get used to changing directions with 45° turns. Usually, it will be necessary to "tack down" the trace by clicking on a point along the expected route. The trace will be locked in place at that point, and we can continue on with the next section of the trace. If you are not happy with the location of the trace, simple hit "Esc" to jump out of drawing mode, select the bad trace, and delete it. (Remember that it is necessary to enable selecting traces before trying to do that!)

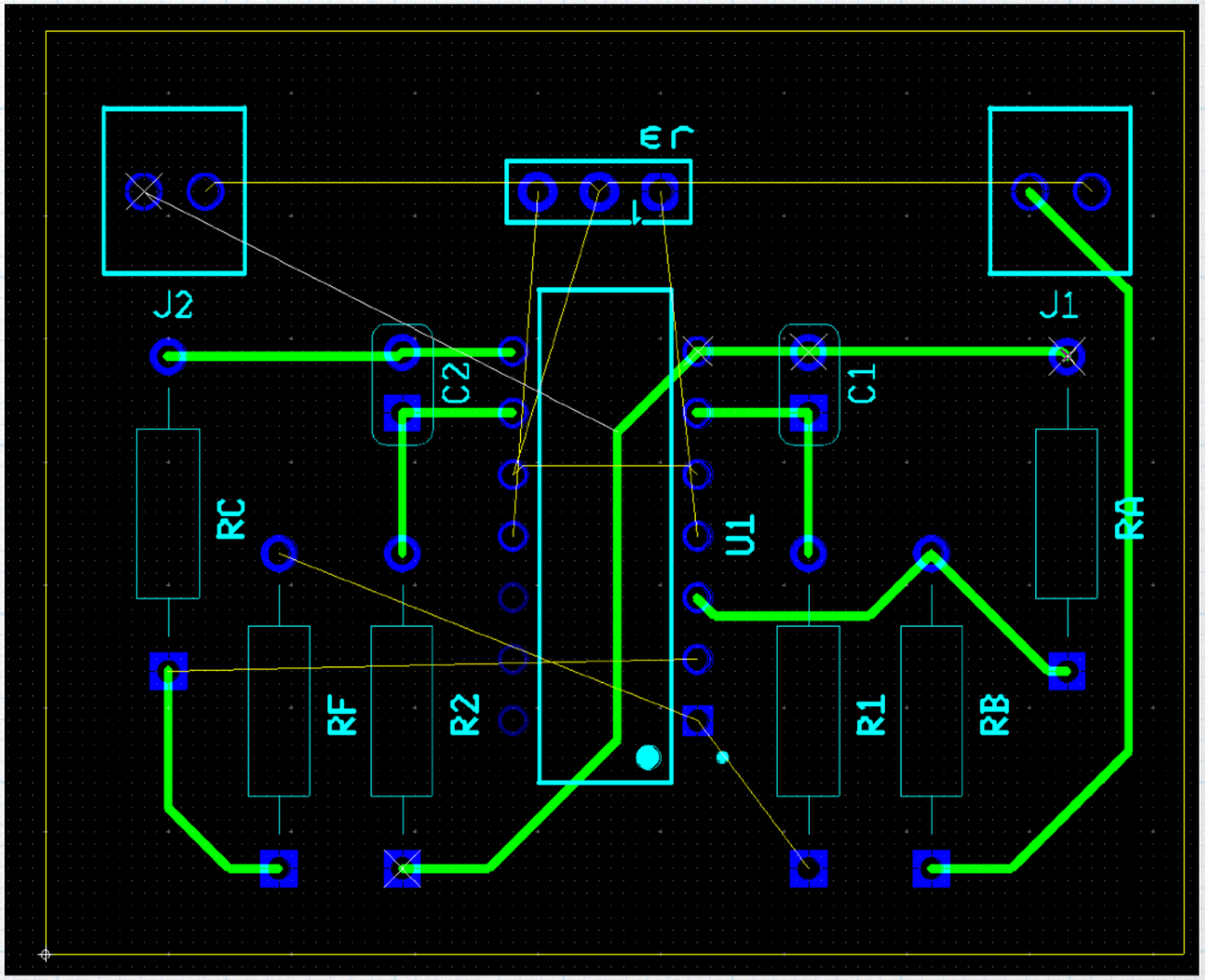
Here are two more traces with some turns.



Note: Even though a single trace should not have a 90° bend, it is OK to have sharp turns at vias, which have a significant amount of metal to facilitate current flow.

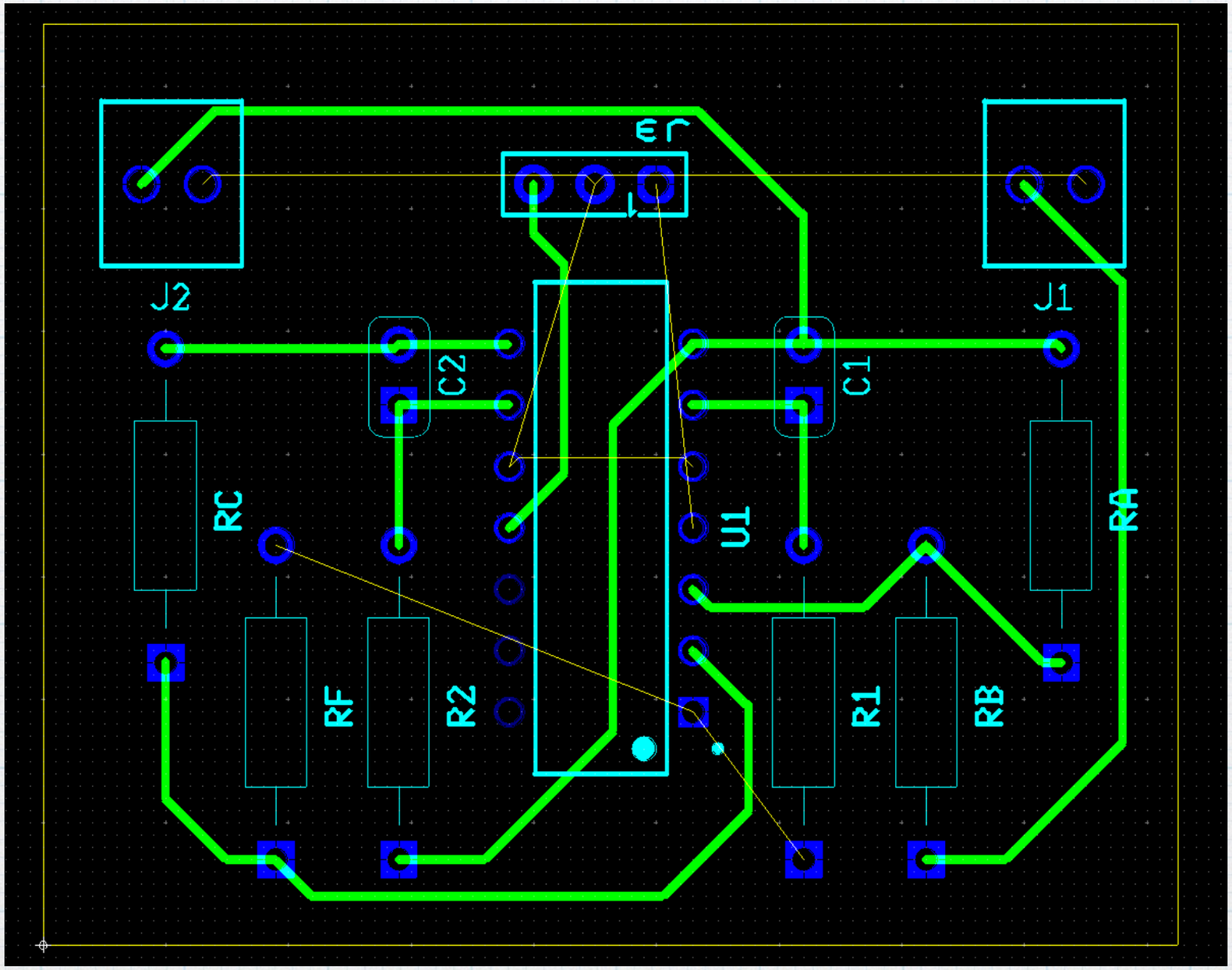
As we complete more traces, it becomes more necessary to route around vias and other traces. (This is one of the reasons that we arranged the component layout to try to minimize the number of crossing rats-nest lines.

