

TINACloud

PCB Design Manual

DesignSoft

www.designsoftware.com

CONTENTS

CREATING A PRINTED CIRCUIT BOARD	3
1.1 Setting and checking footprint names	3
1.2 Invoking TINACloud PCB	6
1.3 Advanced editing functions of TINACloud PCB's Layout Editor.....	11
 CREATING A TWO-LAYER, DOUBLE-SIDED, SURFACE-MOUNT TECHNOLOGY BOARD	 12

CREATING A PRINTED CIRCUIT BOARD (PCB)

Using TINACloud, you've captured the schematic of your circuit and refined the design. It's time to make a prototype of the circuit or to manufacture it. It's good to know that you can continue the process still using TINACloud, since PCB design is now an integral part of the program.

Let's learn how to design a PCB by working through a few examples.

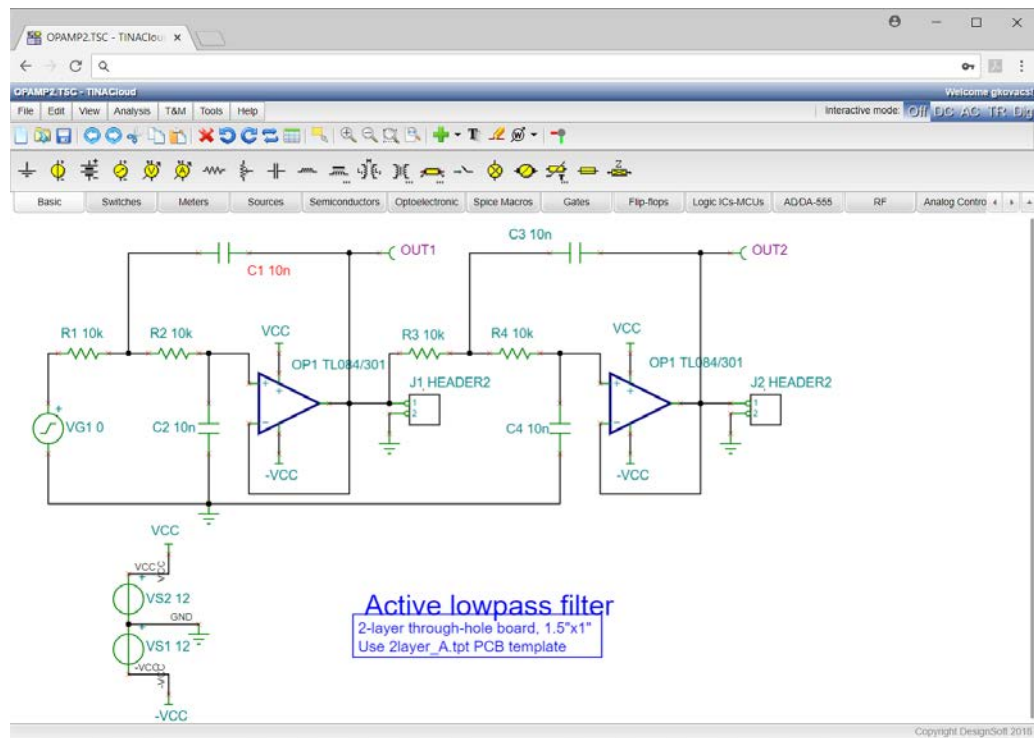
Note that the different phases of the presented design examples have been saved in the 'TINA'Clouds Examples/PCB directory using the following naming conventions:

* origin.tsc	original schematic file
*.tsc	backannotated schematic file (after pin/gate swapping)renumbering)
* placed.tpc	design parameters set, components placed pcb file
* routed.tpc	net properties set and routed pcb file
* finished.tpc	optionally pin/gate swapped and renumbered, routed, silkscreen adjusted, documentation layers finalized pcb file

If you save your versions of these demo examples, be careful not to overwrite the original files TINACloud installed.

1.1 Setting and checking footprint names

As a first example, open the opamp2.tsc project from TINACloud's Examples/PCB folder. The following schematic will appear:



To achieve an accurate PCB design, one that is easy to build, every part in your schematic must have a physical representation with exact physical size. This is realized through so-called footprints: drawings showing the outline and the pins of the parts.

The footprint naming convention in TINAcloud uses as a starting point the IPC-SM-782A (Surface Mount Design and Land Pattern Standard) and the JEDEC standard JESD30C (Descriptive Designation System for Semiconductor-Device Packages, see <http://www.jedec.org/download/search/jesd30c.pdf>). However, the libraries do not conform to any given set of industry or manufacturer standards because standards have difficulty keeping up to date when technology changes faster than the standards. Generally, standards mirror a fixed set of data at a point in time, while new manufacturing capabilities lead to ever-smaller new footprints with more and more pins.

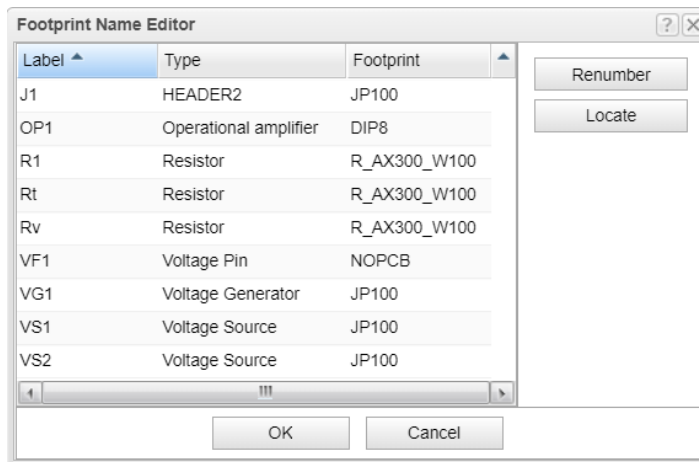
We have already assigned default footprint names to all parts that represent real components.

Note that some parts used for theoretical investigations, controlled sources, for example, do not represent real physical parts and cannot be placed on a PCB. If your design contains such components, you should replace them with real physical parts.

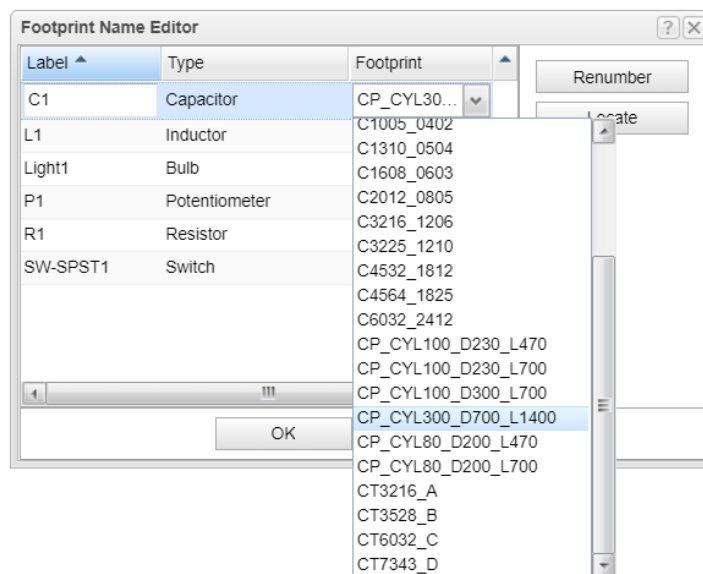
Of course there is no guarantee that the default physical packages of TINAcloud's parts are the same as needed by your design.

