

# PCB Creator Tutorial

© 2016 Bay Area Circuits

## Introduction

This tutorial is recommended for absolute beginners with PCB Creator and as a useful guidebook for the dive into the circuit board design. If you already work with PCB Creator and want a quick answer, please refer to the corresponding Help document ("Help \ ..... Help" from the main menu) or search keywords through this tutorial.

This tutorial allows the user to get started with PCB Creator by designing a simple schematic and its PCB, we will also design some pattern and component libraries in order to gain the basic knowledge about different package features. The PDF book includes a step-by-step design guide and many additional insets. Strictly follow the given instructions and create a simple board at the end of the first paragraph, all the way from schematics to Gerber files.

# Table of Contents

<b>Part 1: Drawing the schematic</b>	<b>5</b>
1 Schematic UI.....	6
Schematic Main Window .....	6
Custom Keyboard Hotkeys .....	7
2 Establishing schematic size and placing titles.....	8
3 Configuring libraries.....	11
4 Designing a schematic.....	12
5 Converting to a PCB.....	30
<b>Part 2: Designing a PCB</b>	<b>31</b>
1 PCB Layout UI.....	31
2 Preparing to route.....	33
3 Autorouting.....	42
4 Working with layers.....	44
5 Working with vias.....	48
6 Net Classes.....	50
7 Manual Routing.....	53
8 Measuring trace length.....	61
9 Selecting objects by type / layer.....	62
10 Placing Text and Graphics.....	65
11 Copper Pour.....	68
12 Locking objects.....	76
13 Design Verification.....	78
14 Design Information.....	82
15 Panelizing.....	83
16 Printing .....	87
<b>Part 3: Board Manufacturing</b>	<b>89</b>
1 Order PCB.....	89
2 Upgrade to DipTrace.....	90
<b>Part 4: Creating Component Libraries</b>	<b>90</b>
1 Designing a pattern library.....	91
Customizing Pattern Editor .....	91
Create/Save library .....	92
Designing a Resistor (pattern) .....	93
Attaching a 3D Model .....	102
Designing BGA-144/13x13 .....	104

Real component Design. SOIC-28 pattern .....	111
<b>2 Designing a component library.....</b>	<b>114</b>
Customizing the Component Editor .....	114
Designing a Resistor (component) .....	116
Designing a Capacitor .....	120
Designing VCC and GND symbols .....	127
Designing a multi-part component .....	130
Using additional fields .....	139
Designing PIC18F24K20 .....	141
Spice settings .....	150
Library Verification .....	152
Placing parts .....	153

## Part 5: Other features and tools

**157**

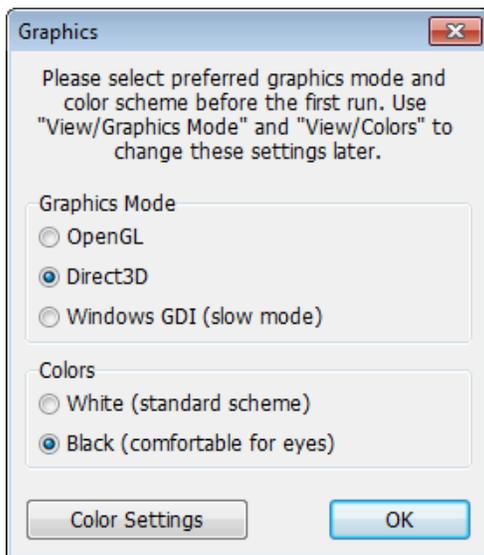
<b>1 Connecting.....</b>	<b>158</b>
Working with Buses and Bus Connectors .....	158
Working with Net Ports .....	164
Connecting without wires .....	165
Connection Manager in Schematic and PCB Layout .....	169
<b>2 High-Speed differential connections.....</b>	<b>170</b>
Create a differential pair .....	170
Differential pair routing / editing .....	174
Verification and phase tuning .....	180
Differential Pair Manager .....	184
Define Paired Pads .....	185
<b>3 Reference Designators.....</b>	<b>188</b>
<b>4 Back Annotate.....</b>	<b>190</b>
<b>5 How to find components in the libraries.....</b>	<b>193</b>
<b>6 Electrical Rule Check.....</b>	<b>195</b>
<b>7 Bill of Materials (BOM).....</b>	<b>197</b>
<b>8 Importing/Exporting netlists.....</b>	<b>199</b>
<b>9 Saving/Loading Design Rules.....</b>	<b>201</b>
<b>10 Spice simulation.....</b>	<b>202</b>
<b>11 Checking net connectivity.....</b>	<b>206</b>
<b>12 Placement and Autorouting.....</b>	<b>209</b>
<b>13 Fanout .....</b>	<b>219</b>
<b>14 Hierarchical Schematic.....</b>	<b>223</b>
<b>15 3D Preview and Export.....</b>	<b>233</b>

## Part 1: Drawing the schematic

In this part of the tutorial you will learn how to create a simple schematic and PCB (Printed Circuit Board) using the PCB Creator software.

Let's start from the schematic. Open PCB Creator Schematic capture, go to "Start \ All Programs \ PCB Creator \ Schematic" in Windows OS.

If this is the first time you launch PCB Creator Schematic, graphics mode and color scheme selection dialog box will pop up.



You can select the graphics mode that works better for you:

**1. Direct3D** is the fastest mode for typical Windows PC, we recommend you use this mode if it works correctly on your system. Direct3D suits machines without high-end graphics system and OpenGL hardware. However, it depends on hardware/drivers/versions, therefore a small percentage of computers (usually with new/very buggy, or very outdated OS/drivers) can have issues (artefacts on the screen or some objects disappearing).

**2. OpenGL** usually works a bit slower than Direct3D, but it is more universal, suitable for different operating systems and less dependent on hardware/drivers.

OpenGL is the best choice for high-end engineering/graphic stations with professional OpenGL equipment. However, you can try both modes on complex projects and choose the best for you.

**3. Windows GDI** can be used as an alternate mode if both Direct3D and OpenGL failed to work properly with your graphics card. It is much slower, but doesn't depend on drivers/hardware/OS. However, this mode is enough for comfortable work on small and medium-sized projects.

We will use a white background as more acceptable for printing this tutorial, but you can select the color scheme you want (classic black background is the most eye-friendly). Notice that you can change the color scheme or define colors any time in the "View \ Colors" main menu item.

Graphics dialog box will appear after the PCB Layout module has been launched for a first time. Component Editor and Pattern Editor use the color settings of Schematic Capture and PCB Layout respectively.

*Notice that relative sizes of program panels in the screenshots may differ from what you see on the screen due to resolution limitations applied in this PDF tutorial.*

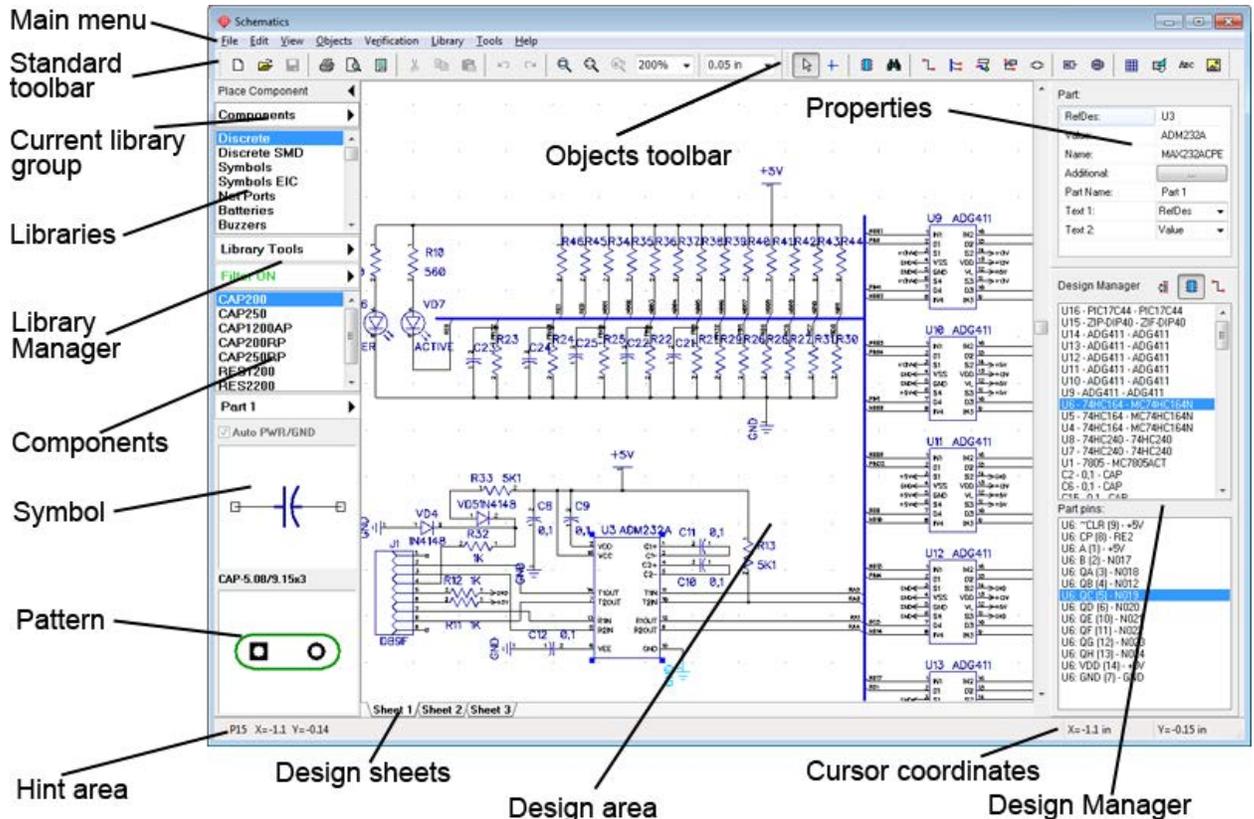
Sometimes we will hide the Design Manager (which is on the right side of the screen) to add more design space, but if you have high resolution screen you don't have to do this. Select "View \ Toolbars \ Design Manager" from main menu to show / hide the Design Manager panel or press "Ctrl+2" default hotkeys.

Some schematic and PCB examples presented in this tutorial are designed exclusively as a demonstration cases of the tools being used, not as working prototype circuit boards.

## 1.1 Schematic UI

### 1.1.1 Schematic Main Window

Schematic main window includes: schematic design area, main menu, toolbars, properties panel, design manager, component placement panel (Library Manager), and the status bar.



**Design area** – create and edit schematic objects (parts, wires, buses, shapes, tables etc.). Access to all common functions of the program via the main menu.

**Standard toolbar** – tools to work with files, cut/copy/paste objects, print, preview and configure titles, change scale and grid size.

**Objects toolbar** – default mode, define origin, find and place components, create and edit wires and buses, place page connectors, define differential pairs, place hierarchy connectors and blocks, place shapes, text, and tables.

**Component placement panel (Library Manager)** – all active libraries, user libraries or only components of the current project. Select library from the list, find and place component. Setup the panel, search filters, multi-part component placement tools, symbol and pattern previews.

**Properties panel** – displays properties of the active tool or selected object/s.

**Design Manager** – navigates you through the schematic (left click highlights an object on the design area, double click pans the circuit to show selected component/net).

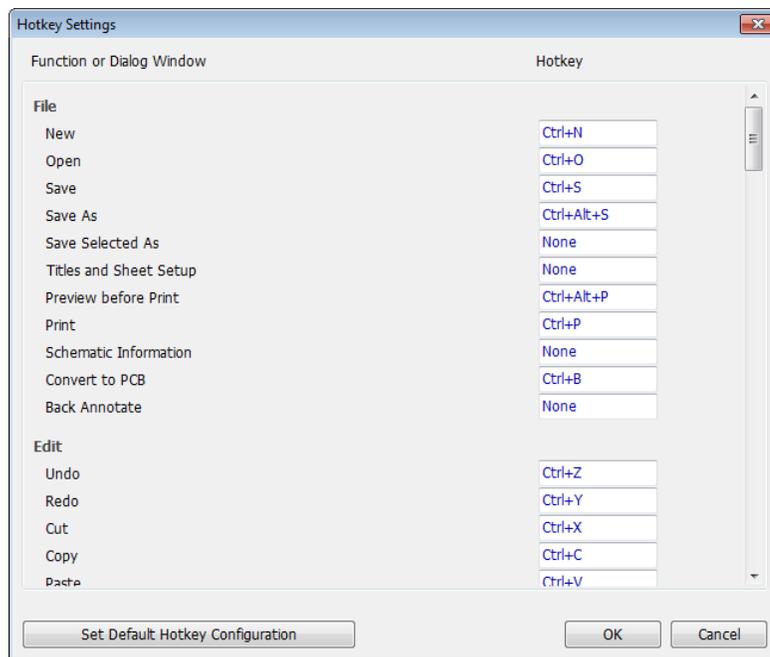
**Status Bar** – left side shows the current hint and the right side – cursor coordinates.

See PCB Creator Help for details ("Help \ Schematic Help" from main menu).

Please go to "View \ Toolbars \ ..." to hide / show any of the UI toolbars (or use "Ctrl+1 through 9" hotkeys).

### 1.1.2 Custom Keyboard Hotkeys

PCB Creator features customizable hotkeys in all software modules. This makes switching to PCB Creator especially convenient and fast, since you can assign keyboard shortcuts that you've used to work with. Go to "Tools \ Hotkey Settings" from the main menu in Schematic, however the same main menu item leads to Hotkey Settings dialog boxes in any of the PCB Creator modules: Schematic, PCB Layout, Component and Pattern Editors.



In our case we have Schematic hotkeys opened, but described here can be applied to customizing shortcuts anywhere in PCB Creator. In the Hotkey Settings dialog box you can assign custom keyboard shortcuts to most PCB Creator tools, program actions, and dialog boxes.

Let's assign another keyboard combination for creating a new file. Left click in the corresponding field and then press a new key or a combination. If the combination is already assigned to another tool, you can either proceed with a new shortcut, or leave the old one.

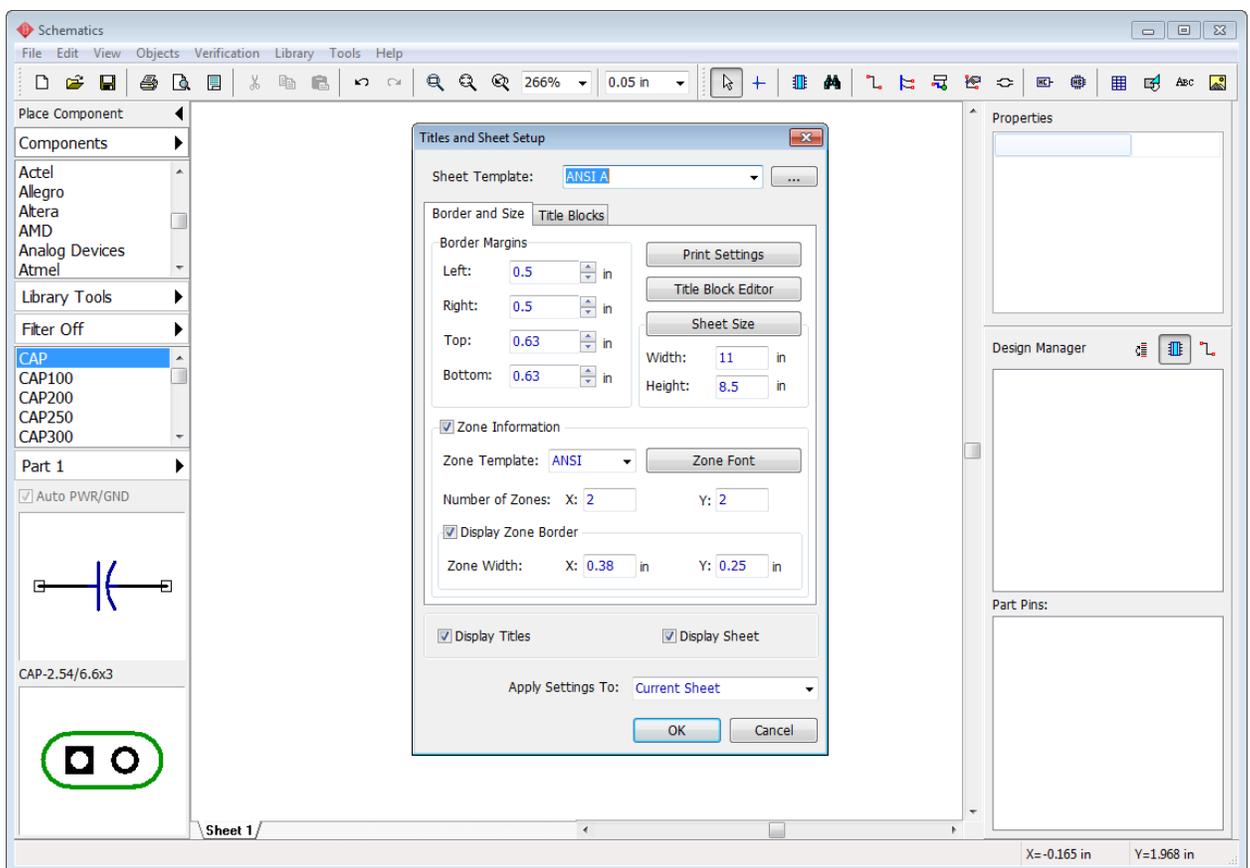
**Notice that all hotkeys mentioned in all topics of this tutorial are default ones.**

Press Set Default Hotkey Configuration button to restore default values.

*Notice that certain keys and combinations are reserved by the system. Please refer to the "Hotkeys" topic in the Schematic Help document ("Help \ Schematic Help" from the main menu) for complete list of reserved key combinations in PCB Creator Schematic. Check out corresponding topics in PCB Layout Help, Component Editor Help, and Pattern Editor Help to get more information about hotkeys in the respective programs.*

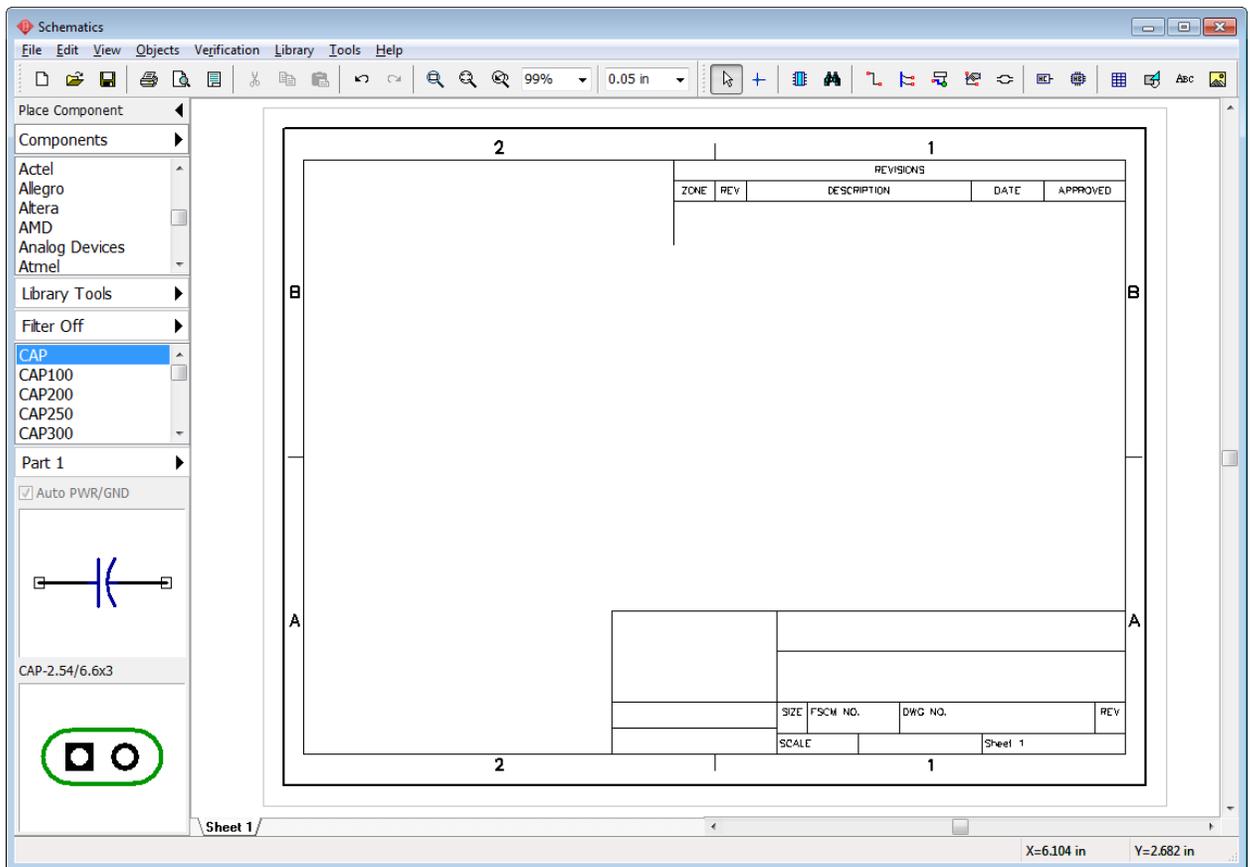
## 1.2 Establishing schematic size and placing titles

Establish schematic size and place the drawing frame, go to "File \ Titles and Sheet Setup", select "ANSI A" in the Sheet Template drop-down box. Then go to the bottom of the dialog and check Display Titles and Display Sheet items. Press OK.

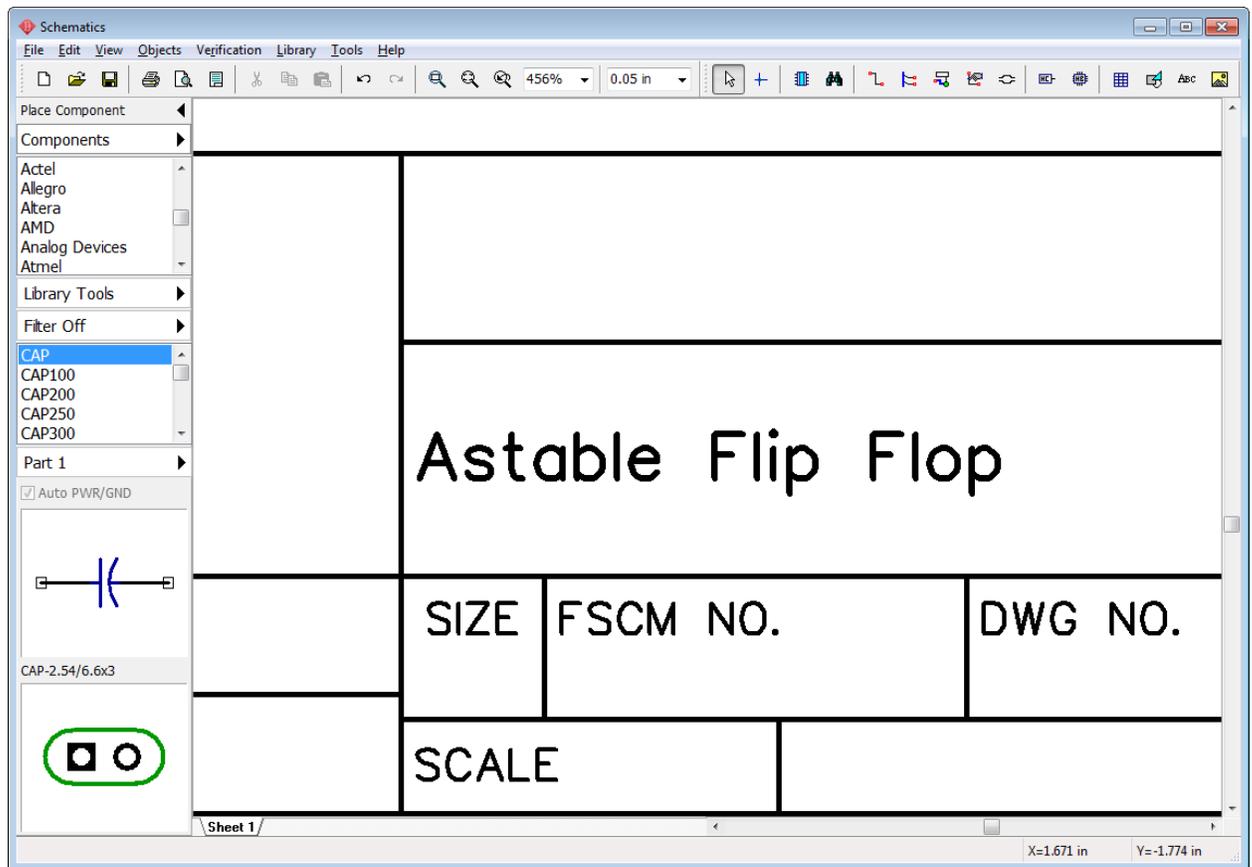


*Notice that you can show or hide titles and sheet frame on the design area by selecting "View \ Display Titles" and "View \ Display Sheet" from the main menu.*

Now press the minus sign (-) keyboard hotkey to zoom out until drawing frame can be seen. Plus and minus signs, the mouse wheel, and the scale box on the instruments toolbar allow for zooming on the schematic. Hover your cursor over a component or selected area for more precise zooming. Notice that we have hidden the design manager panel to the right ("Ctrl+2" hotkeys) to get more space on the design area.



To enter text into the title field simply hover the mouse pointer over that field (it should be highlighted in green), left click it to open the field properties pop-up dialog box, select or type in the text (field content), define alignment (Left, Center or Right) and font. In our case type in "Astable Flip Flop" then press the Font button and set font size to 12. Then click OK to close the dialog box and apply changes. You can enter multi-line text into the title block fields. This text will be saved only for current project. If you need to create a custom title template with predefined texts, please refer to the Schematic Help ("Title Block Editor" topic).



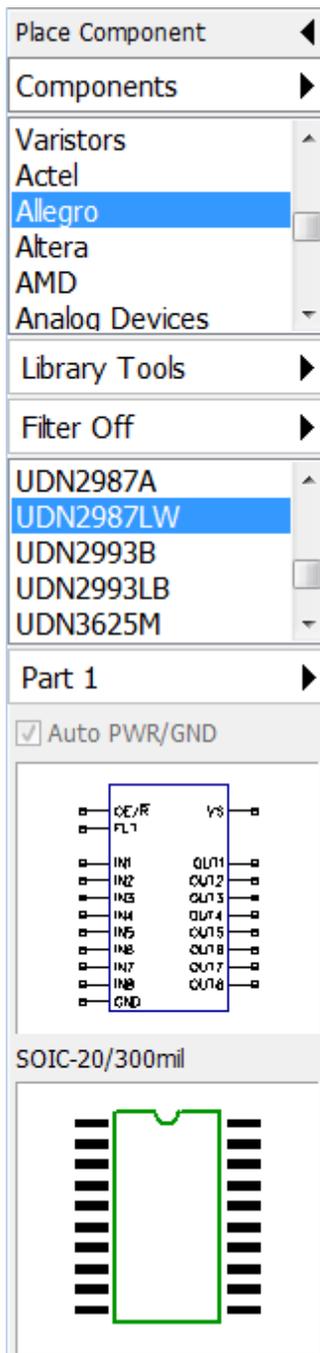
You may zoom on the title block by hovering over it and repeatedly pressing the "plus sign" button or scrolling the mouse wheel.

Let's practice with different zoom options for a while. Click on the  button (Zoom Window) and draw a rectangle on the design area where you want to zoom.

To return to previous scale and position, use  button (Undo Scale).

Go to "File \ Save As", type in a file name and make sure file is in the directory that you need. Click Save.

## 1.3 Configuring libraries



PCB Creator incorporates a single cross-module library management system. Libraries are organized to user-customized groups with multi-level search filters ensuring that correct components can be found quickly. The component placement panel (or Library Manager) is on the left side of the screen. It has all necessary tools for managing groups of libraries and placing single- and multi-part components. The list of libraries is in the upper part of the panel and the list of components of selected library is right below. Symbol and pattern previews are at the bottom.

Press Components (<Current Library Group>) item at the top of the component placement panel, there are three library groups by default:

- 1) Components (all standard libraries, sorted alphabetically by component type and manufacturer);
  - 2) User Components (add / delete libraries to / from this library group), empty by default;
  - 3) Project Libraries (auto-generated library with all components from the current circuit). It is empty if no schematic file is open.
- "Library Setup" – add / delete / copy / move libraries between the groups, add and edit library groups.

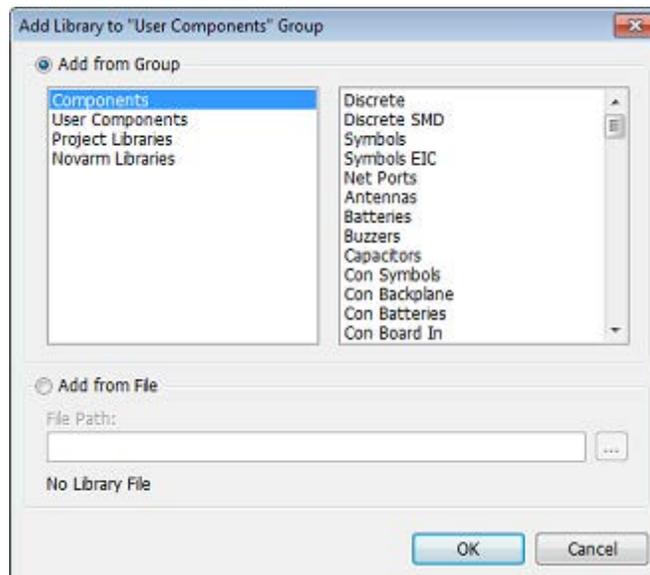
### Configuring Library Groups

Select User Components library group. As you can see it is blank at the moment. We assume that all components and their libraries needed for the project are known in advance, therefore we only need to add them to this library group. Press "Library Tools \ Add Library to "User Components". In the pop-up dialog box select "Components" library group from the Add from Group list.

This group contains all standard PCB Creator component libraries. Select Discrete, Symbols, and Fairchild libraries, use "Ctrl" button for multiple selection, then press OK.

Corresponding libraries appeared in User Components group and we are ready to start designing the schematic.

Go to "Library \ Library Setup" from the main menu or press "User Components (<Current library group>) \ Library Setup" on the component placement panel to get access to the comprehensive library system settings.



*Notice that Library Setup panel allows for configuring both pattern and component library groups, though pattern libraries are invisible in the PCB Creator Schematic.*

More information in the PCB Creator Help ("Help \ Schematic Help" from the main menu), Working with Libraries section.

## 1.4 Designing a schematic

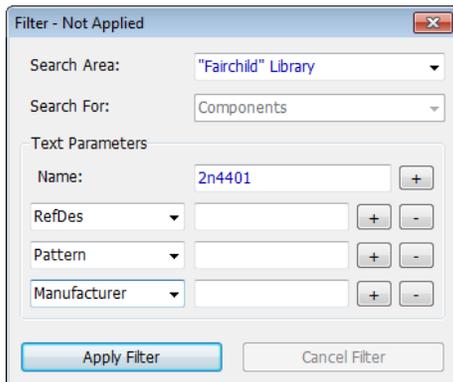
In this section of the tutorial we will show the basic principles of working in the Schematic module of PCB Creator PCB Design Environment.

Please turn ON the grid (if it was turned OFF) with "F11" hotkey. Change the grid size to 0.1 inch, you can select it from the list of grids  on the instruments toolbar, or press "Ctrl +" hotkeys to increase, or "Ctrl -" to decrease the grid size.

*Notice that you can add new sizes by selecting "View \ Customize Grid" from the main menu. Measurement units can be changed in the "View \ Units" main menu item. We note only default keyboard shortcuts in this tutorial. Go to "Tools \ HotKey Settings" to restore the defaults or change shortcuts.*

Now let's start creating the circuit. Select Fairchild library from the User Components library group on the component placement panel.

## Search component in libraries



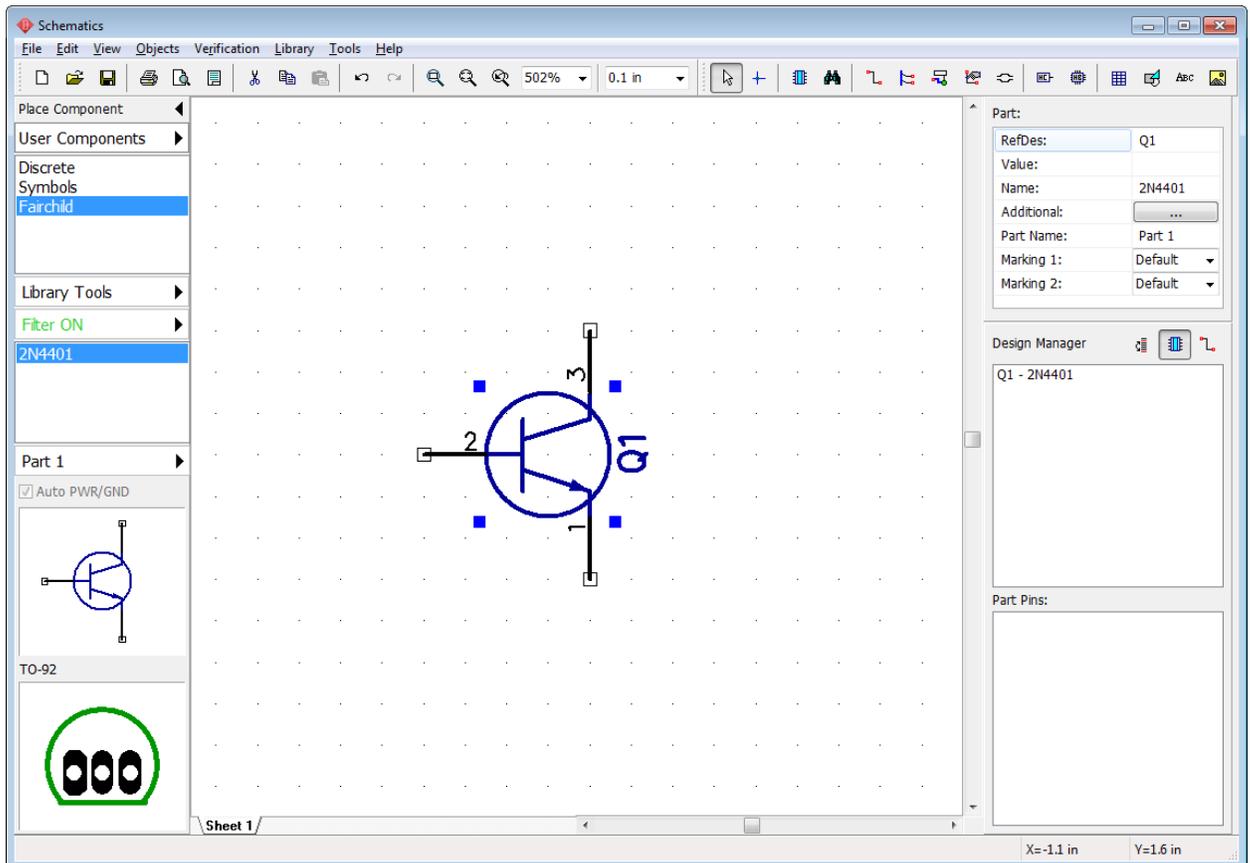
Once the library is selected, scroll down the component list to find 2N4401 transistor, or use search filters. Select "Objects \ Find Component" from the main menu or just press Filter OFF button on the component placement panel.

In the pop-up dialog box select Fairchild library as a search area, then type in "2n4401" into the Name field and press Apply Filter. You will see only the components you have searched in the component list and Filter ON button glowing green (all other components will be hidden). Close the search filter dialog box.

*Notice that you can enter a part of the component name as well as filter components by RefDes, value, pattern, manufacturer, datasheet, or additional fields. Use the "+" and "-" buttons to add or delete search filters.*

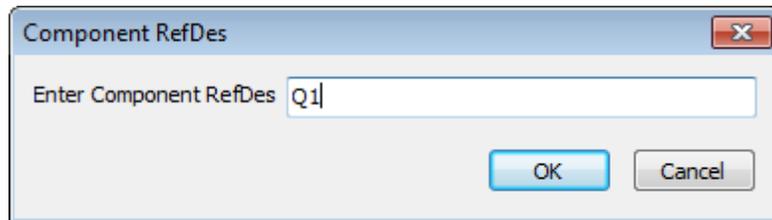
## Placing components

Click on the transistor in the list and move your mouse pointer to the design area. Left click once to place one transistor. Right click to disable component placement mode.

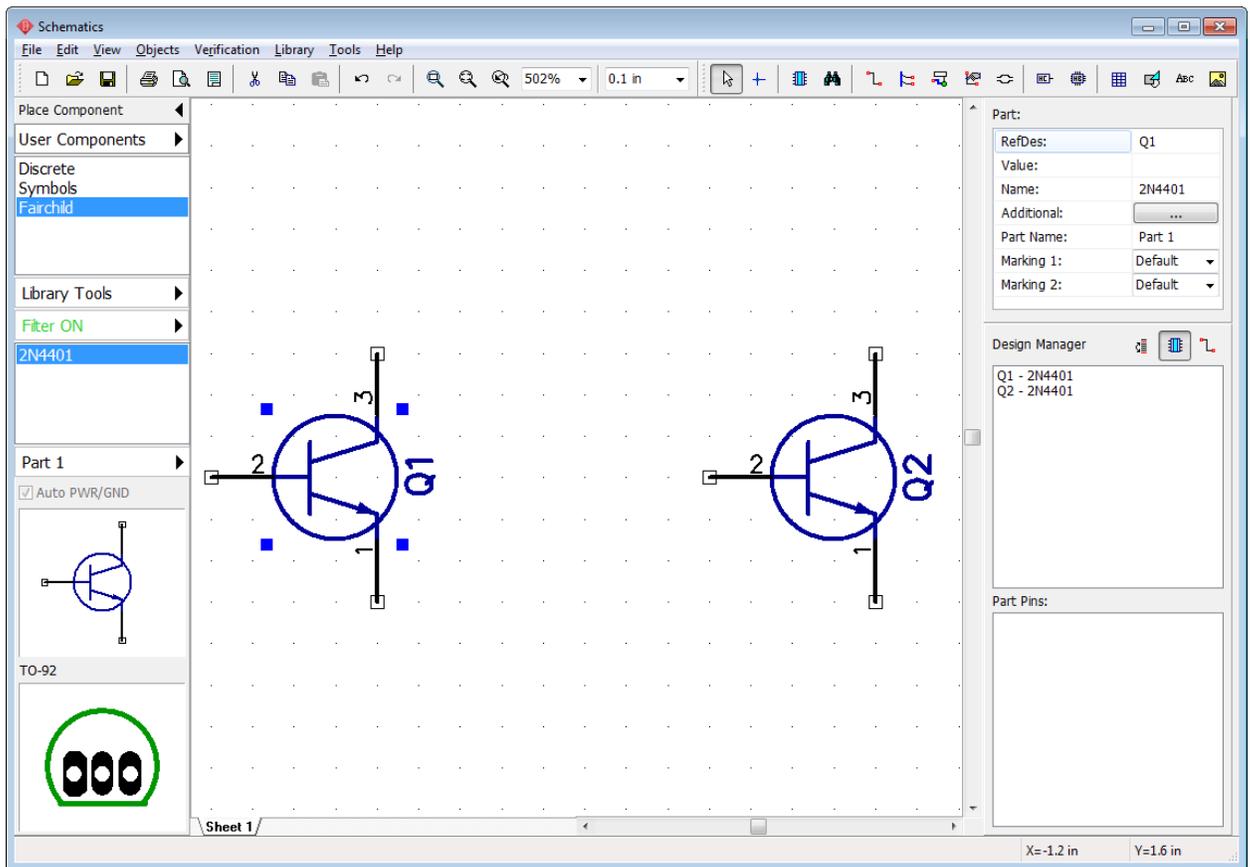


Drag and drop component if you need to move it to another location on the design area. To select several objects, press and hold "Ctrl" button then click on each object that you want to add to the selection or move mouse to the upper-left corner of the group, hold down the left mouse button and move cursor to the lower-right corner. Release the mouse button and objects inside the rectangle will be selected (if "Ctrl" key is pressed, selection will be inverted). Now you can move these objects at a time.

Sometimes it is necessary to change the reference designator of the component. Hover the mouse pointer over the component, right click it and select top item from the submenu. In the pop-up dialog box type in a new designator if needed. We will keep "Q1":



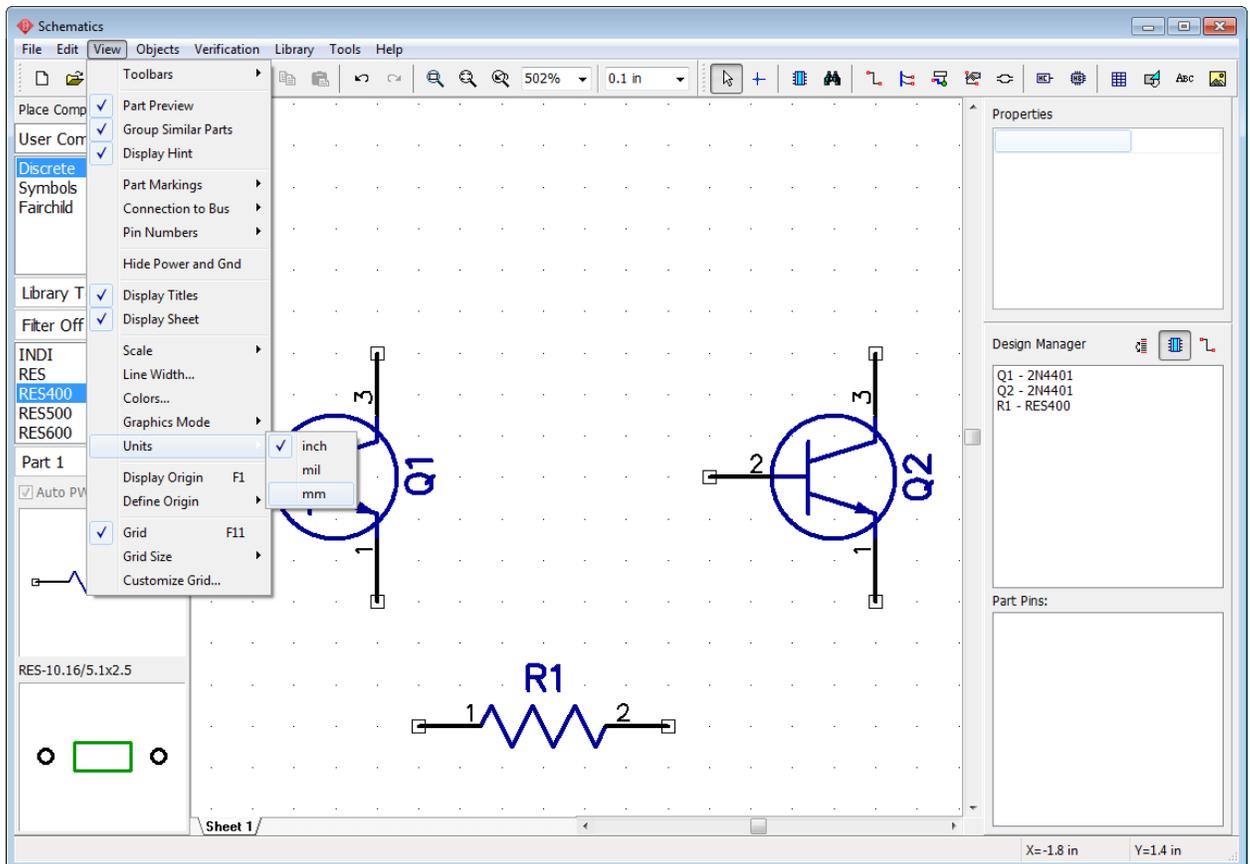
We need two transistors for the schematic, select 2N4401 in the component list again and place it to the design area. You don't need to rename the second transistor, it happens automatically. If you want to rotate the component before placing it on the design area, press "Space" or "R" default keyboard shortcuts.