As we look for route options, it is useful to note that traces can run underneath components. This quite different from prototyping on perf-board or on a solder-less breadboad. Don't be afraid to use of all the available space.



Note the traces routed underneath R1 and the IC.

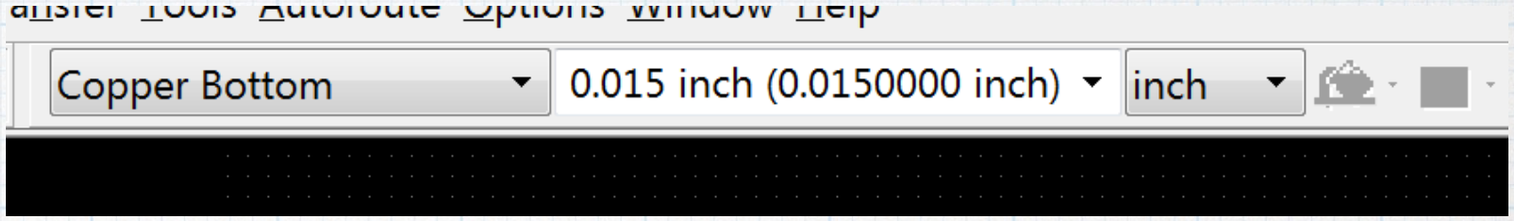
As we continue that exercise of converting rats-nest lines into traces, we may eventually get into a bind — we cannot make any new traces without crossing existing traces. We get out of the bind by remembering that we still have the bottom copper, which has not yet been used at all.

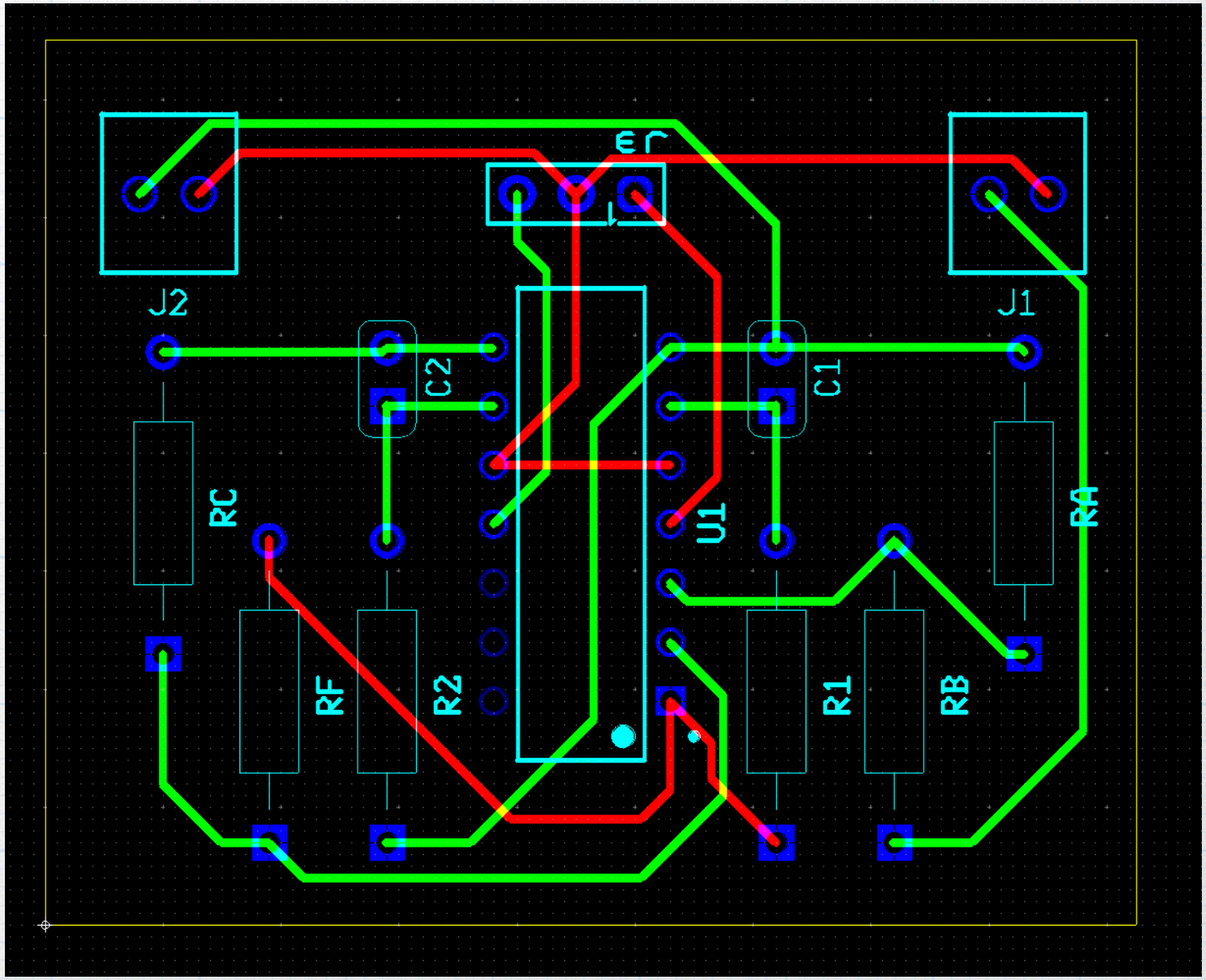


On the draw settings menu, switch over to Copper Bottom. Continue making traces using the bottom layer, which is indicated in red.

Using to the bottom layer, we are able to complete all the interconnections — all rats-nest lines are gone.

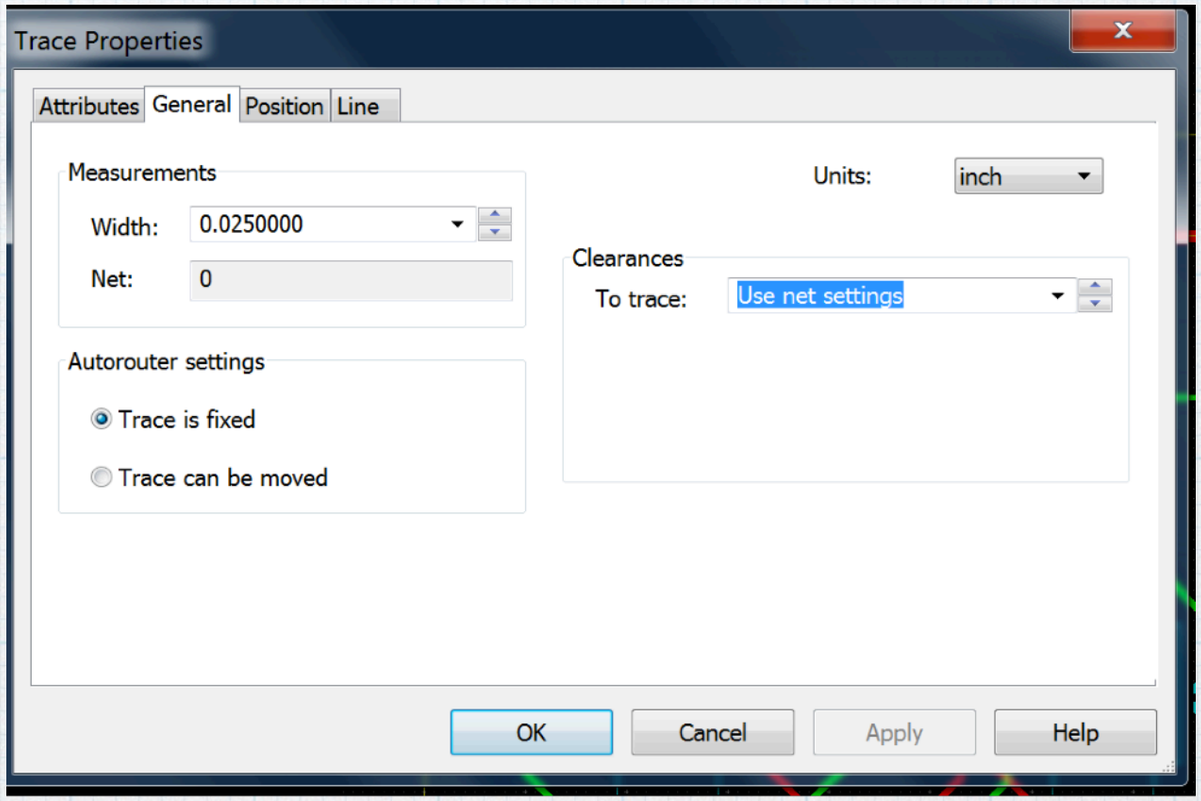
Note that there are many possible solutions to the routing configuration. There is nothing particularly good or bad about this arrangement. As practice, we should probably try some other trace configurations.