There are two ways to check this.

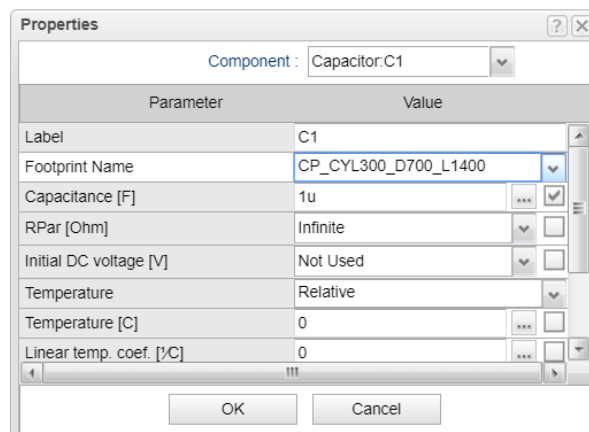
- 1) You can use TINAcloud's "Footprint name editor", which you can invoke from TINAcloud's Tools menu. In this dialog you can see all of TINAcloud's components and their corresponding footprint names.




Select from the available footprint names by clicking on the footprint name fields. In the dialog box, components with no footprint name association are denoted by empty footprint name field.

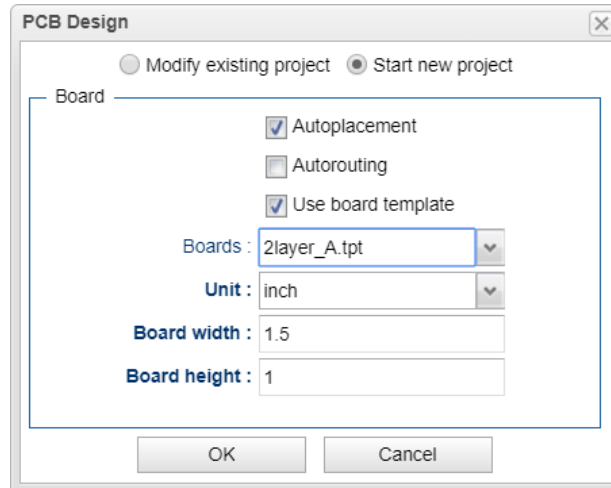


2) The second way to examine the assigned footprints is to double-click on each part and check the Footprint Name in the component property dialog that appears.



1.2 Invoking TINACloud PCB

To begin PCB design, press the  button on TINACloud's toolbar (the last one on the right) or select the "PCB Design" command on the Tools menu. Set the parameters as shown below.



Select "Start New Project," "Autoplacement," and "Use board template." With the Browse button find and select the 2layer_A.tpt template file from TINACloud's Template folder. Using this file will ensure the proper settings for a double-sided PCB.

When starting with a TINACloud template, you are choosing the level of manufacturing complexity of your project. The following three levels of manufacturing technology are defined by the IPC-2221 generic standard:

- Level A : General Design Complexity
- Level B : Moderate Design Complexity
- Level C : High Design Complexity

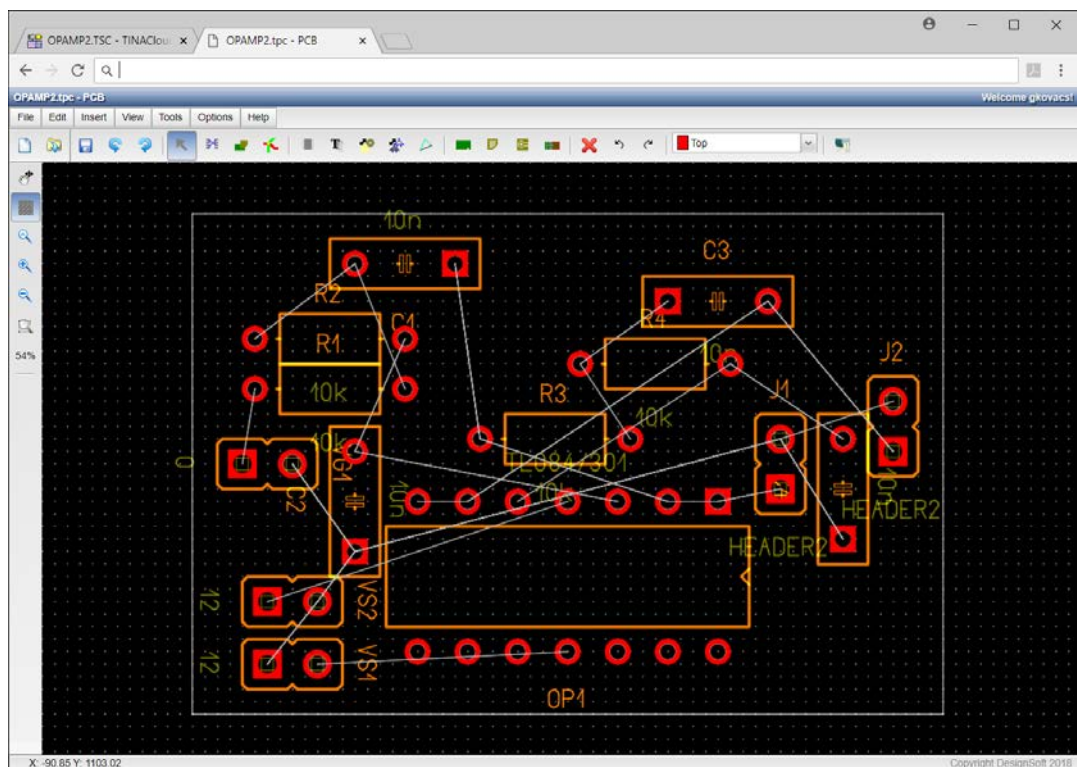
The template file specifies the number of layers, including their properties, system grid size, autorouter settings, spacing and track width. The following templates are included with PCB Designer:

	Level	Routing Layers	Plane Layers	Routing	Spacing	Comments
1layer_A.tpt	A	1	-	25	12 1/2	Allows one track between standard IP IC pin
2layer_A.tpt	A	2	-	25	12 1/2	
2layer_B.tpt	B	2	-	8 1/3	8 1/3	Use for SMT or mixed-technoly board
2layer_B_mm.tpt	B	2	-	0.1	0.2	
4layer_C_mm.tpt	C	2	2	0.1	0.15	For moderate and high density