When search filter is active you can see only certain (filtered) components of a library. Press

Filter ON button on the component placement panel and then press Cancel Filter in the pop-up dialog box to turn the filtering OFF. Close the search filters dialog box.

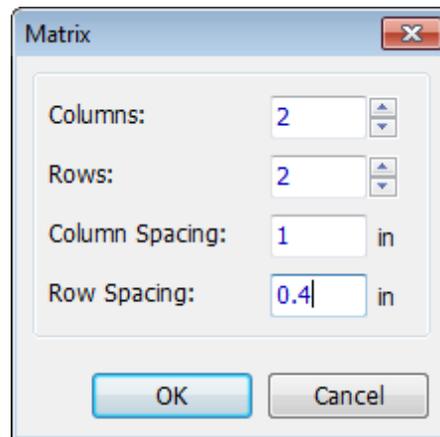
Select Discrete library, find RES400 resistor and place it on the design area. "400" in resistor's name stands for 400 mils of lead spacing. If you prefer metric units, select "View \ Units \ mm" from the main menu, however we will keep inches as these are the most suitable units for current project. We recommend to pay attention to active measurement units to avoid mistakes in the future.



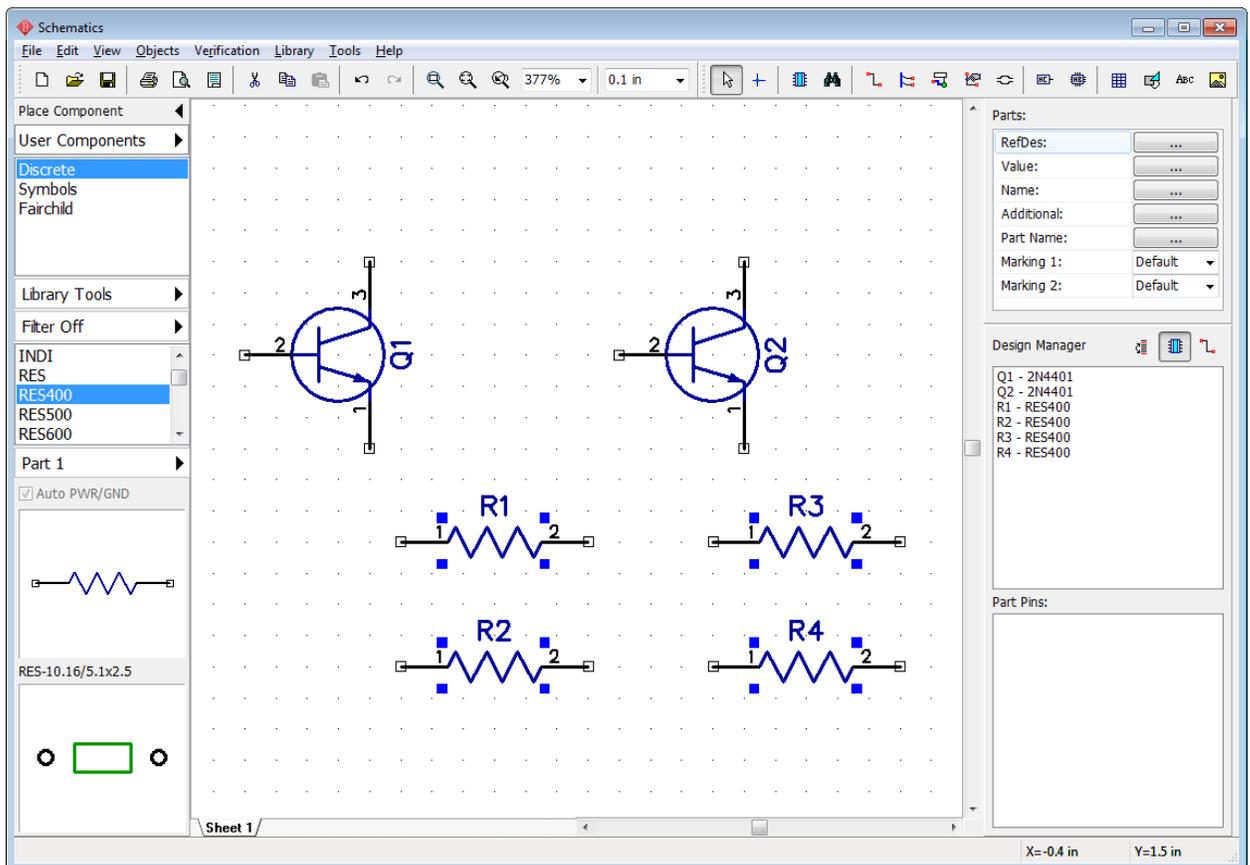
### Copy components

We need 4 resistors for this project. They can be placed manually like the Q1 and Q2 transistors, one by one. But this time we select the resistor on the design area and copy it 3 times. There are two ways to copy a component:

1. Once component is selected, go to "Edit \ Copy" from main menu or "right click on the component \ Copy", ("Ctrl+C" hotkeys can be used as well) then select "Edit \ Paste" 3 times, or right click on the design area and select Paste from the submenu.
2. Copy Matrix, good for bulk copying. Select resistor then go to "Edit \ Copy Matrix" from the main menu (or press "Ctrl+M" default hotkeys).



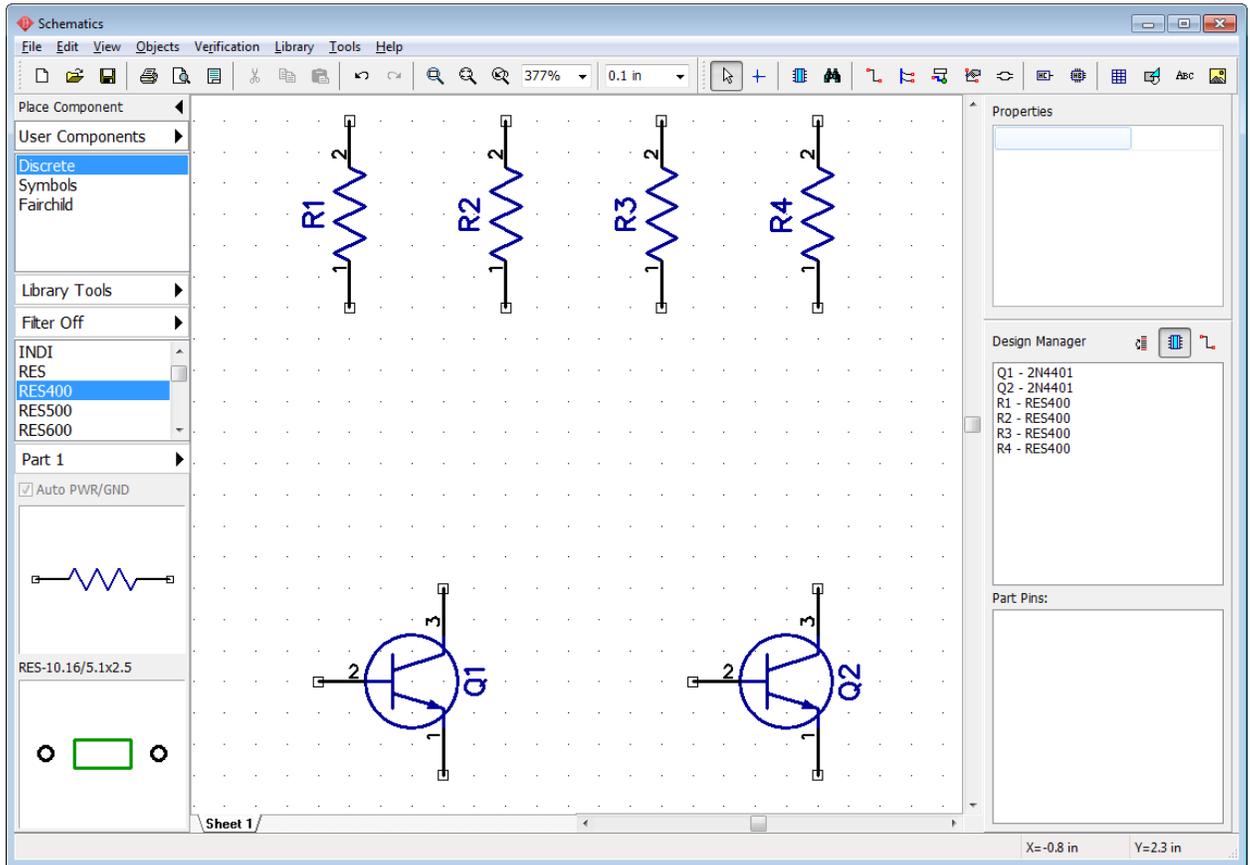
In the Copy Matrix dialog box set the number of columns and rows ("2" columns and "2" rows to get 4 resistors) and spacing (1 inch for columns and 0.4 inch for rows are good enough), click OK. Now you can see 4 resistors on the design area:



Move resistors to a proper location, like on the picture below (use mouse or arrows on the keyboard for moving) and rotate components 90 degrees (use "Space" or "R" hotkeys to rotate selected components). Another method to rotate an object is using "Edit \ Rotate" main menu item or right clicking on an object and selecting Rotate from the submenu.

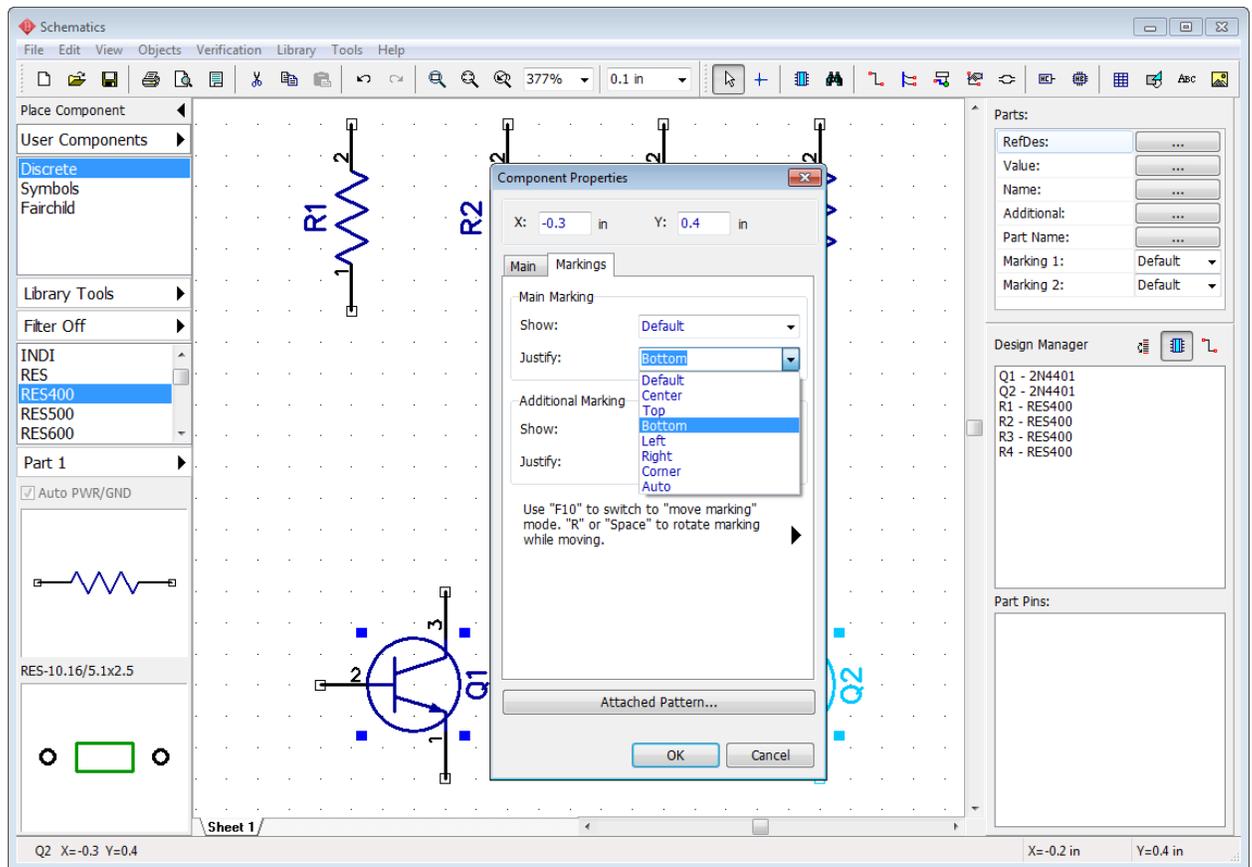
*Notice that you can pan design with right mouse button or the mouse wheel: move mouse arrow*

to the design area, hold right mouse button or mouse wheel and pan.



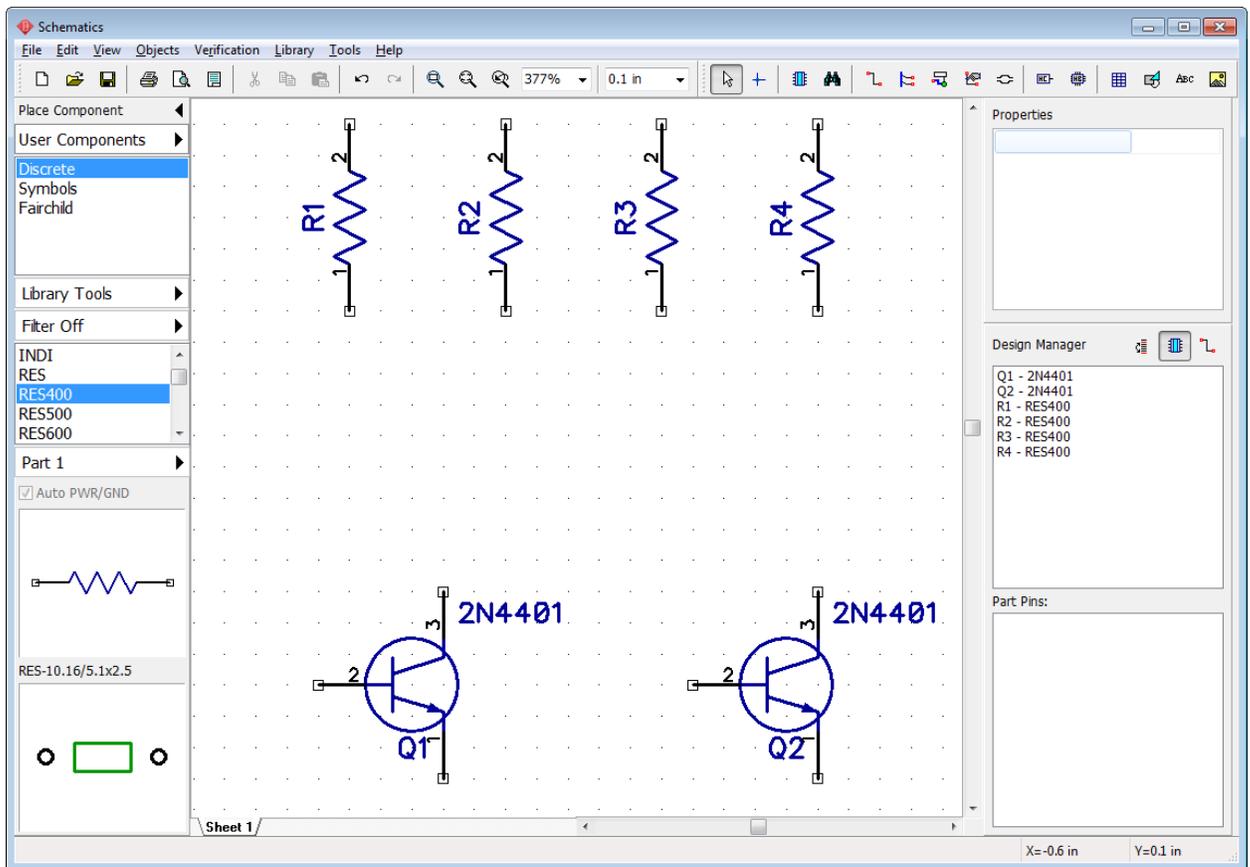
### Components' markings

The reference designators of Q1 and Q2 transistors have inappropriate location, they should be under the component symbols. To change RefDes location, select both transistors, right click on one of them and select Properties from the submenu. Choose the Markings tab in the component properties dialog box and specify: "Justify: Bottom" in the "Main Marking" section. Press OK.



Now we see the reference designators. Let's display component names for the transistors. Select the transistors again (if not selected already) and set "Show: Name and Part" and "Justify: Corner" in Additional Marking section in the Markings tab of the Component Properties dialog box. This will show the names of selected components. Notice that Reference Designators are already displayed as primary markings. Default means using common Schematic settings for all components. Displaying RefDes is a common property.

If you want to edit precise markings' positions, press the right arrow button under the Additional Marking section. The Component Properties dialog box will become wider.

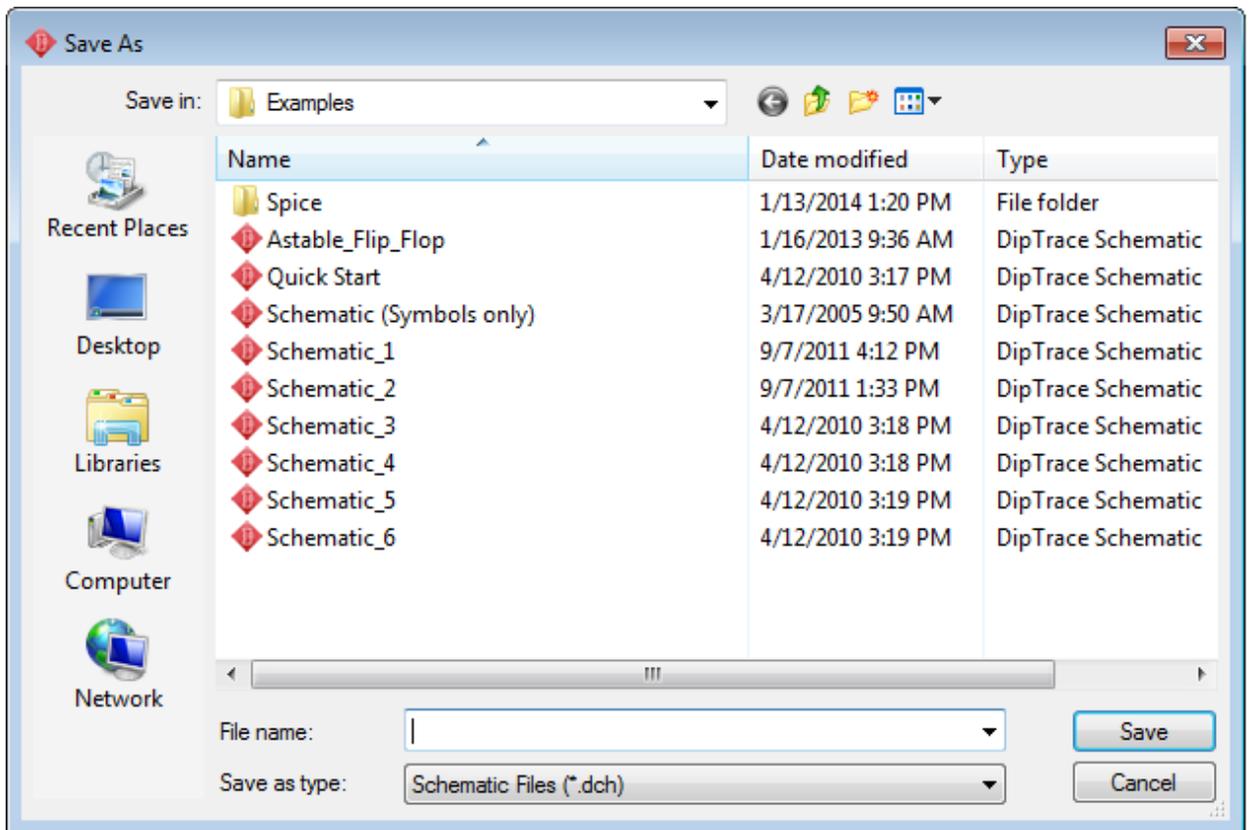


You can show or hide pin numbers for the entire circuit by selecting "View \ Pin Numbers \ Show", if they are not displayed yet. To change pin display settings for selected part, right click it and select Pin Numbers from the submenu.

However, if you're still not satisfied with location of RefDes, numbers, pin names or any other marking objects, you can easily move them around visually with a simple move tool. Select "View \ Part Marking \ Move Tool" from the main menu or press "F10". It is recommended to turn OFF the grid for precise moving ("F11" hotkey). You can move and rotate part markings like separate objects with "R" or "Space" keys.

"View \ Part Marking" menu item allows the user to change common settings of part markings. Common settings are applied to all schematic parts except those with custom properties.

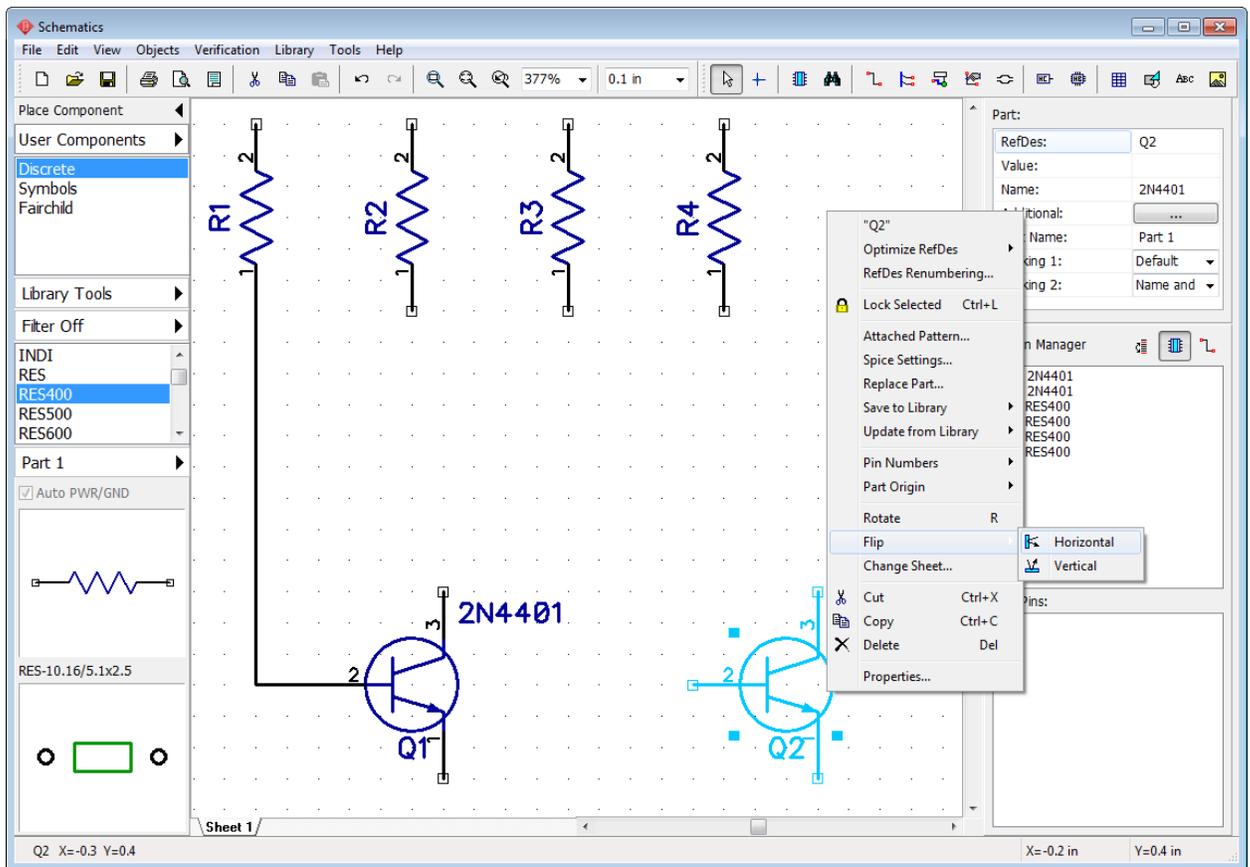
Use Undo or Redo buttons (   ) if you are not satisfied with the changes you've made, "Edit \ Undo" or "Edit \ Redo" from the main menu. The PCB Creator saves up to 50 steps. Remember to save schematic into the file. Select "File \ Save" from the main menu or click Save button on the standard toolbar. If current schematic has never been saved, the Save As dialog box will pop up to define file name and location. If file exists, clicking the Save button or pressing "Ctrl+S" hotkeys is enough. "File \ Save As" item can be used for changing the name of the file, for example, for backup etc.



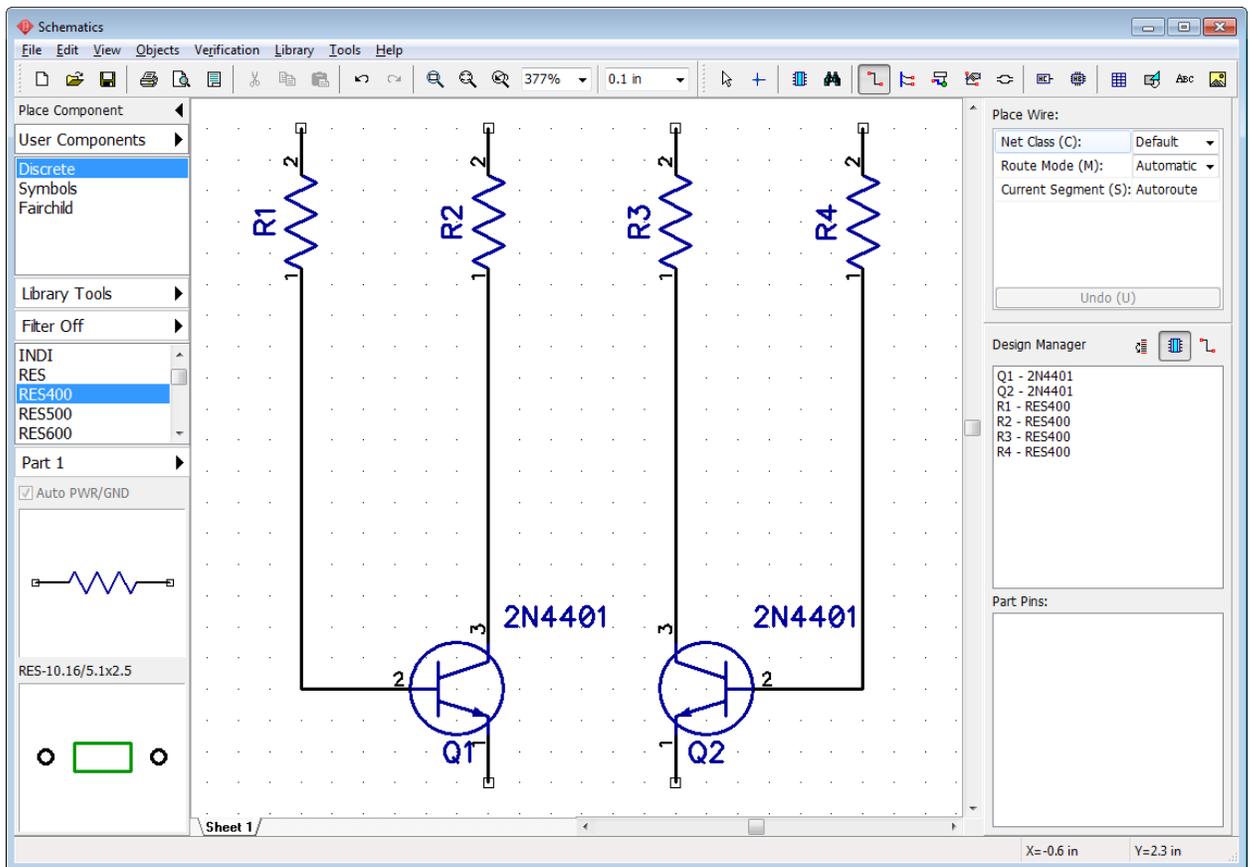
### Create connections

Connect pin 1 of R1 resistor to pin 2 (base) of transistor Q1. You need to make sure that you are in the default mode (  button is pressed). Hover with the mouse arrow over the bottom pin of R1 resistor and left click it. Then move mouse arrow down to the base pin of transistor Q1 and left click it to connect wire and create the connection between R1 and Q1.

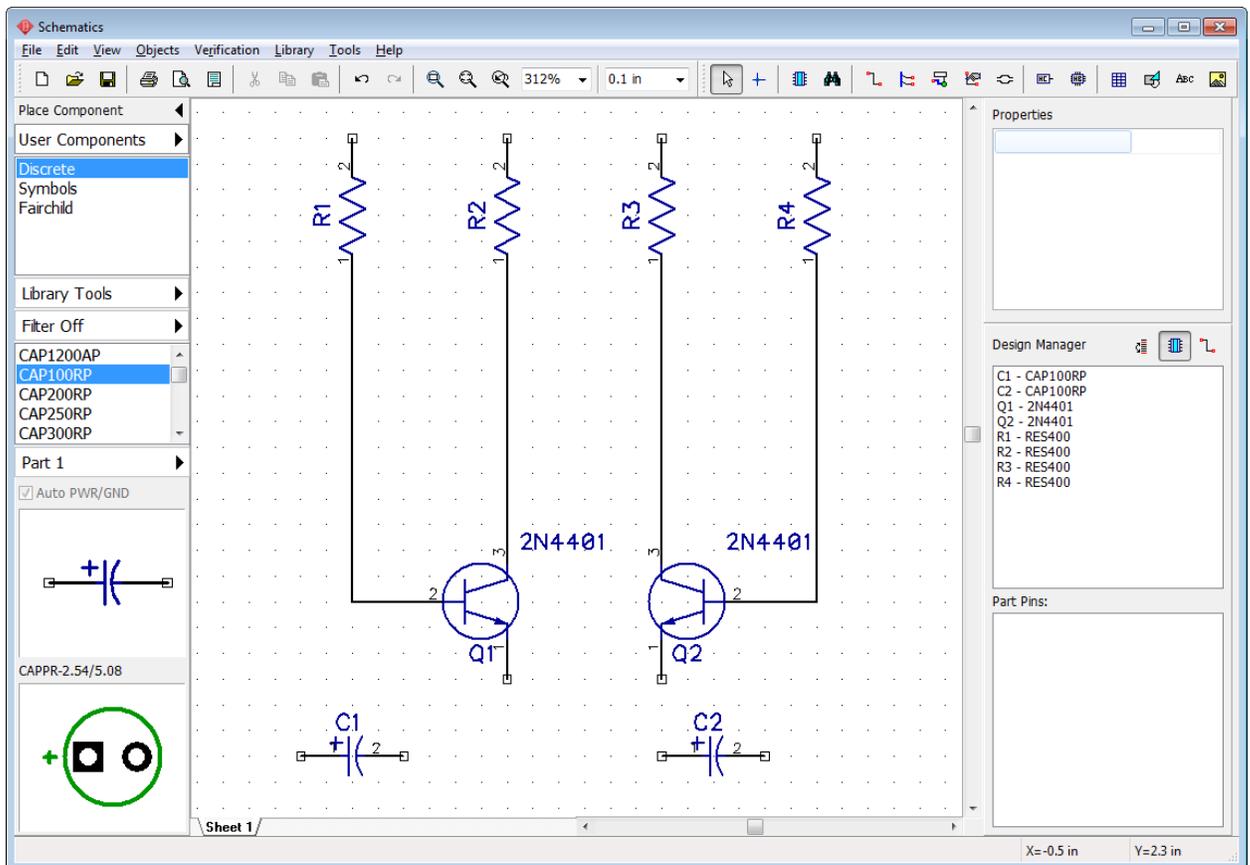
Now we need to mirror Q2 transistor, this will make schematic more easy to understand. Right click it and select "Flip \ Horizontal".



Connect R4 to pin 2 (base) of Q2, R2 to pin 3 of Q1, and R3 to pin 3 of Q2. Like on the picture below. If some wires are not straight, you can move parts or wires. This is not important for electrical connectivity but for esthetic pleasure and to make schematic organized and easy to understand. If you don't like automatic wire placement feature you can turn it OFF in the Place Wire panel on the Design Manager to your right-hand side, set Manual in the Route Mode section or just press "M" hotkey. You can see the Place wire panel only when you are in the wire placement mode. You may need to move some components for a good-looking schematic.



Now select CAP100RP from the Discrete library and place it twice to the design.

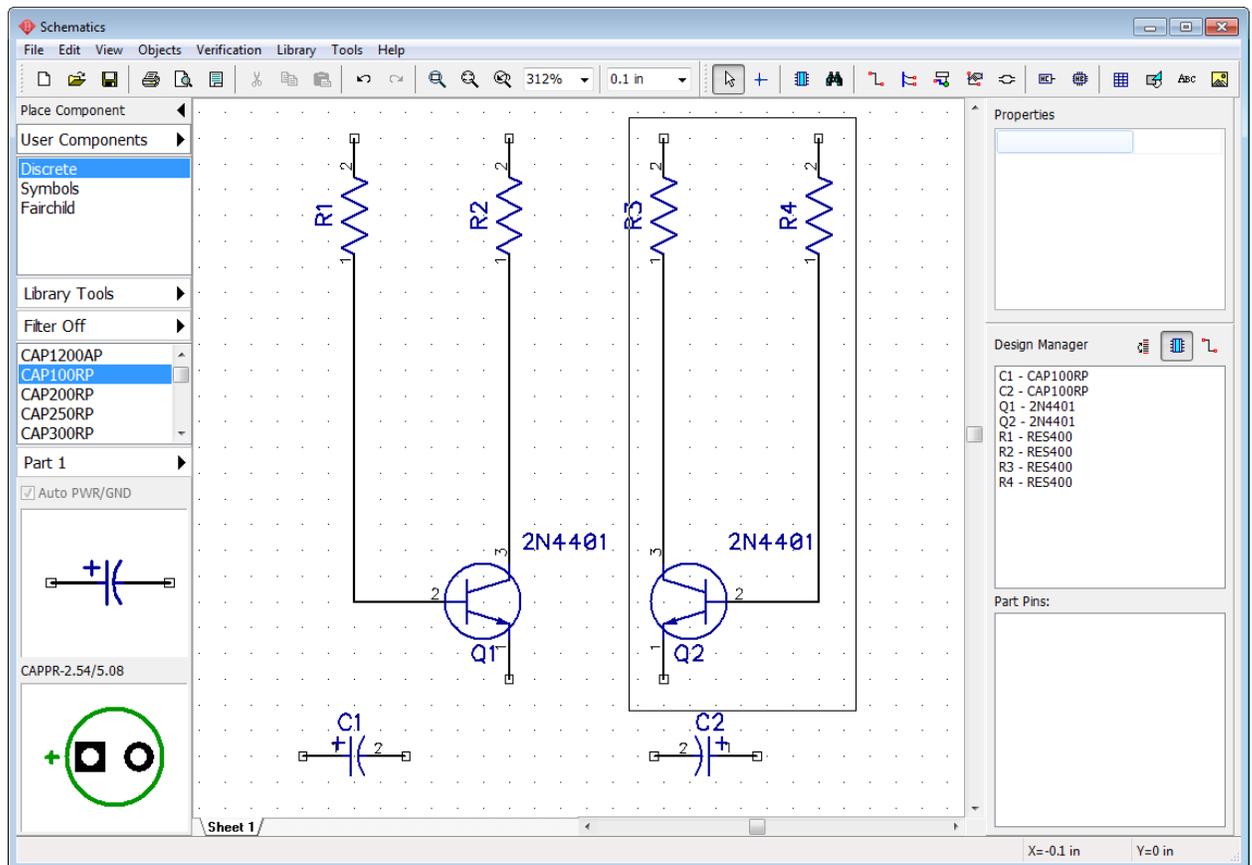


Flip C2 capacitor, select "Flip \ Horizontal" from the right-click submenu. C2 capacitor's plus sign should be on the right side.

We need to place two capacitors between the transistors Q1 and Q2 with respect to their polarities.

You might need to move some components to give enough space for the capacitors and connections.

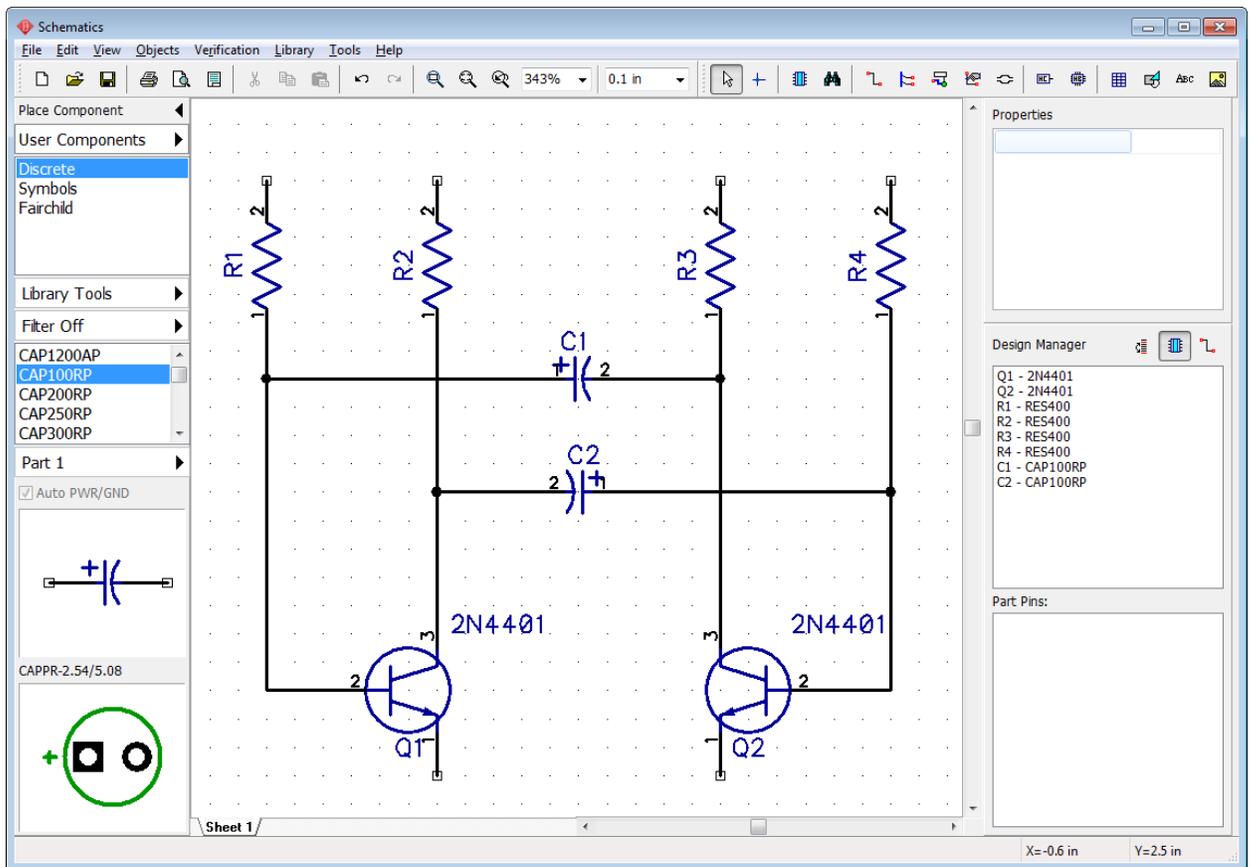
Move resistors a bit upwards and then select Q2, R3, R4 and related wires and move them to the right a little bit. Place mouse arrow in the upper-left corner of the selection, hold left mouse button and move to the opposite corner of selected objects. All components and wires inside the rectangle will be selected when you release the mouse button.



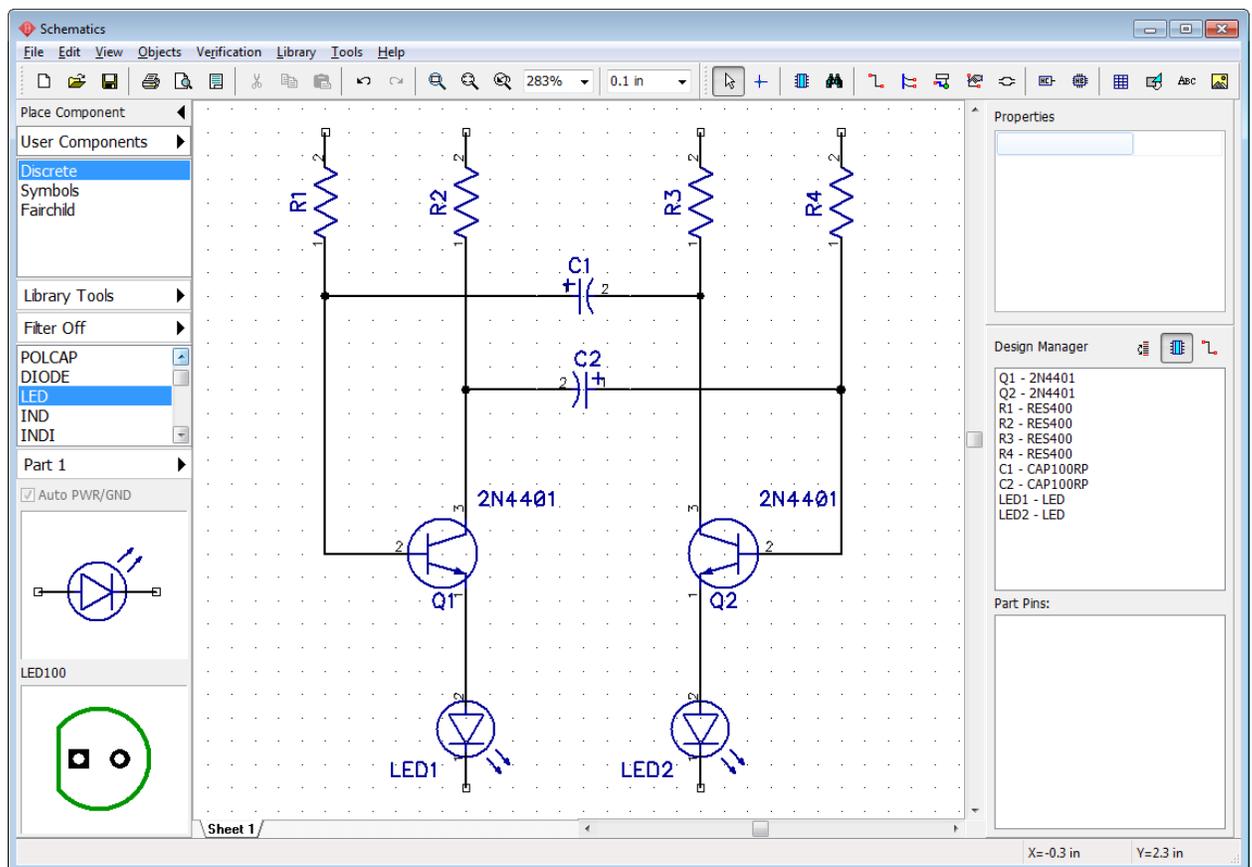
Use the right click to deselect all if you are in default mode, or double right-click if you are in another mode (first click to disable an active mode and the second click to deselect all).