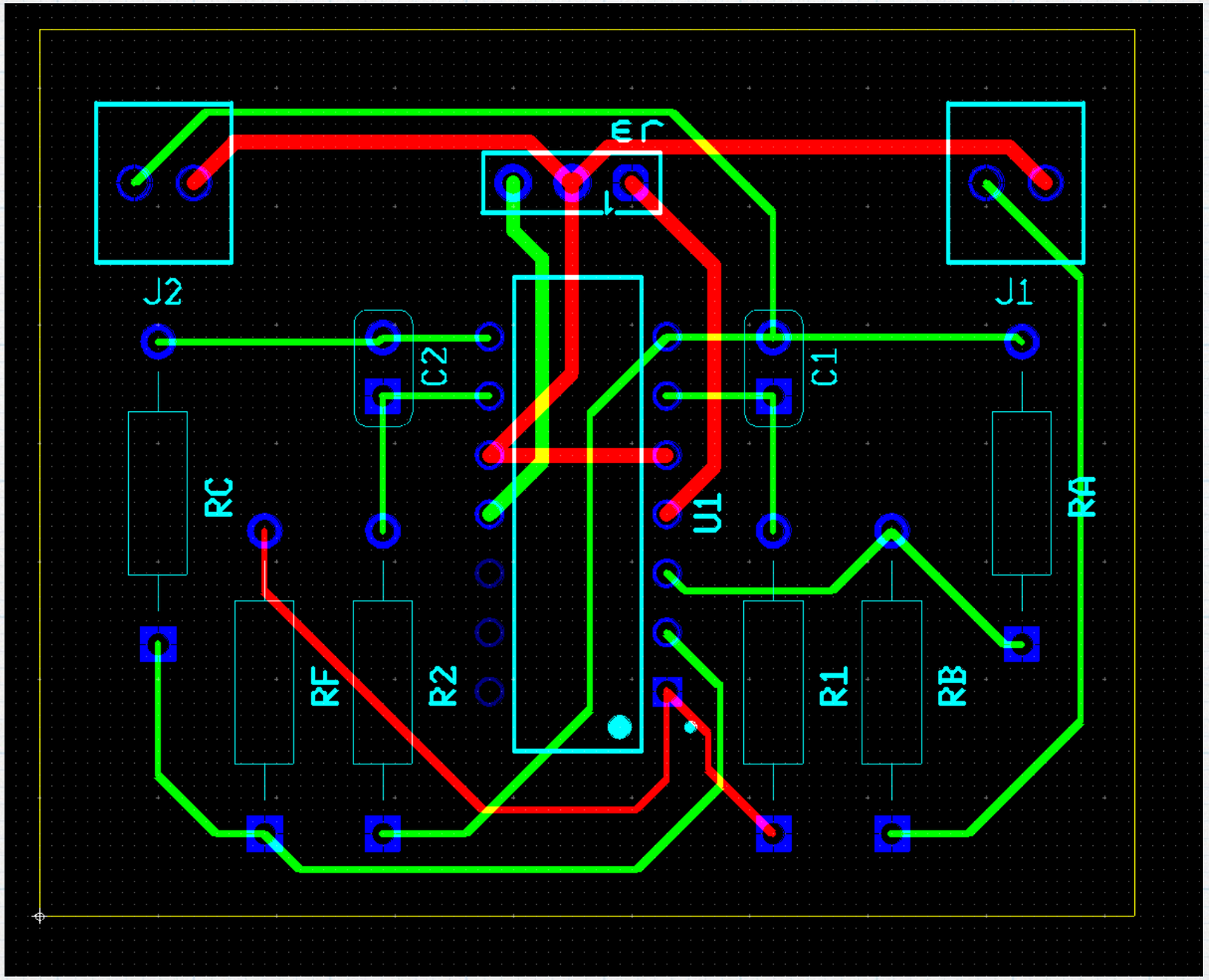
The traces can be edited easily. To move a segment, simple click on it, and then move it with the cursor. You can also delete segments or entire traces and start over.

To change the widths of any traces after they have been routed, double-click on a segment to bring up the "Trace Properties" dialog. Change the trace width under the "General" tab. (Also, it is possible to shift-click to select a group of segments and then double click to edit the group.)



For this design, we increased the widths of the VS+, VS–, and ground traces (i.e. all the power) to 25 mils.

"Trace Properties" can also be invoked by right-clicking on a segment and choosing the "Properties" item from the pop-up menu.



There is an alternative mode for drawing traces manually, called “Follow me” under the “Place” menu. It is similar to the “Line” mode, but the program picks the route between the two vias. It adds a little bit of automation to the manual process. Try it — you might like it.

Ground planes (power planes)

An electronic circuit often has many components connected to the ground and power nodes. Of course, this means that there might be many ground or power traces, as well. (This circuit is an exception, having only one connection to each of the power leads and only a few ground connections.) To limit the number of ground traces that need to routed, we can use a ground plane. Essentially, a ground is giant trace that covers one side (or both sides!) of the PCB. Since the ground plane is available over a large area, connecting vias to it is easy. And we do not to worry about trace widths because it as wide as possible. Also, the larger amount of metal in the ground plane means that it is easier to dissipate heat generated by the components.

As a general rule, we should use ground planes on one or both sides of our PCB designs. It is not an absolute requirement, but there are enough advantages to including a ground plane should be the default approach. We can also use power planes for the positive and negative power supplies. Typically, this is done in multi-layer boards where one or more internal copper layers are devoted to power supplies.