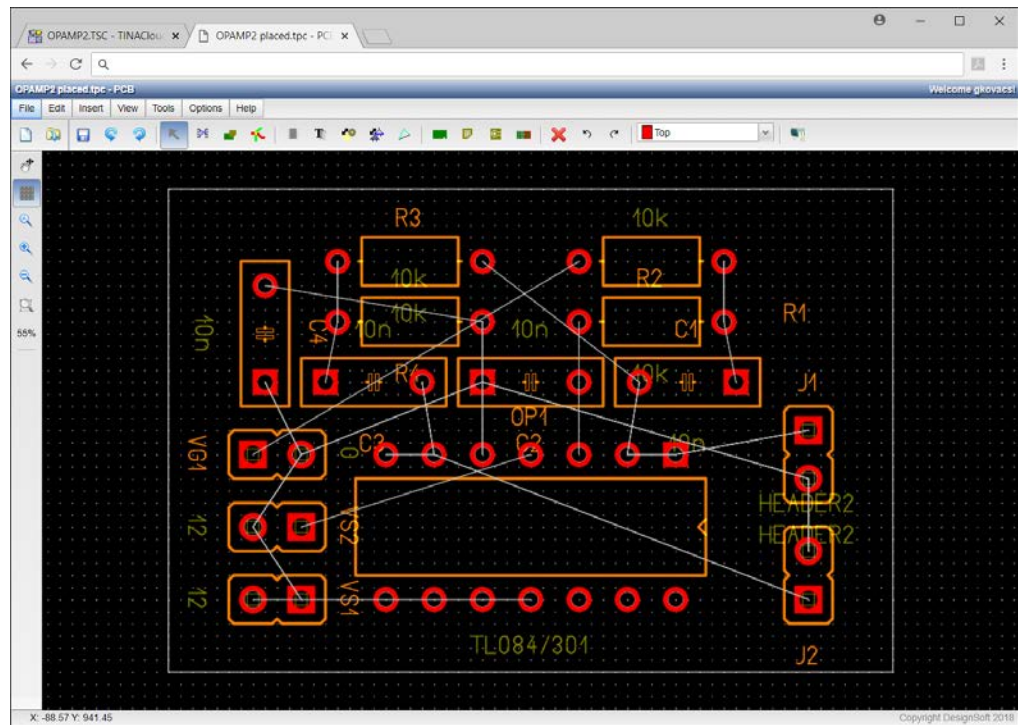
In choosing a PCB template, you should take into consideration technology, density, and package pitch.

To complete the set up, set the PCB size in inches or mm depending on the measurement unit settings in the View/Options dialog of TINACloud.

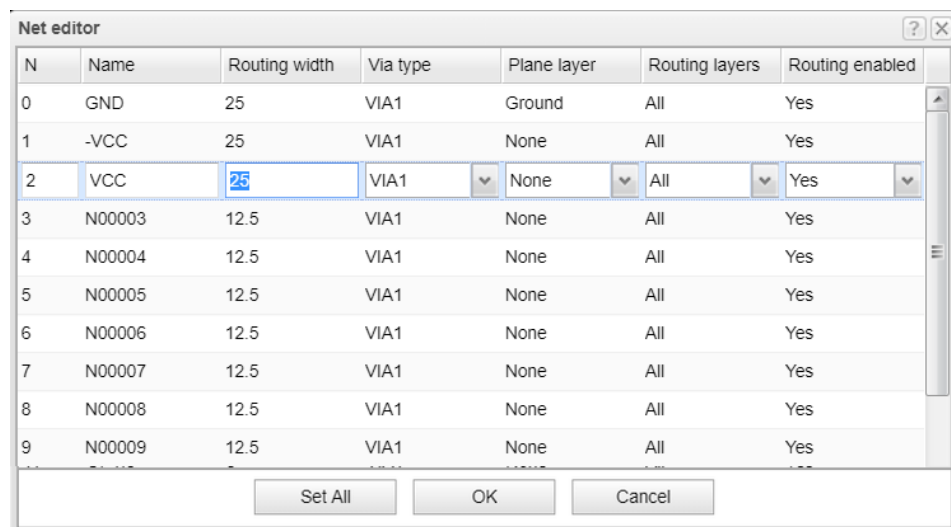
Now that everything is set properly press the OK button and the PCB layout design will appear with all the components automatically placed on the PCB board.



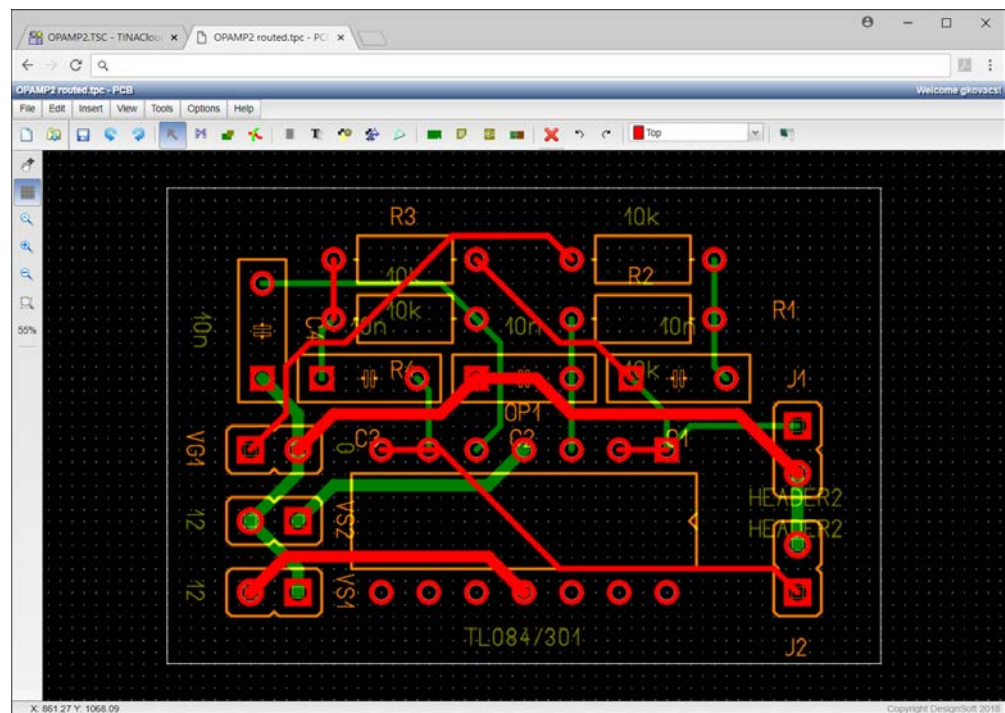
While all the parts and nets are placed, we need to adjust their positions for good placement and easier routing. Click and drag the parts to the position as shown on the figure below. (Find “opamp2 placed.tpc” to check your results.)



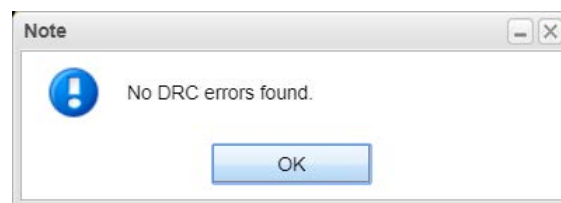
Click on Tools main menu item to reach the Net Editor and set net routing width. First, enter 12.5 into a “Track width” field then press “Set All”. Then select power nets (Ground, VCC, -VCC) and set their track width to 25mil.