Connect C1 (+) to pin 2 of the Q1: move mouse arrow to C1 (+) pin, left click it and move to the wire between R1 and Q1. Left click on the wire to connect. Small circle should appear if wires are properly connected.

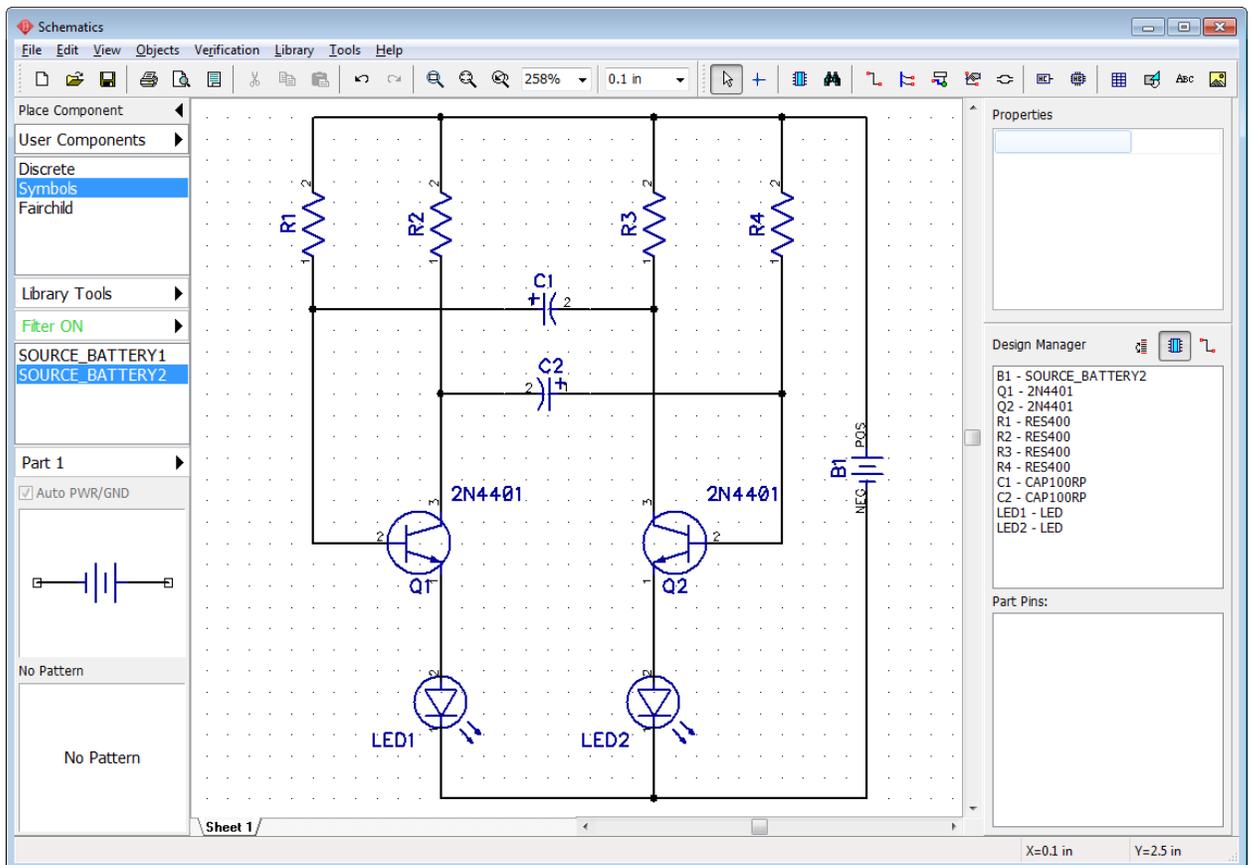
Then connect C1 pin 2 and C2 pins like on the picture below.



Scroll down the components list (Discrete library) on the Library Manager (component placement panel) to find LED component and place two of them onto the schematic. Then change the reference designators to "LED1" and "LED2" (right click on the component and select first item from the submenu), rotate these parts with "R" hotkey or Space bar pressed three times. Probably, you'll need to move and rotate RefDes a little bit with Move tool ("F10"). Then connect LEDs to transistors like on this picture.



Place a battery symbol SOURCE\_BATTERY2 from the Symbols library, change its RefDes and complete the circuit by connecting all the wires (see the picture). Make sure that you see small black circles where two wires are connected, if not, then wires are not connected.



If you want to move an existing wire, hover with your mouse over it (the net should be highlighted, mouse cursor shows possible moving directions) then hold the left mouse button and move wire to a new position.

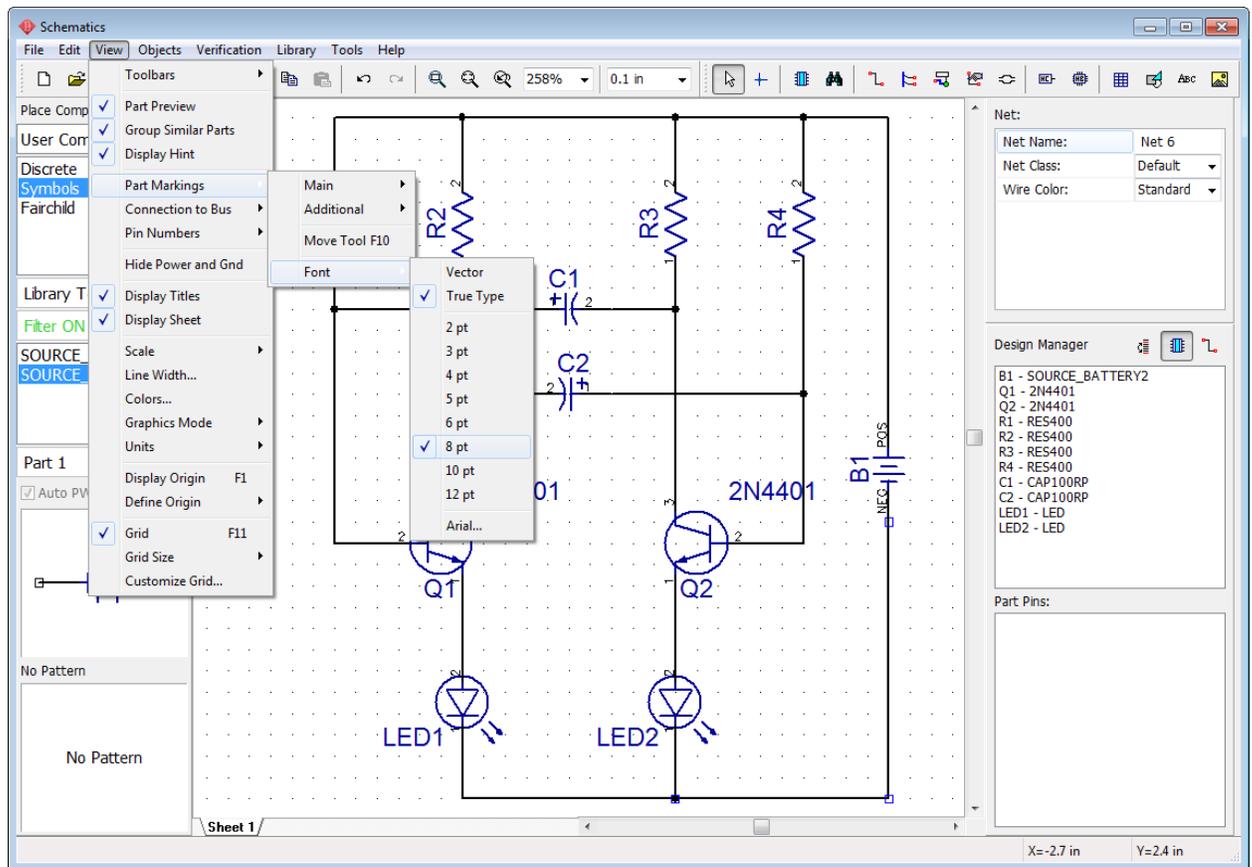
*Notice that if you are in the Place Wire mode and you click on the existing wire, you start to create a new connection, not editing an existing one.*

(Place Wire mode is enabled automatically when you try to place wire by clicking on some component's pin).

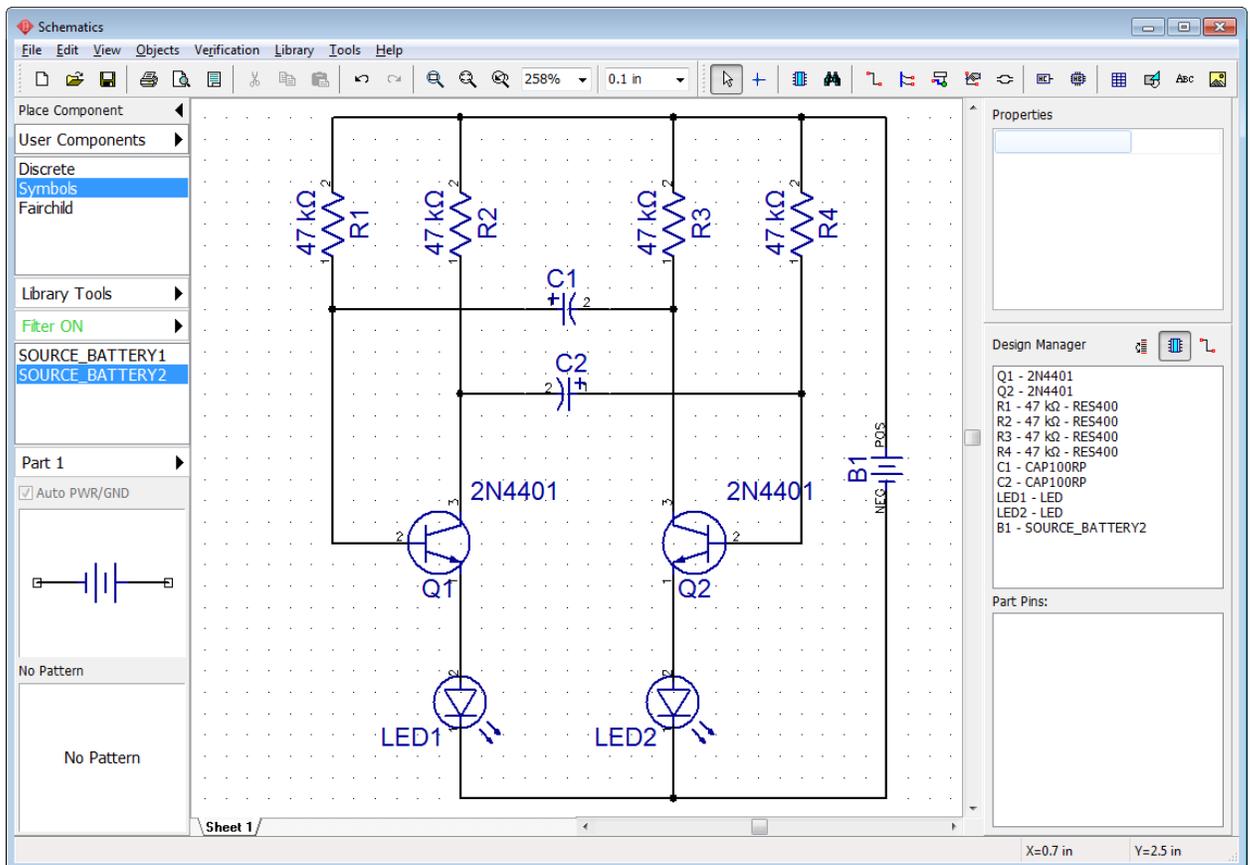
If some objects are not highlighted when you move mouse arrow over them, right click on a free area to activate the default mode. If you want to delete a wire, right click it to open the submenu and select Delete Wire. To delete a wire segment, select "Delete Line" from the same submenu. You can use Undo to return to the previous state(s) of the circuit.

Now we will add resistance values "10k " for all resistors on this schematic. Since " " is a Unicode character, it doesn't work in vector fonts, which are set by default in PCB Creator. We need to activate TrueType font for part markings in order to use Unicode characters. Go to "View \ Part Marking \ Font \ TrueType". Since TrueType characters look a bit different from Vector on the design area, you might need to change font size. We made it a little bit bigger (8 pt).

*Notice that there are various ways to enter special characters. We recommend to copy symbol from the character map ("Start \ All Programs \ Accessories \ System Tools \ Character Map" in Windows OS) and paste them in the PCB Creator.*



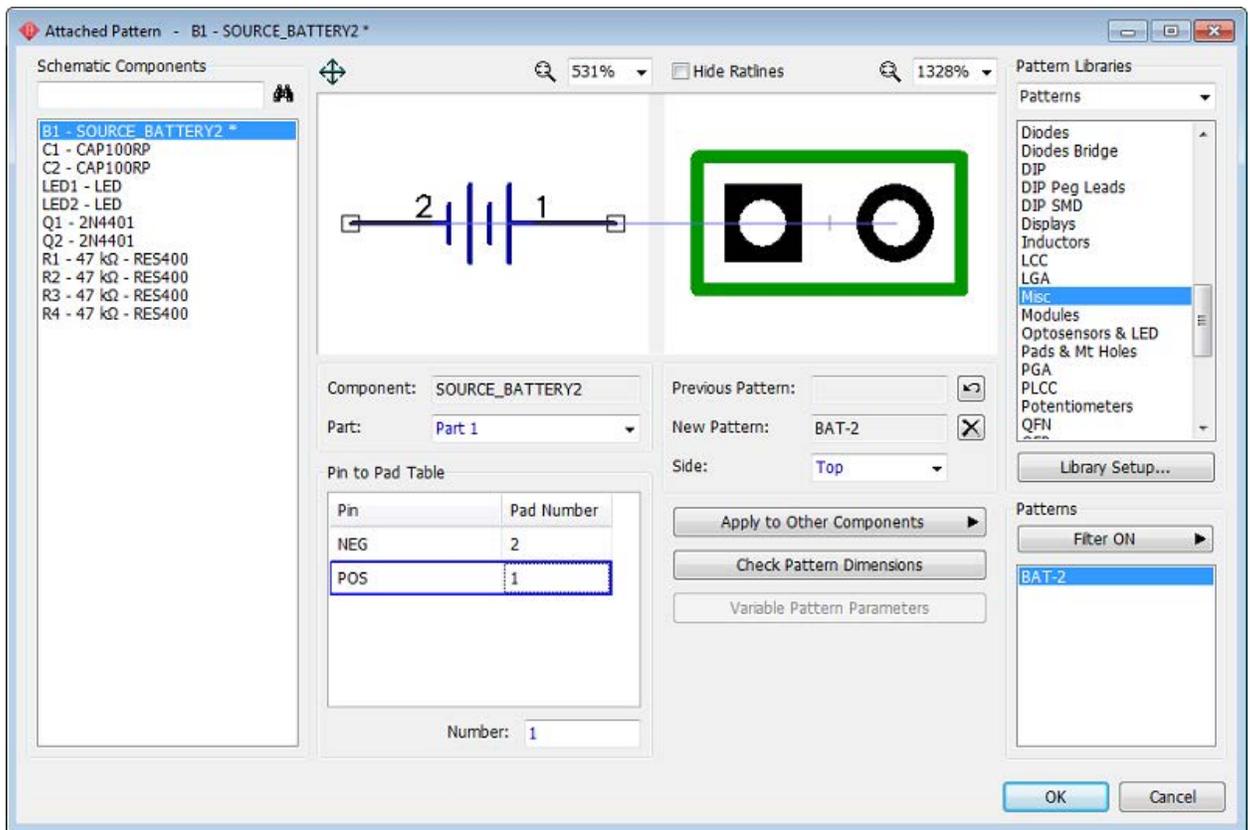
Now select all resistors then right click on one of them and select Properties from the submenu. Select the Main tab and type "47 k " into the Value field. Click on the Markings tab and specify "Main Marking \ Show \ Value", then go to "Additional Marking \ Show \ RefDes" and click OK.



As you remember, we took battery component from the Symbols library. All components in this library don't have patterns, it's just symbols (pattern preview field on the component placement panel says "No Pattern"). But in order to proceed to PCB layout stage you should attach related pattern to this symbol. If left blank, PCB Creator will not be able to show this component on the board and error dialog box will pop up.

Hover with your mouse over the battery symbol, right click it to show the submenu and click Attached Pattern. In the pop-up dialog box you can see the list of all components of current circuit in the left part of the dialog box, make sure B1-SOURCE\_BATTERY2 is selected (you will see the battery symbol in the preview field). Select the Patterns library group from the Pattern Libraries drop-down list on the right. This library group contains all standard PCB Creator pattern libraries separately from the symbols. Select "Misc" library from the list and find BAT-2 pattern in the pattern list at the bottom-right of the dialog box (use search filters if you want). Pin-to-Pad connections are usually defined automatically by pad numbers, but this is not the case with this battery symbol.

Positive pad is usually square shaped and negative is round. Click on the corresponding row in the Pin to Pad Table and type in related pad number in the Pad Number column (NEG pin should refer to Pad #2, POS pin – to Pad #1). Pin to pad connections can be defined visually, just left click on the pin in the symbol preview field and then on the corresponding pad in the pattern preview field.



Click OK when you are done to close the Attach Pattern dialog box.

*Notice that some symbols in the libraries are intentionally made without attached patterns (for example, VCC, GND or other logical connectors – Net Ports).*

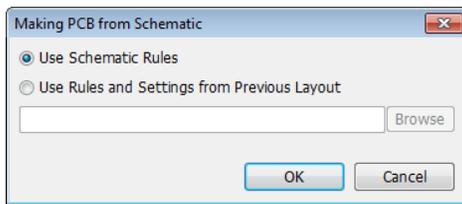
Our schematic is now ready to become a PCB. Do not forget to save the schematic, select "File \ Save" from the main menu or click on the Save button or simply "Ctrl+S".

Schematic can be printed or saved in BMP or JPG file. To print in PDF you need to install any of the free PDF printers widely available online and select it in the printer selection dialog box.

Select "File \ Preview" from the main menu, customize it and press Print All to print all schematic sheets, use Print Current Sheet button – to print selected sheet or the Save button to create a BMP/JPG file with defined resolution.

## 1.5 Converting to a PCB

You can open PCB Creator schematic files (\*.bch) in the PCB Layout module, but if you want to save your time, select "File \ Convert to PCB" or press "Ctrl+B" hotkeys directly in the Schematic. In the pop-up dialog box you can use Schematic rules or load rules from any other PCB layout file. When you press OK, the schematic will be opened in PCB Layout.



In case of incorrect exit from the program or if you somehow forgot to save your design, it is possible to recover the latest schematic by selecting "File \ Recover Schematic" in the Schematic or "File / Recover Board" in the PCB Layout module.

If you want to hide the layers panel and the Design Manager to get more space, press "Ctrl+3" hotkeys or uncheck "View \ Toolbars \ Design Manager" main menu item.

### Renew from Schematic

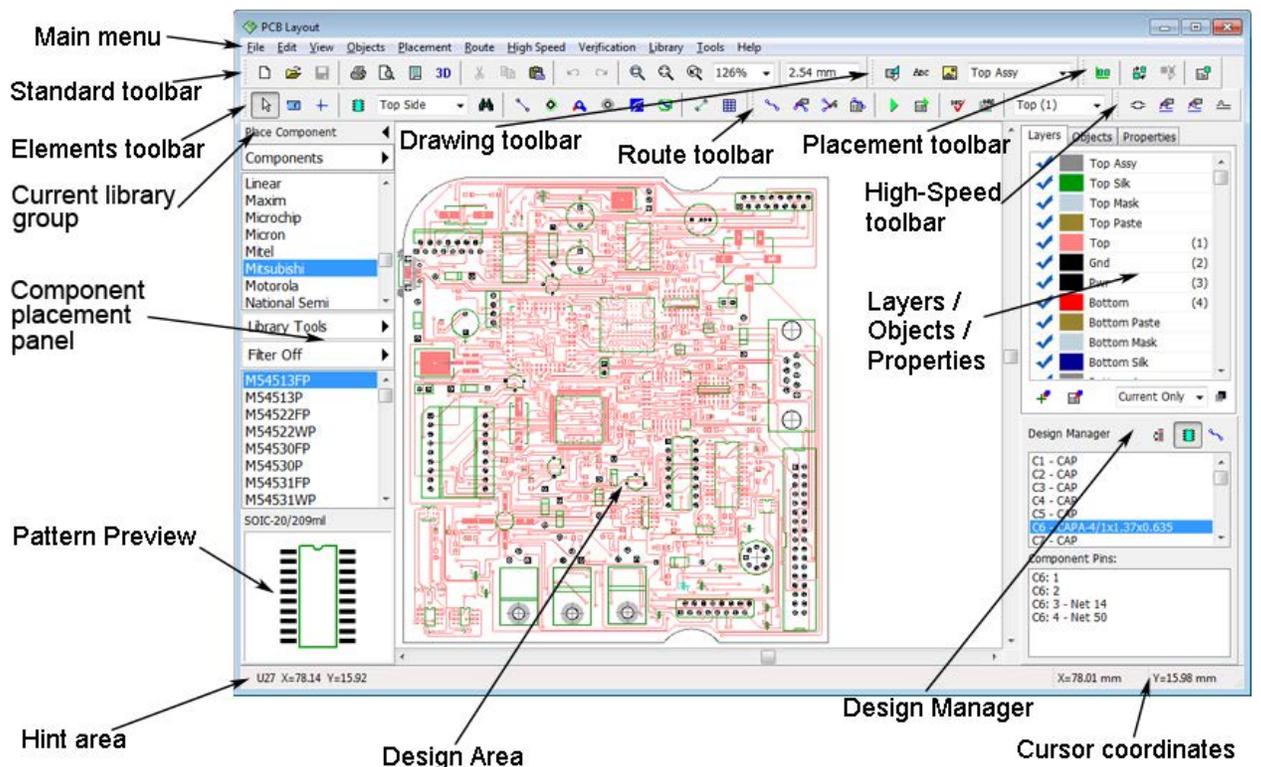
If you've made changes to the schematic, they do not automatically appear in the PCB Layout. You need to manually renew board from the schematic file. Select "File \ Renew Design from Schematic" then open updated schematic file. PCB Creator will keep component placement and current routing on the board. Renewing from schematic allows for several options:

- 1) **Renewing by components** means using hidden IDs to determine component-to-pattern links – this will work only if circuit was created in PCB Creator Schematic. Renewing by components doesn't depend on RefDes, therefore they can differ on the schematic and PCB.
- 2) **Renewing by RefDes** means that component-to-pattern links are determined only by RefDes.
- 3) **Updating from Related Schematic** means renewing by components from related schematic file (go to "File \ Layout Information" in main menu if you don't remember the source-schematic file).

## Part 2: Designing a PCB

### 2.1 PCB Layout UI

PCB Layout main window includes: design area, main menu, toolbars, component placement panel (Library Manager), Design Manager with layers / objects / properties panel and the status bar (bottom).



You can place and edit different objects (components, ratlines, traces, copper pours, shapes, tables etc.) on the design area. Main menu provides access to all common program features.

Other interface elements:

**Standard Toolbar** – tools to work with files, cut / copy / paste objects, print, preview and configure titles, run 3D Preview, change scale and grid size.

**Elements Toolbar** – tools to switch to default mode, measure, change origin, place components, ratlines, pads, vias, mounting holes, copper pours, dimensions and tables.

**Route Toolbar** – tools to create and edit traces, create board, run and setup autorouter, check design rules, select current signal layer.

**Placement Toolbar** – placement and auto-placement tools.

**Drawing Toolbar** – tools to create shapes, text, insert images, and select shape and text placement layer.

**High-Speed Toolbar** – define differential pair, edit single trace of a differential pair, free single trace editing mode, phase tune tool.

**Component placement panel (Library Manager)** – work with libraries and components: library groups, library tools, search filters, pattern preview. Select library, select component and place it to the design area.

**Layers / Objects / Properties** panel (active tab depends on current selected object / tool / mode).

The Layers tab allows working with layers (show / hide, add / delete / edit, change position and color) and changing layer display mode;

The Objects tab allows the user to show / hide different objects on the design area and block certain objects from being selected;

The Properties tab shows properties of selected tool / object and allows for their editing.

**Design Manager** – navigates user around the layout. Left click in the list of components or nets (selected with buttons) highlights object on the design area, double click - pans to selected component or net.

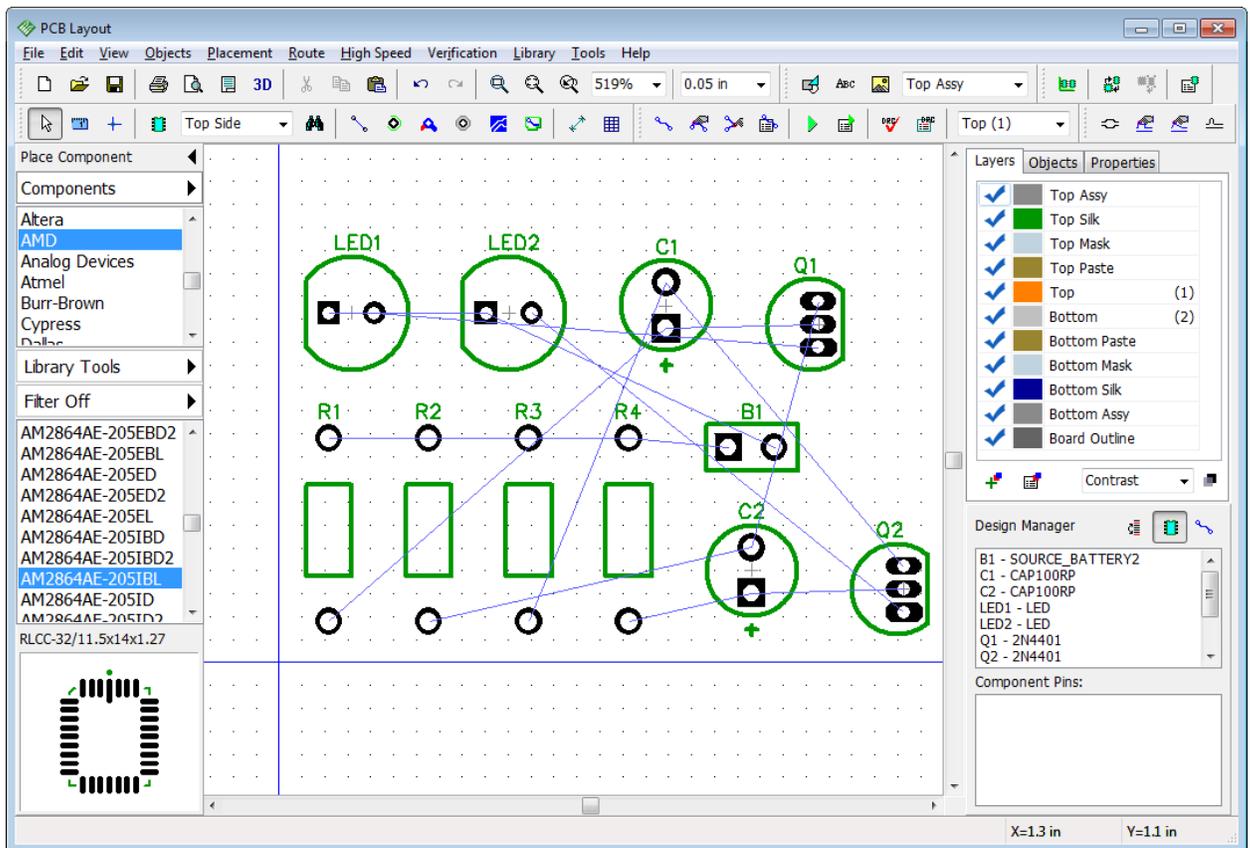
**Status bar** – shows current hint on the left and cursor coordinates on the right.

Go to "View \ Toolbars \ ..." to hide / show any of the UI toolbars (or use "Ctrl+1 through 9" hotkeys). See the PCB Creator Help for details ("Help \ PCB Layout Help" from the main menu).

## 2.2 Preparing to route

Routing itself is one of the final stages of board design. But quality of routing greatly depends on the preparation procedures.

Right after conversion layout looks chaotic. Press Arrange Components button on the placement toolbar or select "Placement \ Arrange Components" from the main menu, the components will be placed near the design center (blue line cross) and arranged according to the placement settings. Automatic arrangement is not necessary for this design, but you need to know how to do this when dealing with real-life projects.



You can use Auto-placement or Placement by list after converting to PCB. These are very convenient and useful tools which allow the user to get advantages of both automatic and manual placement modes. We will try to place components automatically in [Part III](#)<sup>[209]</sup> of this tutorial with more complex circuits.

### Component markings

Make sure that reference designators are visible. Select "View \ Component Markings \ Main \ RefDes" if you don't see them. This command shows the reference designators of all components, except those with individual settings. If marking's text location doesn't look acceptable, you can justify it. Select "View \ Component Markings \ Main \ Justify \ Auto" or select another mode. To increase the size of the marking text select "View \ Component Markings \ Font \ 5pt" from the main menu. You can choose another size, but don't make it too big.

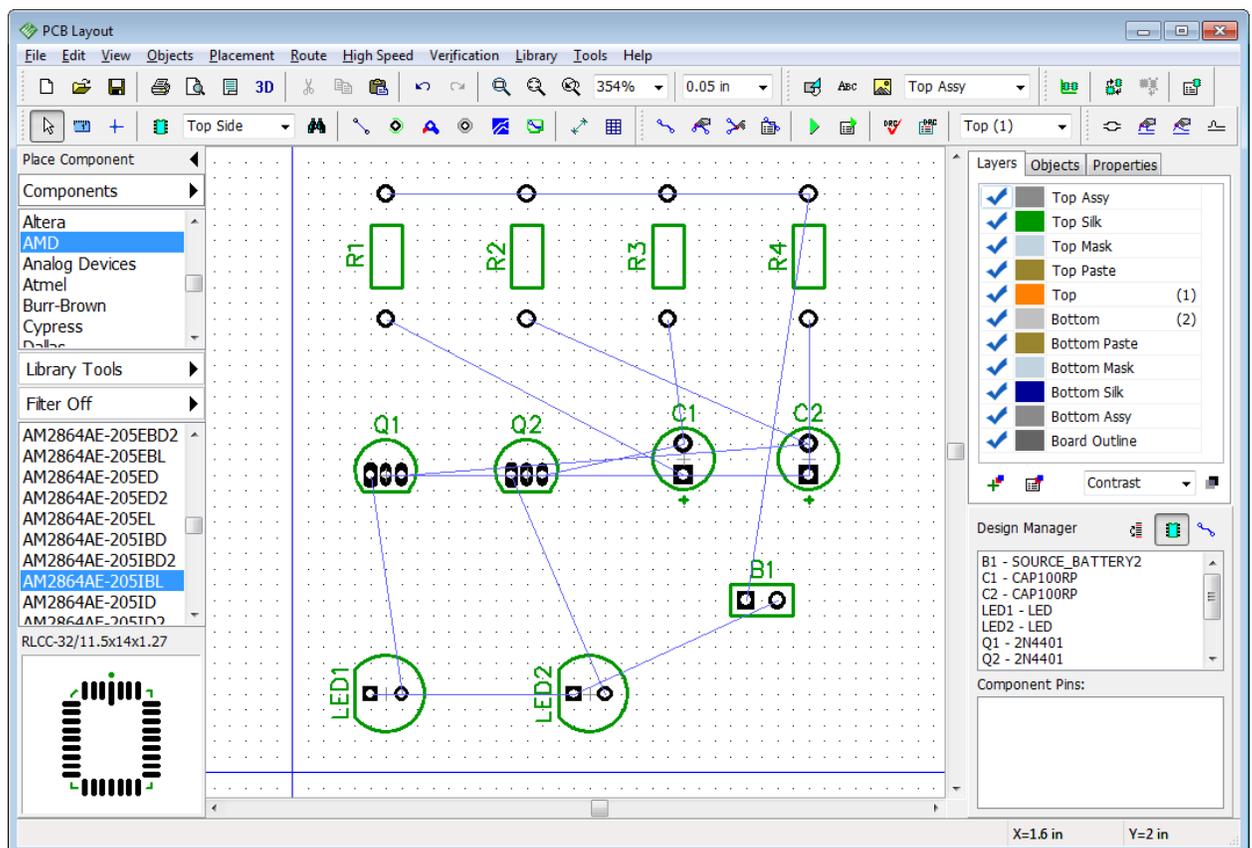
For PCB Layout we recommend vector font type, however **only TrueType fonts support Unicode and non-English characters**. Font settings – "View \ Component Markings \ Font \ ..." main menu item. Vector and TrueType fonts look a bit different.

To define custom component marking parameters for selected components right click on one of them and select Properties from the submenu. Then open the Markings tab in the pop-up dialog box. Remember that you can use the move tool – "F10", or "View \ Component Markings \ Move Tool". This option allows for moving and rotating (90 degree step) any text object on the board.

## Manual placement

Now place components manually, according to your preferences and design rules. It is a good practice to keep power supply components in one area and functional blocks in another part of the board. Apply appropriate layout rules and differential signaling for high frequency circuits. Notice that we use 0.05 inch (1.27 mm) grid, change it if you need with a drop-down list on the instruments toolbar. Select "View \ Units \ Inch" main menu item to change measurement units.

Create the layout similar to the picture below, with resistors at the top and LEDs at the bottom of the board. Drag and drop components to move them on the board. Press "Space" or "R" default hotkeys to rotate selected components 90 degrees. If you need to rotate to different angle, select components then right click on one of them and choose Define Angle or Free Rotate for precise and visual rotation respectively.



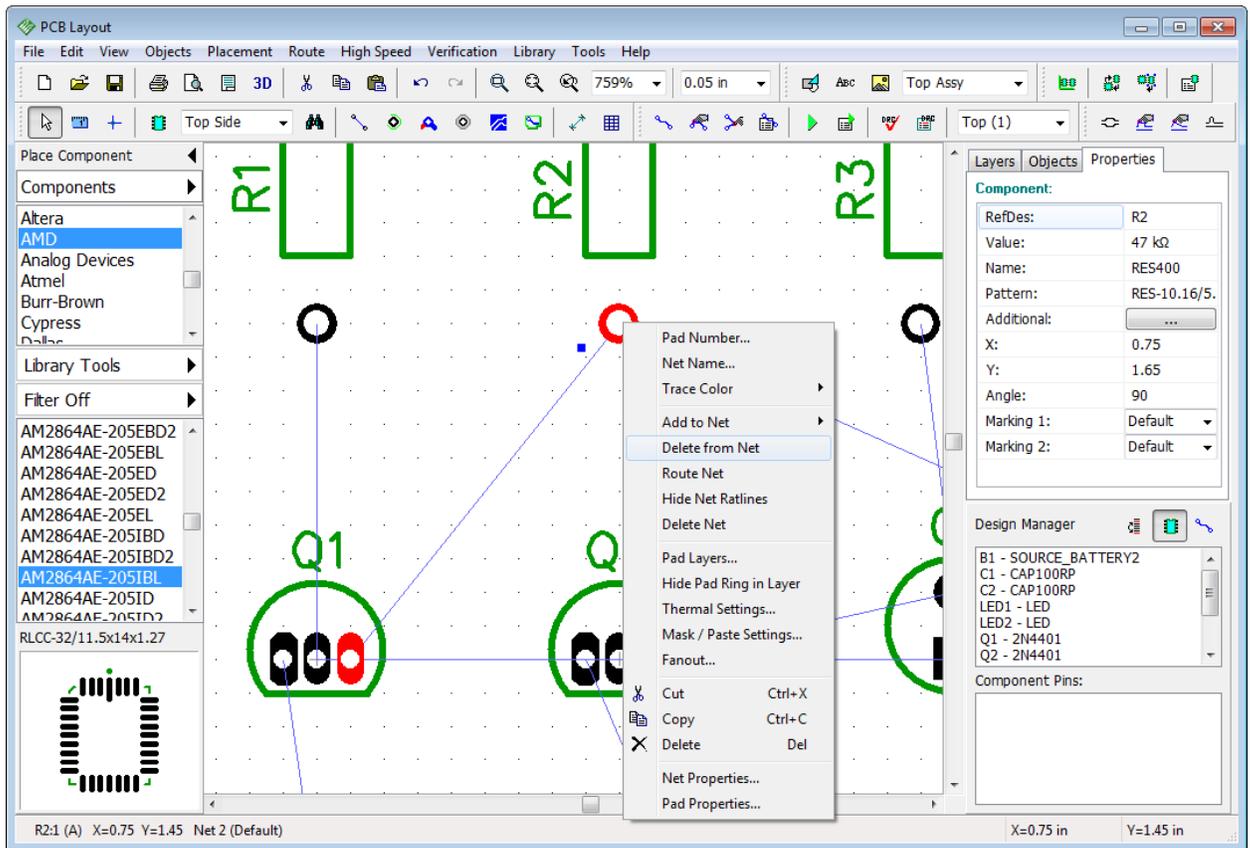
Press "F12" to optimize how connection lines visually appear on the screen (this does not change net structure).

## Changing net structure

We're going to practice in changing the net structure on the board, adding and removing connections. Blue thin lines between pads show logic connections, these lines are called "ratlines". Changing the net structure is not necessary for this board, but you should know how to do it.

Hover with the mouse over any pad, right click it and select Delete from Net and the pad will

disappear from the net. This pad is no longer connected with a blue line.

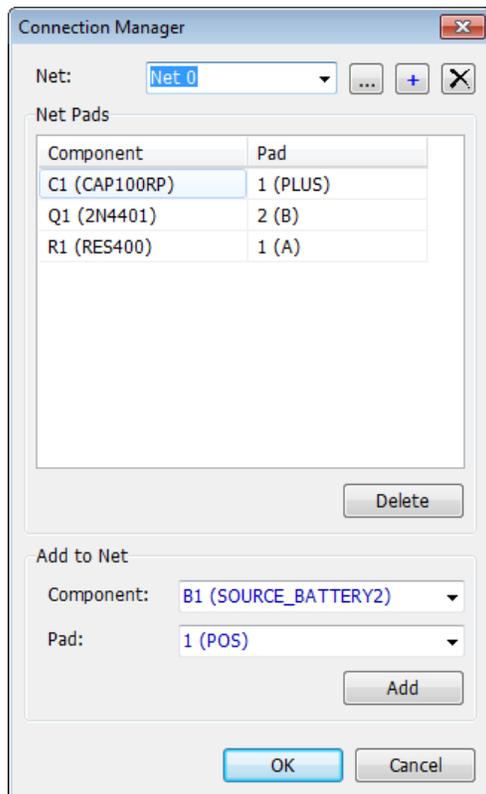


You can visually create a pad-to-pad connection. Select "Objects \ Place Ratline" from main

menu or press  button on the objects toolbar. Then hover the mouse pointer over unconnected pad, left click it and move your mouse to any other pad (connected or unconnected), and left click it. A new wire (or a new net in the case with both previously unconnected pads), represented as a thin blue line (ratline) will appear. By the way, existing connection can be deleted the same way, just select Delete Connection from the right-click submenu on the pad.

If you would like to add selected pad to some net without creating a connection visually on the design area, hover over the pad, right click it and select "Add to Net \ ...", then select net from the list of all nets of the project or point it with the mouse cursor on the design area.

However, the most convenient way to add, delete or rename nets, as well as add or delete pads to/from the nets is Connection Manager. Select "Route \ Connection Manager" from main menu to open it. Connection manager is easy and friendly.



Select net from the "Net:" drop-down list and you will see all pads of the net in the table, they can be deleted anytime. If you want to connect some pad to the net, select component and its pad, using drop-down menus at the bottom of the dialog box and press Add button.

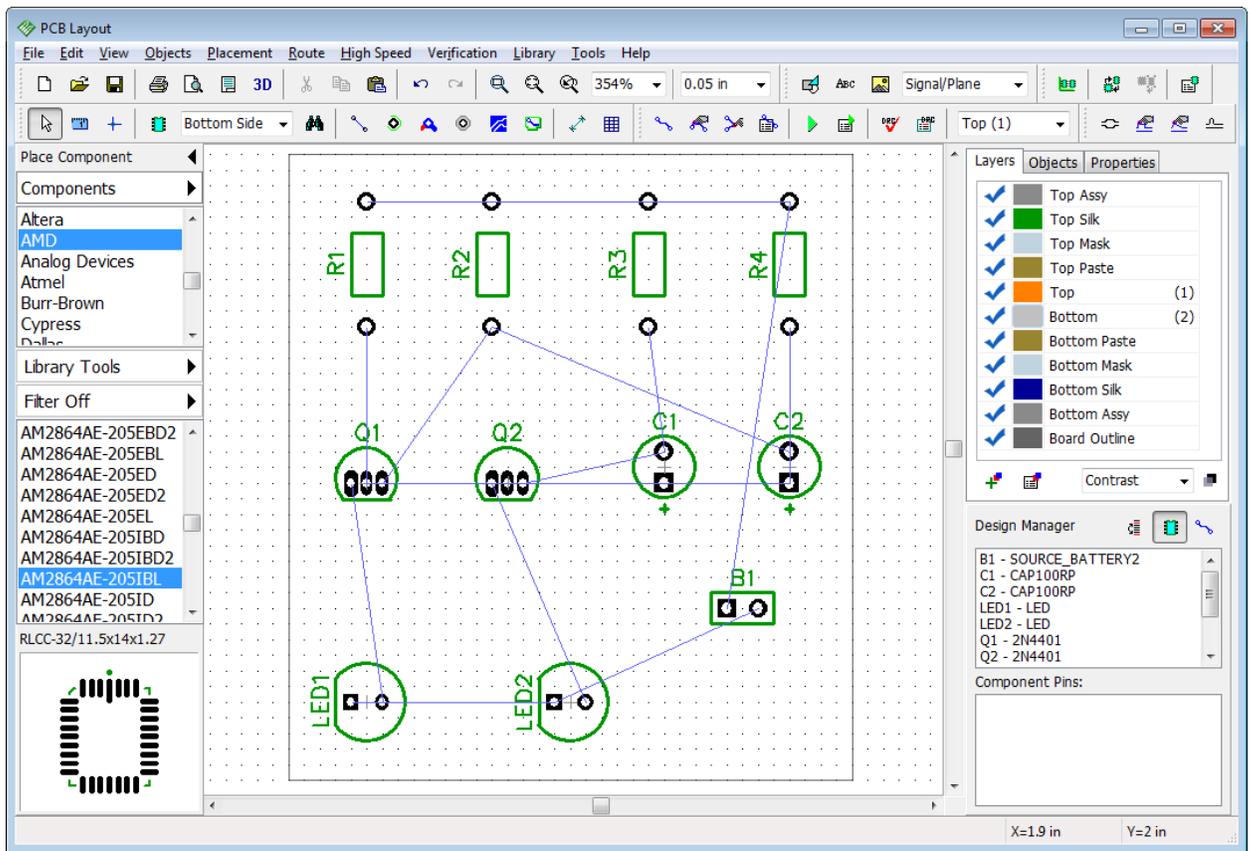
If you have changed the net structure, please press Undo until previous design is restored. Close the connection manager. By the way, if you lose design or schematic because of incorrect exit from the program, use "File \ Recover Board" in PCB Layout and "File \ Recover Schematic" in Schematic to recover the latest project version.