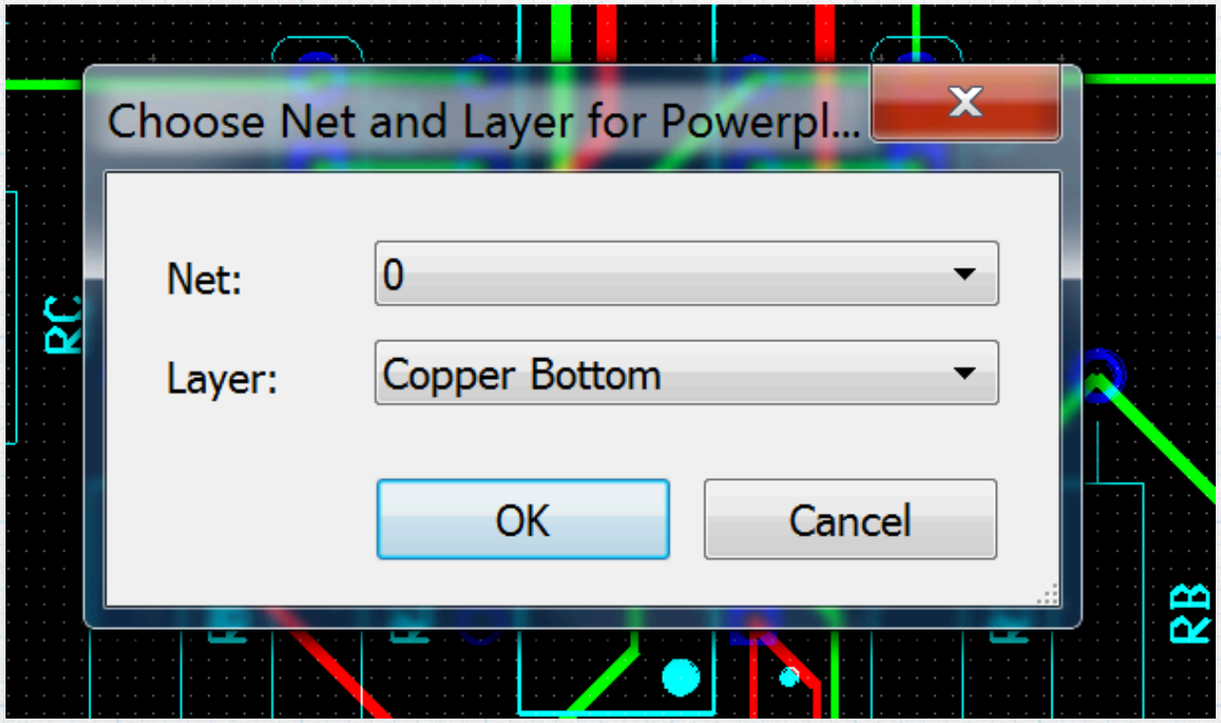
One aspect of using ground planes is that any other (non-ground) traces must be isolated from the ground plane. This is done by routing the trace in the usual manner, and then removing a strip of the ground plane on either side of the trace. (Think of the ditches on either side of the roadway, isolating the road from the surrounding countryside.) The "ditches" in the PCB are generated automatically by the program.

Typically, if we want to use a ground plane, we would specify that *before* we start laying out any traces — that saves on some of the work. In our example, we have already defined ground traces. In principle, we could just leave these — the program will simply "cover up" those traces with the ground plane. However, there may be some artifacts of the ground traces that may show up, even with the ground plane, so we will remove them.

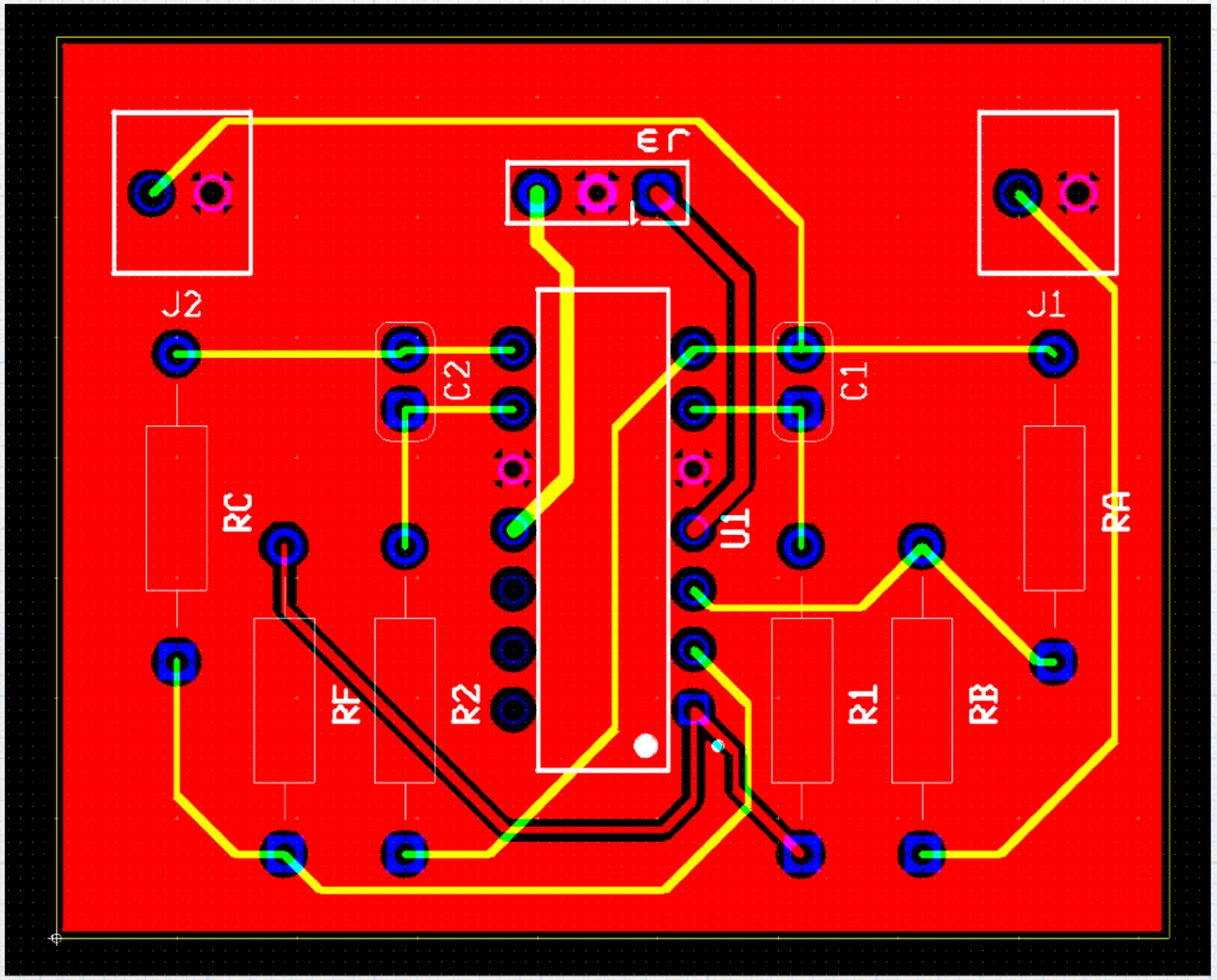
In this example, the ground connections are fairly simple, and a single ground plane — top or bottom — is probably sufficient. We will put the ground plane on the bottom.

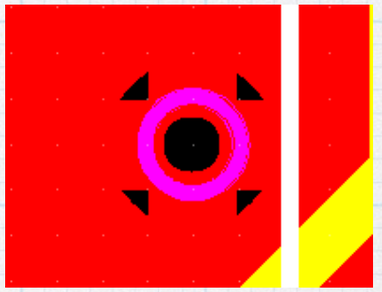
To include the bottom ground plane, first go back and remove the ground connections. (Select each section and then “delete” or “cut”.)
Then:

- Select “Power Plane…” from the “Place” menu.
- In the dialog opens, choose “0” for the Net and “Copper Bottom” for the Layer.
- Click OK.



Now the entire bottom is covered with copper, except for the areas around the through-hole via and the “ditches” on either side of the bottom traces. The ground vias are now connected to the ground plane with “thermals”.