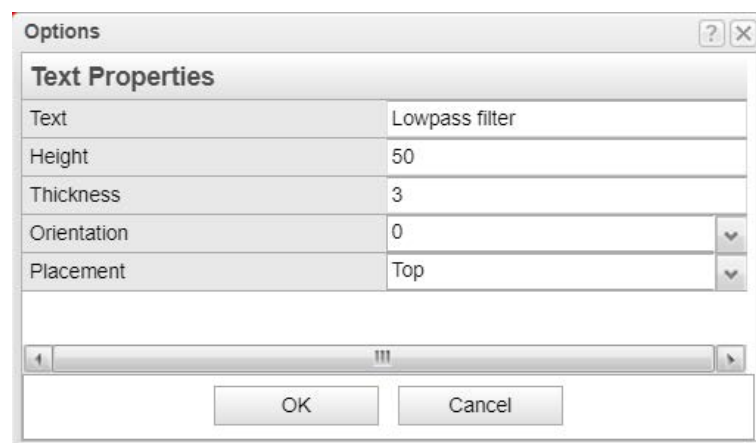
To automatically route the PCB, press select “Autoroute board” command from the Tools menu. The following screen will appear:



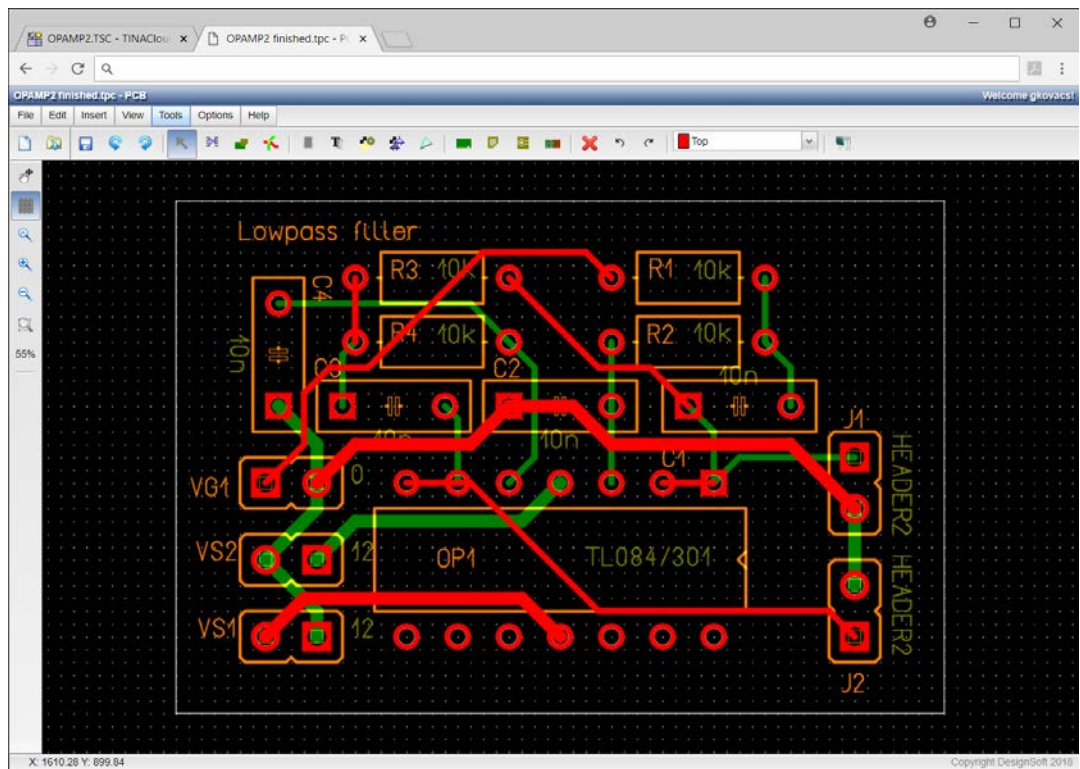
To see if everything is routed correctly select DRC (Design Rule Check)/Run DRC from the Tools menu. The following message will appear:



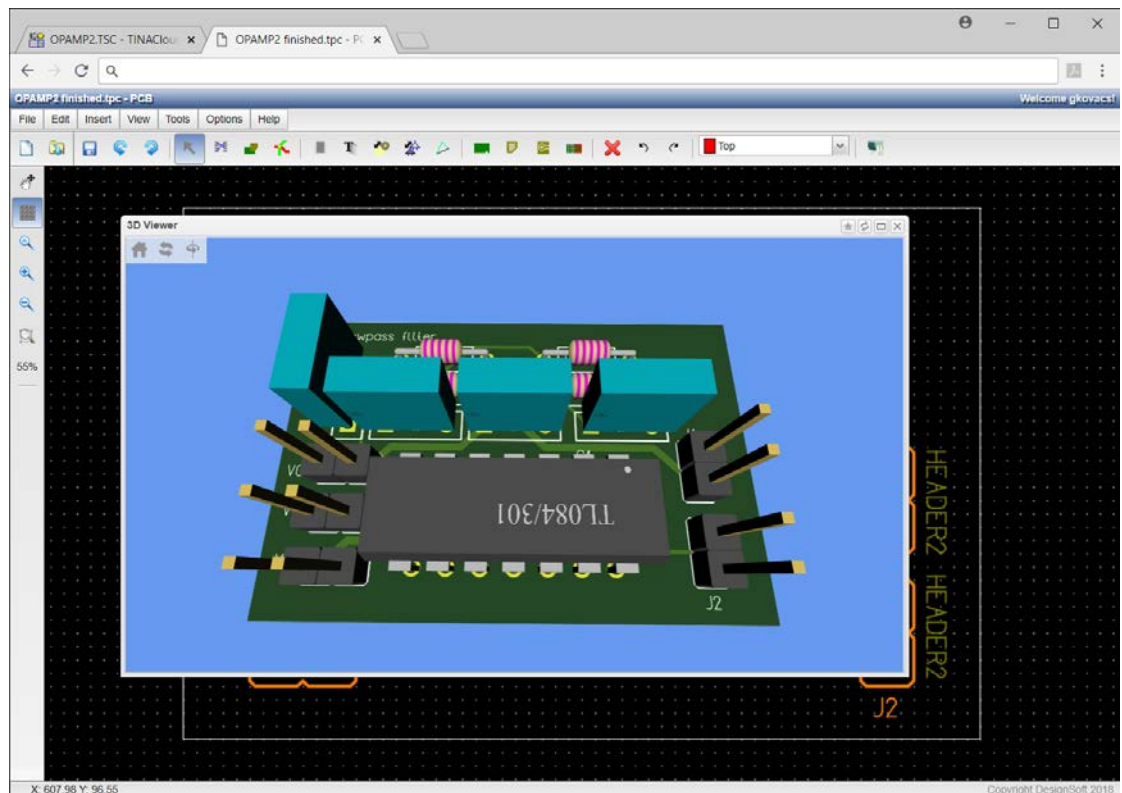
To finish our first simple design, let's add text to the silkscreen/assembly layer. Click the Text button on the toolbar and make the settings in the dialog that appears:



Enter the text into the empty upper field and press the OK button. The text will be attached to the cursor. Move it to the location shown on the picture below.



Finally, you can check your design in full 3D. Press the 3D View button on the toolbar's right end or select "3D View" from the View menu. The following window will appear.



You can rotate the 3D model to any direction by holding the left mouse button and moving. Hold down the middle button and move the mouse to shift the viewpoint. You can also move the "camera" forward or backward to set a large view showing the complete PCB or to zoom in for more details. Use the mouse wheel for this operation.

At this point, you will probably want to create Gerber files for a manufacturer.