To protect the net structure from accidental change go to "Route \ Lock Net Structure".

## Board outline

We haven't determined the board outline yet. If you will launch autorouter, it will create an appropriate board automatically (rectangle for simple boards). But in real life electronic designer usually has certain board requirements well before starting the project. Board polygon can be created directly in PCB Creator or imported from the DXF file (if this is a complex shape).

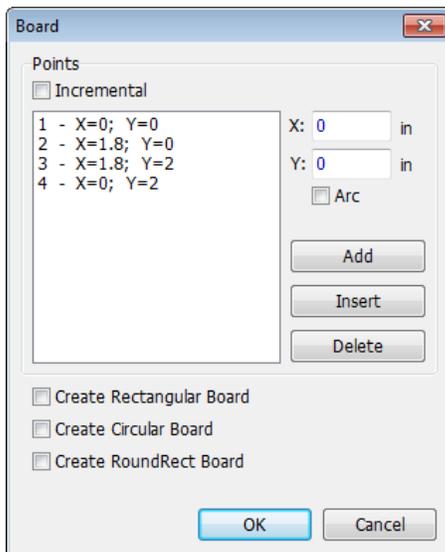
Select "Objects \ Place Board Outline" or press  button on the routing toolbar, then place board outline by left clicking the key points on the design area. Right click on the final point of the polygon and select Enter from the submenu or press "Enter" on the keyboard. For this design we require a simple rectangle board near 2 inches in size, see the picture below (origin point is hidden – "F1" hotkey).



Notice that you can create arcs in board outline by selecting the Arc Mode from the right click submenu while drawing the polygon.

You can insert new points into ready board outline polygon, move each point or entire polygon on the design area. Point coordinates are shown as a hint when cursor hovers over it.

There is another way to create board polygon that does not involve drawing it on the design area. Select "Objects \ Board Points" from the main menu.



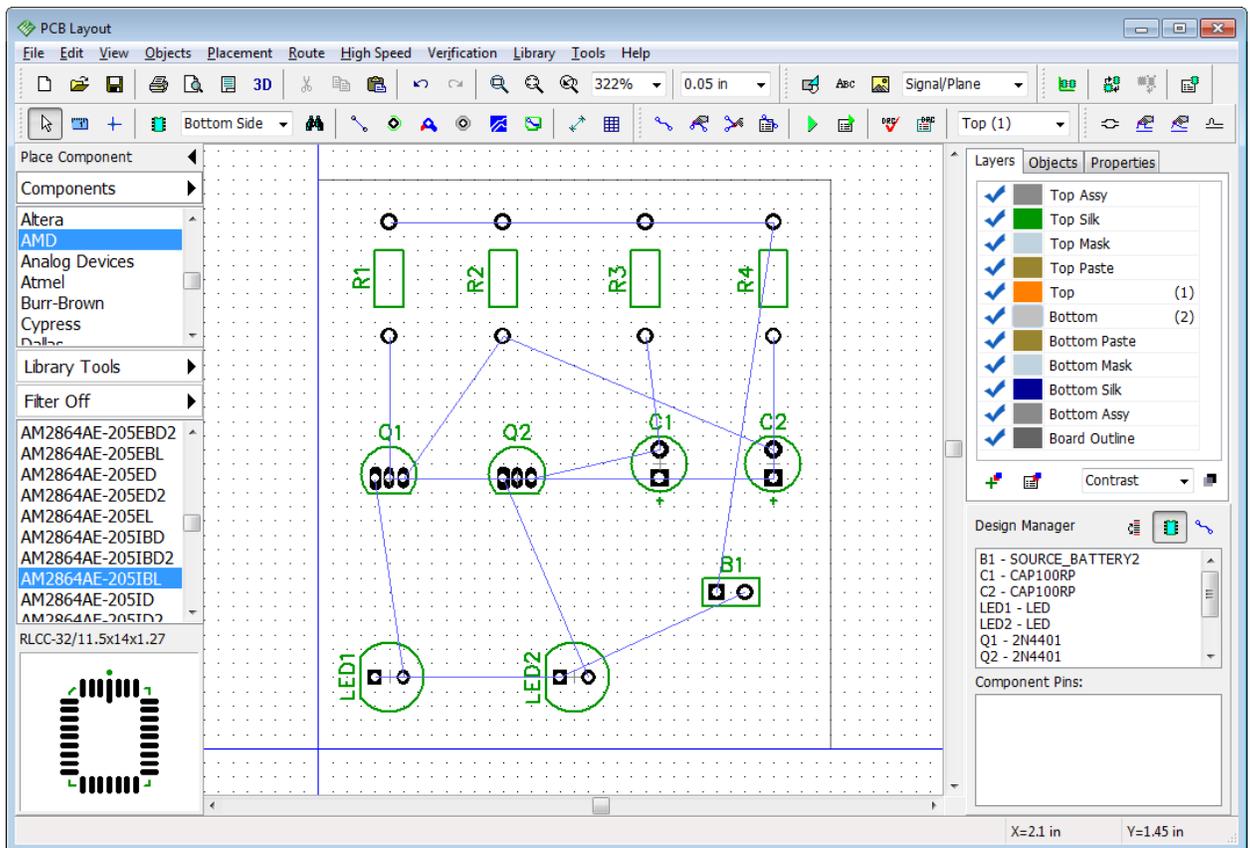
In this dialog box you can Add, Insert, and Delete key points. Coordinates appear and can be edited in the absolute or incremental mode. If you check the Arc box for some point, that point will become a middle point of an arc and the neighboring points – arc's beginning and ending. For rectangular boards check Create Rectangular Board box and simply define the first point (base), width and height of the board. It is also possible to make circular and rectangle boards with rounded corners automatically.

Press OK to apply changes or Cancel to close the dialog box.

Notice that you can use "Objects \ Delete Board" from main menu if you want to delete the outline polygon.

Board origin should be defined, bottom-left corner of the board outline is the best place for the reference point. If you strictly followed the instructions given before you should see two blue lines crossing in the correct place. However, in the case you do not see the origin point or it is not in the bottom-left corner of the board polygon, select "View \ Origin" from main menu or press "F1" hotkey to show it. If the position is wrong, go to "View \ Define Origin \ By Mouse Pointer" from

the main menu or press  button on the instruments toolbar and left click on the design area (PCB Creator helps to target on the key points).

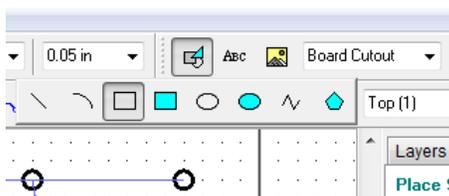


Now all coordinates in the PCB Layout will be displayed and edited relatively to the origin. But you can change the origin's position at any moment.

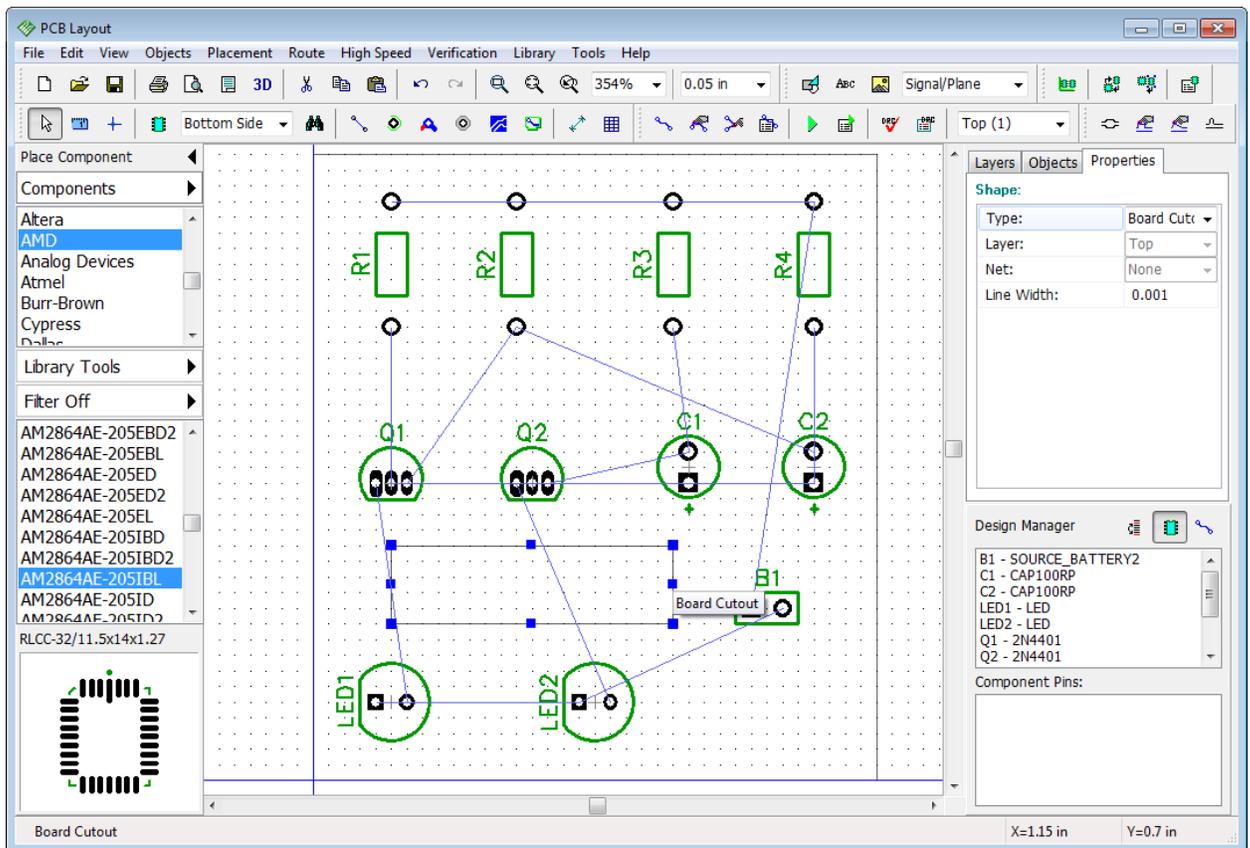
*Notice that coordinates of components on the board are calculated by pattern's origin point. It is defined in Pattern Editor. To show or hide origin of selected component/s, right click on one of them and select Pattern Origin from the submenu.*

### Board cutout

PCB Creator allows the designer to create board cutout polygons. You can create cutout of any shape, but we will create a simple rectangle cutout between LEDs and transistors just to show you how to do this.



Select Board Cutout layer in the drop-down list on the drawing toolbar then choose rectangle drawing tool and create a rectangle cutout on the board in the design area. Pan, zoom and change grid size for precise drawing. Board cutout is ready.

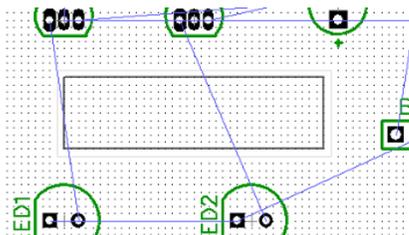


There is another way to create cutouts. Just draw a shape on any layer of the board or import it from the DXF file, then right click on the shape and select Properties from the submenu. In the pop-up dialog box select Board Cutout from the Type drop-down list and press OK.

*Notice that board cutout does not visually differ from the board outline, you should be careful not to place a cutout instead of an outline.*

### Route keepout

Route keepout is an area on the board not intended for any copper inside it. Autorouter does not draw traces there, and the program will report errors if you add them manually. Board cutout shape does not have a clearance parameter like board outline.

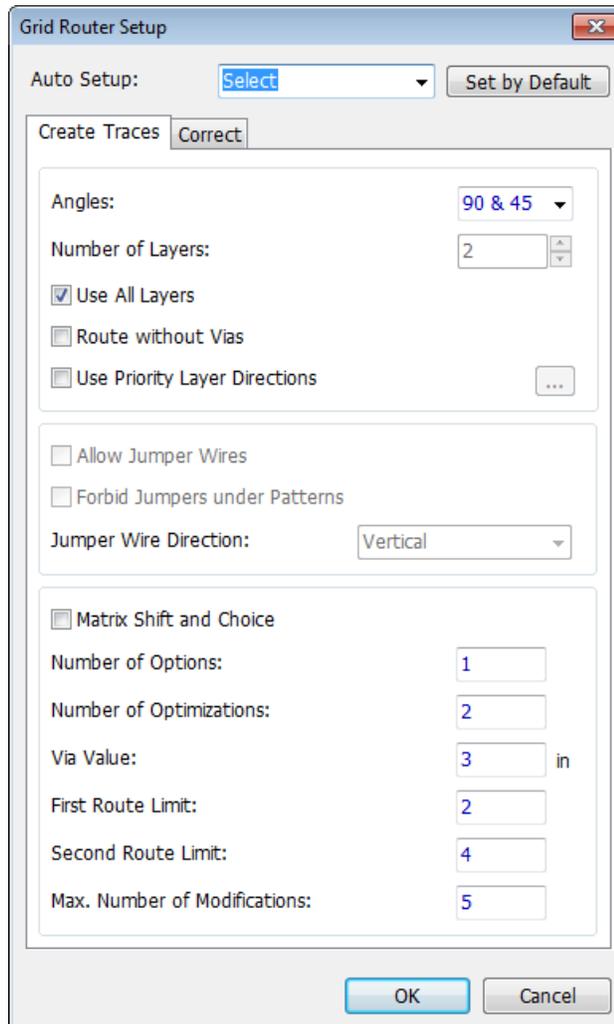


Please place route keepout around the board cutout. This will allow for the clearance between the copper and the cutout. Since we plan to have copper traces only on the bottom layer, select it by pressing "2" hotkey. Then select Route Keepout layer in the drop-down list on the drawing toolbar and draw a shape on the bottom layer of the board. Create a rectangle which is a bit bigger than the cutout, like on the picture to the left. Change the grid size to 0.025 in for comfortable drawing. Then switch back to the top layer (press "1" hotkey).

## 2.3 Autorouting

Now it is time to route the circuit board. PCB Creator has a Grid Router with jumper wires support. Most of the time, a simple PCB can be routed on a single layer (usually a bottom side). This presents many benefits, such as efficiency and speed of having a prototype. Traces might be a bit longer than on the two-sided board, but it's OK for simple designs.

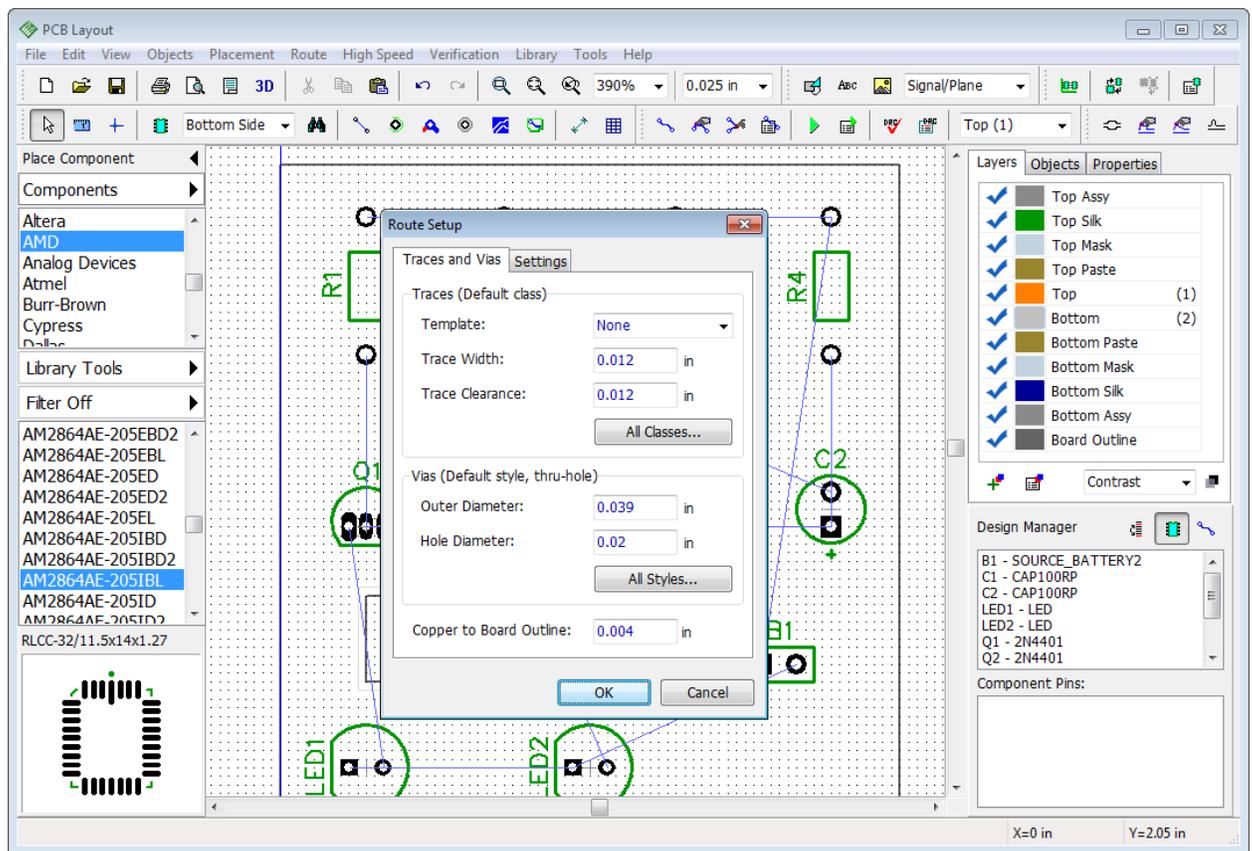
First you need to setup the Grid Router, go to "Route / Autorouter Setup".



This dialog box allows the user to customize Grid Router, define layers for routing, trace directions on layers, via placement options, etc. Check Use Priority Layer Directions box and then press "..." button to set up each layer (PCB Creator allows for up to 4 signal layers. If you need more layers or want to use advanced shape-based autorouter, then purchase [DipTrace Extended or Full editions](#)). Spend some time playing with different options. For less experienced users we recommend to use ready presets from the Auto Setup drop-down list.

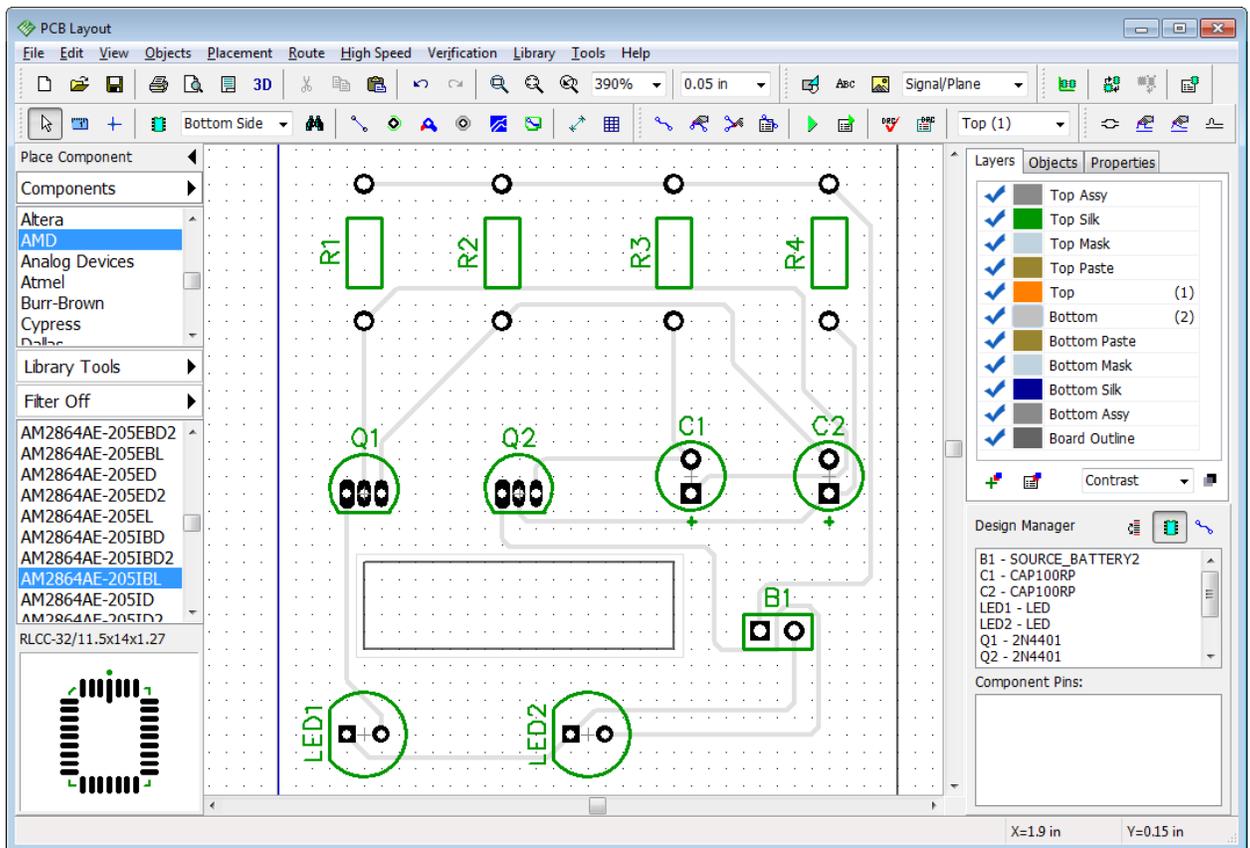
Press OK to accept settings and close the setup dialog box. Then select "Route \ Route Setup" from the main menu. In the pop-up dialog box you can change trace width and clearance between traces for default Net Class, and the diameter of vias for default Via Style. You can

press All Classes... and All Styles... buttons to access Net Classes and Via Styles panels respectively.



We recommend to use 0.012 inch traces for this project. Press OK to close this dialog box and apply changes.

Now it's time to route the circuit board. Select "Route / Run Autorouter" from the main menu. Board will be routed and you'll get something like on the picture below. Your layout doesn't have to be exactly like this one, so don't be confused if you are new to electronic engineering and some routes doesn't coincide with the picture. As you remember trace color depends on a layer color.



DRC (Design Rule Check) runs automatically after autorouting. If there are some errors, it will show them (red and magenta circles). Design Rule Check in the latest PCB Creator version operates in offline and online modes. Online DRC is turned OFF by default, but if it is ON, you've probably noticed some red circles while moving components.

This project is very simple and you should not get any error reports, still if there are some errors, correct the layout and rerun DRC by selecting "Verification \ Check Design Rules" from the main menu or pressing the corresponding button on the instruments toolbar. To change design rules select "Verification \ Design Rules" from the main menu. To hide the error circles, select "Verification \ Hide Errors". You can disable automatic DRC after autorouting.

Do not forget to uncheck Use Priority Layer Directions box in the Autorouter Setup dialog box if you plan to route 2+ layer boards.

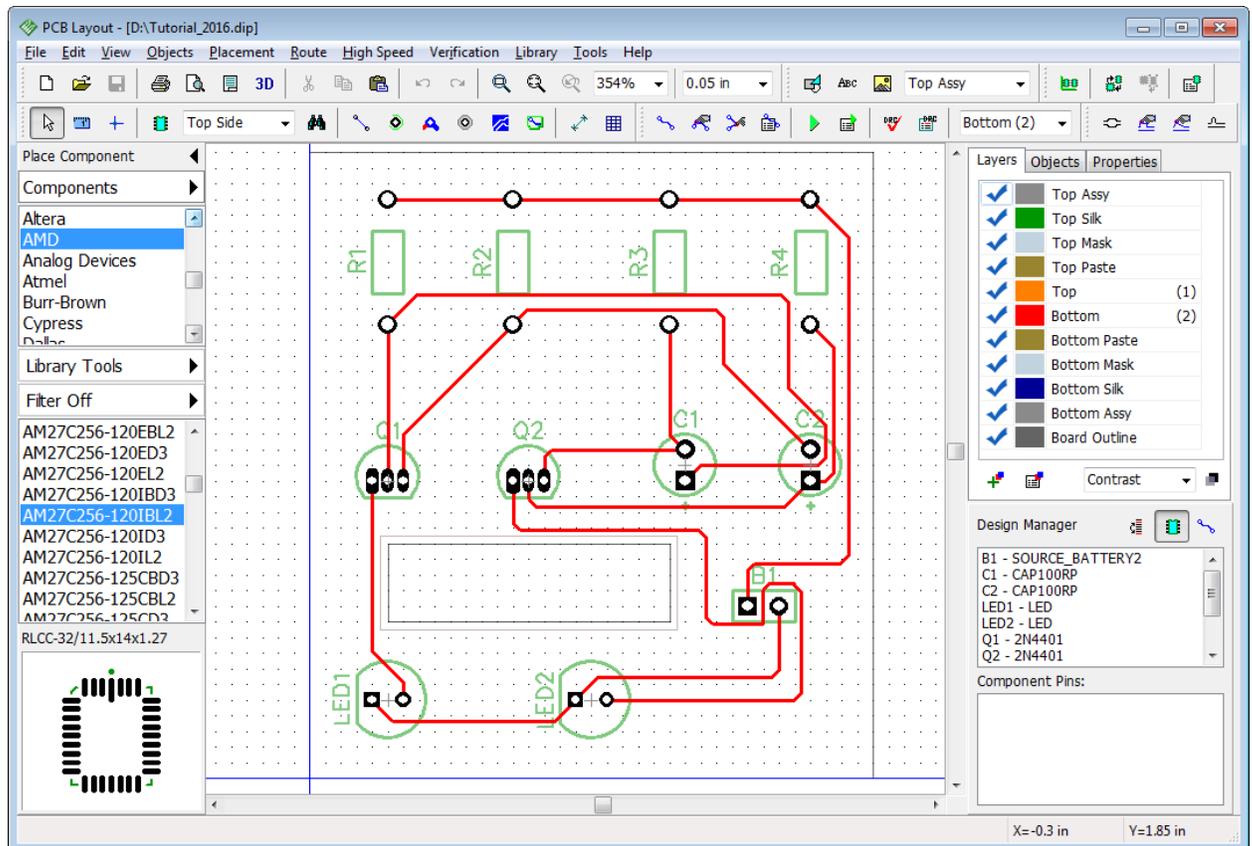
Notice that if you want to finish your project faster, you can skip all topics through ["Printing"](#)<sup>87</sup>, because your PCB is actually ready to output. But if you want to learn more, we recommend not to skip it.

## 2.4 Working with layers

Traces you see right after autorouting are in low contrast. This means that they are not in the top layer of the board. In our case it is the Bottom layer and Contrast layer display mode is active (Top layer and 50% opacity contrast mode are default settings). Look at the Layers tab on the Design Manager (press "Ctrl+3" hotkeys if the Design manager panel is hidden). If you want to

change the active layer, double click it in the list or press corresponding hotkey (they are noted in the brackets next to the name of each layer), or you can use "T" and "B" keys for top and bottom layers respectively. It's also possible to change active layer in the list box near the DRC control buttons, just find what way is the most comfortable for you.

We double click the Bottom layer in the list to activate it. Make sure you click on the layer name (text) in the list, not on the blue check mark or elsewhere. Grey color is quite dim itself, click on the colored rectangle right next to the Bottom layer and select red color in the pop-up dialog box. Press OK to change the bottom layer color to red.



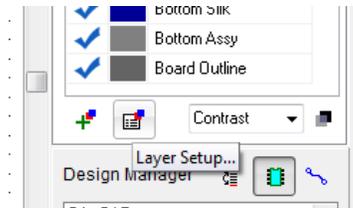
All layers can be divided by two basic types: Signal layers and Non-Signal layers. PCB Creator user can easily add, delete, and edit both. Our project is a simple circuit board with two signal layers: Top and Bottom. But as you can see in the list, there are much more of them. Assy, Silk, Paste, Mask are non-signal layers. PCB Creator creates them automatically on both sides of the board (and gives corresponding names to each of them depending on their side of the circuit board – Top Silk, Bottom Paste etc.). Each layer carries a special type of information.

Top / Bottom Silk are silkscreen layers, all texts and graphical information are automatically added there. Top / Bottom Mask / Paste layers carry information about solder mask and solder paste. Some non-signal layers may be necessary for correct board manufacturing, the others provide additional functionality. If you have any questions, contact [Bay Area Circuits](#).

### Signal layers

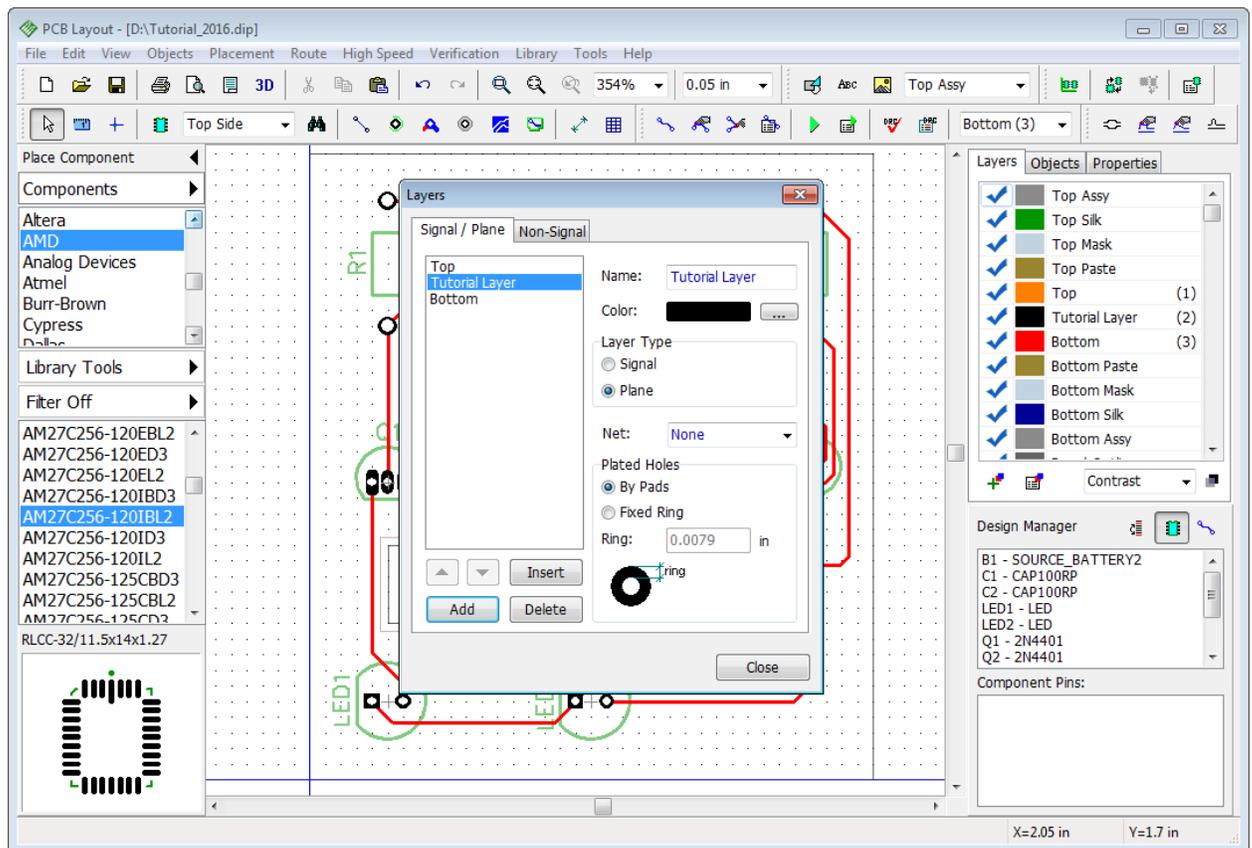
Traces and copper pours can be created only in signal layers. There are two types of signal

layers: Signal and Plane. Signal layers usually contain traces and sometimes copper pours, while Plane layers are inner (inside the board), they contain one or several copper pours. Autorouter can create traces only on signal layers.



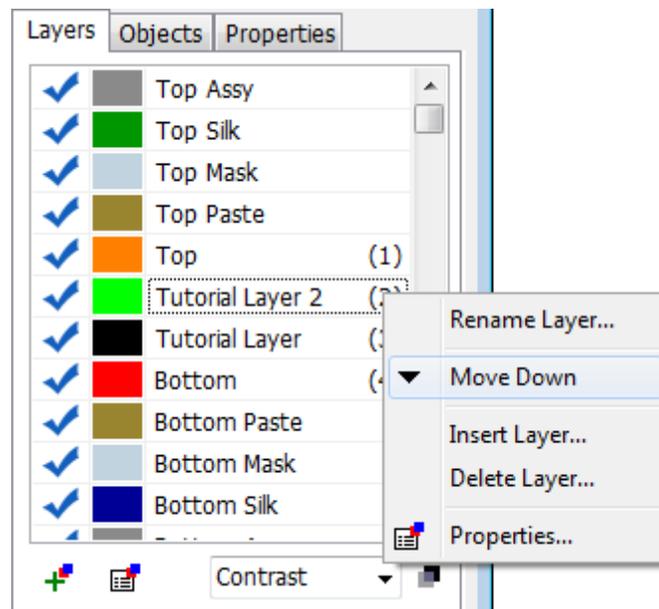
If you want to add, edit, create, or delete layer, go to "Route \ Layer Setup" or press  button in the Layers tab on the Design Manager. In "Signal / Plane" tab of the pop-up dialog box you can specify the name, type, color etc. of each signal or plane layer. Notice that you can't change some parameters for certain layers.

We will add a new plane layer called "Tutorial Layer" just to show you how it works. Press Add button in the Layers Setup dialog box, then enter the layer's name, select type and color. Plane layers can be connected to one of the project nets, usually it is Ground or Power, in our case it is unconnected. You can also specify details of plated holes by pads or choose a fixed ring and set its size. All pads on inner layers are round. Press OK to create a new layer.

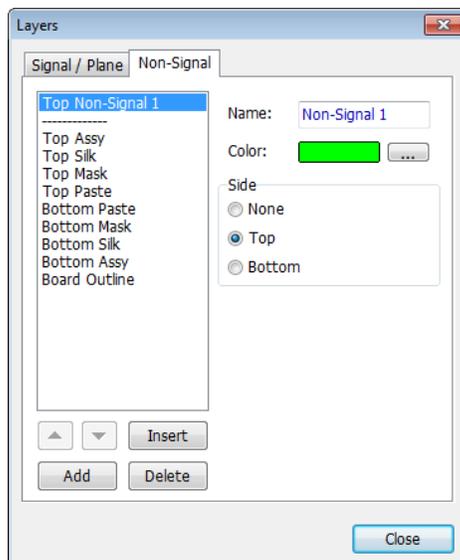


Press the Close button to close the Layers dialog box, as you can see, the "Tutorial Layer" is already on the layers panel, between Top and Bottom layers.

Create one more **plane** layer, because board should always have even number of layers. Right click it in the list and select Move up or Move down in the submenu to change the board stackup. Click on the color rectangle to quickly change the color of the corresponding layer.

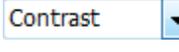


### Non-Signal layers



Customizable non-signal layers are very convenient, they can be used for various engineering purposes. They improve speed and total convenience of electronic design. If you need to create a non-signal layer select Non Signal tab in the Layer Setup dialog box ("Route \ Layer Setup" from main menu). Press Add button, enter the layer name, select color and layer side: None, Top or Bottom. "None" means that layer will not be locked to some specific side of the board.

We do not need any custom non-signal layers. Close this dialog box.

There are some quick-access buttons on the Layers tab of the Design Manager:  – Add Layer,  – Layer Setup,  – layer display mode, and contrast level setup – .

Remember to use 1 through 4 keyboard shortcuts to quickly change an active layer.

Notice that in the PCB Creator you see the bottom layer of the circuit board like if it was transparent. Pick "View \ Mirror" from the main menu to mirror entire circuit board. Now you see how the bottom layer actually looks like.

Delete non-signal layer, if you have created one, because it's not needed for this project. But do not delete custom plane layers, we'll use them later for practicing with vias. Save the design.

## 2.5 Working with vias

PCB Creator supports through and blind/buried vias (if divided by physical properties). Vias are also divided by two logic types that don't depend on their physical properties:

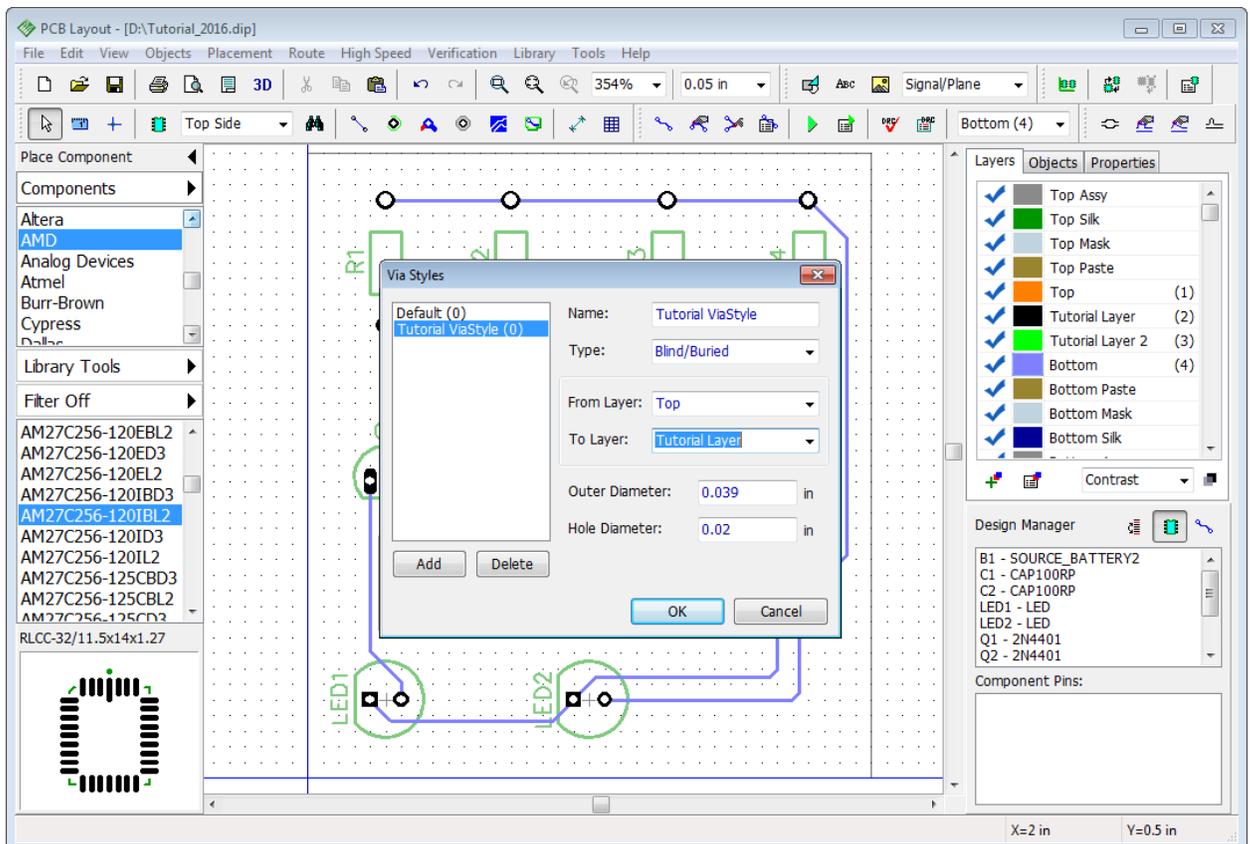
**Trace Vias** (regular vias), which are technically parts of traces and appear automatically when you move a trace segment to another layer;

**Static Vias** (similar to pads), which are placed manually, they have much more variable properties than trace vias.

*All vias in the PCB Creator, disregarding their logical type, are organized to Via Styles.*

We don't need a lot of different via styles for current project, but we will add some in order to teach you the basic principles of working with them.

Let's create two additional via styles: one with blind/buried vias of the same size as the Default via style (0.039 inch diameter and 0.02 inch hole), and the other via style for through vias of bigger diameter. Go to "Route \ Via styles" and check if Default via style has aforementioned parameters. Change them if you need. Then press Add button to add a new via style. It will appear under the Default one. Left click it and type in the name, change via type to Blind/Buried, and specify the layers involved (top and bottom layer of the via). In our case we make blind vias from Top layer to Tutorial Layer. Specify via properties like on the picture below. Blind vias are impossible on printed circuit boards with only two layers, that is why we did not delete tutorial layers from the previous topic of this tutorial.



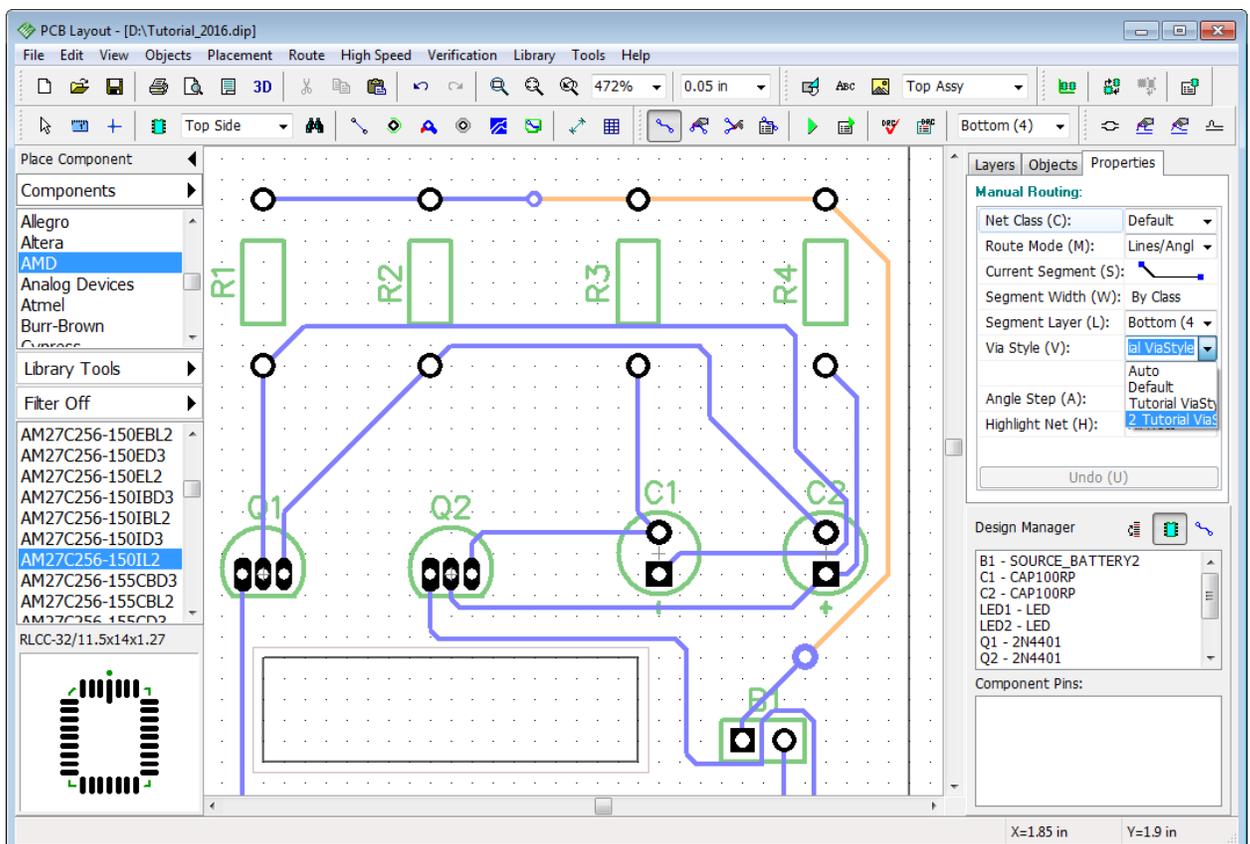
Now add one more through via style called "2 Tutorial ViaStyle" and enter 0.065 inch outer and 0.03 inch hole diameters. Press OK.  
Notice that we've changed the bottom layer color to violet.