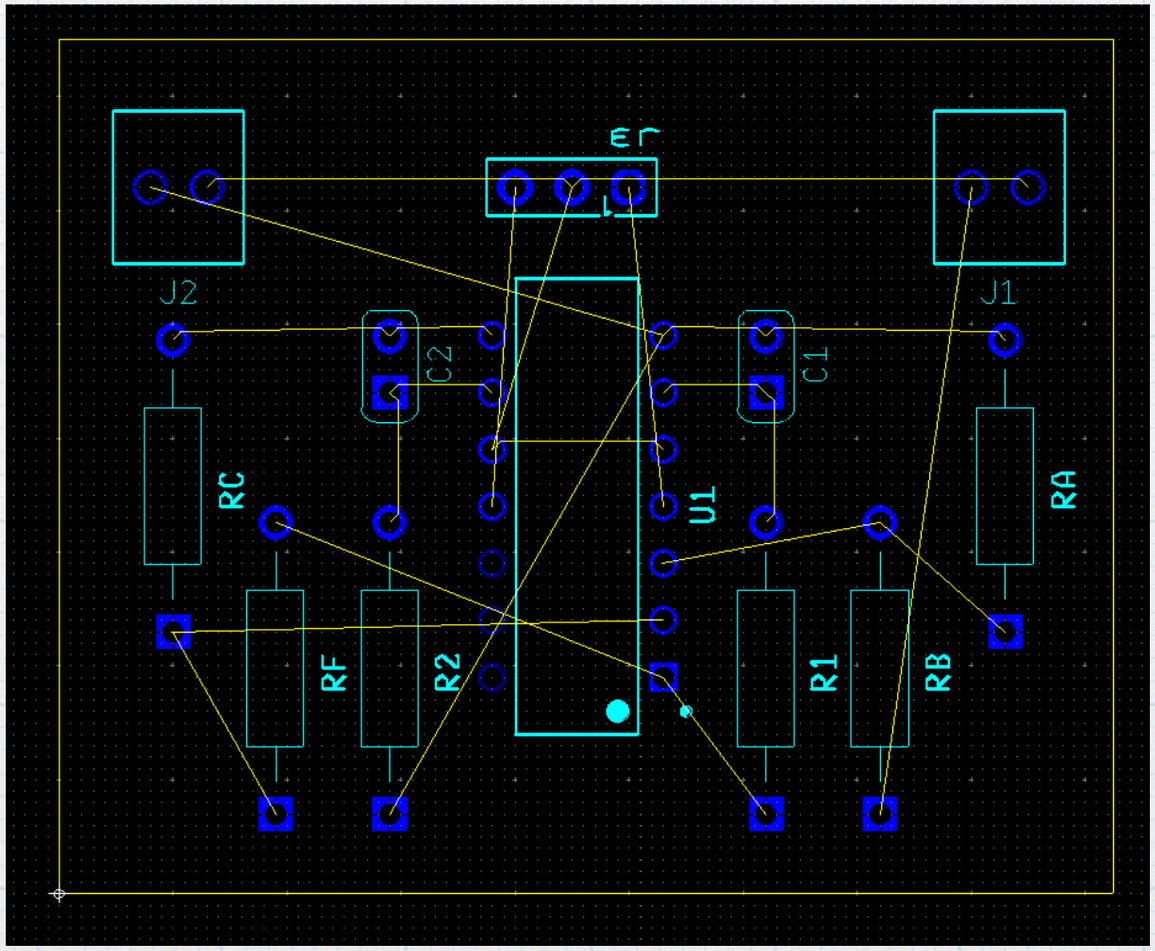
“Thermal” ground plane connection.

We might want to change some aspects of the grounds — in particular the clearance (“ditch” width), which have a default of 10 mils, and the size of the thermals. We will cover making such changes in part 3 of the tutorial.

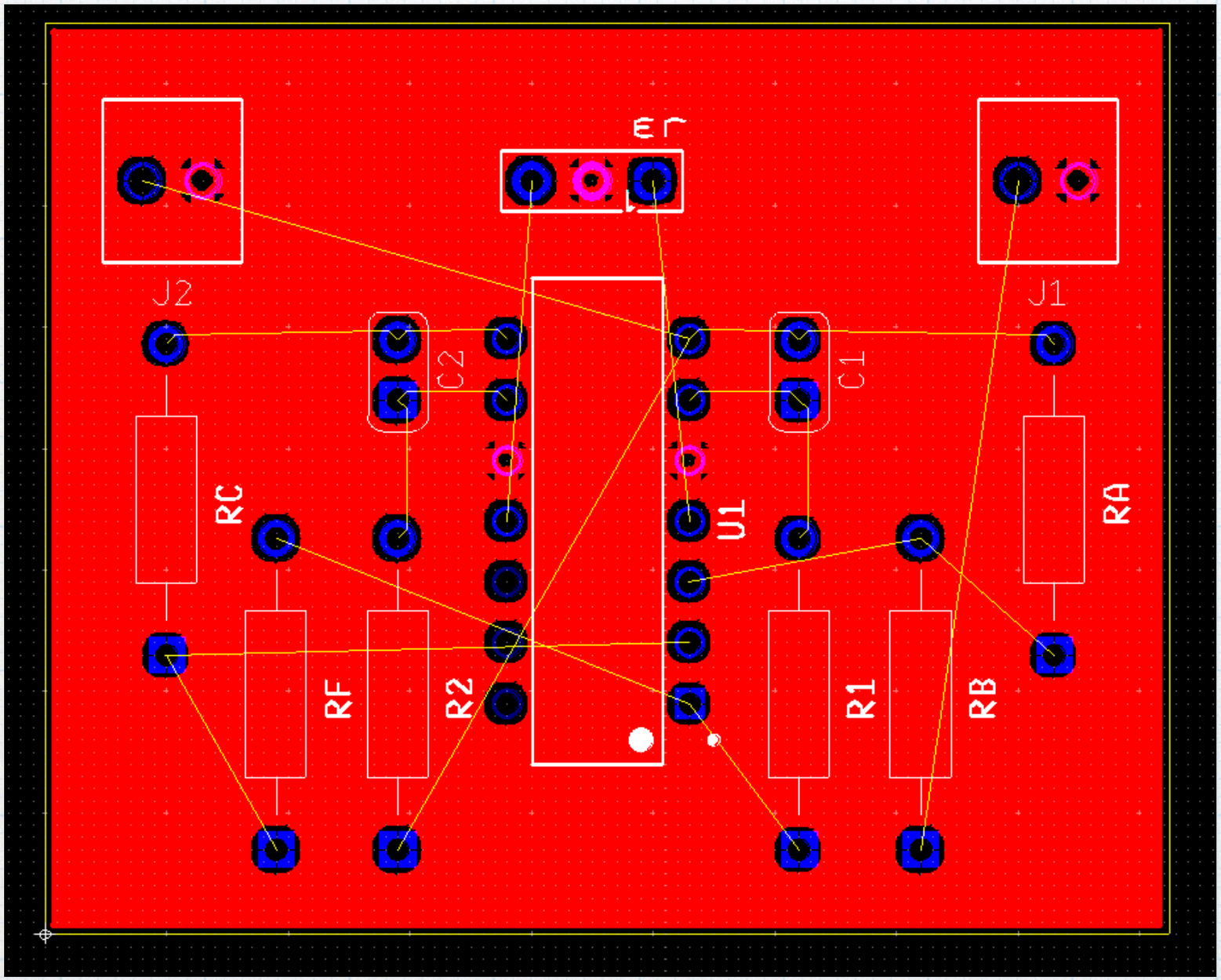
Autorouting traces

For a simple circuit, it is not difficult to route the traces manually. However, all PCB design programs offer the possibility of routing the traces automatically. The advantage is speed. A possible disadvantage is that trace patterns may become overly complicated. It all depends on the algorithm that controls the auto-placement of the traces.

To see how auto-routing works, we go back to the design at the point where all of the parts have been placed, but no traces have been drawn.