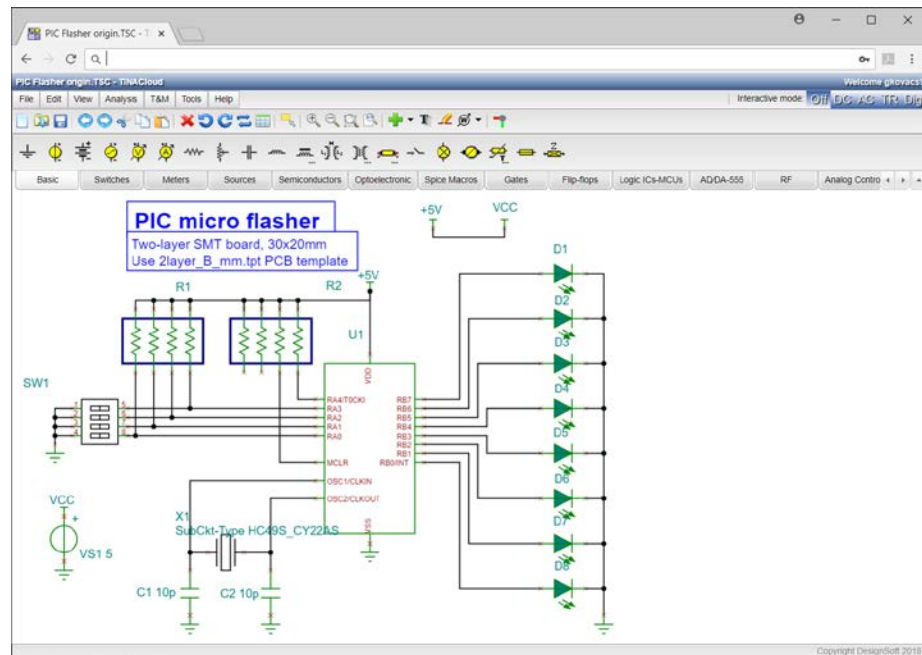
To obtain Gerber (RS - 274 X format) files to direct a photoplotter, choose ExportGerber file from the File menu. (Various Gerber options can be changed using the dialog for Gerber output settings under the Options menu.)

1.3 Advanced editing functions of TINACloud PCB's Layout editor

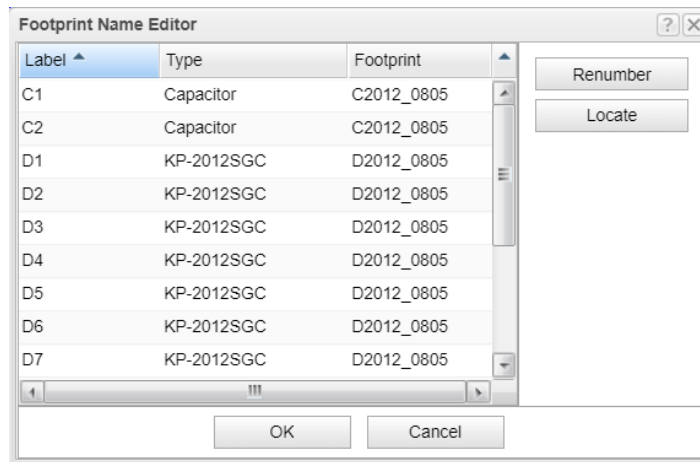
In practice, you may need to do some editing using TINACloud PCB's advanced editing features. This is described through examples in the next section.


CREATING A TWO-LAYER, DOUBLE-SIDED, SURFACE-MOUNT TECHNOLOGY BOARD

To get into TINACloud in more detail, open the second example, the file PIC flasher origin.TSC project from TINACloud's Examples/PCB folder.



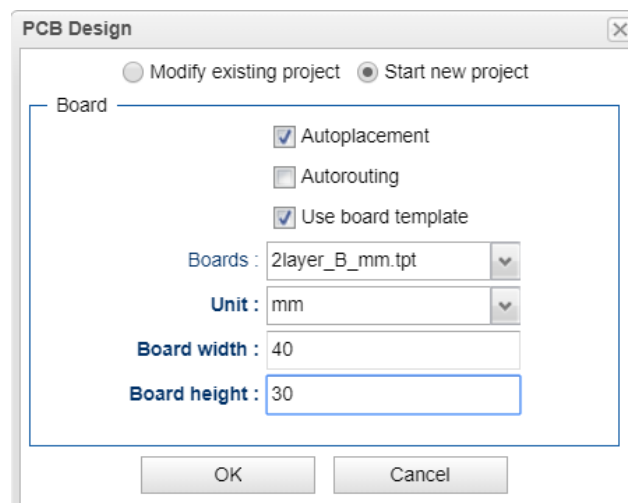
The schematic is PCB-ready, every part has a surface mount device (SMD) physical representation. Now, before we start the PCB wizard, click Tools/Footprint Name Editor... to check the footprints list.



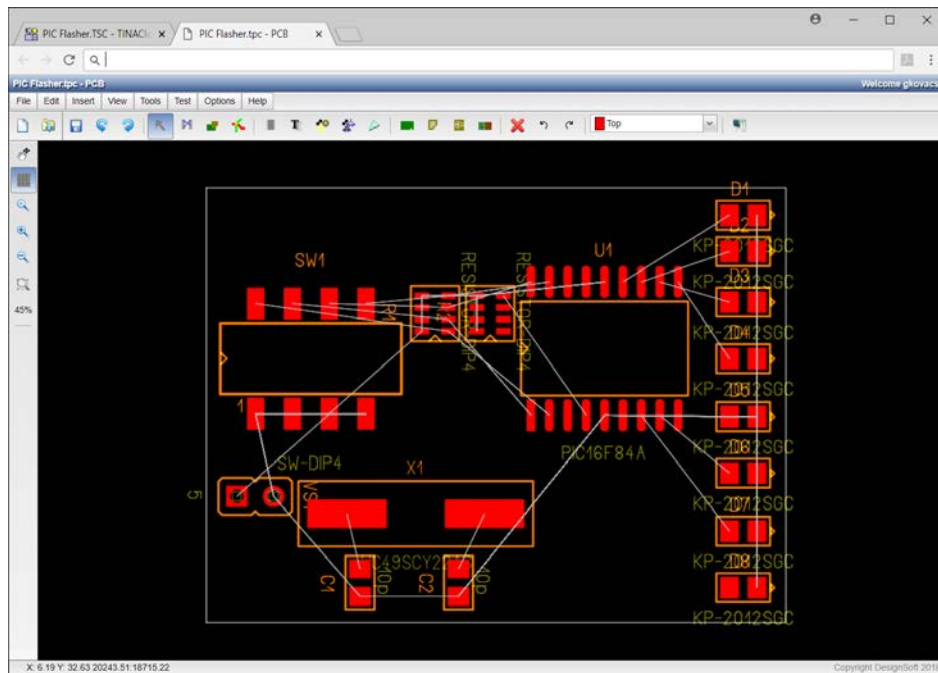
Since all the components appear to have a valid footprint name, we can start using the Tools/PCB Wizard. Click on “PCB Design...” in the Tools menu or on the  button on TINACloud’s toolbar.

Set the “Start new project,” check “Autoplacement” and “Use board template.” Select the template file “2layer_B_mm.tpt.”

Review actual physical parts, if possible. Be sure to allow for the area of all the components, mounting holes, and keep-away zones and make your best estimate of the values for Board width and Board height. Moreover, it is important to provide enough space between the components to allow for the placement of vias and tracks during routing. Enter 40mm length and 30mm width.