### Trace vias

Now unroute one of project nets (we will route it manually). We have chosen the net that connects resistors' pads with a battery.

Return to the Bottom layer then right click on the net that you want to unroute and select Unroute Net from the submenu. Go to "Route \ Manual routing \ Add Trace" from the main menu or press "~" hotkey. Left click on the first pad (R1:2), then on the second pad (R2:2) to create a trace. Now left click on R2:2 again, but do not create a trace all the way to R3 resistor. Just route it to some point between the R2 and R3, left click to set a trace segment between the pads, then right click and select "Segment Layer \ Top" from the drop-down submenu (if you're routing on the bottom layer and vice versa). Trace via will appear automatically, now we can continue routing on the opposite side of the board to another pad and then left click it. Create one more segment between the R3 and R4 components. Notice that trace color is defined by the layer color. Do not route entire net all the way to the battery.

Start creating the trace heading from the R4 resistor. Select "2 Tutorial ViaStyle" from the Via Style drop-down on the Manual Routing panel on the right side of the screen to apply selected via style to the vias being created. "Auto" means that PCB Creator will use a via style that takes less space on the board. But in our case we select the style with a bigger vias (2 Tutorial Via Style). Then create a trace via (go to the opposite board side) and finish the trace to the B1 component.

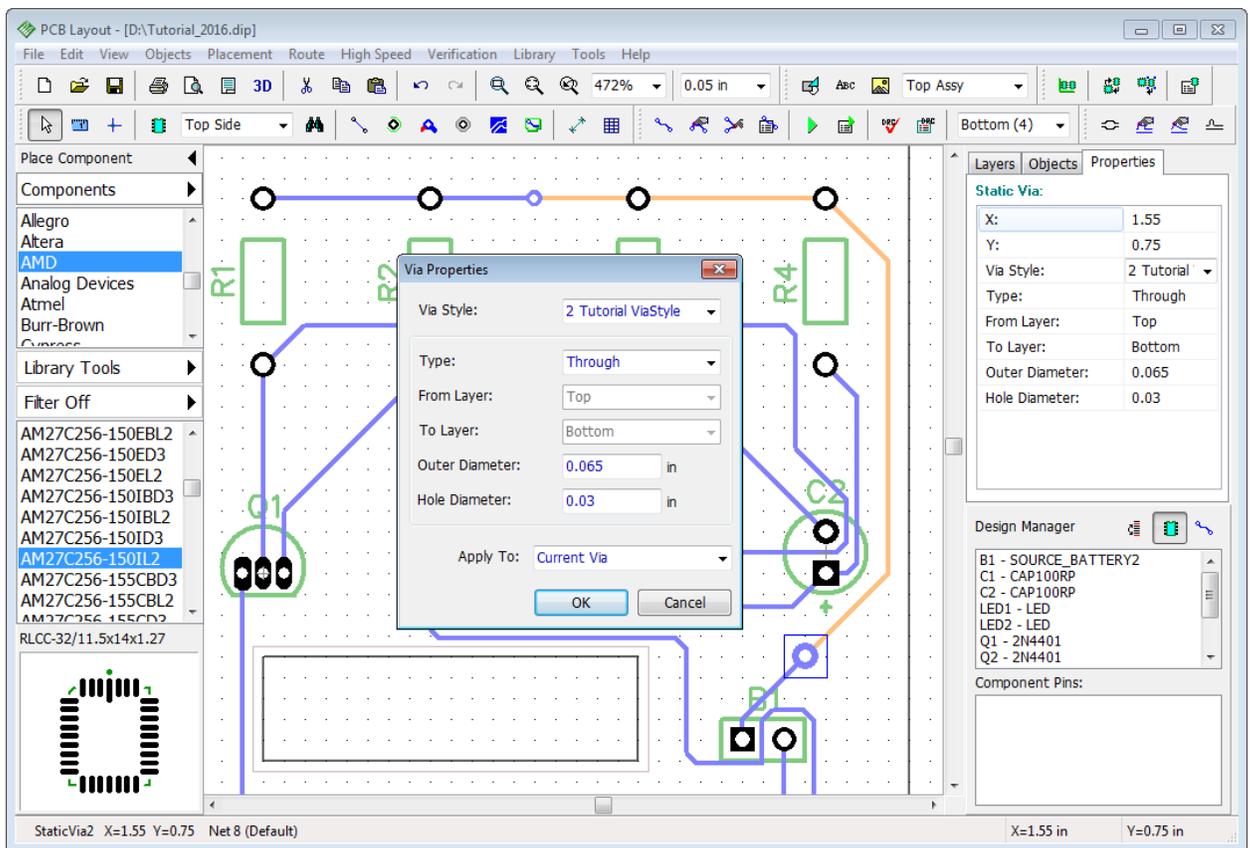


## Static vias

Press  button to place a new static via on the design area or you can make it directly from the trace via, just right click on it and select Convert Via to Static. Then specify which vias to convert: Current via, Selected segments etc. Static vias behave almost like pads.

*If you change the parameters of some via style, all vias of that style, even those on the design area, will change automatically.*

We can change style, type, diameter of the static via and apply new settings to current or selected vias or nets at the via properties panel. Right click on one or several vias, and select Via Properties from the submenu, make some changes and press OK. If there is no via style with the parameters you've entered, PCB Creator will ask if you want to create a new via style.



Static vias can be converted back to trace vias. Right click on the static via and select Convert to Trace Via from the submenu and select which vias to convert. If you've placed a static via directly on the design area, you can not convert it back to trace via. Delete all vias and route the trace on a single layer. But do not use Undo feature, because this will delete custom via styles.

## 2.6 Net Classes

All nets in PCB Creator are organized to Net Classes. This feature allows the user to apply certain parameters to any net/s with nearly a one click. Net Classes can be used while routing

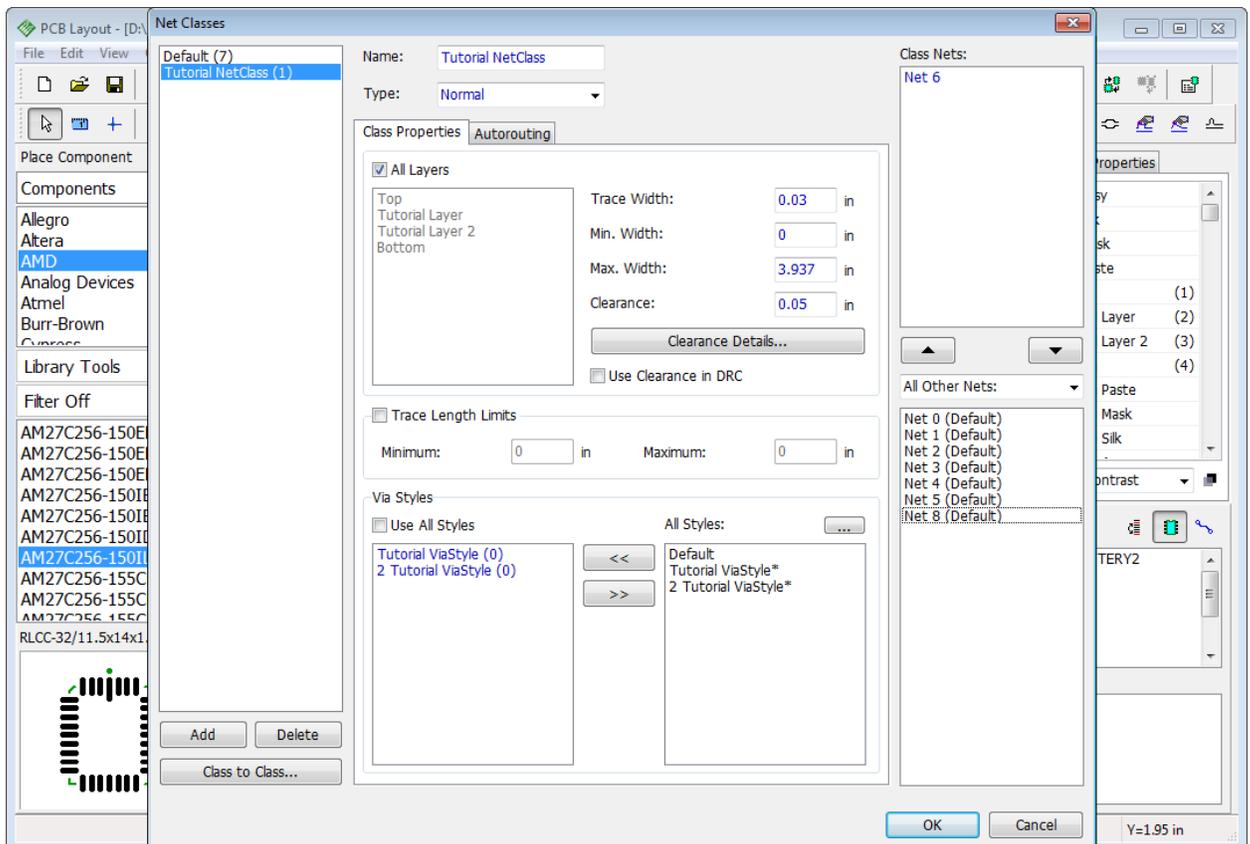
the board manually or automatically (Autorouter). Parameters of net classes should be specified **before** routing.

We are going to practice in working with net classes using the same project, therefore we need to completely unroute it first. Go to "Route \ Unroute All" from main menu. Then select "Route \ Net Classes" to open Net Classes panel. In the pop-up dialog box you can see that only the Default net class is available and all nets belong to this class. Press Add button and a new net class will appear in the list of all net classes, right under the Default. Left click it and type in the name.

In Class Properties tab specify trace parameters and clearance between the traces. In our case we will make traces of a new net class significantly larger (0.03 in) with 0.05 clearance.

*Notice that "\*" symbol in the input field means that it is a default parameter.*

If you uncheck All Layers, the list under that checkbox will become active, allowing different trace parameters on different layers of the board. We don't need this now. Uncheck Use All Styles in the Via Styles section and choose which via styles will be used in this net class. Just press "<<" and ">>" buttons to add or delete via styles to/from the list of active. The "..." button allows the user to preview parameters of each via style. We have allowed only custom via styles for this net class (see the picture below). New net class exists, but it does not have any sense if no nets belong to it. So we're going to add some. In the lower-right of the Net Classes dialog box you can see the list of all project nets and the name of the net class of each net in the brackets. In our case it is Default net class. Select one or several nets with "Ctrl" key and press arrow up to add them to the net class (Class Nets list right above).

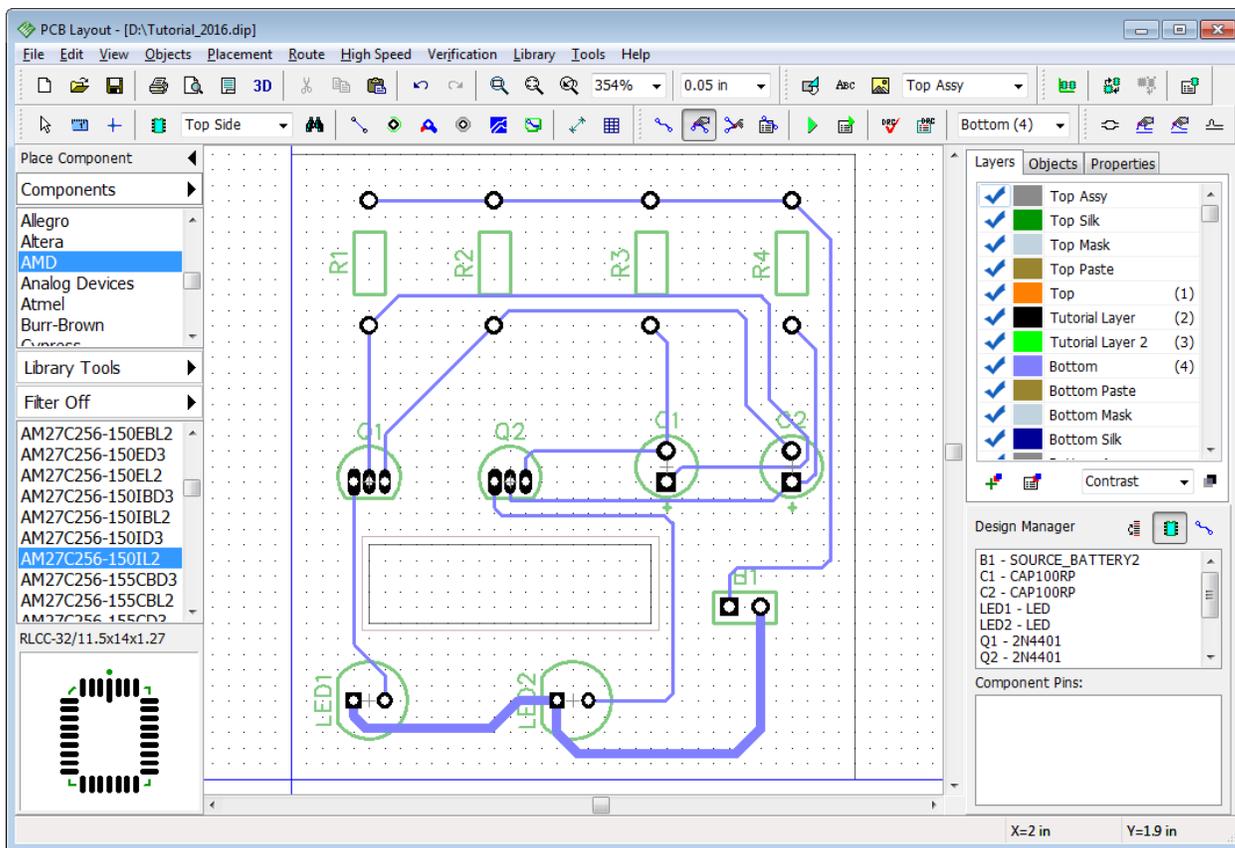


As you can see Tutorial NetClass contains one net (Net 6 in our case) with .03 inch trace width.

The Clearance Details button allows you to set clearances between different objects. The Class to Class – specifies clearance between the nets of different net classes. Class to class clearance is used by DRC and has priority over regular net class clearances. Make sure Use Clearance in DRC item is unchecked and press OK button to close the panel and save changes.

### Autorouting with Net Classes

Now you have two different net classes, one net belongs to Tutorial Net Class and the rest – to Default. It's time to route the board with autorouter, select "Route \ Run Autorouter" from main menu or press "Ctrl+F9" hotkeys and you'll get something like on the picture below. As you can see, the traces on the PCB have different width, because they belong to different net classes with different parameters.



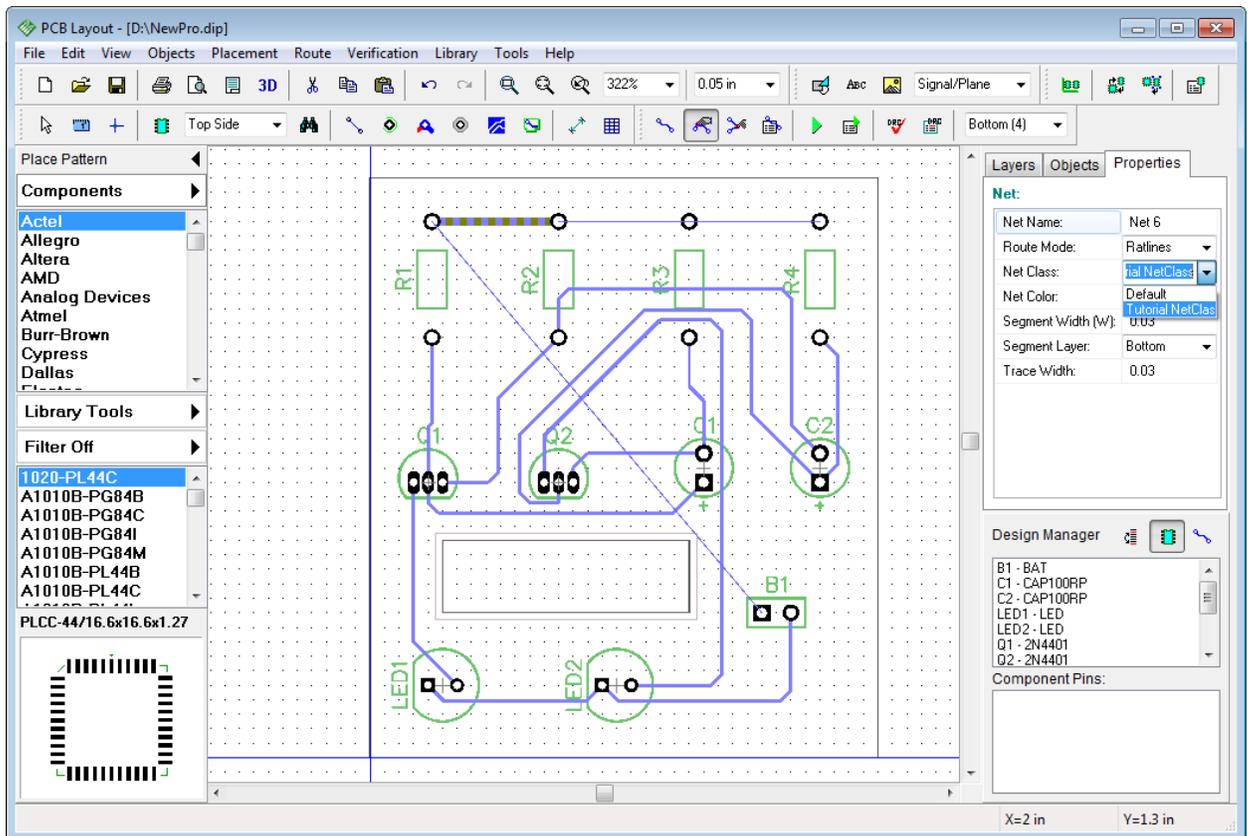
If everything is clear, unroute the board again ("Route \ Unroute All"), open Net Classes dialog box and reassign Net 6 from Tutorial NetClass to Default class (use down arrow button below the Class Nets list). Press OK then launch autorouter again and you'll get the circuit board with all traces of the same width. Tutorial NetClass still exists but it doesn't do anything, because no nets belong to it.

### Manual routing with Net Classes

Select the Bottom layer and left click on one of the nets (for example, Net 8 between the resistors and a battery), you'll see the Net Properties panel on the Design Manager to your right-

hand side. In the Net Class drop-down list select Tutorial Net Class. Then right click on the same net and select Unroute Net from the submenu. Now press the tilde hotkey "~" to activate manual routing mode. Left click on the first pad (R1:2) and create a trace to another pad (R2:2) and left click it to create a trace segment. You'll notice that the trace is much wider because it is in another net class.

*Notice that PCB Creator allows for changing net class of the routed net, but in order to apply changes net has to be unrouted and routed again.*



We don't need that trace width diversity on the board. Please **Undo ("Ctrl+Z")** several times or manually delete all custom net classes, via styles, and inner layers to get the layout right after [autorouting](#)<sup>42</sup>.

Save the project ("File \ Save" from the main menu).

## 2.7 Manual Routing

Simple projects like ours can be routed automatically, but for more complex boards manual routing is inevitable. A combination of two methods is the best choice. This allows the designer not only to get a well-working prototype, but to succeed in reasonable terms. Critical nets are routed manually and the rest – with autorouter.

Our simple board is already good enough, but we're not done with practicing yet. Moreover, sometimes you may need to correct traces after the autorouter.

## Editing modes

You should already know how to create traces – select "Route \ Manual Routing \ Add Trace" from main menu or press  button, then left click on the first pad to start routing, and click on the next pad to create a trace. Just make sure that correct layer (Bottom) is selected.

Editing traces is a bit different. Go to "Route \ Manual Routing \ Edit Traces" from the main menu () or simply left click on the trace (Edit Traces mode will be activated automatically), and drag & drop trace to another location.

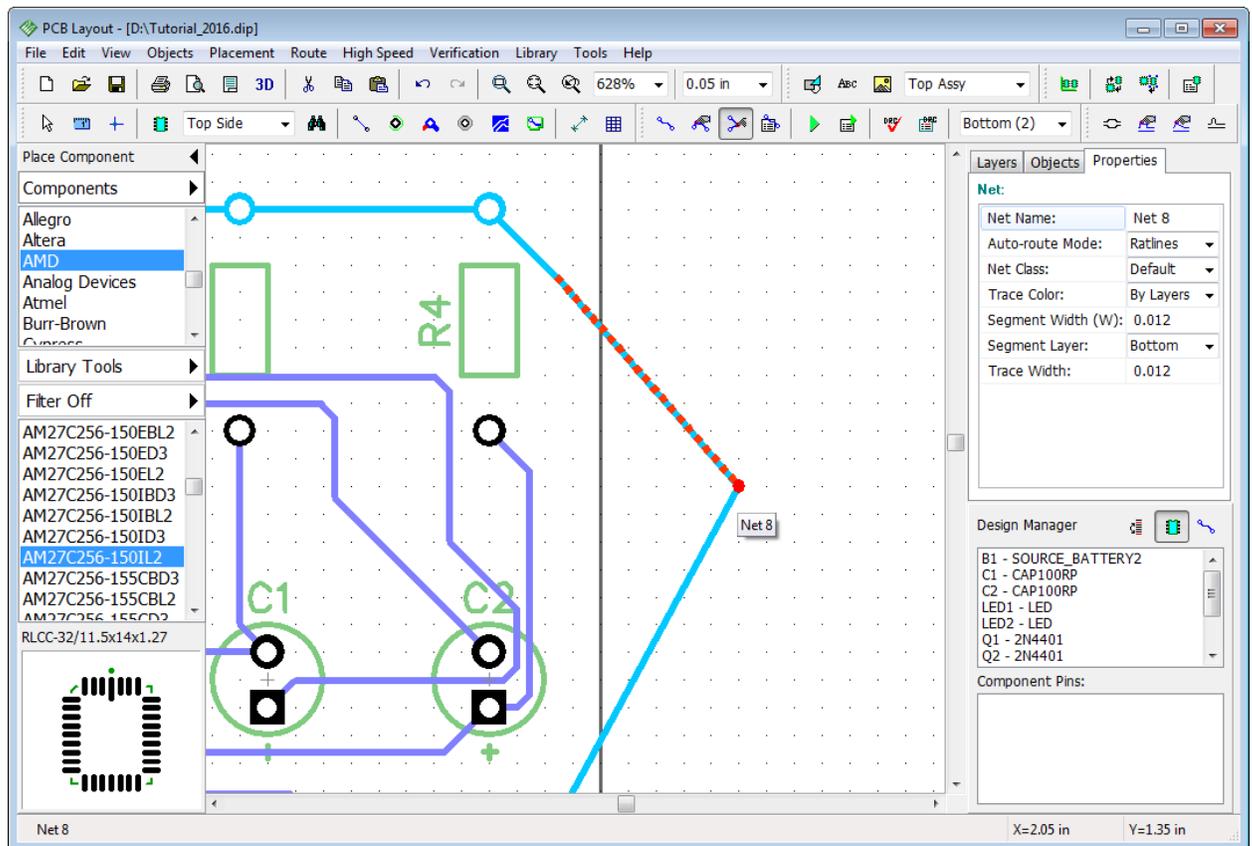
Edit Traces mode allows the user to move traces with 45 or 90 degree angles. This is very convenient for almost any designs, but sometimes you might need trace editing tool with more capabilities. Go to "Route \ Manual Routing \ Free Edit Trace" or press  button on the route toolbar. Now you can edit traces without any restrictions.

Don't forget to change the grid size (with the button on the standard toolbar or "Ctrl+plus sign" and "Ctrl+ minus sign" hotkeys). To configure the list of available grids, select "View \ Customize Grid" from main menu. "F11" hotkey hides / shows the grid.

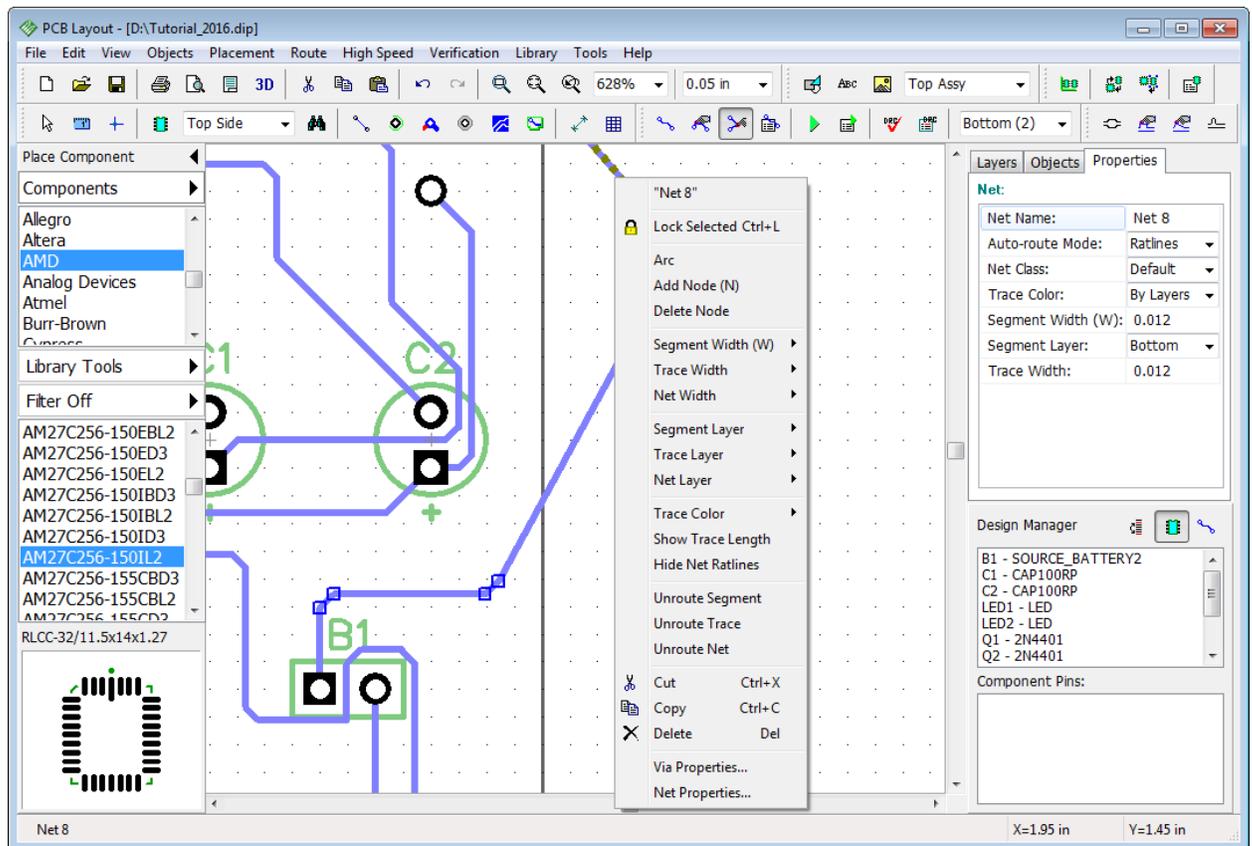
*Remember, if you don't know which tool you are working with, right click a couple times on a free spot of the design area and the PCB Creator will return to the default mode.*

## Track Nodes

Any routed net is divided to traces (often called "tracks"). Trace is the copper track between two pads of the net. Trace (track) consists of segments. Segment is the route between two nodes. Node is a point on the route which divides a trace to segments (red dot and a small square on the picture below). Designer can move existing nodes, add new ones or delete them. This gives more opportunities while editing traces. Left click on the trace segment and press "N" hotkey to add a new node in selected place, then select it and move to some point outside the board outline (Free Edit Mode on the picture below).



If you don't need some node any more you can delete it – right click on the node and choose Delete Node. In the same submenu you can change the name, color, width and the layer of the net, trace or segment etc.

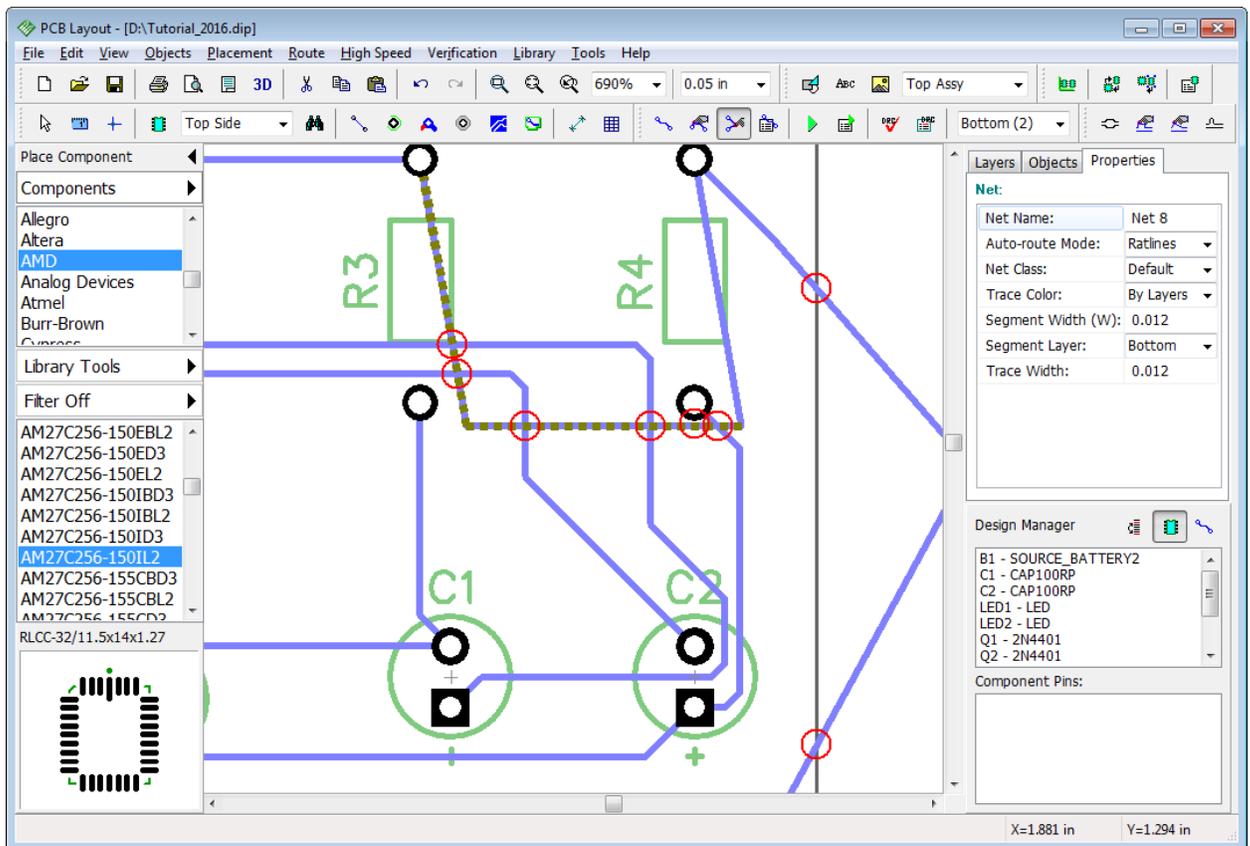


Manual routing offers greater opportunities and more chances to commit an error. Fortunately, the PCB Creator has Real-time Design Rules Check which verifies board in real time and enables user to see errors **before** actually making them. We will have a detailed discussion about DRC [later](#)<sup>78</sup>.

### Real-time DRC

Let's turn ON Real-Time DRC and continue. Go to "Verification \ Design Rules", in the pop-up dialog box check Enable Real-time DRC then select Real-time DRC tab and check Manual Routing and Moving Objects options (if not already selected). Press OK to close the Design Rules dialog box. You can see that two red circles have appeared at the points where the trace crosses the board outline, these circles shows errors.

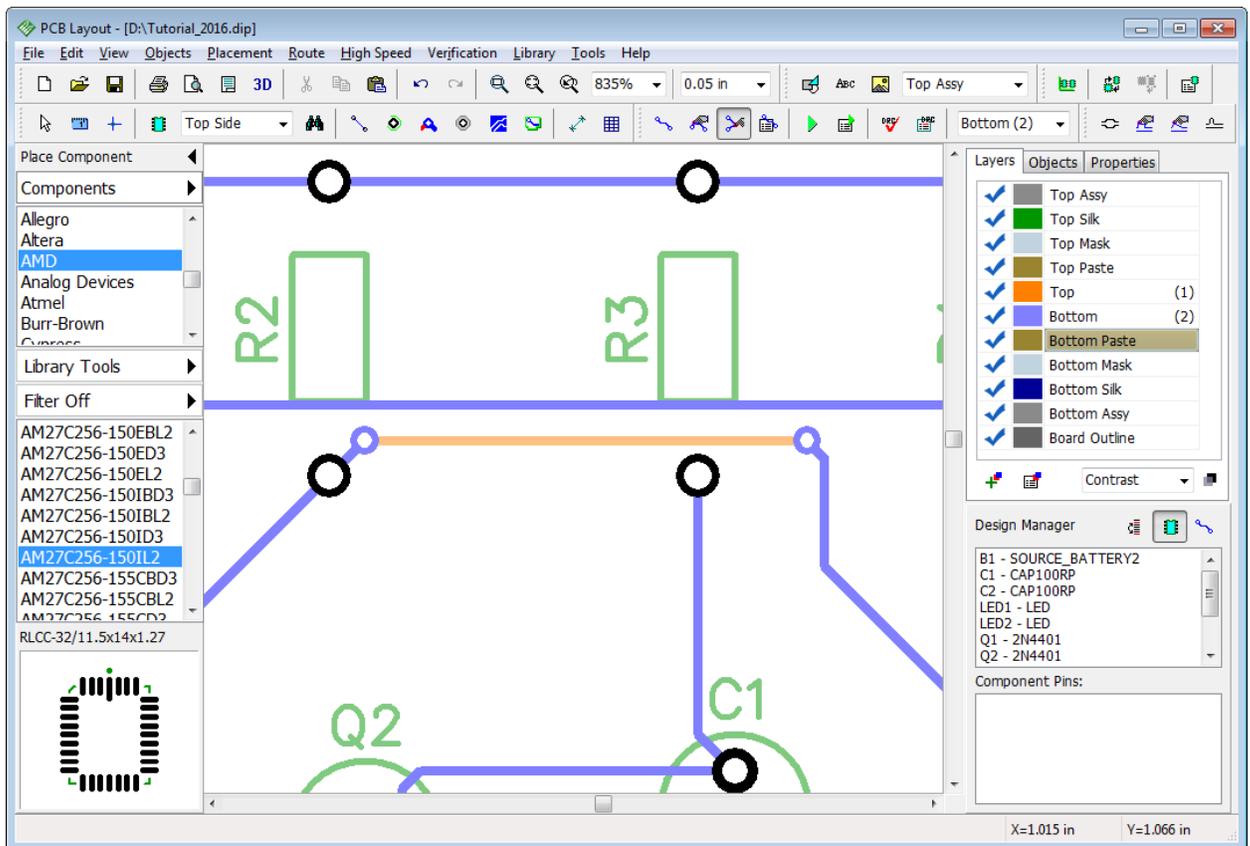
Now select a random trace and intentionally move it too close to another trace or object. Red circles report a clearance error **before** you place the trace to a new position, the same happens when you move objects or edit components if corresponding items in the Real-time DRC tab have been checked.



Notice that we have hidden the grid on the picture above ("F11" hotkey) for more precise editing. Now return the trace (between R3 and R4) to its initial position.

### Change layer

PCB Creator allows you to move existing net (trace or trace segment) to another layer. Right click on the trace segment of the net and select "Segment Layer \ Top" or use Segment Layer drop-down list in the Net Properties panel to your right-hand side. Two trace vias will be created automatically. You can choose several segments of the same or different nets with "Ctrl" or "Shift" buttons and change their properties at a time.



Now return that segment back to the Bottom layer and select the bottom layer on the Design Manager.

### Manual routing panel

It's time to practice more in manual routing. Right click on one of the nets and select Unroute Net from the submenu. We have selected Net 8, but you can choose another one. The Unroute Net command from net submenu applies to all selected nets. Then press  button on the route toolbar.

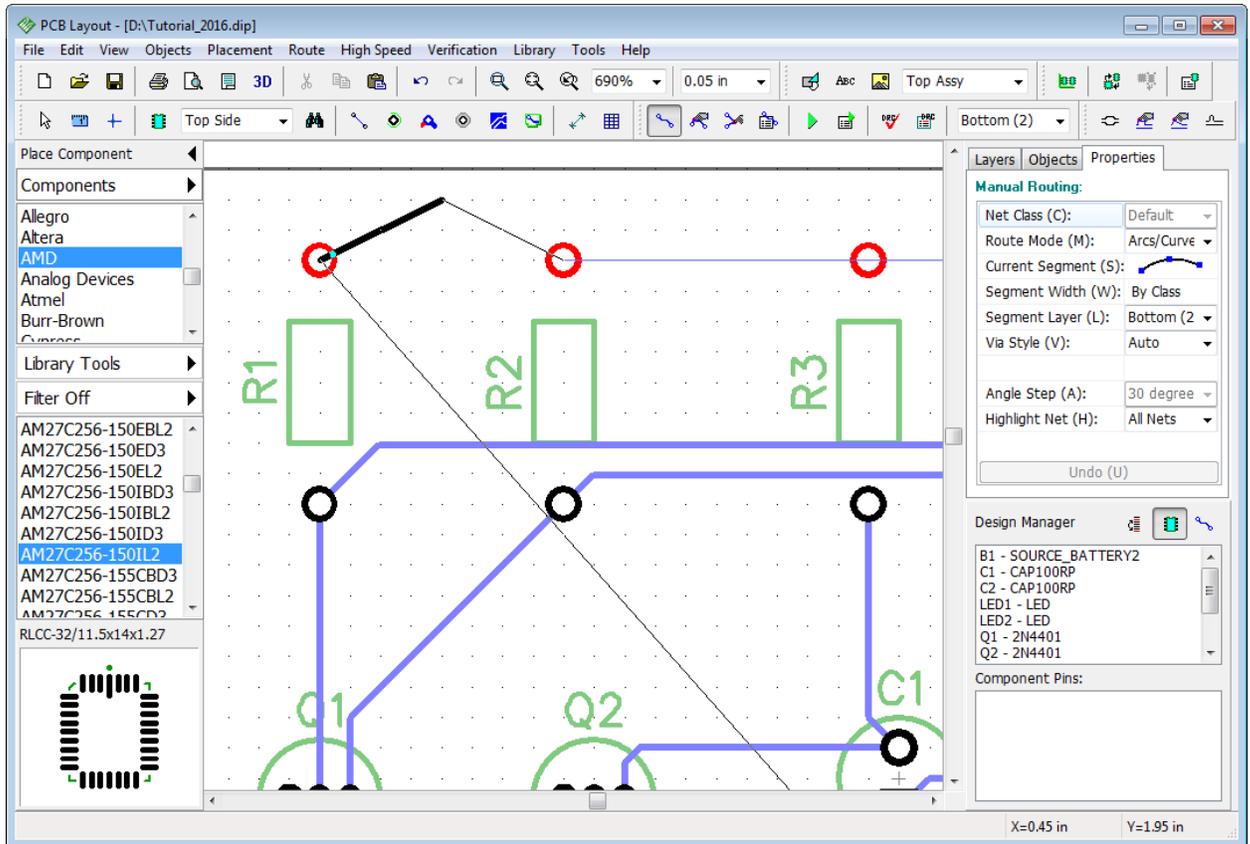
Manual Routing panel is on your right-hand side (Design Manager panel). Do not try to change the net class of existing net in there. Net Class should be defined in the Net Properties or Net Class dialog box **before** routing.

*Notice that you cannot change the net class of existing net on the Manual Routing Panel. This change will be ignored and applied only to the new net. Don't forget that net exists not depending on whether it is routed or unrouted.*

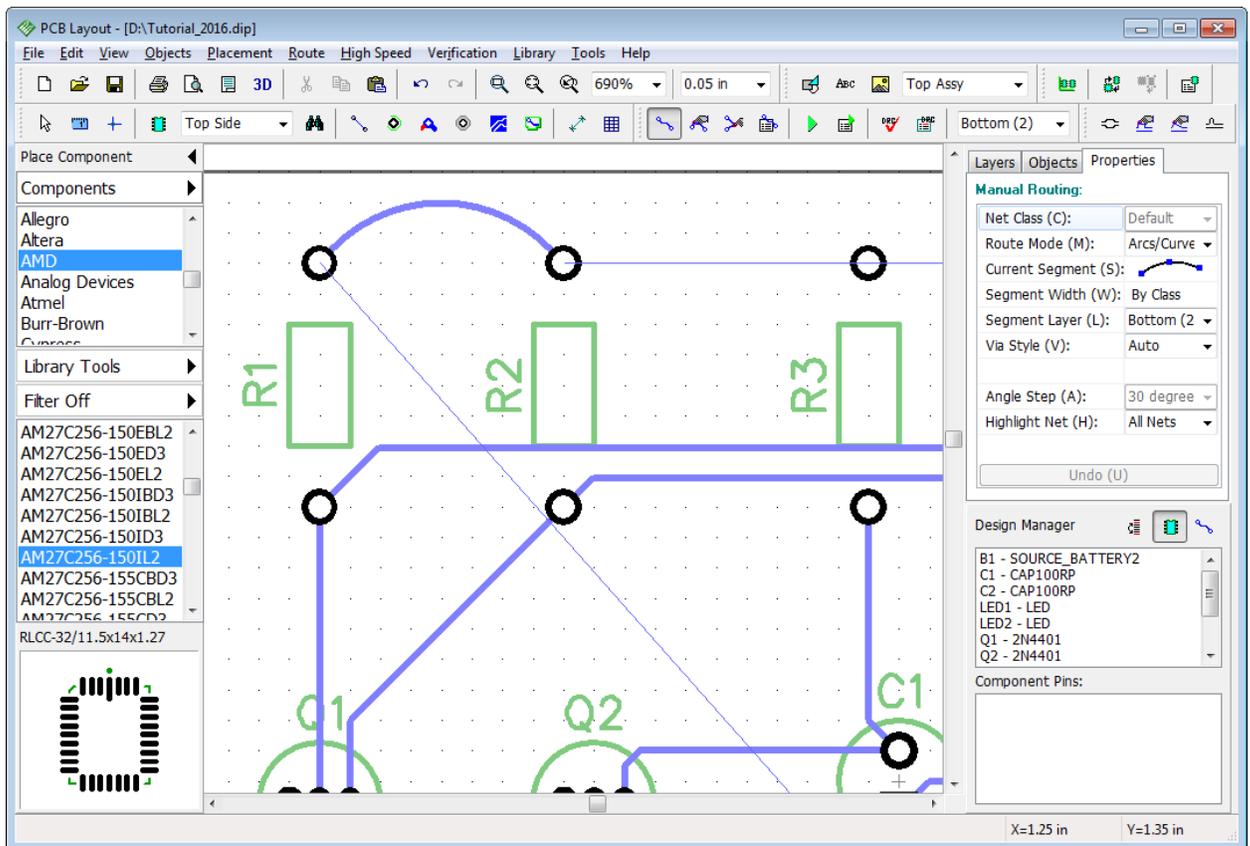
Fortunately, we have only one net class to avoid confusion and concentrate on the subject.