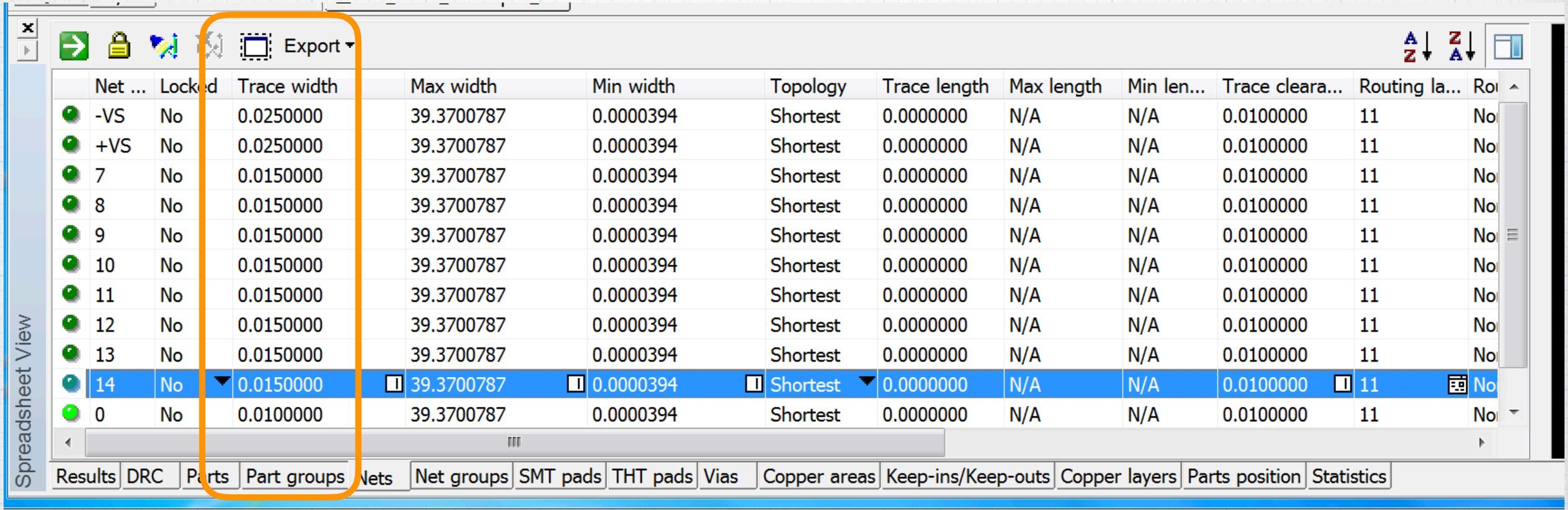


1. Create the ground plane, if using one. See the earlier instructions to create a ground plane on the bottom.



2. We would like to control the thicknesses of the traces that the auto-router will define. We could let the auto-router use the default widths and then change them afterwards, but it is better to set up everything prior to running the auto-router.

   - Choose "Spreadsheet View" from the "View" menu — a region with lots of text opens at the bottom of the window. (The size of this spreadsheet area can be adjusted.)
   - Click on the "Nets" tab. A spread-sheet-like collection of information about each of the circuit nets (nodes) appears. Using this, we can see — and change — many of the parameters for each individual net.
   - Change all of the traces widths to the values used previously during manual routing — 25 mils for -VS, and +VS, and 15 mils for all other nets. (Since we are using a ground plane, the width for Net 0 — ground — is irrelevant.)
   - We also note that we can see — and could change — the trace clearances. The 10-mil default value is OK for now. (We will see shortly that the traces clearances won't be needed now, anyway.)



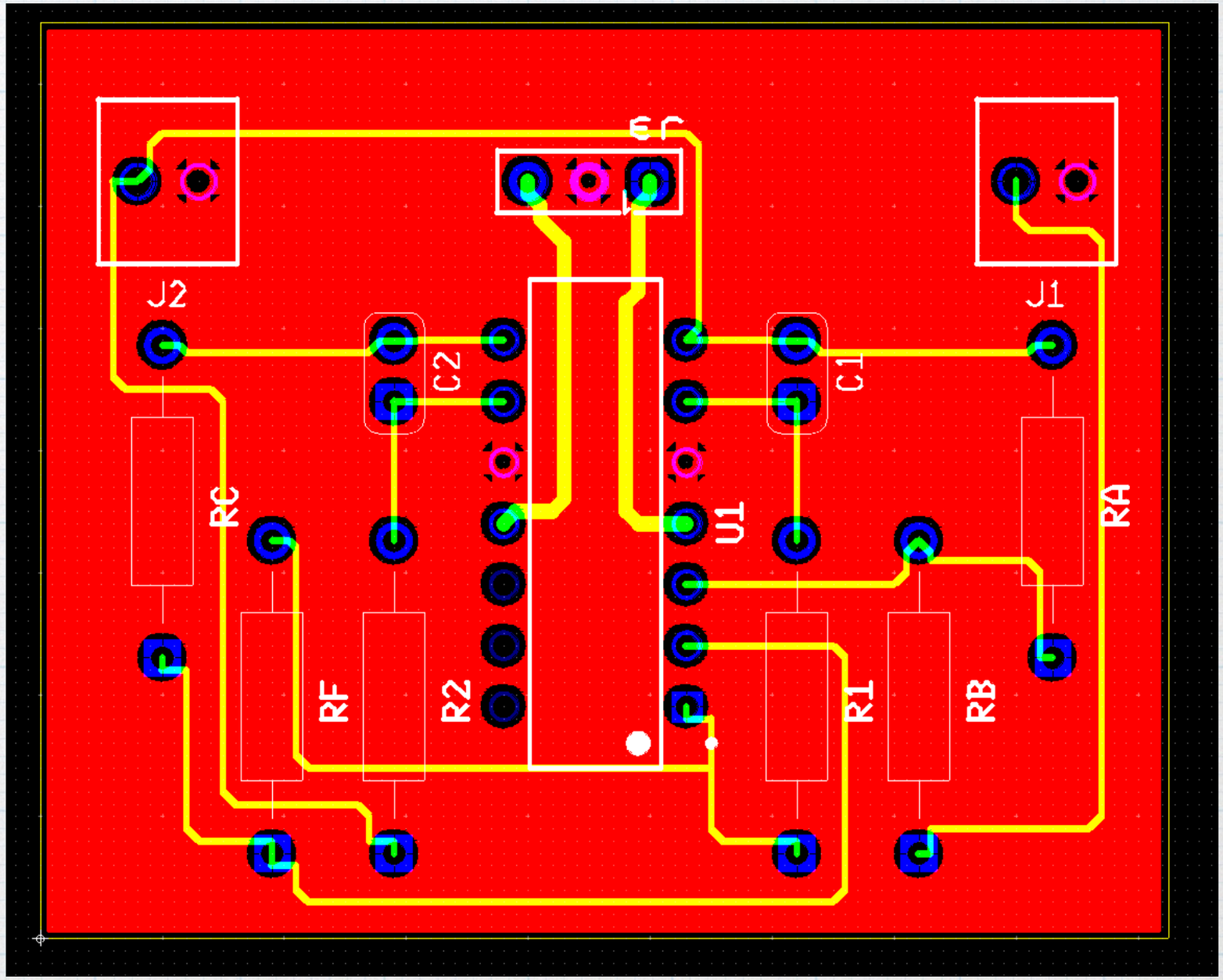
The “Nets” tab of the spreadsheet view, showing the altered values of the traces widths.

3. Close the Spreadsheet View (just to get it out of the way for now). Then choose “Start/resume autorouter” from the “Autorroute" menu.

4. In about a second or so, the autorouter algorithm will determine an “optimum” pattern for routing all of the traces and then draw the traces onto the layout.

Auto-routed traces.

There is information about the operation under the “Results” tab of the “Spreadsheet View”, which, of course, must be re-opened.



Autorouter [BP_filter_example_1] 9/13/2018 5:00:28 PM
Routing completed successfully.
24 of 24 connections (100%) routed with 0 vias. Time: 0:00