Press the OK button, the PCB Editor window will appear.




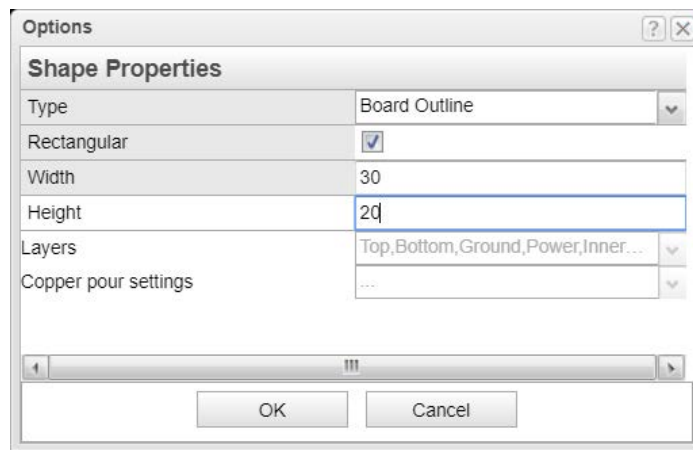
The components are placed in the close proximity to minimize the connection lengths, in a topology similar to that of the schematic, while still respecting the design rule settings. However convenient the result may be for autorouting, the designer usually has to make adjustments to the component layout to satisfy electrical, mechanical and other characteristics. Some of the considerations are—

- the ohmic effect of a long and/or thin power trace the length of a track from the signal source to the load in high-speed digital systems introduces reflections
- in analog situations, poor placement can lead to increased noise coupling
- allowance for automated parts placement clearance
- future serviceability of the PCB
- aesthetic values

These considerations influence the components' position and could be critical—not only in complex designs—but even in the simplest ones. For these reasons, one must still adjust parts placement manually.

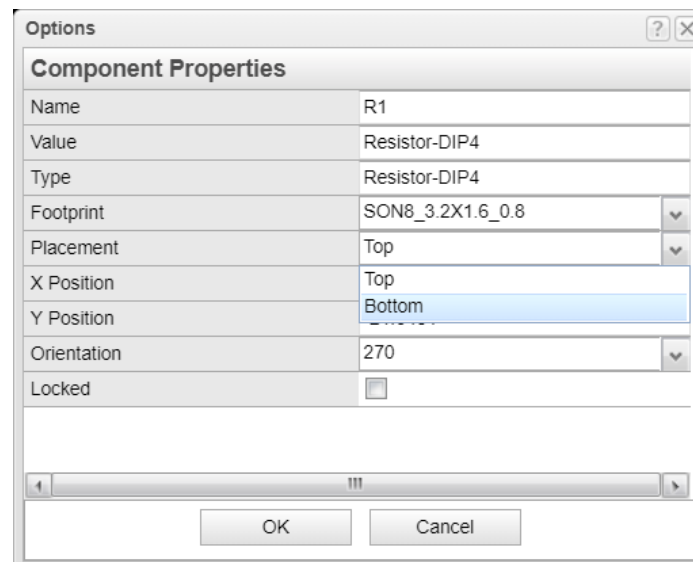
Our circuit, although it is small and not very dense, has a few special requirements, namely to put the crystal closer to the microcontroller, to position the power supply connector, and to adjust the LEDs along the board.

If you want to change the board size, click the “Draw/modify shapes” button . Let's try making the board smaller. You can double click on the middle of the board and enter the following values into the fields:

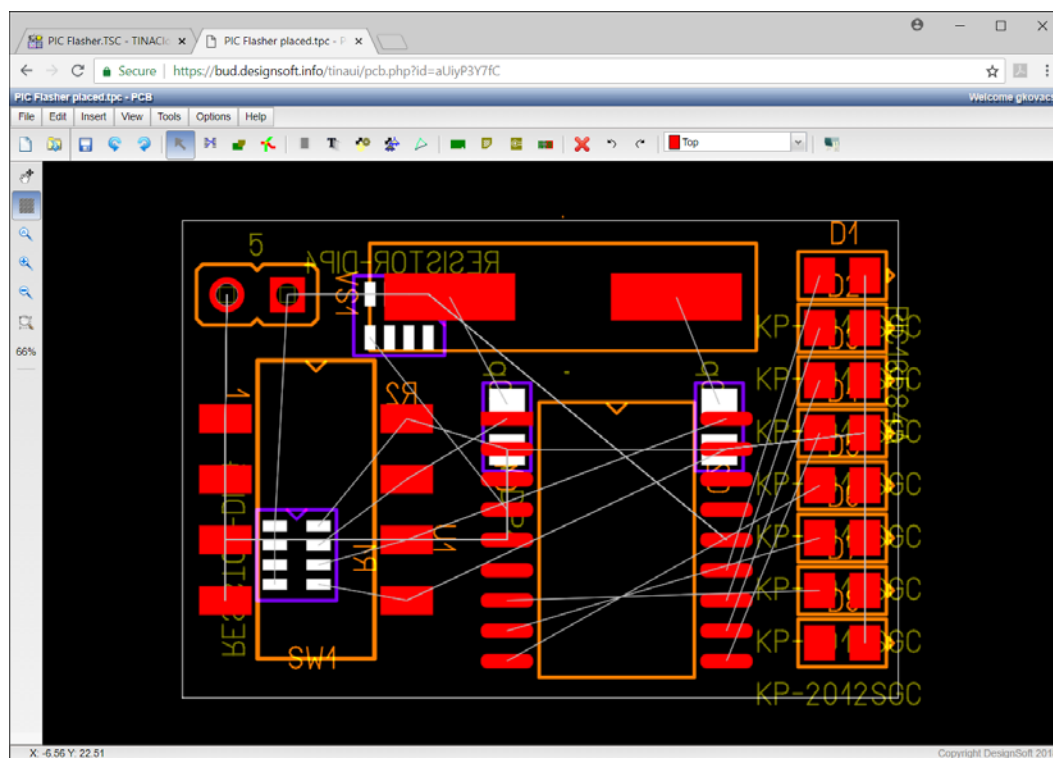



When you press OK, the board outline will shrink.

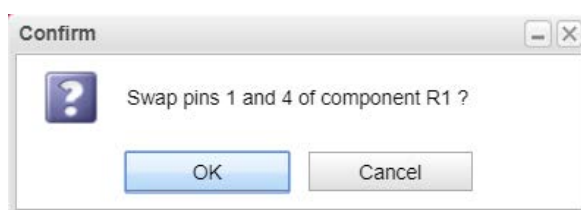
Now, before we begin routing, let's set the position of the components; place the power connector and the DIP switch on the left hand side and the LED bar on the right hand side. Place the capacitors and resistor networks on the bottom side by double clicking on the components and choosing Bottom Placement side on the dialog.



Compare your result to the file \Examples\PCB\PIC flasher placed.tpc.