In the Route Mode drop-down list (on the Design Manager) we can specify the group of trace segments that we are going to need for routing, therefore we will be able to select current segment not from the list of all segments but from the list of segments of one mode. Custom route mode is available.

Select "Arcs/Curves" route mode then left click in the Current segment field and select "3-point Arc". Left click on one of the pads of unrouted net (this will be the first point of an arc), then left click on some point between the two pads higher than the blue connection line (this is the second point). Like on the picture below.



Then move mouse cursor to the second pad and left click it. You will see an arc.



On the Manual Routing panel you can choose highlighted nets. If you highlight only current net, no other nets will glow, even if you'll touch them with a new trace.

You can undo by pressing "U" hotkey while routing. Notice that there are hotkeys that will make manual routing really easy and quick.

"M" – switch between the routing modes.

"S" or "Space" – change current segment.

"W" – set trace width.

"T" – switch to Top layer.

"B" – switch to Bottom layer.

"L" – segment layer.

"J" – switch to jumper wires or back (if you are in Bottom layer, jumper will be placed to the Top and vice versa).

"A" – angle step.

"H" – highlight net.

"1" – "0" in the top of keyboard – switching between layers (up to 10).

Go to "Tools \ Hotkey Settings" from the main menu to view and change hotkeys. Refer to PCB

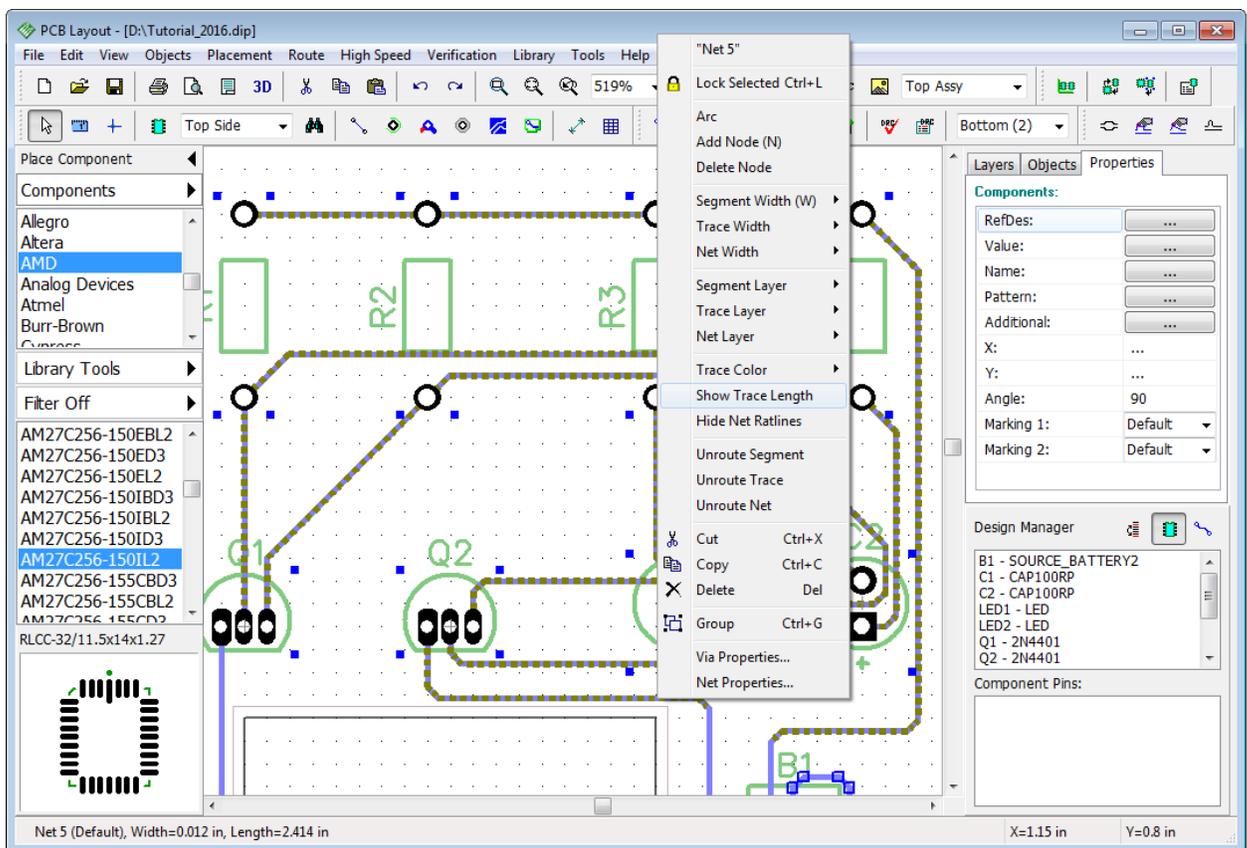
Layout Help document ("Help \ PCB Layout Help") for more details about custom keyboard shortcuts.

Now please **Undo** ("Ctrl+Z") several times or change the layout to the state like after autorouting (no Net Classes, Via Styles, new layers etc.).

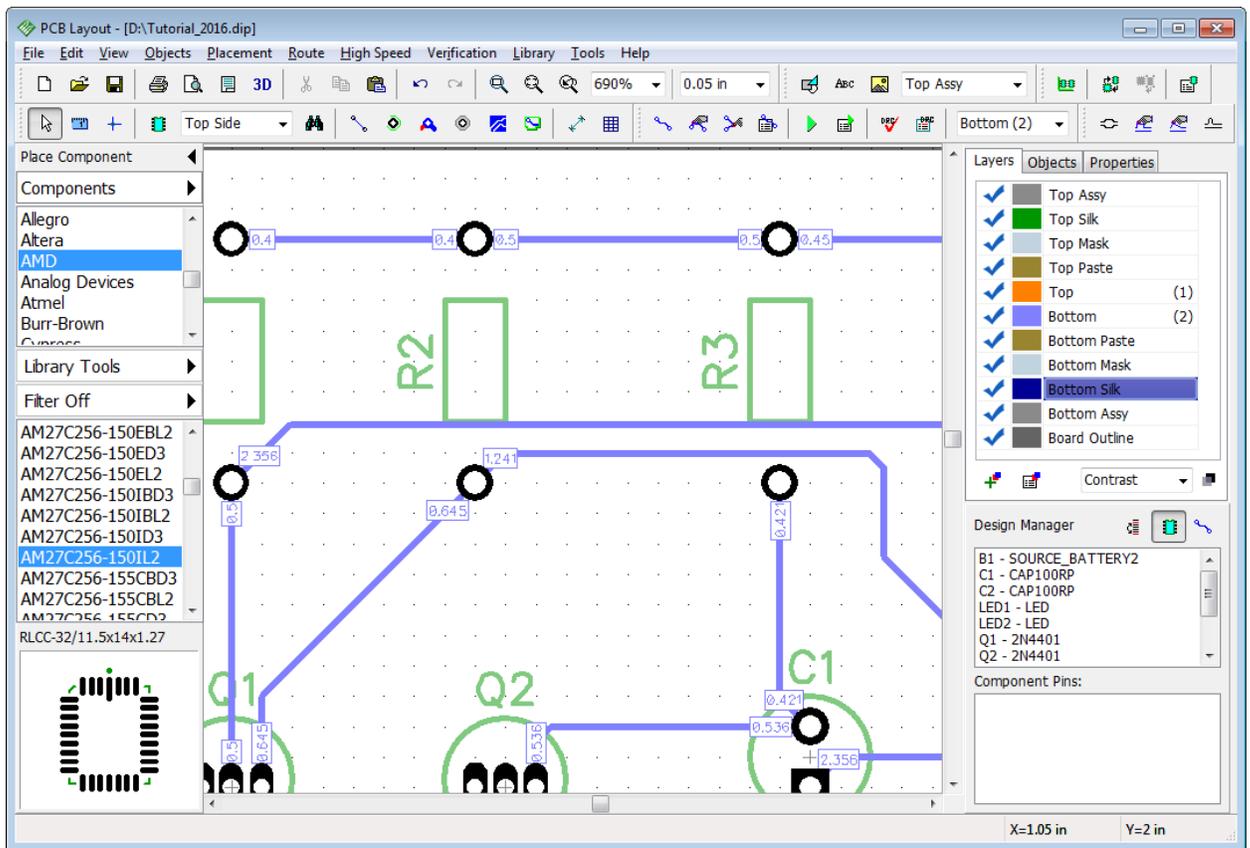
## 2.8 Measuring trace length

PCB Creator allows for easy and convenient trace length measuring. Current project is simple, therefore we don't need to use this tool, but if you design high-speed circuits with differential pair signaling, trace length becomes very important.

Please select several traces (you can use box selection or "Ctrl" button). Right click on one of selected traces and choose Show Trace Length from the submenu.



You will see the small boxes with trace length values near all pads of selected nets, they are also highlighted when hovering over the trace with the mouse cursor. Values are shown in current measurement units (inches in our case), they change in real-time when you edit the layout. Notice that in some situations you may be unsatisfied with current color. Trace color depends on the layer color.

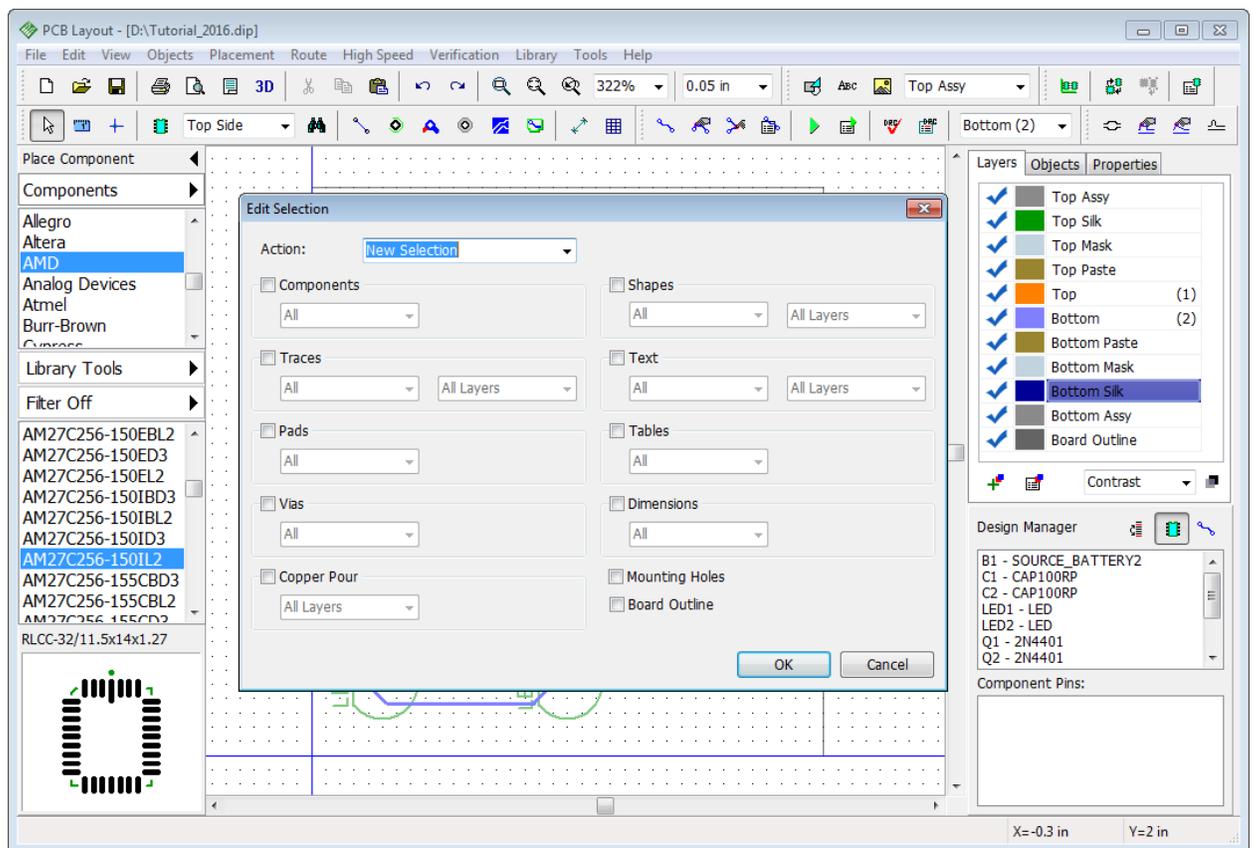


Now please hide the trace length boxes, using the net submenu (select the same item) or **Undo** tool.

## 2.9 Selecting objects by type / layer

Sometimes it is necessary to select all objects on one layer or exclusively components, nets, vias etc. With current layout it is very easy and can be achieved visually with mouse and "Ctrl" key. But what if the layout is very complex?

Select "Edit \ Edit Selection" from the main menu.



Check the Components item and click OK – all components will be selected.

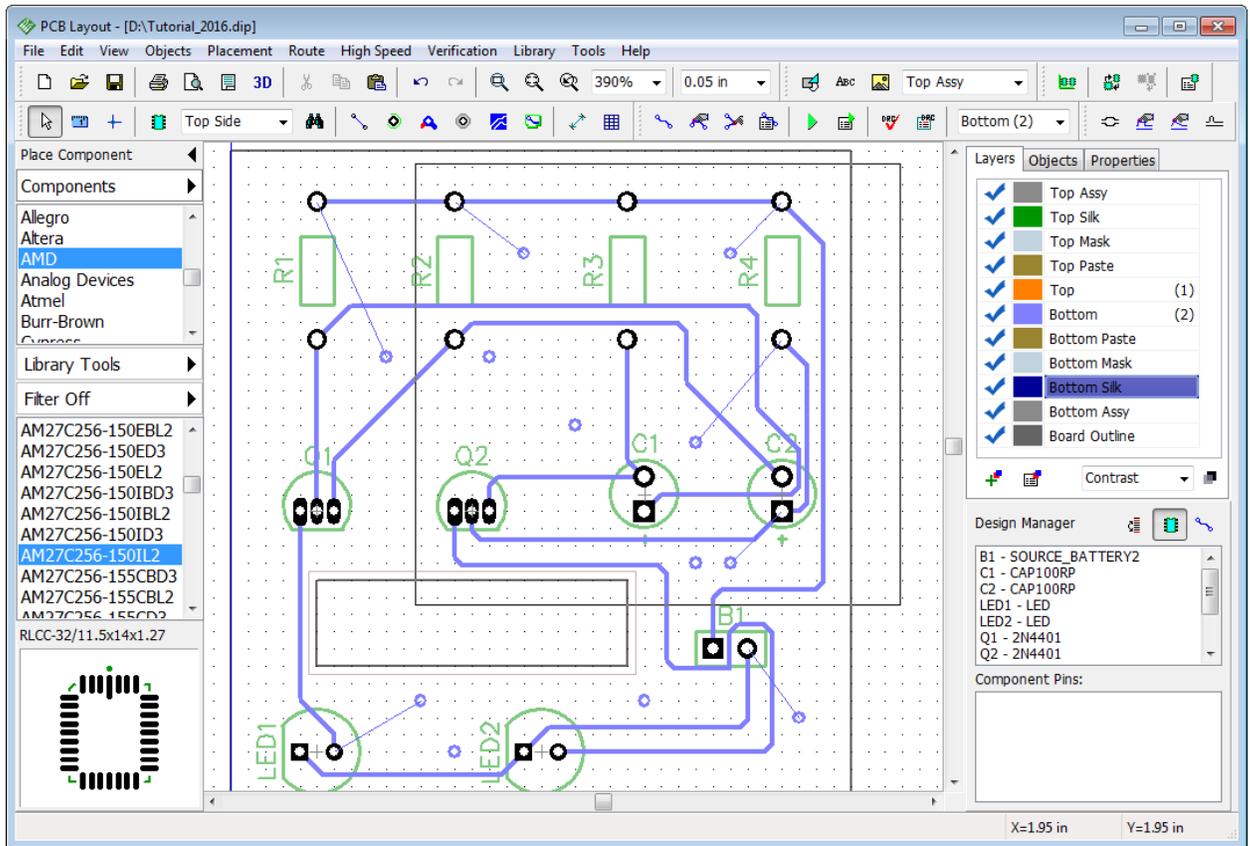
Now let's make it a bit harder and model a real-life situation when we need to select only unconnected vias in the predefined area of the board.

First of all deselect components with a right click on an empty area. Then place several static vias and connect **only some of them** to nets randomly, while leaving a couple of vias

unconnected. Use "Objects \ Place Static Via" from the main menu (or press  button on the

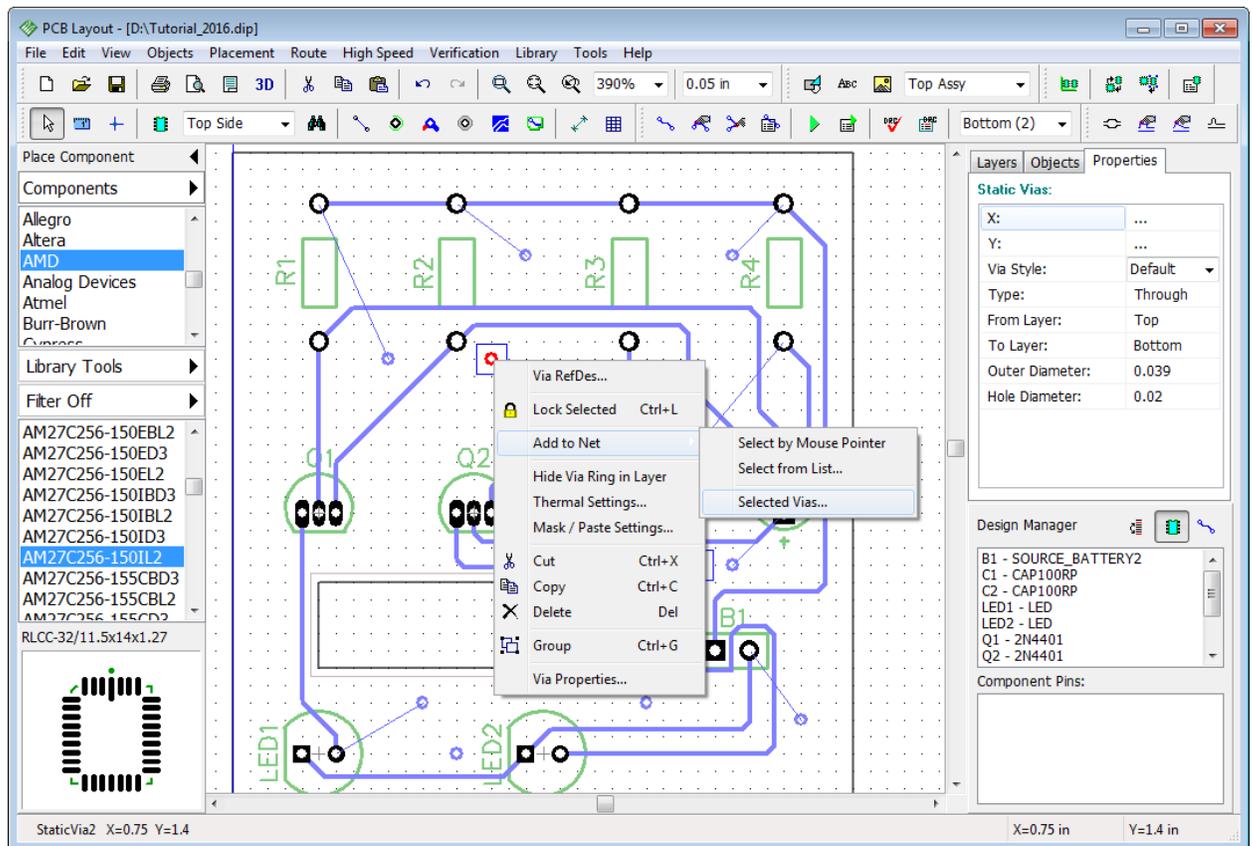
objects toolbar) to place via and "Objects \ Place Ratline" ( button) to create connections visually. Left click on the via and then left click on the pad to add via to the pad's net.

Now define the area using the box selection. This box represents an area where we plan to select unconnected vias so we will not include all vias of the layout to this selection. Notice that we are in the bottom layer, where we have all traces.



All objects in the box are selected. We need to extract only non-connected vias from the selection. Open "Edit \ Edit Selection", choose "Mode: Keep Selected", check only the Vias box (other boxes should be unchecked), and then select Not Connected from the Vias drop-down list. Click OK and only unconnected vias will be selected now.

The next step could be connecting these vias to some net, all at a time. In real life this is used to connect ground net to plane / copper pours. Right click on one of selected vias (it should be highlighted in red) and select "Add to Net \ Selected Vias". Specify the net in the pop-up dialog box.



Choose some net from the list and click OK. All vias will be connected.

Remove all static vias from the board and return project to the previous state (select all vias and press "Del" key) or **Undo**.

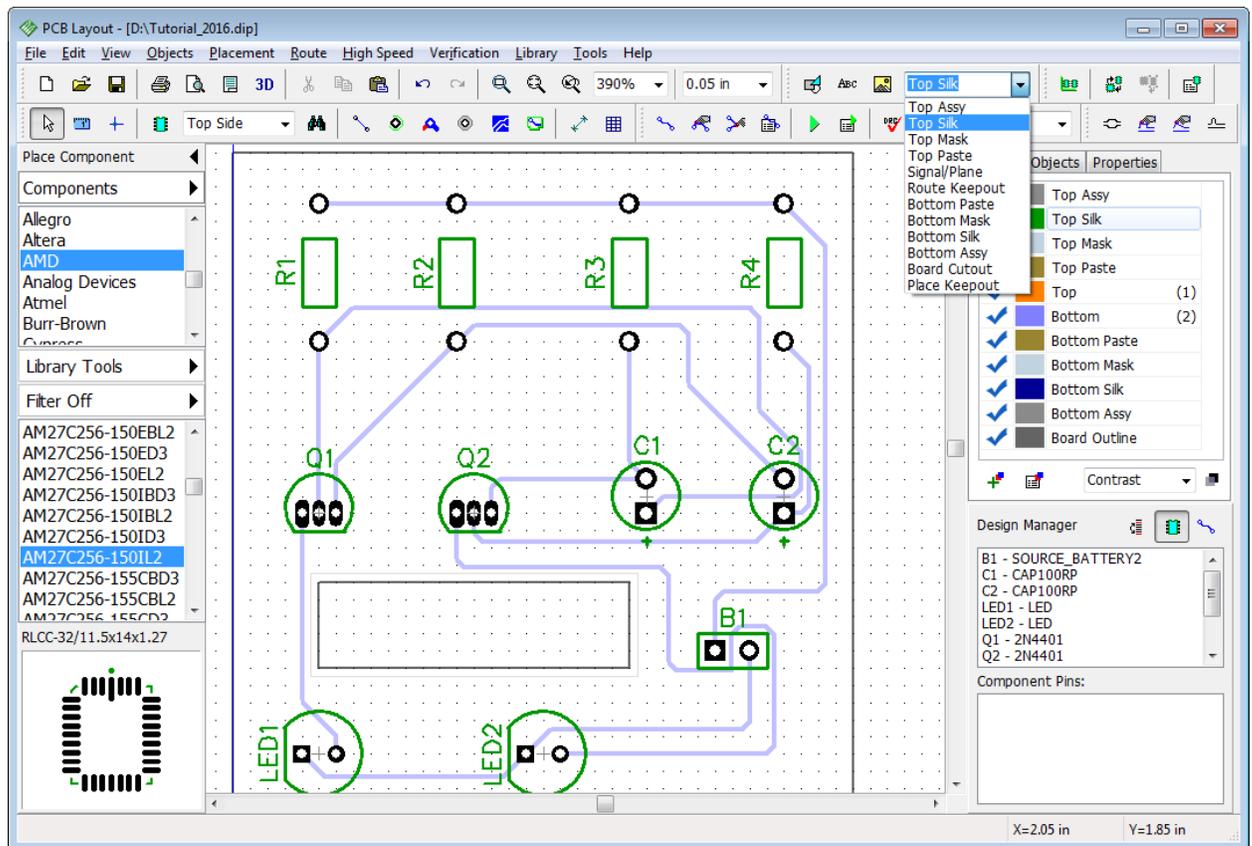
## 2.10 Placing Text and Graphics

With PCB Creator you can add texts, shapes and logos in BMP, DXF, or JPEG formats directly on the board.

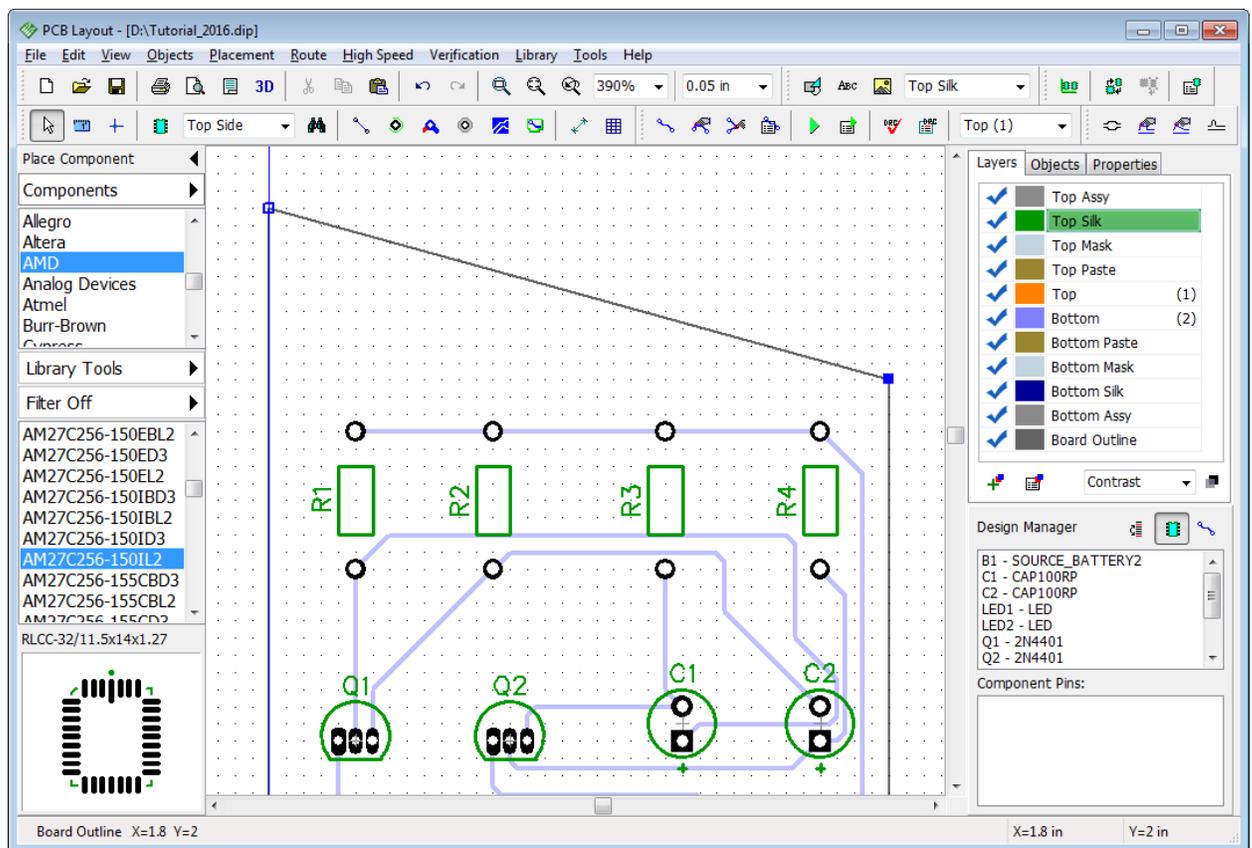
Select the layer where you're going to place graphics, usually it's a silkscreen layer (Top Silk in our case), double click it in the Layers tab of the Design Manager panel.

PCB Layout allows the user to change layers with two drop-down lists on the instruments toolbar and in the Layers tab on the Design Manager panel.

Drop-down list on the drawing toolbar allows you to select any non-signal or Signal / Plane layer for placing graphics. If you have selected Signal / Plane layer, all shapes, texts and logos will be placed on current signal/plane layer, which can be specified with the drop-down list on the route toolbar.



Let's make the board polygon a little bit bigger to place additional text at the top. Drag and drop the upper-left and upper-right vertices of the board outline a little bit upwards. Make sure you click on the vertex point not on the outline. PCB Creator makes visual editing very easy with appropriate grid size.

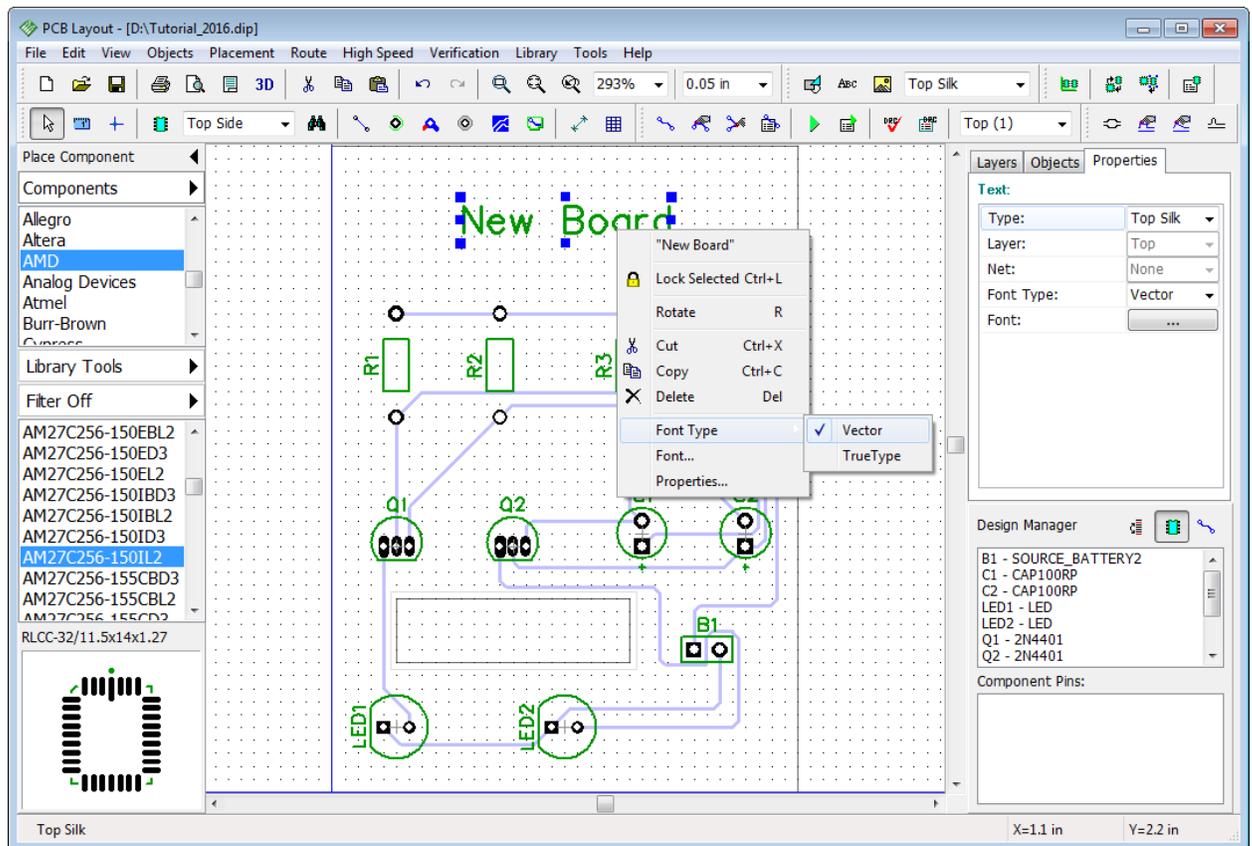


You can move the board outline. Left click on the line (not the vertex), then drag and drop it.

Remember that if you can not highlight and edit certain objects, probably, you are not in the default mode. Therefore right click on a free area to cancel current mode. Objects located on inactive layers of the board can't be edited.

Select the Text tool on the drawing toolbar (button with "Abc"), left click at where you would like to place text, type in the text and press Enter to finalize.

Use mouse to move text around the design area until you'll find a good location. When text object is selected, font settings, font type (Vector, TrueType), and the text layer can be changed in the Text Properties panel on the Design Manager, or with the right click submenu. Use vector font in most cases. Use TrueType fonts only for Unicode and Non-English characters.



Text object is on the silk layer, it inherits the layer's color. If you need to change the text color, move the text object to the "Top Assy" layer and then change Top Assy color.

You can change the layer of the graphics and text objects any time. Select graphic object, right click it, and select Properties from the submenu. In the pop-up dialog box change Type (Layer for signal layers) field to move object to another layer or define different properties (for example, Route Keepout used for autorouting, etc.).

In our case we just leave that text on the Top Silk layer.

Notice that you can add shapes to mask, paste, signal, route keepout, and board layers. These properties can be defined on the drawing toolbar or in the shape properties dialog box.

## 2.11 Copper Pour

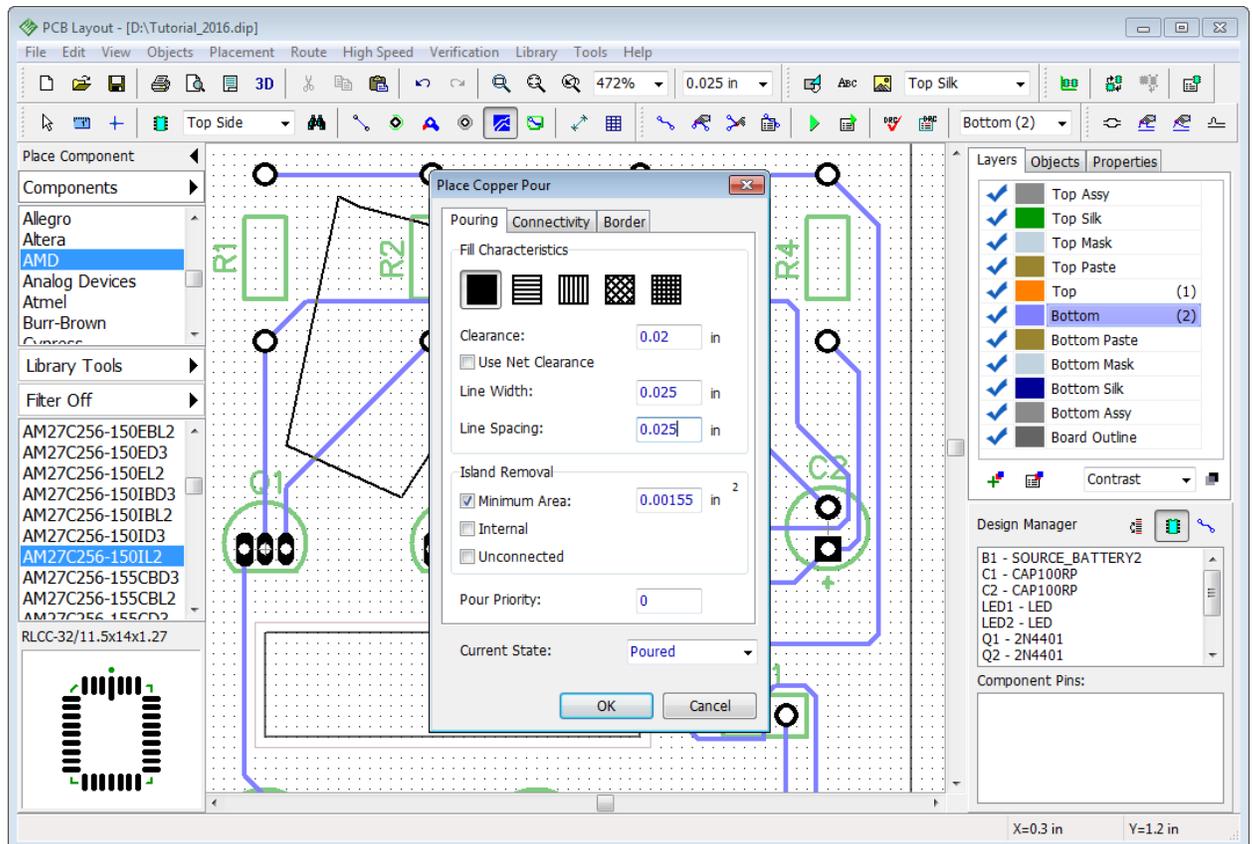
Copper Pour is used as a low-impedance conductor for Power and Ground nets. Pours are usually located on inner layers of the board, but can be placed on top and bottom sides as well.

### Place copper pour

Change the grid size to comfortable 0.025 inch. Select the Bottom layer then go to "Objects \

Place Copper Pour" from the main menu or press  button on the elements toolbar. Now you can draw a copper pour polygon borderline by defining its key points, then right click on the last polygon's point and select Enter from the submenu to finish drawing. But we need a copper pour that covers the entire bottom layer. You can draw a precise polygon manually or create a random

shape (for example, like on the picture below) and use Depending on Board feature which will pour the entire layer automatically (regardless of the initial shape). When you select Enter from the submenu Place Copper Pour dialog box pops up.



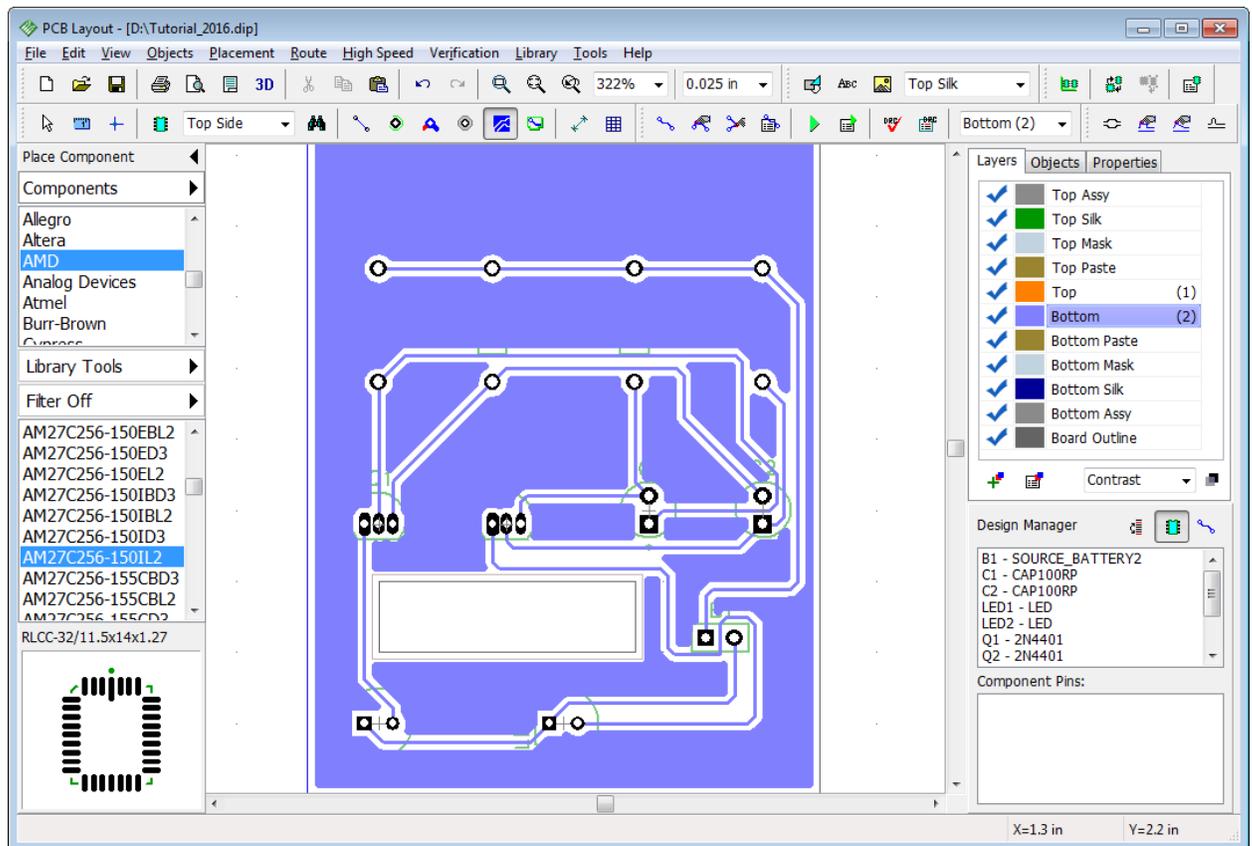
This dialog box has three tabs: Pouring, Connectivity, and Border.

The **Pouring** tab allows you to specify different non-solid fills for the copper pour, clearance width, line width, line spacing, island removal options, pour priority, and current state (poured or unpoured). You can also apply net clearances as copper pour clearances by checking the corresponding item. PCB Creator has shape-based copper pour system.

The **Connectivity** – here you can connect copper pour to the net, select thermals and change their settings. PCB Creator supports separate thermals for SMD pads. Hide Net Ratlines regime can automatically show ratlines only for unconnected traces or other if specified.

The **Border** tab allows you to define border points. Use Depending on Board check box to save your time and build copper outline automatically; "Snap to Board" option means that copper pour will resize depending on the board outline.

Check the Depending on Board item and keep all other settings like on the picture above. Click OK to place a copper pour.



Board outline clearance specified in the copper pour settings is not applied to board cutouts. Always use route keepout to allow for a certain clearance between the copper pour and the cutout, like we did [before](#)<sup>33</sup>.