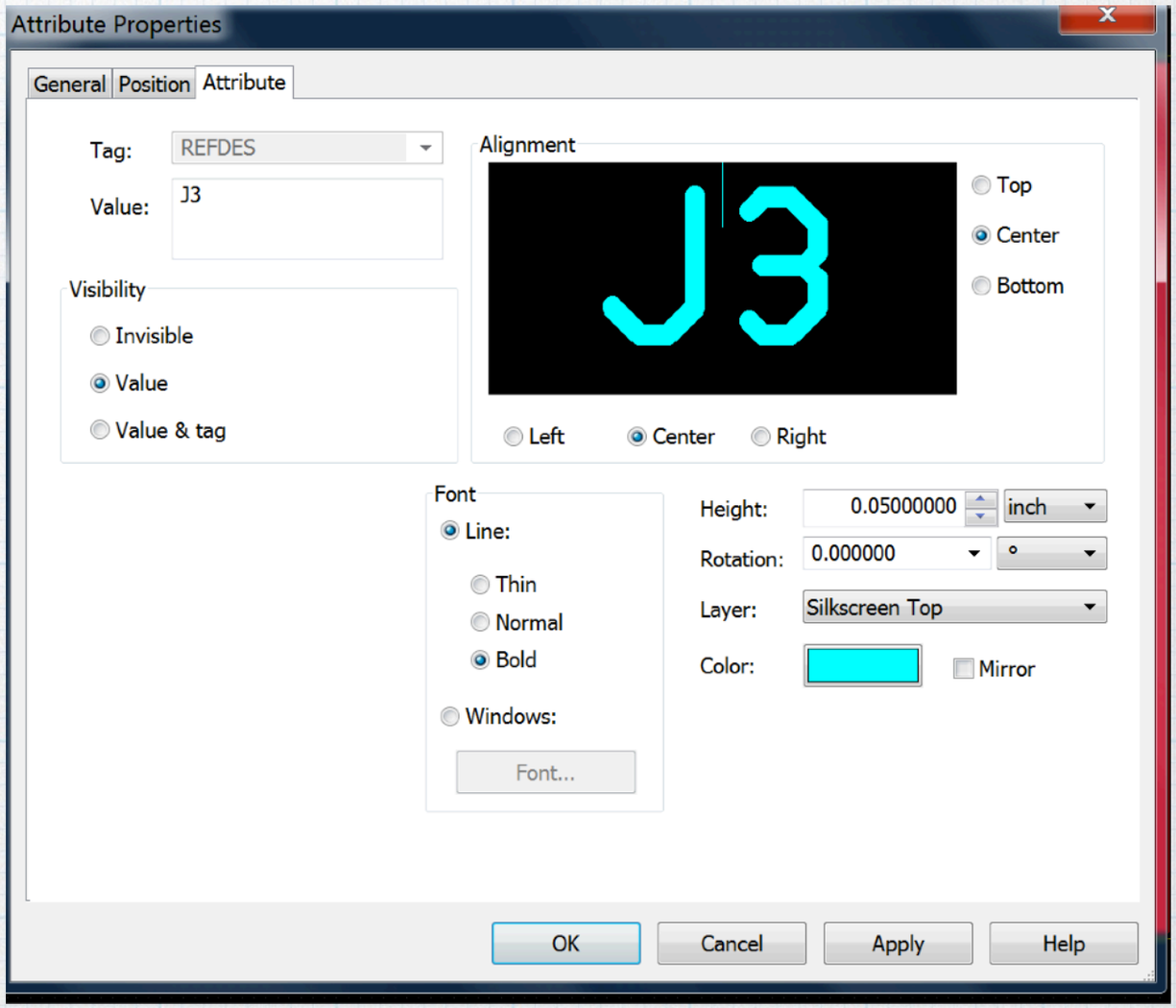
Autorouting observations/comments

- For this example, the auto-router was able to find a solution in which all non-ground traces were on the top copper. That a feature of the algorithm — it will try to put everything on one layer if possible and only use the other side copper if necessary.
- Obviously, auto-routing is very fast.
- It is difficult to quantify as to whether auto-routing is "better" than manual routing. We would need some sort of metric to use in making a comparison.
- The algorithm seems to "prefer" north-south and east-west runs. There are very few diagonals, except where needed to make a turn.
- Sometimes it seems to use many small-angle turns when a longer stretch at an angle of 45° might been have cleaner.
- We can always make manual adjustments to the auto-routed traces to "fix" anything that we don't like.
- Sometimes the auto-router fails, meaning that it cannot make all of the connections. For instance, in our example, if we use top and bottom ground planes, the auto-router fails miserably. Failures are also reported in the "Results" section of the "Spreadsheet View".
- Almost all aspects of how the auto-router works can be modified. See the manual for all the details.

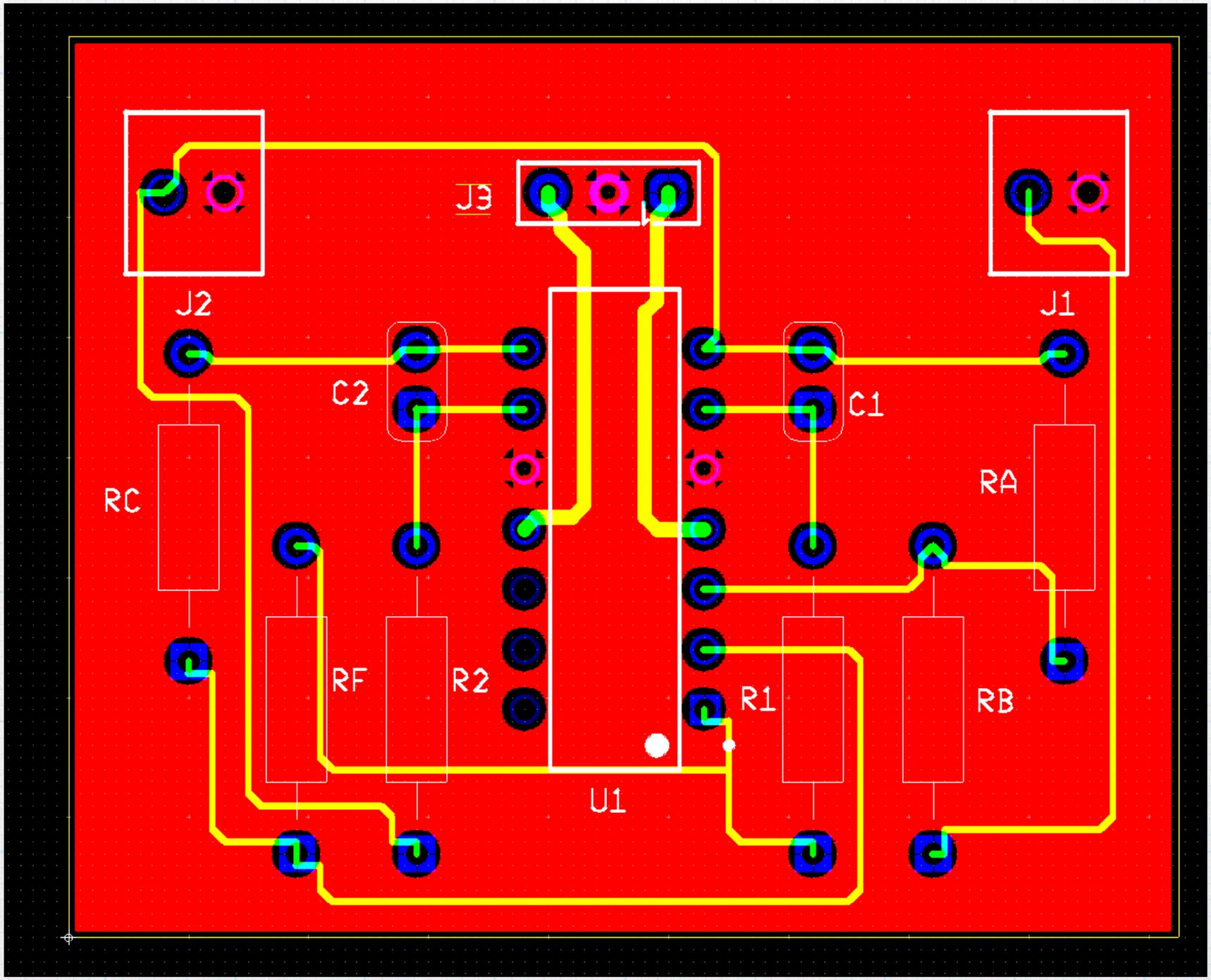
Silkscreen layers – labels and documentation

All of the labeling in the silkscreen layers can edited. It is not necessary to change anything, but often the lettering can be cleaned up to make it easier to read. To change the location, orientation, font, or any aspect of the silkscreen objects:

1. Click on "Enable selecting attributes" from the "Select" toolbar.
2. Click on a label to select it and move it to a new location.
3. Double-click on label to bring up a dialog that allows for changing the orientation, size, and font. (In this example, we have made all of the lettering face the same way with a height of 50 mils.)



Labels are “cleaned up”.