In order to decrease route length, swap R1 pins connected to SW1. Pin swapping is allowable if identically functioning pins are to be exchanged, such as pins of resistors, capacitors, etc. Click on the Pinswap button  on the toolbar to pick up the tool, click on pad 1 of R1 (the upper right), and—finally—pad 4 of R1. The following window should come up:

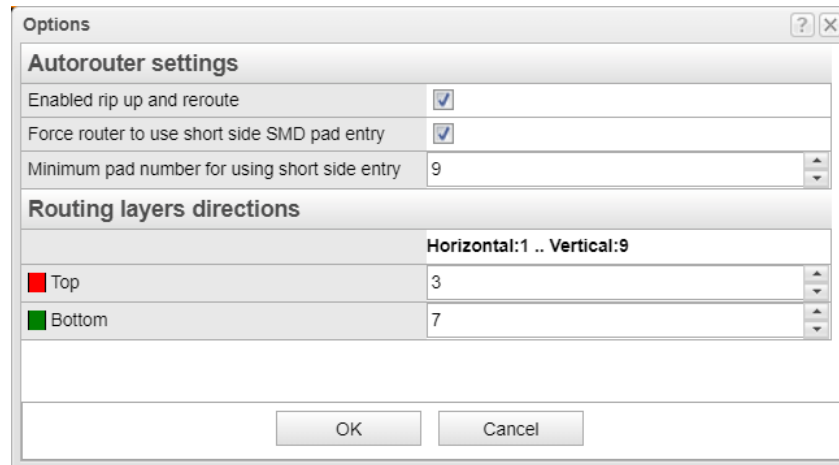


Press the Yes button, then do the same with R1 pad 2,3 and R2 pad 3,4.

Note that a pin swap changes the original connections, so we must update the original schematic later to maintain the correspondence between the schematic and the board taking into account the changes we made to the board while using PCB Designer.

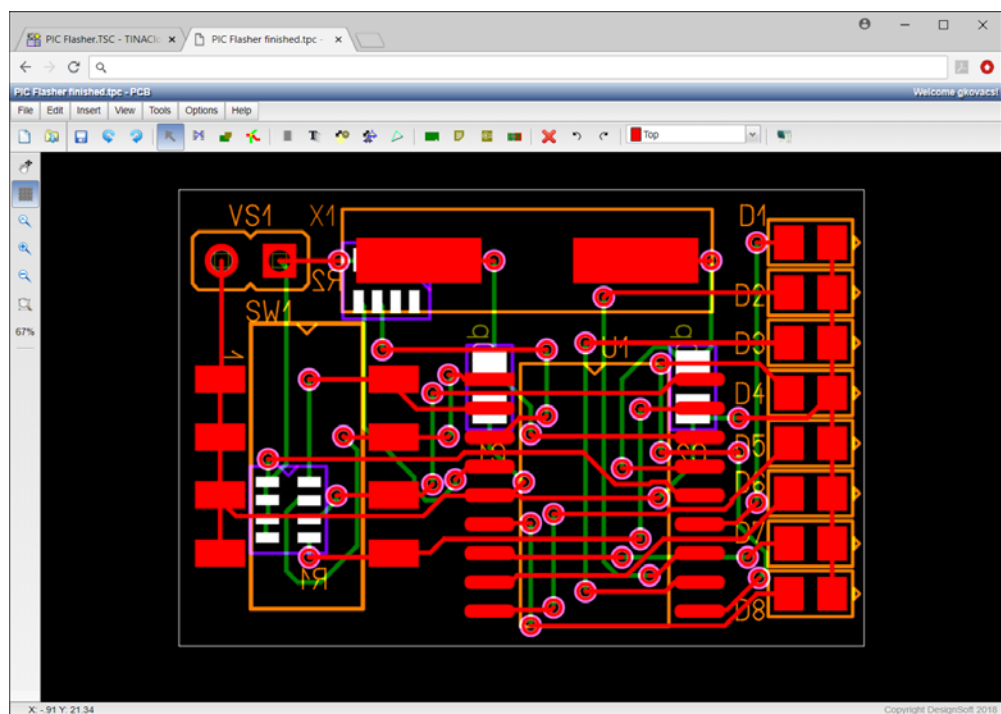
This process, called back annotation, has to be completed whenever a pin/gate swap and/or component renaming have been performed.

Signal planes (in our present example, the top and bottom layers) generally are given preferred trace directions up and down in one layer and left and right in the other layer. To set these trace directions, click on Options/Autorouter settings. The Autorouter settings are set to our preferred direction by choosing an integer number from 1 to 9. A value of 1 forces the router to heavily emphasize horizontal lines, while a value of 9 forces it to use vertical lines. 5 tells the router not to care about horizontal or vertical preference. Choosing the extreme values (1 and 9 for a pair) is usually too strict, so we choose to enter 3 for the top layer and 7 for the bottom.



Also on this panel, check “Force router to use short side SMD pad entry” with pad number 9 constraining the router to connect SMD pads on their short side if the component has at least 9 pins. This option can be very used to preserve the gaps between SMD pads for track routing. Because of these choices, R1, R2 and SW1 are allowed to route freely, as half of the pins are connected together, while U1 pads connect on the short side.

After these moves, click on the main menu Tools/Autoroute board to route the PCB automatically. First the power and ground nets are connected, then signals are routed. If necessary, the program will rip-up tracks and reroute the unconnected nets. The following result will appear:

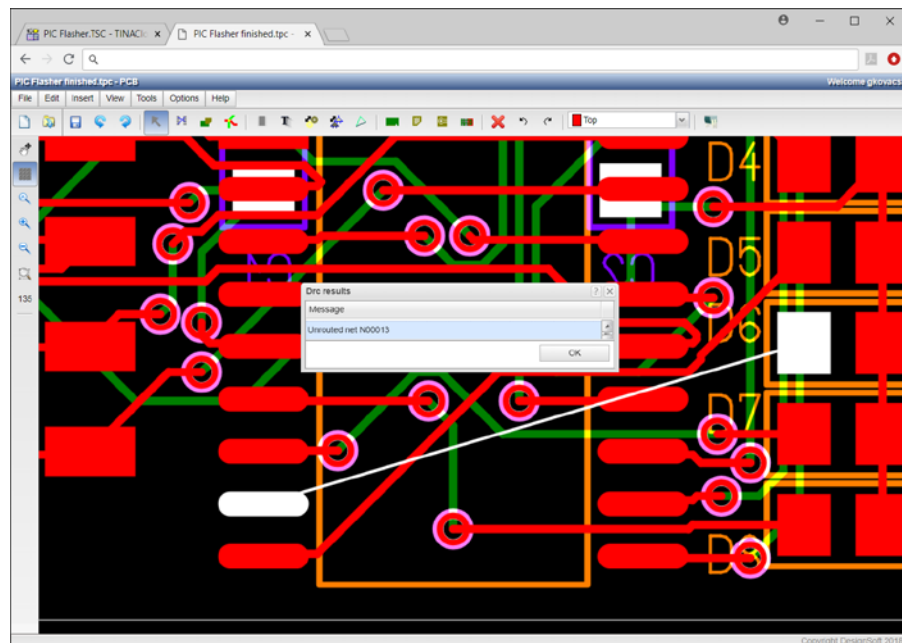


In some cases you might see that there is a net left unconnected. Fortunately, the DRC utility will examine your design and reveal any unconnected traces. Run DRC by clicking on the menu Tools/DRC/Run DRC.

The DRC Results window will appear, highlighting the corresponding nets.