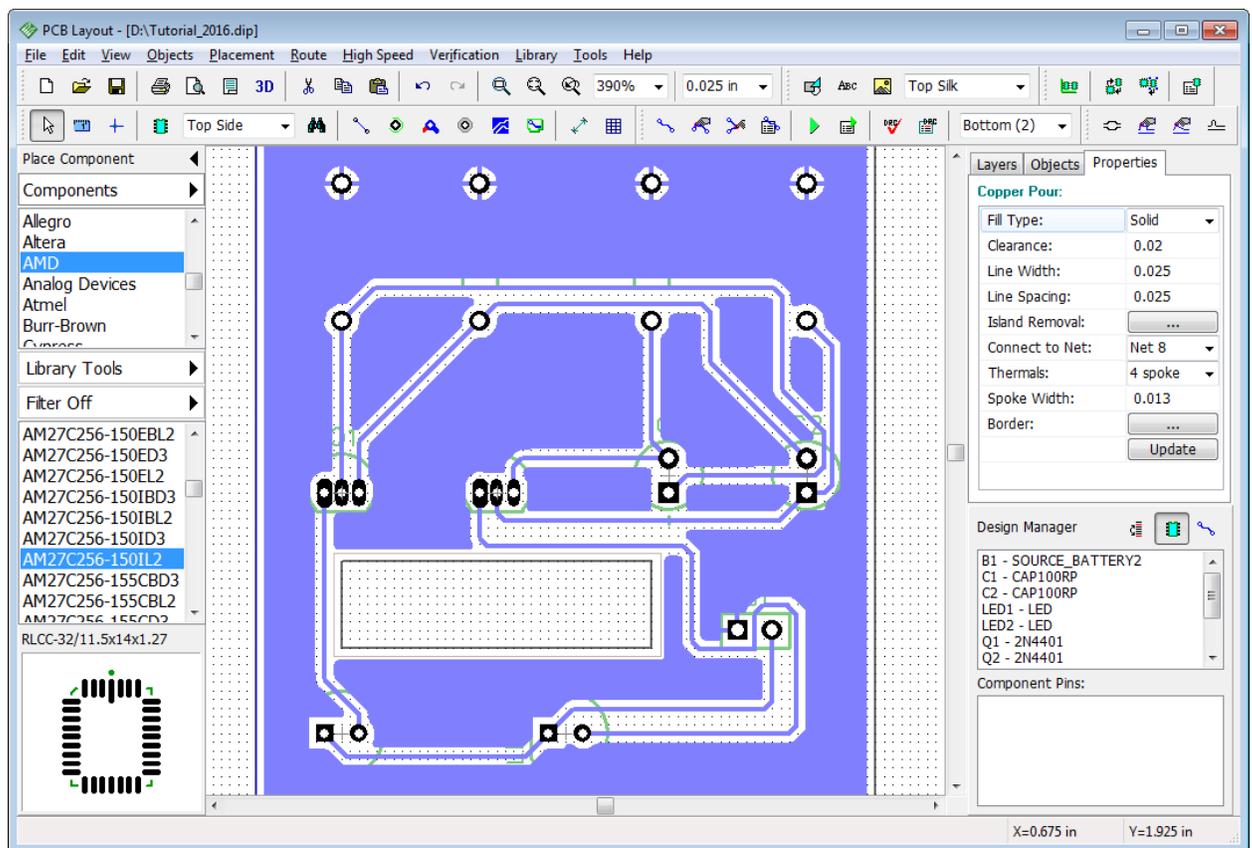
Copper Pour object can be in two states: Poured and Unpoured. The second state is often used for editing objects, because only the copper pour border is visible. To change the copper pour state **right click on the copper outline (not on the copper pour's body)**, select State and choose the item you need.

As you can see we have a copper pour, but it is not connecting any net. Now we will practice and connect two different nets using two copper pours on the Bottom layer. Copper pour priority option will help us to achieve our goals.

### Connect copper pour

Unroute one of the nets (for example, Net 8 which connects resistors to the battery), right click on the trace and select Unroute Net from the submenu. Remember the net name ("Net 8"). Right click on the copper pour border and select Properties from the submenu. Go to the Connectivity tab and select Connect to Net: Net 8, then select appropriate thermals (for example, 4 spoke) and press OK to update the copper pour.

*Notice that you should click directly on the copper pour border (not on the copper body or the board outline) in order to open copper pour properties dialog box.*



You can see that connection lines (ratlines) are hidden now and the net (Net 8) is connected to the copper pour with thermals of selected type (4 Spoke thermals).

Now we will place the second copper pour. Select another net that we will connect with a copper pour (for example, Net 3 that connects R3:1, C1:2, and Q2:3), and unroute it. Then right click **on the edge** of existing copper pour and open Pour Properties dialog box. Select "Current State: Unpoured", but do not close this panel.

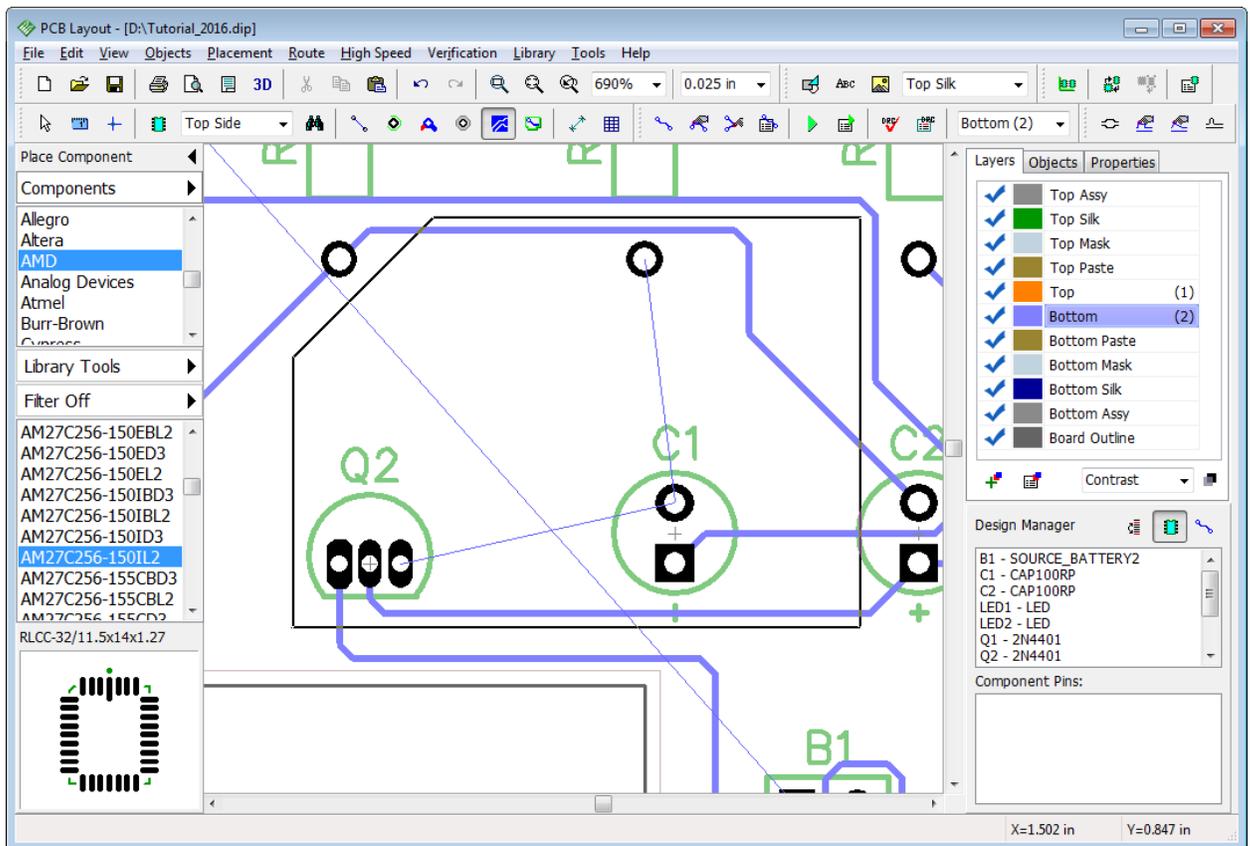
### Pour priority

Now it's time to change pour priority for existing copper polygon. Specify: "Pour priority: 1" in the Pouring tab.

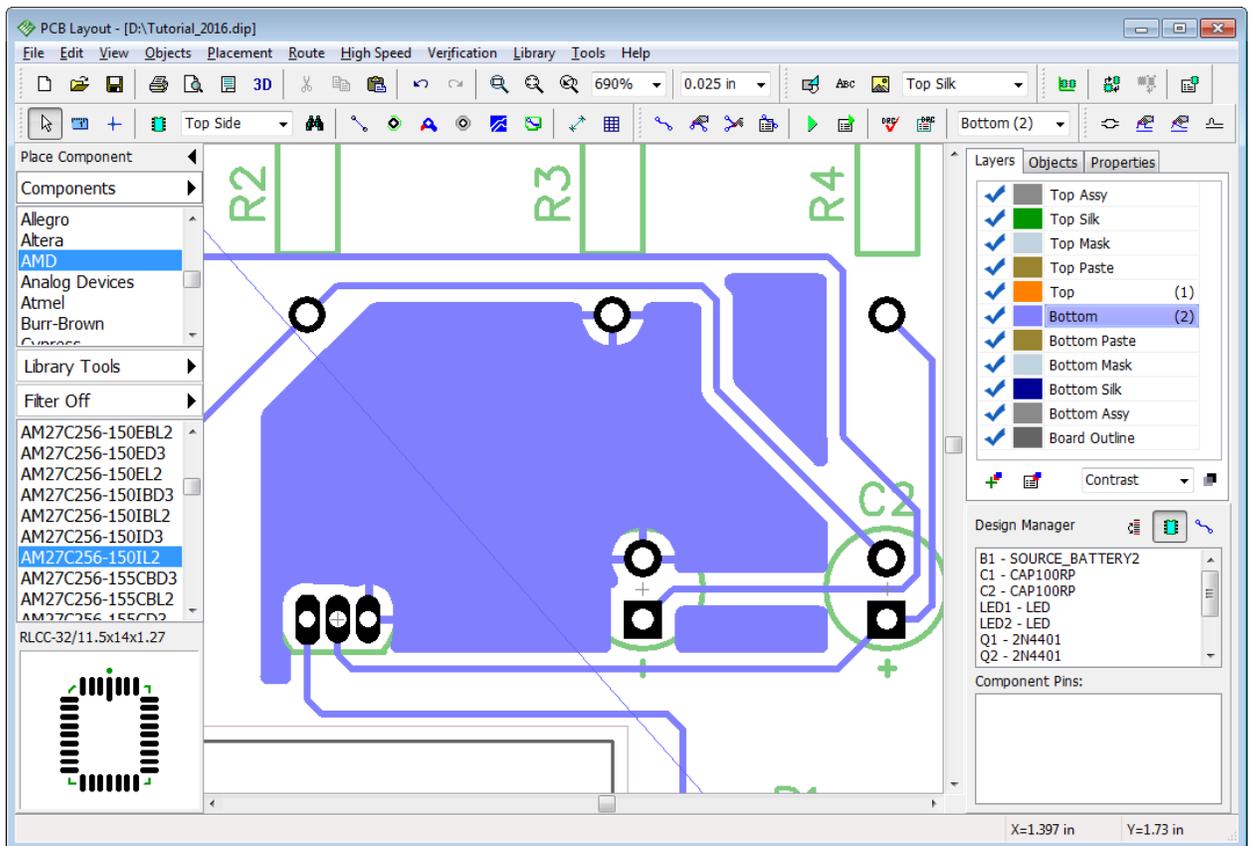
You can enter any value, depending on how much copper pours you plan to have on this layer. Lower value means higher priority, therefore copper pour with "Pour Priority: 0" will have higher priority than "Pour Priority: 1".

*Notice that two different-net copper pours with the same priority level will intersect. Real-time Design Rule Check will show numerous errors in this case.*

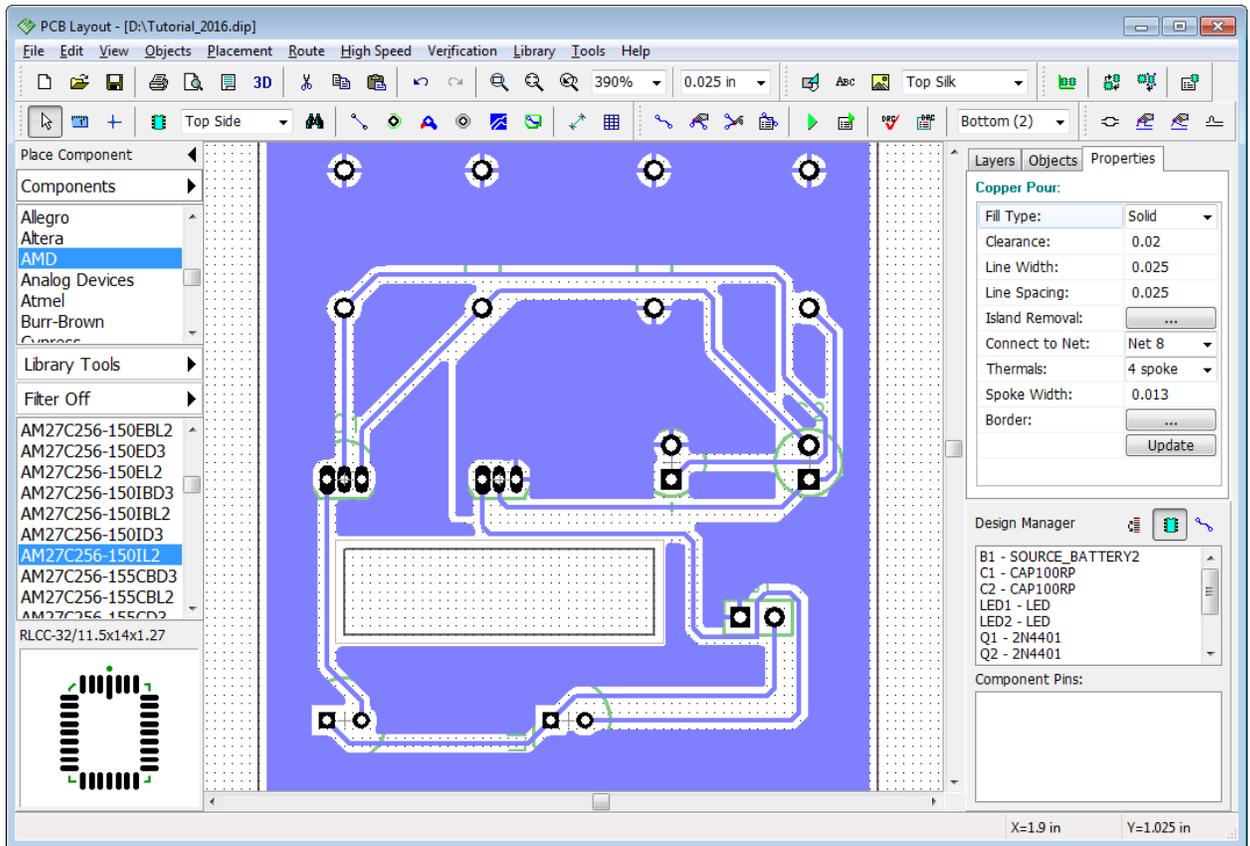
Unroute Net 3, then select copper pour placement tool ("Objects \ Place Copper Pour") and draw the second polygon that covers pads of the Net 3, like on the picture below. Press OK to apply new settings. Notice that in unpoured state ratlines are displayed automatically.



In the pop-up dialog box connect the second copper pour to Net 3 and specify the thermal type (4 Spoke should be fine). Press OK to close the dialog box and create a copper pour polygon.



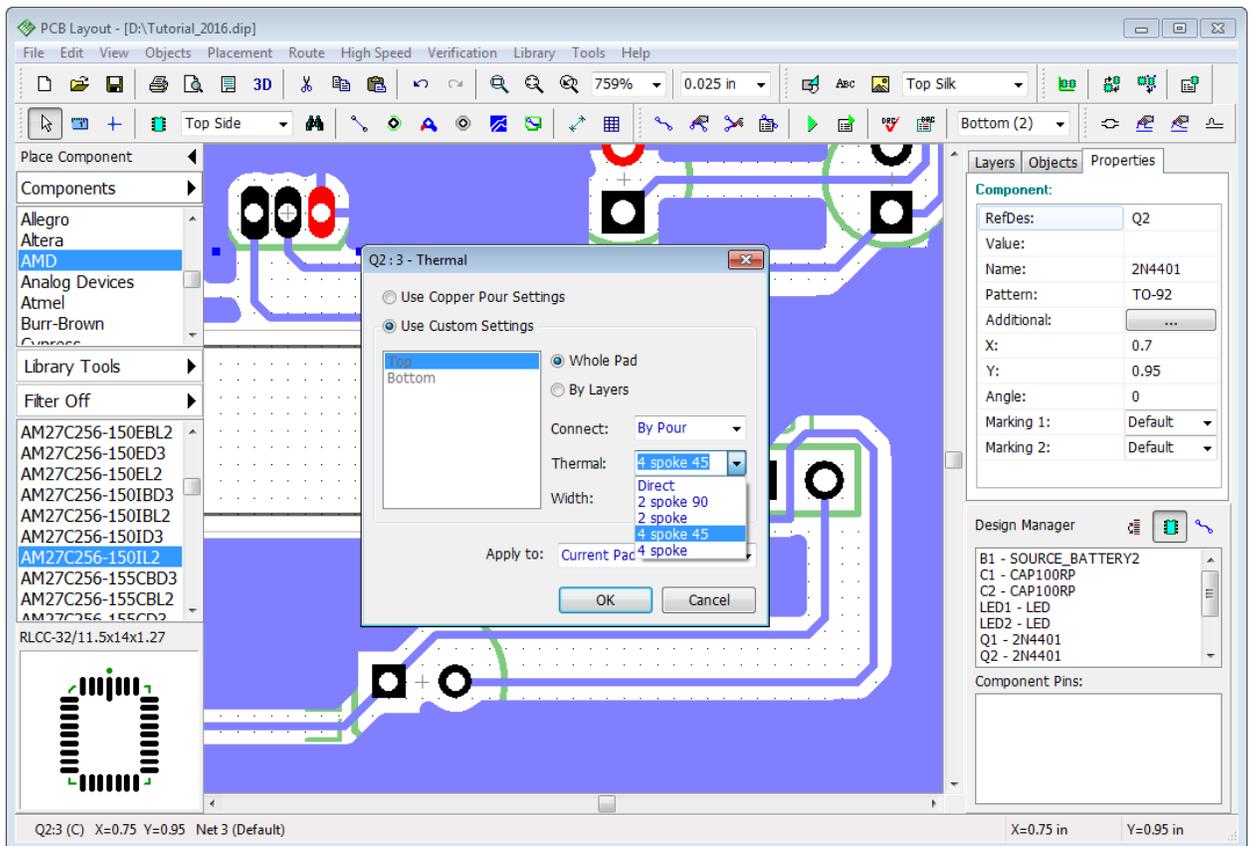
Now select Net 8 copper pour, which is unpoured now. Right click on its border and select "State \ Poured" from the submenu. You will see that two copper pours connecting two different nets are independent and Net 8 copper changed its shape according to the Net 2 polygon which has higher priority level.



### Thermals

Some pads require custom thermal connections that will be different from the copper pour's thermals. Right click on the pad (when the pad is highlighted) and select Thermal Settings from the submenu. Then check Use Custom Settings and select new thermal connection.

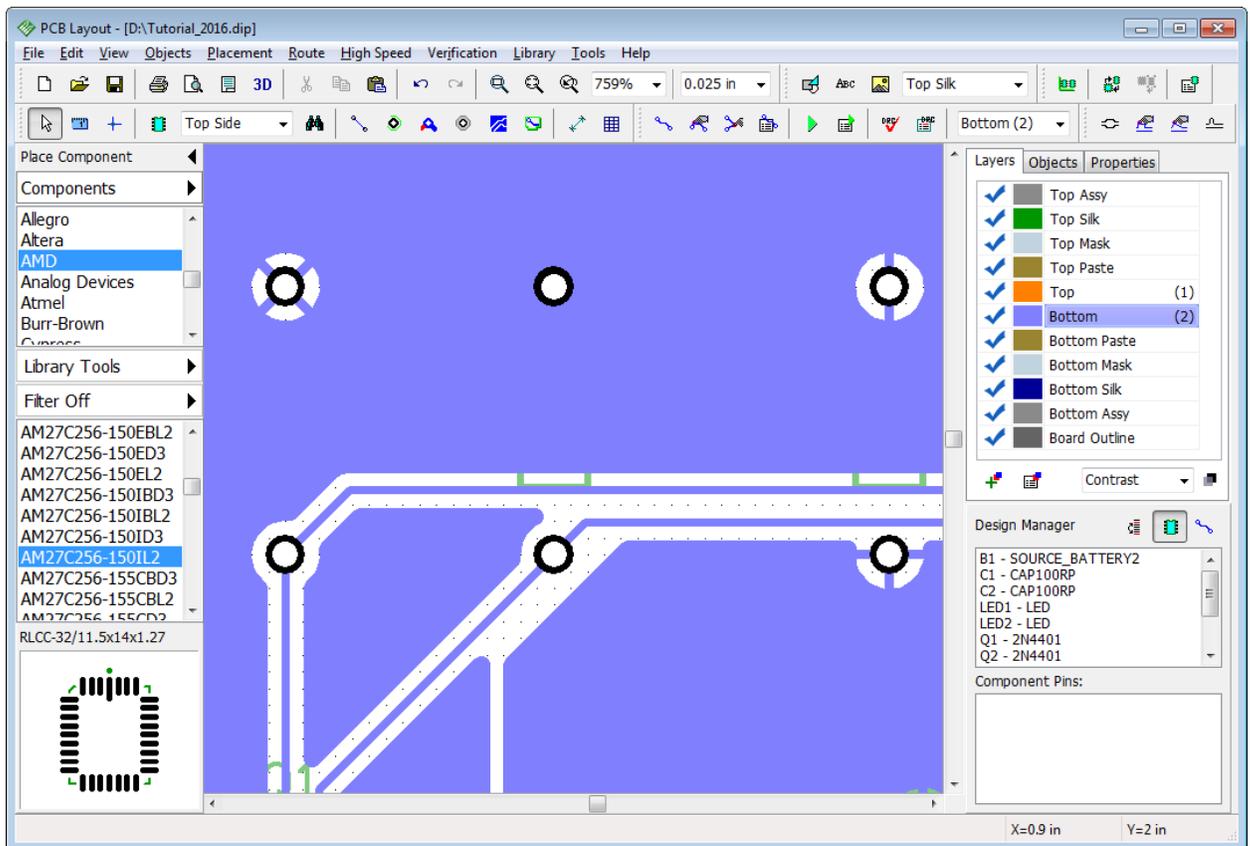
Some pads can become unconnected after placing a copper pour, because of selected thermal type and the layout structure (net connectivity check will report this), so selecting separate thermal settings for pads will help you to fix those issues.



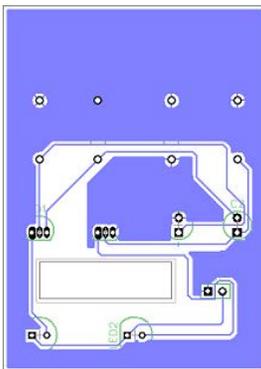
After changing thermal settings click OK to close the dialog box, then right click on the copper pour border and choose Update from the submenu to see these changes applied.

Select "Objects \ Update All Copper Pours" from the main menu to update all pours at a time.

We'll try different thermals for pads to show you how it works.



On the picture above you can see that one pad has 4-Spoke 45-degree thermal, another one is connected directly, and the third pad has 2-spoke 90-degree thermal.



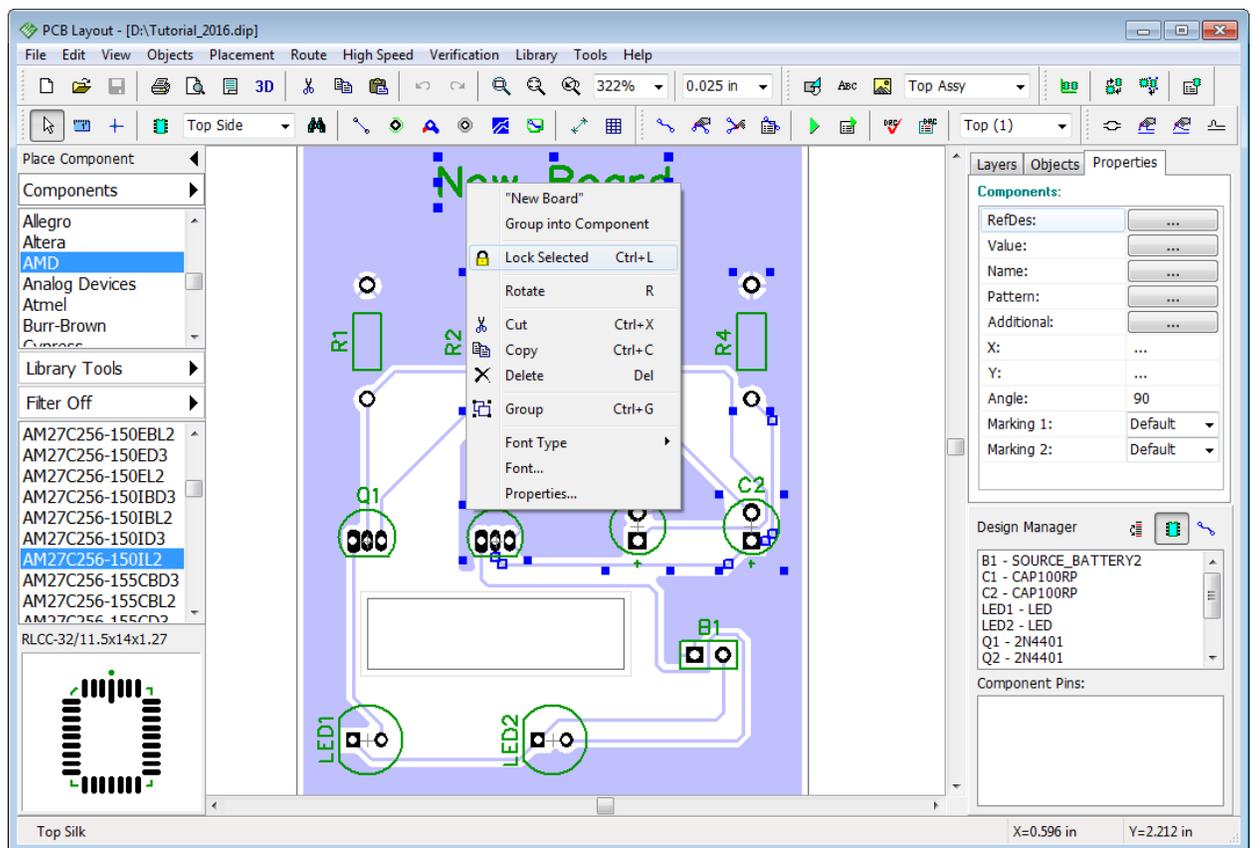
When copper pours are used as Ground and Power planes, SMD vias are connected to them by fanouts. Fanout can be made manually with [Fanout](#)<sup>[219]</sup> feature or automatically by [Shape Router](#)<sup>[42]</sup>.

We have decided to remove all unconnected parts of both copper pours. Go to properties of each copper pour and check Unconnected item in the Island Removal section of the Pouring tab. Press OK and Save the project.

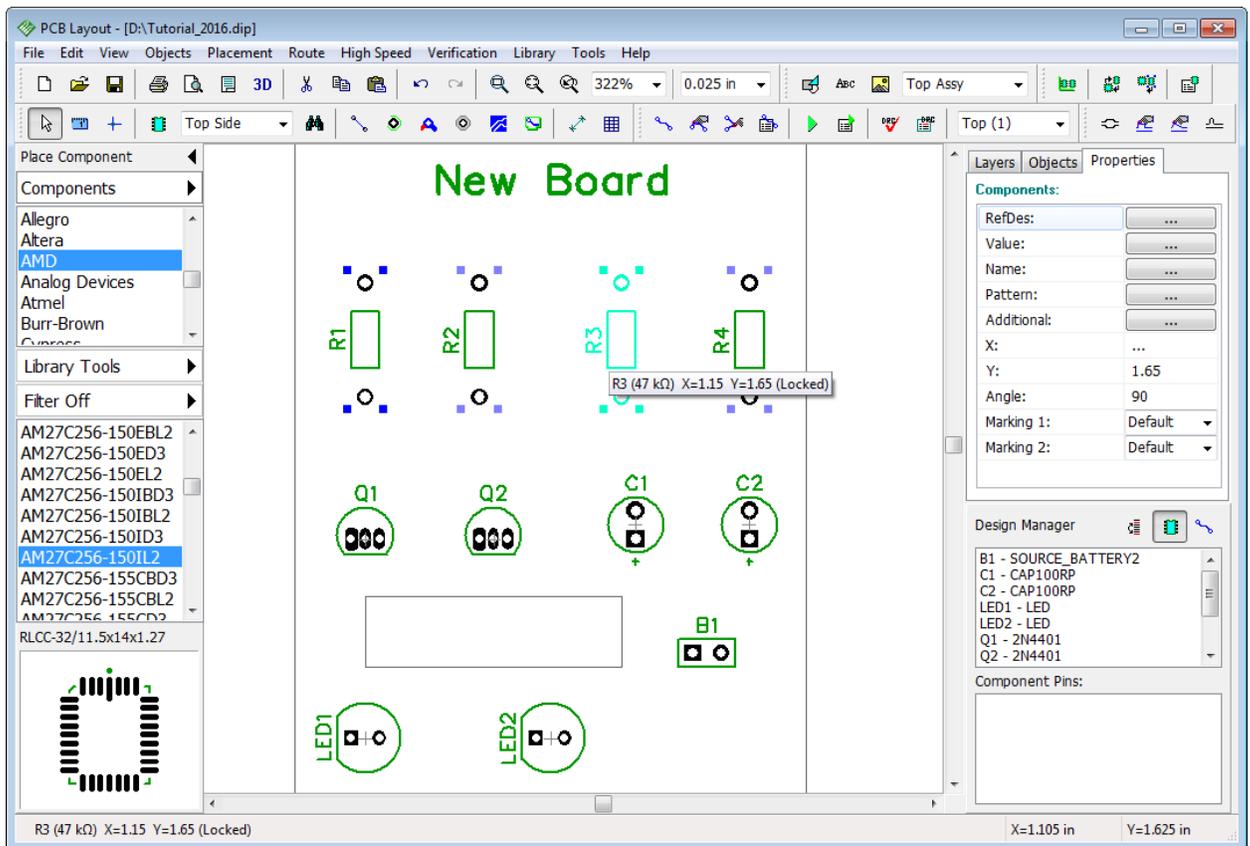
## 2.12 Locking objects

Sometimes, while making changes to the schematic or PCB, you need to lock objects so you don't change them accidentally. Let's see how to do this in the PCB Creator.

Switch to the Top layer, select several objects, right click on one of them, and click Lock Selected from the submenu.



Notice that locked objects have lower contrast of the selection rectangles (in our case color is similar to the copper pour, so we've made only current layer visible (with the drop-down list on the Layers panel). "Locked" text appears in the hint of the locked objects, you are unable to move, resize or edit them without unlocking first.



Now please unlock all objects. Select all with "Ctrl+A", and unlock ("Edit \ Unlock Selected" from the main menu or "Ctrl+Alt+L" keyboard combination).

Components can be locked after placing them on top or bottom side of the board. Select "Edit \ Lock Components \ Top" to lock top components. Using this mode you can do the routing and do not worry about accidentally changing something. Select "Edit \ Unlock Selected" from the main menu to unlock components on the board. Then return to the contrast layer display mode using the drop-down on the Layers tab (Design Manager).

## 2.13 Design Verification

PCB Creator has several verification procedures united in the Verification main menu item. We recommend to use all three of them: DRC, Net Connectivity Check, and Compare PCB to Schematic.

### DRC (Design Rules Check)

This feature is one of the most important verifications. It allows the user to check clearances between the objects, allowable object sizes, and differential pair parameters according to the set of design rules. DRC works in regular (offline) and real-time modes. Real-time DRC checks all user's actions on the go. For example, when you move some component or create a new trace too close to another object, Real-time DRC shows red circles which mean that the clearance between these objects (trace, pad, copper pour) is smaller than the specified target value. If Real-time DRC is completely turned OFF, you would not see errors until you start DRC manually

in regular mode by selecting "Verification \ Check Design Rules" from the main menu or by pressing "F9" hotkey. Errors list or "No Errors Found" message will pop up. Most likely current PCB doesn't have any errors, because it is very simple.

Now select "Verification \ Design Rules" to setup design constraints. There are 4 tabs in the pop-up dialog box: "Clearances", "Sizes", "Real-time DRC" and "Options".

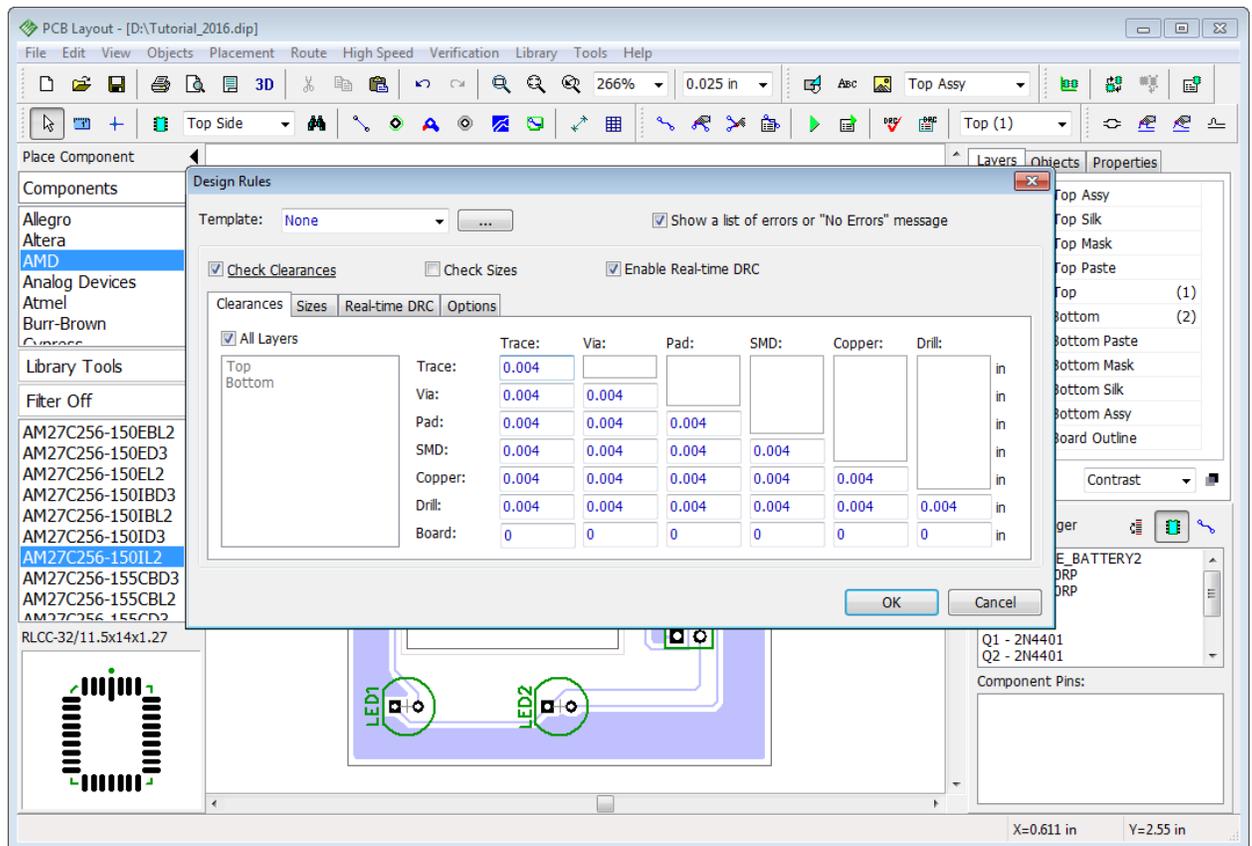
**Clearances.** Specify object-to-object clearances. Uncheck "All Layers" item, select layer from the list below, and define object-to-object clearances applicable on the particular PCB layer.

*Notice that clearance settings do NOT apply to the nets with custom net class clearance ("Use Clearance in DRC" option in [Net Classes](#)<sup>[50]</sup> dialog box) or Class-to-Class settings.*

**Sizes.** Specify minimum and maximum allowed sizes for different objects on different layers.

**Real-time DRC.** Customize real-time DRC. You can turn it ON/OFF for specific actions, for example, manual routing, creating / editing, and moving objects. If you uncheck "Enable Real-time DRC" item, real-time verification will be completely turned OFF. But if you uncheck all secondary items in the corresponding tab and leave only "Enable Real-Time DRC" item active, you will see errors right after completing a certain action, not while performing it. For example, if "Moving objects" item is checked, you will see errors before moving component to a new position, if this option is unchecked - you will see errors right after moving it. Still no need to launch DRC separately. If "Enable Real-Time DRC" is completely turned OFF, you would not see any errors, unless you start DRC manually.

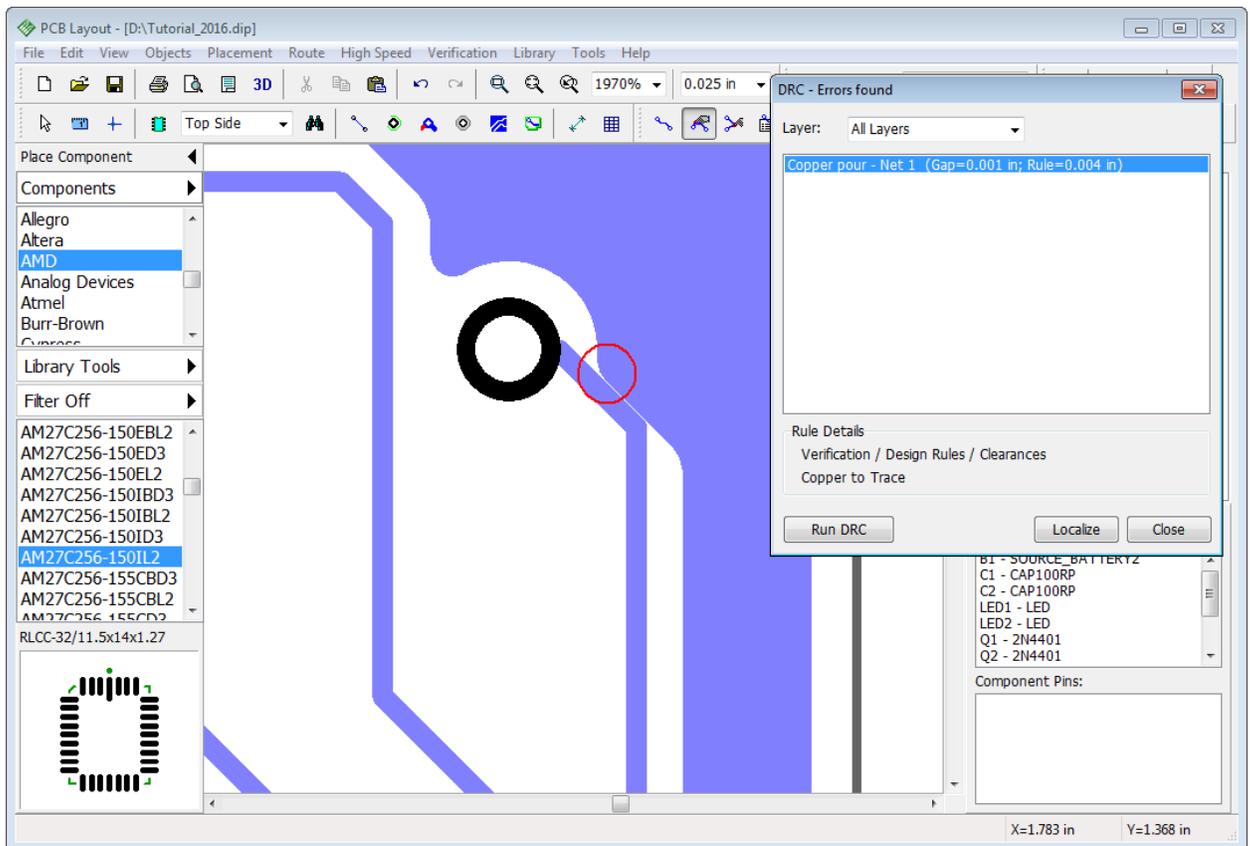
**Options.** Setup other verification options.



For this tutorial example, please turn OFF the size verification and Real-time DRC by unchecking the corresponding items. Make sure that Check clearances and Show a list of errors... are checked. Keep settings like on the picture above (these values are quite small, but still within technical capabilities of most PCB houses).

Open the Options tab and make sure that Class-to-Class Rules, Trace by Length, Copper Pours, or whatever objects you want to be verified are checked. Now let's try to see how DRC works in a regular mode. Even if the real-time DRC was ON during the design process, we recommend to verify project with regular-mode DRC at least once before sending out for manufacturing. Press OK to apply changes and close the dialog box.

Our project does not have errors, therefore we will create them intentionally. Select the Bottom layer ("B" hotkey), switch OFF the grid ("F11" hotkey) and move some trace until it touches the copper pour or another trace. Go to "Verification \ Check Design Rules" or just press "F9" hotkey to launch DRC. The list of errors will pop up automatically.



Errors can be displayed by layer. You can see the description of each error including current value and the target value. Left click on the error in the list and PCB Creator will show where to change the constraint value in the Rule Details section. Press Localize to see selected error on the design area and fix it. Red circle means a clearance error, magenta circle – size error.

Move trace back to it's original location without closing error-report panel and then press Run DRC button. This time everything is good and "No Errors Found" message appears.

### Net Connectivity Check

This verification allows the user to check if all nets are properly connected. For such simple design this feature is not necessary, but if you have larger board with many layers, pins, copper pours, and shapes, net connectivity verification becomes really useful. It checks entire design and displays the list of broken and merged nets. Please select "Verification \ Check Net Connectivity" and click OK. Most probably your design will not have connectivity errors and you will see the "No Errors Found" message. More information about [Net Connectivity Check](#)<sup>[206]</sup>.