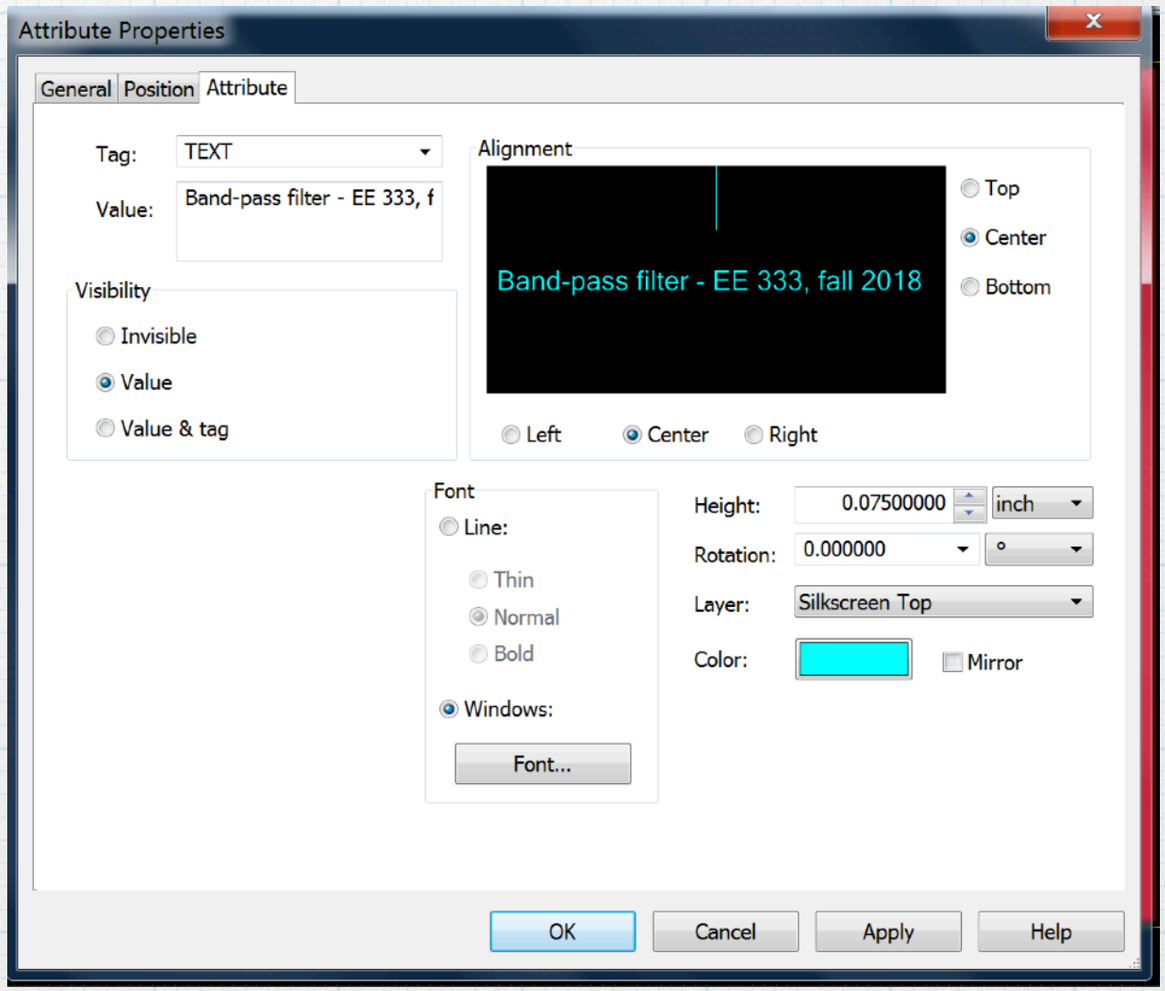


Other text

It is reasonable to add a title and other documentation to the board.

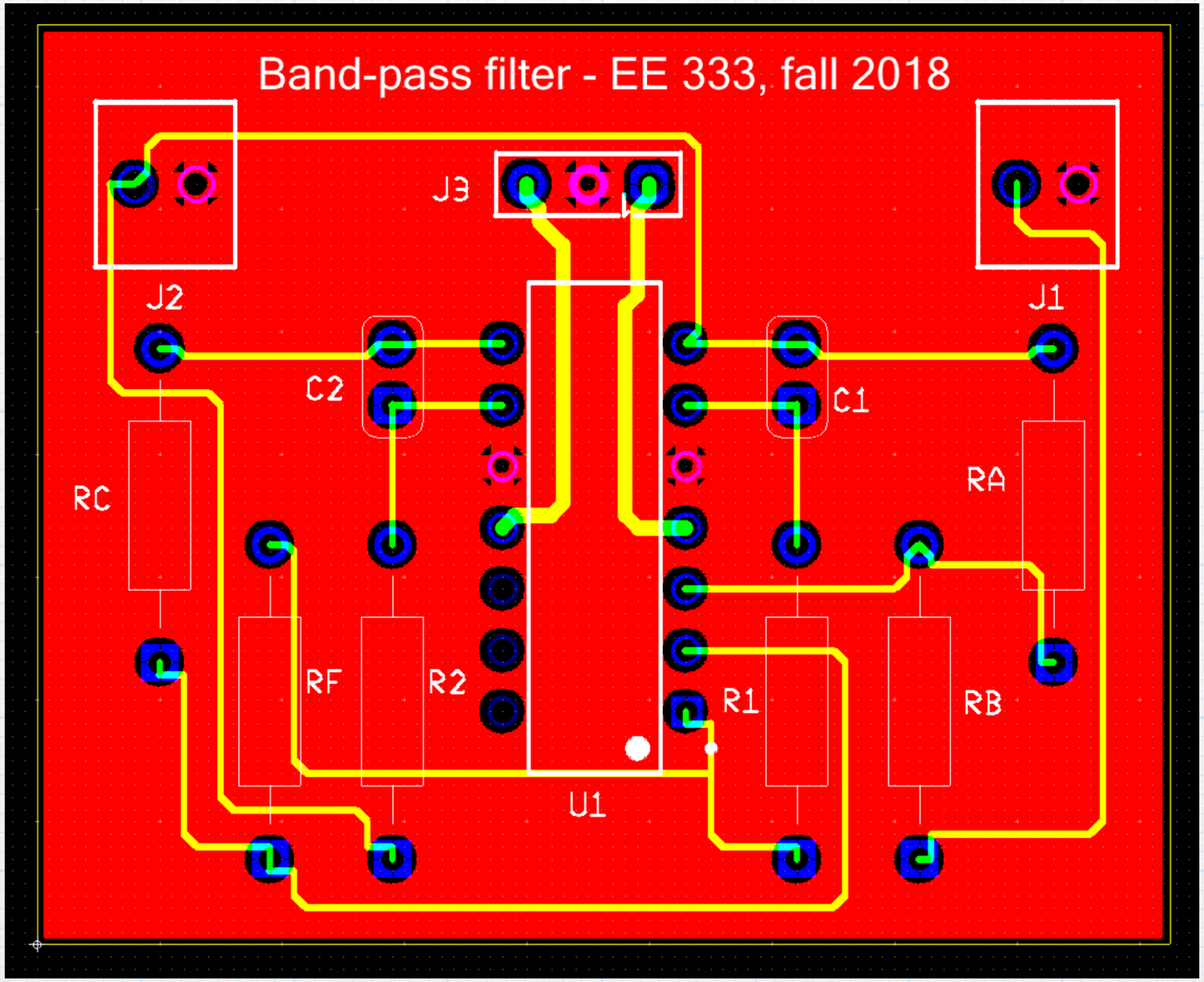
1. Choose the "Graphics –> Text" under the "Place" menu.
2. In the dialog that opens, enter a "value" for the text (i.e. the actual text). The size and font can be changed. Also note which layer the text will be in — it should be one of the silkscreen layers. (It could be put into one of copper layers, but then the solder mask should be also be removed in order to see the lettering.)
3. Click OK and move the text to the desired location.

The text could also go on the back side (Silkscreen Bottom). In that case, be sure to check the "Mirror" option, otherwise the text will be reversed.



The completed board

Whew!



In the part 3 of our epic PCB saga, we will cover error checking, making some specific adjustments, and generating Gerber files to submit to the manufacturer.