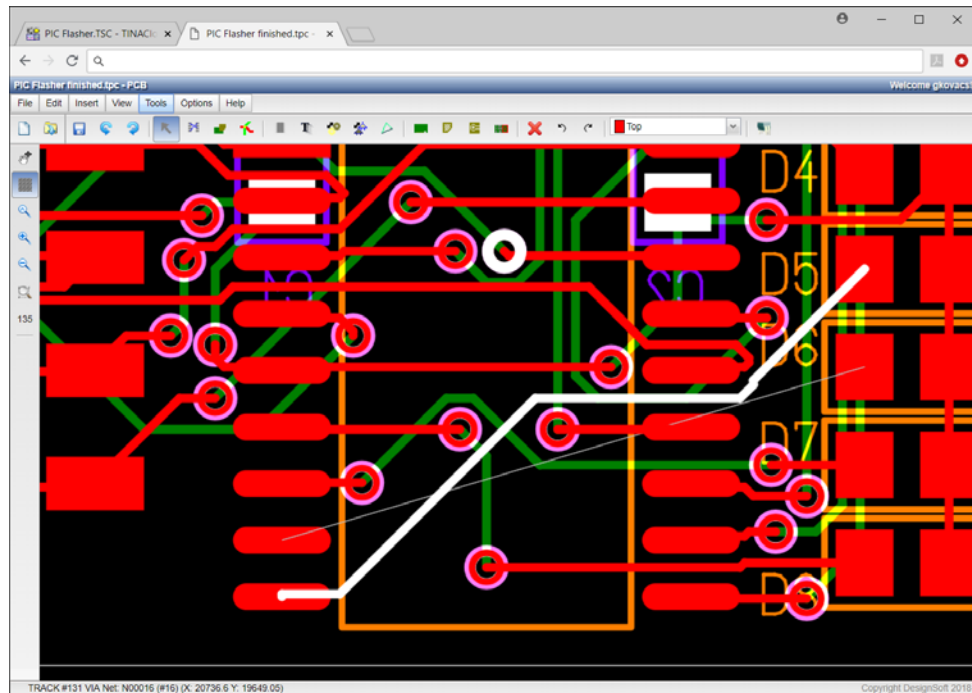
You should browse and correct any errors DRC reports. If you double click on the text 'Unrouted net N00013,' the program magnifies and highlights the selected net while centering the position of the selected objects.



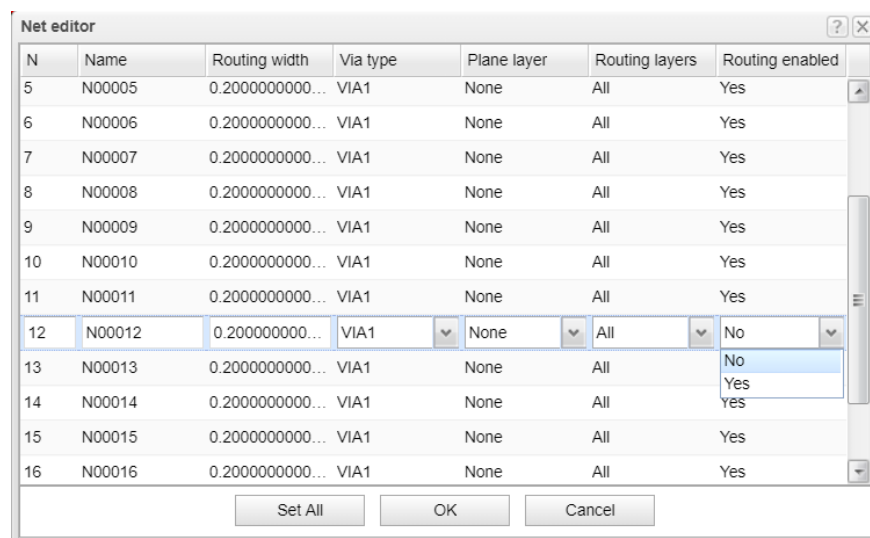
In order to fully route the board, we can use the manual route modes to make space for the unconnected tracks or delete crossing tracks and reroute them in a different order. Working in manual route, you can direct routing wherever necessary. You can move existing segments of tracks, remove segments, or create new segments.

Proceeding step by step, we will use autorouting to route the last trace.

First, press the , the select button on the toolbar, then hold down the Shift key and click the track that connects U1 pad 9 to D5 (see picture below), then press the Delete key to remove the track. This makes room for the unconnected net.

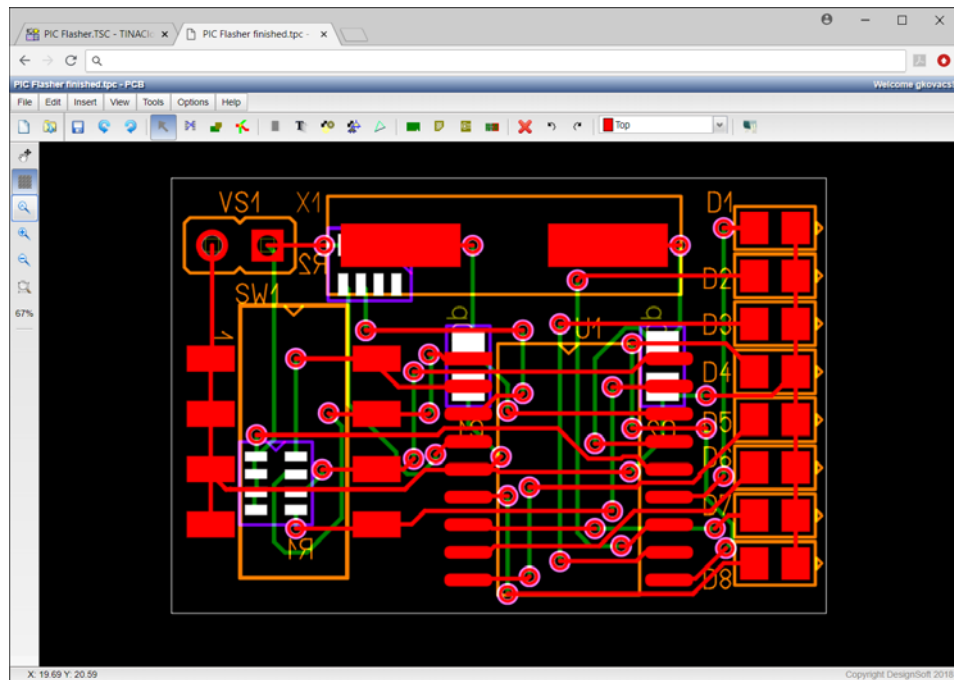


Now click on the menu Tools/Net editor to invoke the net editor. Disable N00012, the previously removed connection. This will prevent the autorouter from reconnecting the net. Leave N00013 routing enabled.



Click OK and the menu Tools/Continue autorouting and the autorouter will connect all the enabled nets. Next, enable N00012 in the Net editor, checking Routing Enabled, clicking OK and click again on Continue autorouting menu to autoroute the last net.

Let's see if our manual intervention has worked. Call again Run DRC to verify that there are no errors.



Now, let's synchronize the PCB and schematic files. First, choose TinaCloud backannotation from the File menu to save the changes back to the schematic Editor. In the other browser window TINACloud will read and update the original schematic.

Take a close look at the resistor networks—the pin order reflects the result of the pinswap.