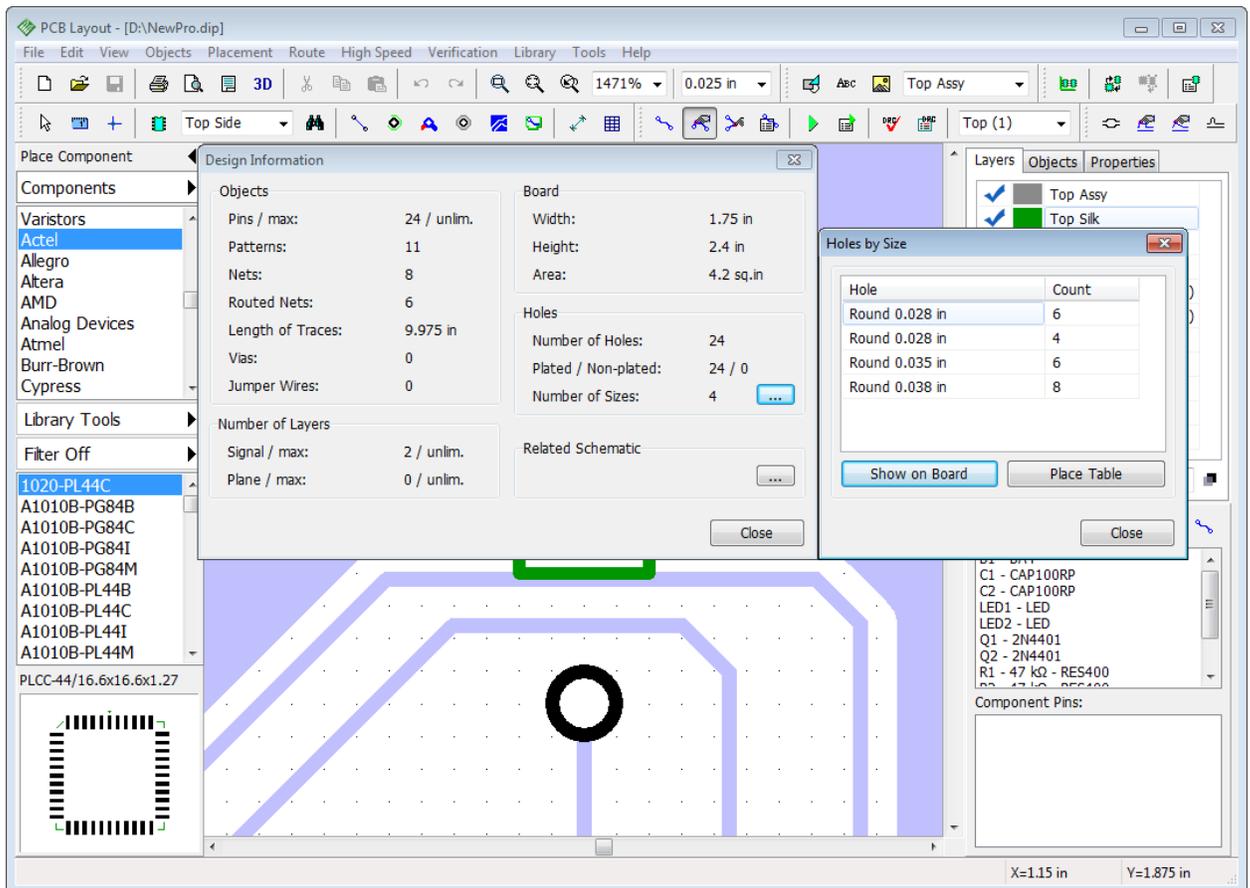
### Comparing to Schematic

This procedure allows you to check if the PCB corresponds to the source Schematic file. Verification shows the net structure errors and unknown components. Select "Verification \ Compare to Schematic" from the main menu then choose the source Schematic file and press OK. If your net structure was not changed and has no errors, you will see the "No Errors Found" message, otherwise the list of errors will pop up.

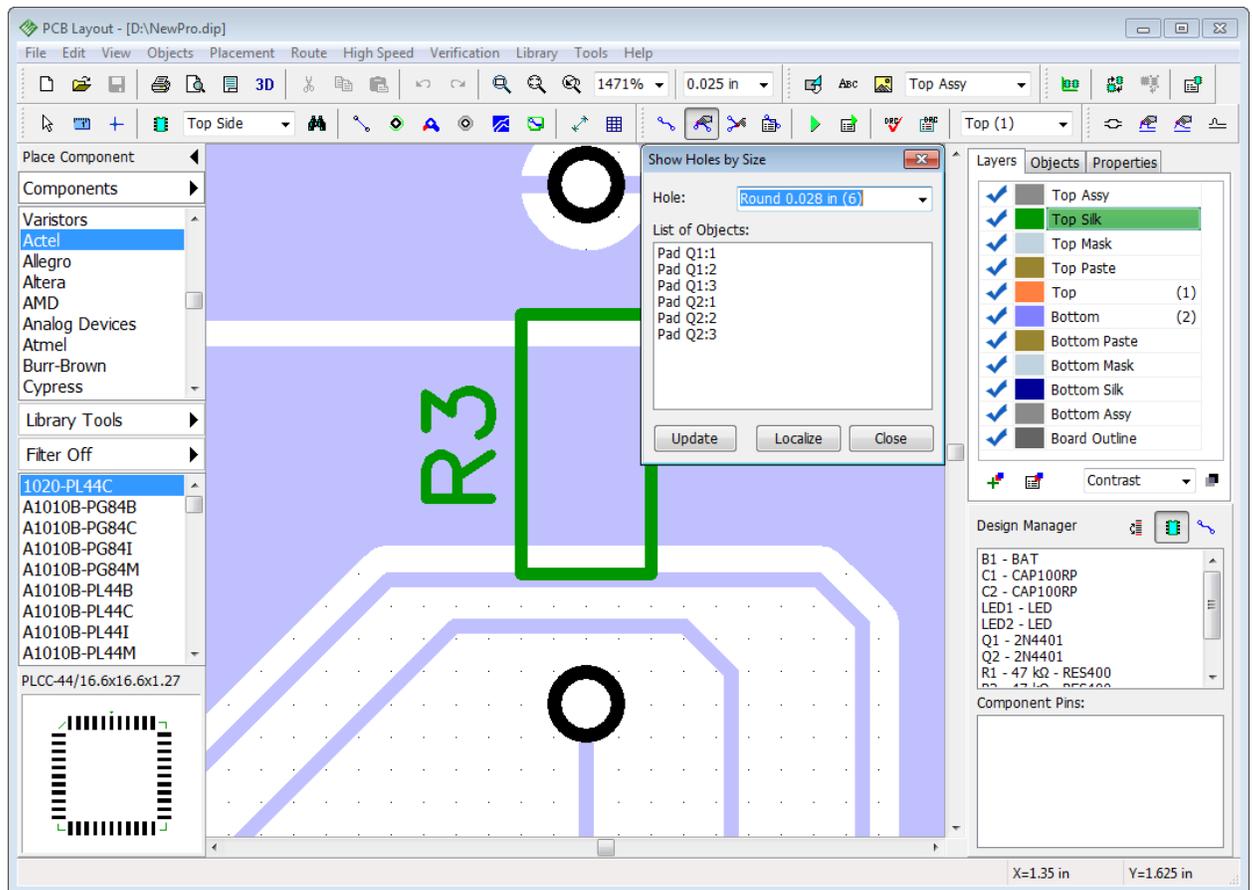
Net connectivity check and Comparing to Schematic features work fast and provide an easy-to-understand user interface with reliable functionality.

## 2.14 Design Information

How about counting the number of pins or the board area? Select "File \ Layout Information" from the main menu in the PCB Layout module.



In the design information dialog box you can preview the number of different objects, layers, board and hole sizes. Press "." button in the Holes section to open Holes by Size panel. Press Show on Board button to highlight holes by size directly on the board.



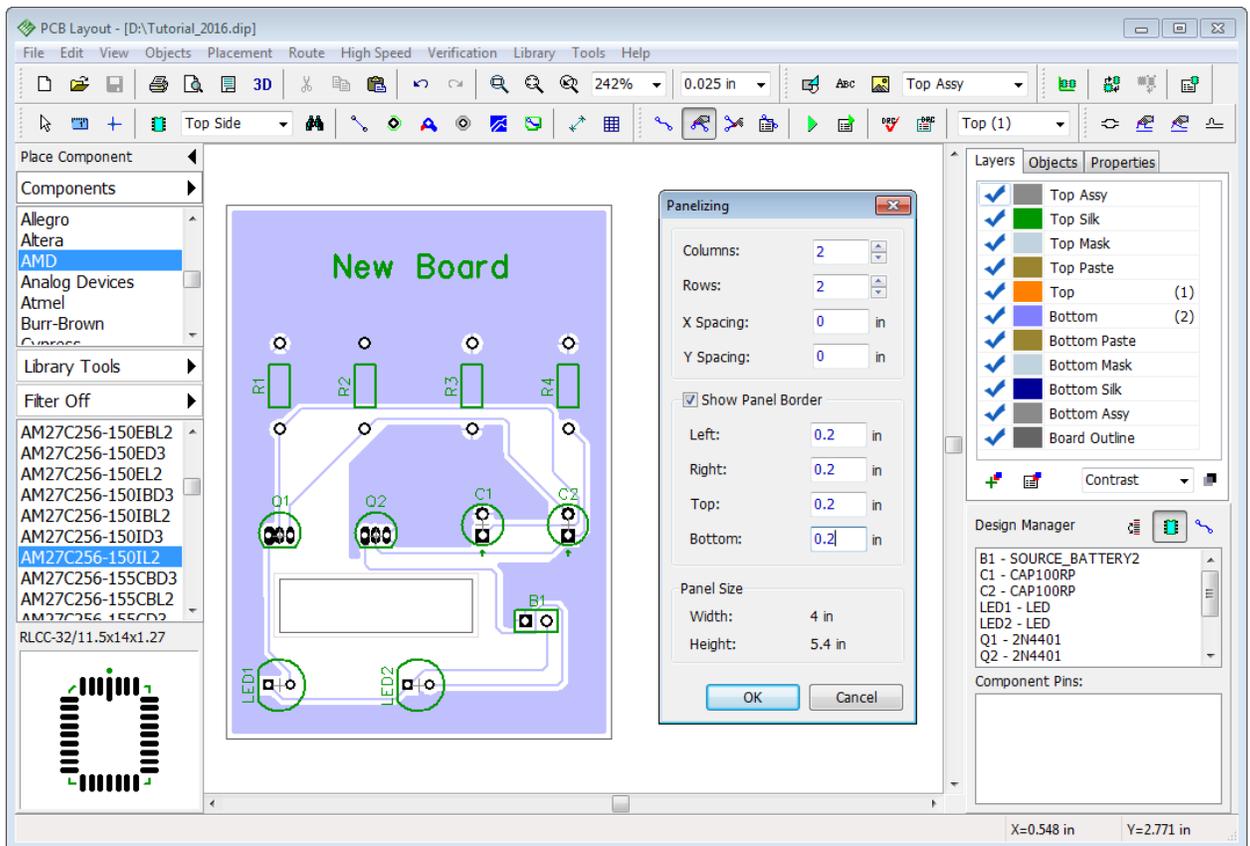
Close the dialog box and save the layout.

## 2.15 Panelizing

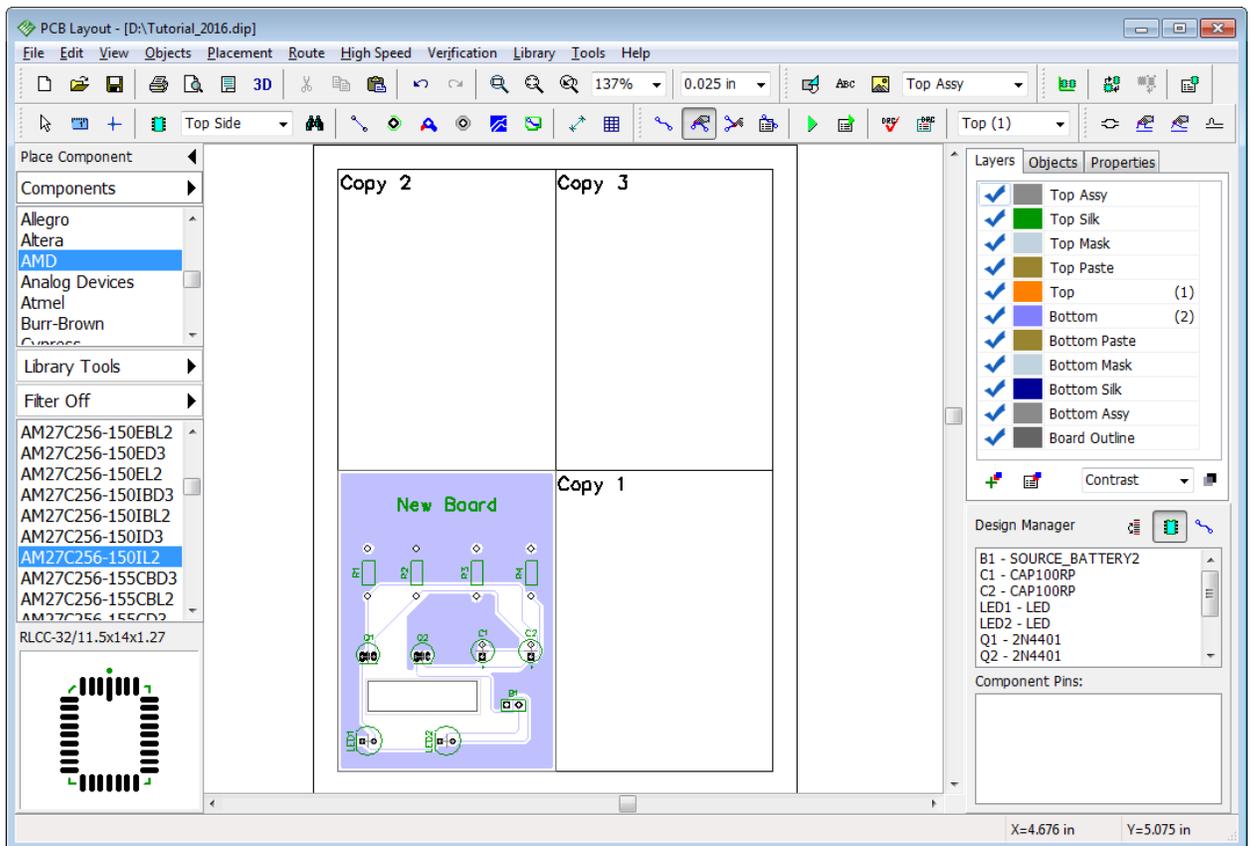
With PCB Creator you can panelize similar or different boards on a single layout.

### Panelize the same layout

If you need several copies of the same PCB select "Edit \ Panelizing" from the main menu:

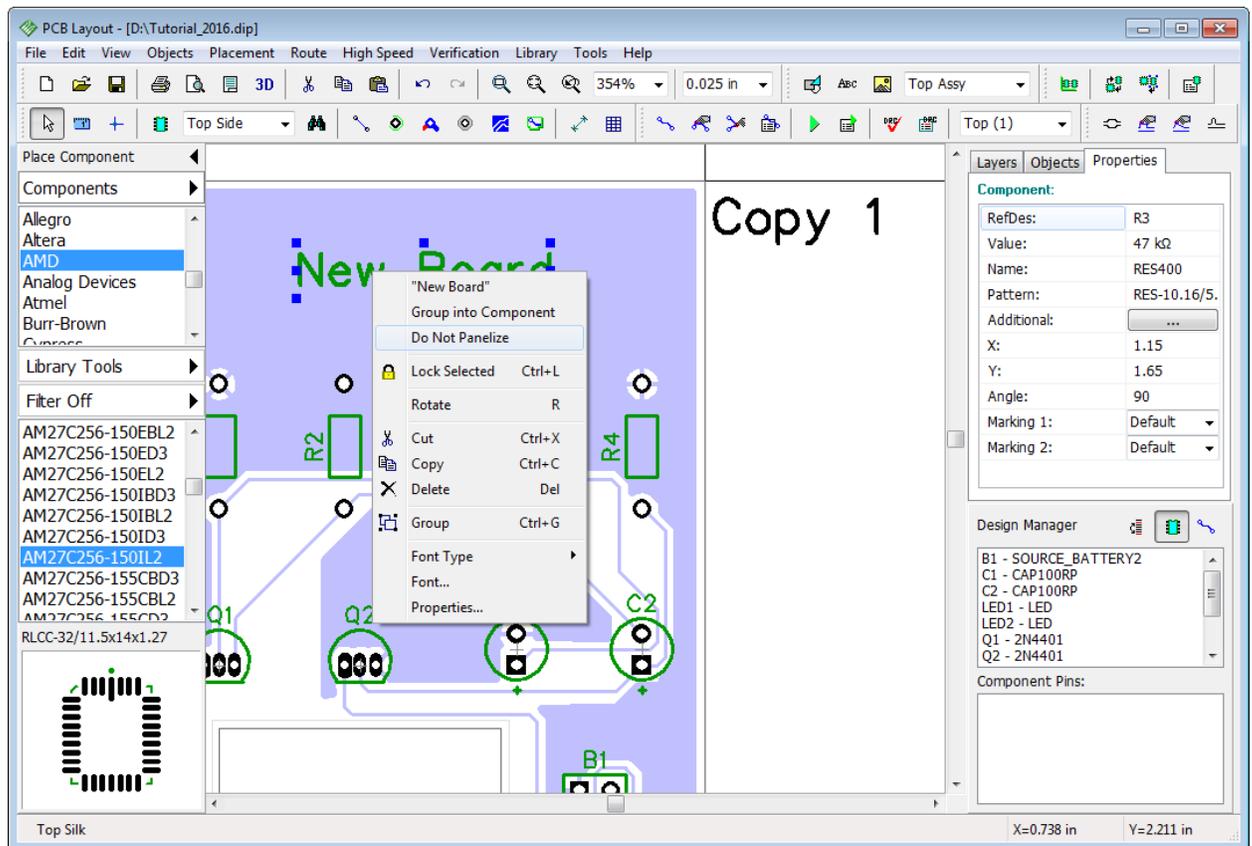


We will make 4 copies of the PCB, 2 columns and 2 rows. Spacings between the boards will be zero. Showing panel border may help us to determine the approximate panel size. Board to panel border distance (sometimes called "Rail Edges") will be 0.2 inch for all sides. Some manufacturers need panel border in the board outline layer, so we will also check Show Panel Border box. Click OK.



We can see only the boxes with "Copy #" text on the design area, but in the Print Preview dialog box ("File \ Preview" from the main menu), while printing or exporting Gerber, DXF, or N/C Drill file, you will see a final layout.

It is possible to exclude some objects from panelizing (for example, holes or shapes). To exclude any object from panelizing, right click it and check Do Not Panelize box. This item is available only if panelizing is ON.

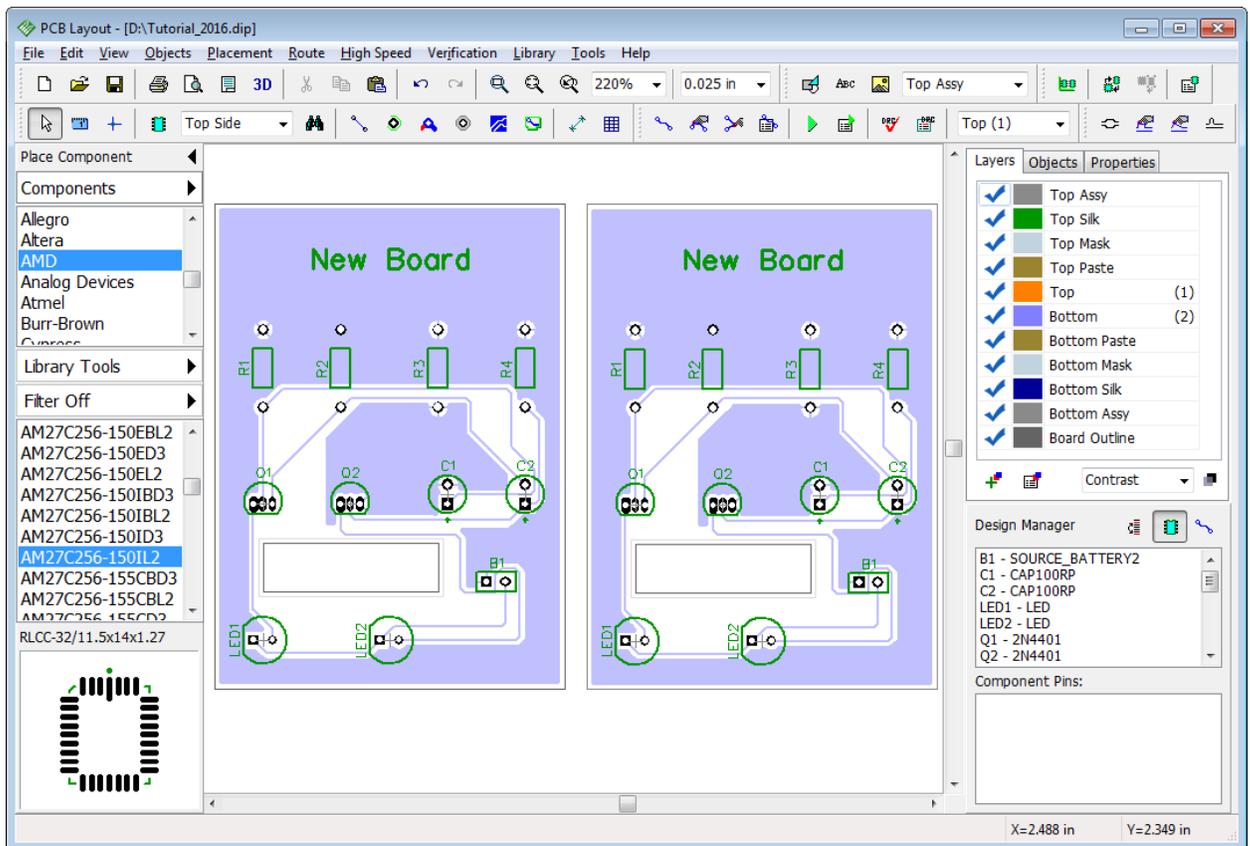


*Notice that panelizing works only if PCB has a board outline.*

Now open Panelizing dialog box and change number of columns and rows to "1" to remove copies and let us practice with panelizing different boards on a single layout.

### Panelize different layouts

Check "Edit \ Keep RefDes while Pasting" item from the main menu. Then select all objects ("Ctrl+A") of your second layout and press "Ctrl+C" to copy it. Then right click in the empty area (this will be an upper-left corner of the second board) and select Paste from the submenu.



We got the second copy of our PCB (but it can be a different PCB if you want). Reference Designators will not change.

*Notice that common board outline should be created, board cutout shapes may be required for correct manufacturing. Contact Bay Area Circuits for details.*

## 2.16 Printing

We recommend to use print preview dialog box to print the PCB, select "File \ Preview" from the main menu or press  button on the standard toolbar. Notice that we didn't describe creating Titles in Designing PCB section. If you want to display titles, select "File \ Titles and Sheet" from the main menu and select "ANSI A" in the "Sheet Template" box, check Display Titles and close the dialog box before opening the Print Preview panel.

More detailed information about Titles and Sheets as well as creating and editing titles with Title Block Editor in the PCB Creator Help ("Help \ PCB Layout Help" from the main menu).

You can customize the view of your PCB by checking / unchecking boxes in the Objects group. Sheet is automatically scaled to fit the preview panel. To change preview scale from 10% to 600% use the Scale box. Use your mouse to move the sheet in the preview area.

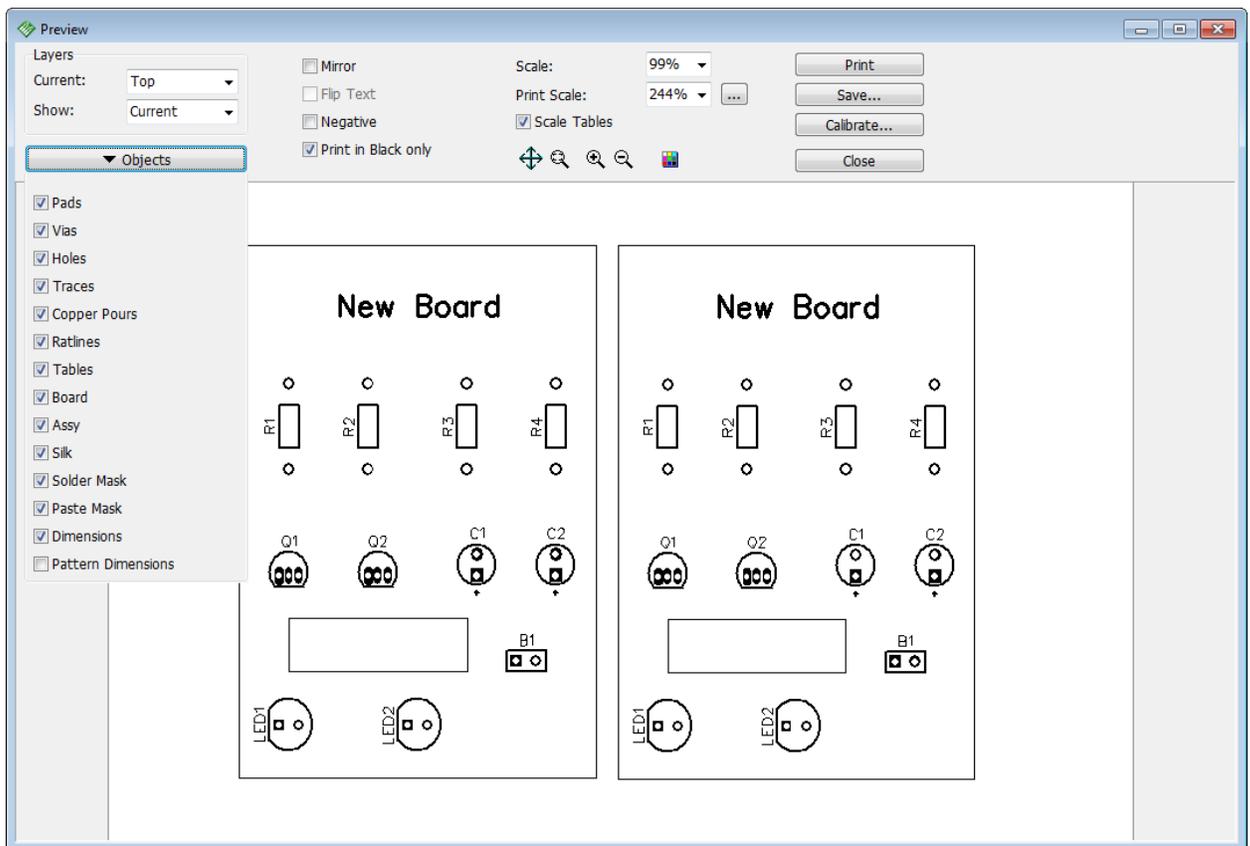
Use Print Scale drop-down list or "Zoom In" / "Zoom Out" buttons to change the scale of the layout on the sheet and press  button to move the design on the sheet.

In the upper-left you can select current Signal/Plane layer and layer display mode. If you want to get a mirrored PCB and/or text, check the Mirror and/or Flip Text boxes (Flip Text box is disabled if "View \ Flip Text Automatically" main menu item is ON).

Press Print button to print the layout. Press Save if you want to save image in BMP or JPEG file. Small button with colors to the left from the Zoom Out tool allows the user to define print colors ("View \ Colors...\ Print Colors" from the main menu).

*Notice that only layers with default color depend on the color scheme.*

For printing all in black and white without changing layer colors, check Print in Black only box.



**For hobbyist's attention.** Be aware that laser printers often introduce some degree of dimensional distortion due to heat expansion of the paper. It depends on your laser printer and the quality of the paper used. For most people it's not important. However, one way to cope with this issue is to preheat paper by running it through the printer, without printing on it (for example, you can print just a dot in the corner). For ink-jet printers this is not the issue, since ink-jet technology does not heat paper. Laser printers do not always distort image visibly, but you have to be ready that it can happen. You can use Calibration feature in the print preview dialog box to minimize heat distortions.

There are two methods of prototyping a PCB at home: using a TT (Toner Transfer) or UV exposure. TT is definitely a method for a laser printer and UV exposure is better with ink-jet printers.

Close the Print Preview dialog box and **Undo** several times to remove the second PCB and return the board without panelized copies. Then save the project.

## Part 3: Board Manufacturing

### 3.1 Order PCB

Order PCB manufacturing from the California-based [Bay Area Circuits](#). No need to bother with various manufacturing formats, just enter few details and manufactured board will be delivered to your place. Go to "File \ Order PCB from Bay Area Circuits" from the main menu. Review board parameters, specify quantity, manufacturing time, shipping address, name, phone, email, and some additional details. Price will be automatically calculated. Notice that more prototypes and longer manufacturing period significantly reduce cost per unit.

Order PCB from BayArea Circuits

Section	Field	Value
Board Parameters	Board Size:	1.8 x 2.5 in
	Number of Layers:	2
	Solder Mask Color:	Green
Standard Options	- 0.062 thick	
	- FR4 (standard temp for 2 layers, high temp for 4 and 6 layers)	
	- 5/5 min trace and space	
	- 0.010 smallest finished hole size	
	- HASL Finish	
	- Solder Mask Both Sides	
	- Silk Screen Both Sides if wanted	
Additional Options	<input type="checkbox"/> No Solder Mask (2-layers only)	
Comments		
Quantity	Quantity:	5
	Turn Time:	10 days
Price	Per Unit:	\$11.75
	Total:	\$58.73
Order Details	First and Last Name *	John Doe
	Company	*****
	Phone *	+1*****
	Country *	United States
	City *	Sacramento
	Address *	
	State *	California
	Zip Code *	
	Email Address *	John_Doe@*****.com
	Confirm Email Address *	John_Doe@*****.com

Close Place Order...

Press the Place Order button to open the order page in your web browser, review the total cost, including shipping. Online payment via PayPal.

DRC checks your PCB automatically. If there are errors, we recommend to carefully review and fix them. Please do not allow for any ambiguity, especially related to Solder Mask and Solder Paste settings. Describe it in the Comments section, or contact [Bay Area Circuits](#) to clarify any of the question that you think may arise related to your design.

Bay Area Circuits will send you the Gerber files for the circuit board with your order.

**Email: [support@bacircuits.com](mailto:support@bacircuits.com)**

**855-811-1975 FREE (toll free)**

**510-933-9000 (local)**

**510-933-9001 (fax)**

**Corporate Headquarters**

**44358 Old Warm Springs Blvd**

**Fremont, CA 94538**

Congratulations! You have finished designing a simple project with PCB Creator.

Please, save your Schematic and PCB files if you want. It took longer to read this tutorial than to actually create a project. :-)

## 3.2 Upgrade to DipTrace

If you want to build the circuit boards out of 1000 pins limit, or need a direct Gerber / Gerber X2 / ODB++ manufacturing export capabilities, go to "File / Export / Add Gerber and Export Capabilities" from the main menu and then select "[Upgrade to DipTrace](#)". [DipTrace Standard](#), [DipTrace Extended](#), and [DipTrace Full](#) editions provide: wide export capabilities, high-density and high-speed shape-based autorouter, as well as other features for a truly unlimited PCB design.

DipTrace supports all PCB Creator schematics, boards, and libraries, so you will not lose your previous work.

Switching to DipTrace means absolutely no learning curve, because both software packages are very similar.

## Part 4: Creating Component Libraries

In this part of the tutorial we will teach you how to create component and pattern libraries with PCB Creator Component and Pattern Editors.

*In this tutorial we often refer to components as patterns or footprints (if talking about the PCB Layout), while in the Schematic components are often called symbols, component symbols, or schematic symbols, all means the same.*

*A regular component in PCB Creator consists of schematic symbol, pattern drawing and, possibly, a 3D model, all three represent the same entity, but on different stages of the design: schematic capture, PCB Layout, and 3D visualization / export respectively. Components and attached patterns are saved in files with \*.eli extension. PCB Creator also allows access to patterns separately from the components. Pattern libraries are stored in files with \*.lib extension. \*.wrl, \*.3ds, \*.iges, and \*.step files store 3D models. Patterns and schematic symbols can exist as separate stand-alone parts, but correct component always have all of them properly connected.*

*Different schematic symbols can have the same pattern (footprint) attached to each of them and vice versa, for example the same resistor can be in regular and SMD packages.*

*Component Editor is used to manage components - draw schematic symbols and connect them with pattern drawings (footprints). However, Component Editor does not edit footprints, use Pattern Editor for drawing them.*

In this section of the tutorial we will create a new component, starting from the pattern design.

In most cases you can find appropriate pattern in the standard libraries and attach it to a new component, but for educational purposes we will create it from a scratch.

## 4.1 Designing a pattern library

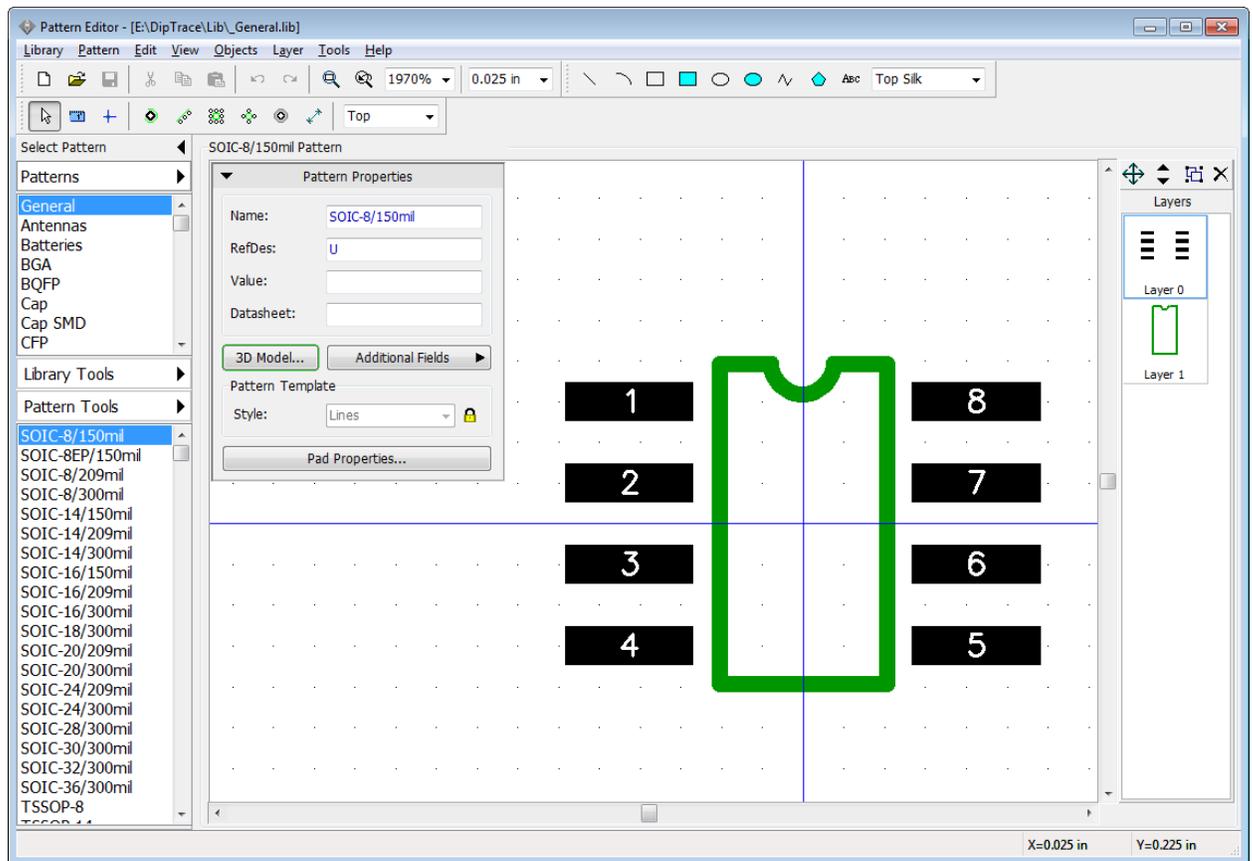
Open PCB Creator Pattern Editor, i.e. go to "Start \ All Programs \ PCB Creator \ Pattern Editor" on Windows machine.

### 4.1.1 Customizing Pattern Editor

First of all we need to show a center point and X,Y-axes, select "View \ Display Origin" from the main menu or press "F1" hotkey (if it is not displayed yet). Notice that you can change origin at

any time while designing a pattern ("View \ Define Origin \ ..." from main menu or press  button on the instruments toolbar). Origin is a zero point of the pattern when you place, rotate, or change pattern's position by coordinates in the **PCB Layout**.

Pattern Properties panel in the upper-left of the design area allows the engineer to design patterns by types or templates, define pattern attributes, attach 3D model, and change default pad settings. Panel can be hidden, minimized or moved during pattern design. To minimize the panel click the arrow in its upper-left corner. To hide / show this panel select "View \ Toolbars \ Pattern Properties" from the main menu.

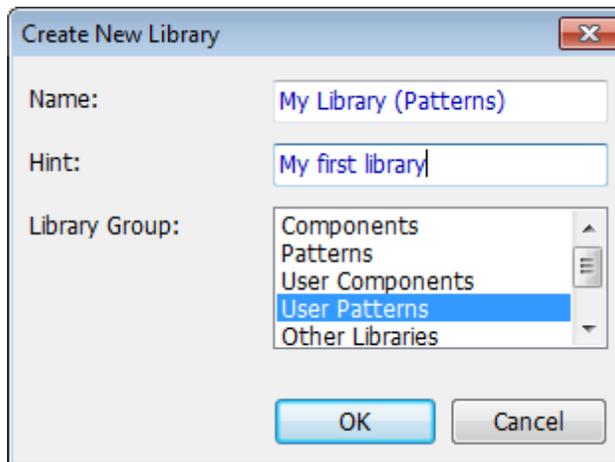


Use plus sign "+" and minus sign "-" keys or the mouse wheel for zooming in and out in the Component and Pattern Editors or change scale in the scale box on the instruments toolbar. Go to "View \ Pad Numbers \..." to show / hide pad numbers.

#### 4.1.2 Create/Save library

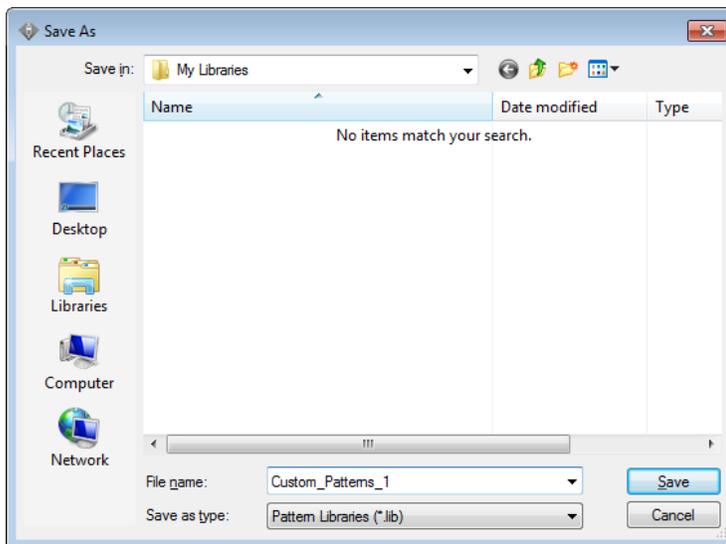
##### Create a library

Press "Library Tools \ New Library " on the Libraries panel on the left side of the screen. In the pop-up dialog box enter the library name, hint, and select the library group. We recommend to create this library in the User Patterns library group offered by default. Press OK.



The name of your library will appear on the Libraries panel, User Libraries library group will be selected automatically. Now please save newly created library in the separate file.

### Save library



Select "Library \ Save" from the main menu. We recommend to save user libraries in the My Libraries folder (Windows OS), which is offered by default, however you can select another location.

*PCB Creator does not allow for saving user libraries in the folder with standard libraries.*

Type in the file name (not shown in the PCB Creator) and press Save.

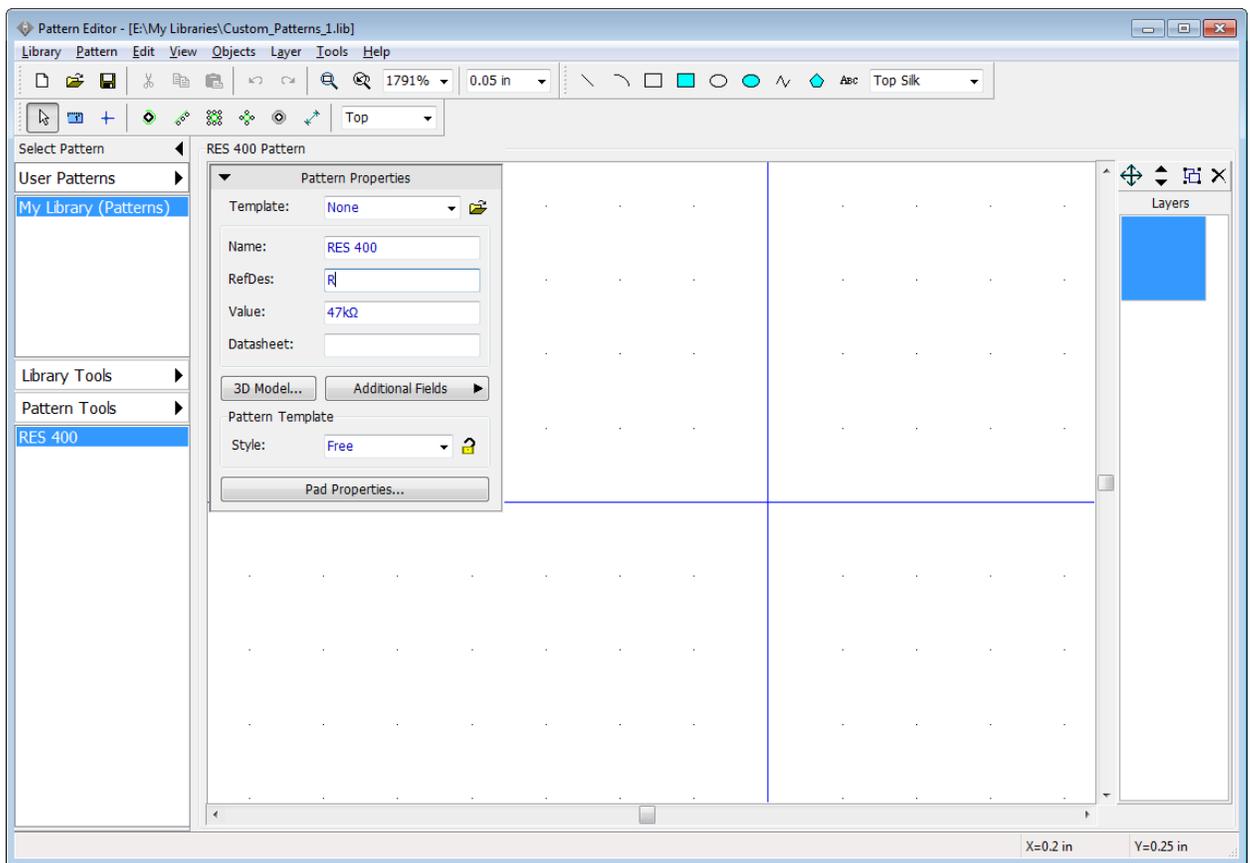
Now we have My Library (Patterns) pattern library, which belongs to the User Patterns Library Group and is saved on your computer. Notice that file has ".lib" extension, saying that this is a pattern library.

#### 4.1.3 Designing a Resistor (pattern)

We will design the first pattern of our library, it is going to be a resistor with 400 mils lead spacing.

First we need to name the pattern and fill up the basic description fields. Type "RES 400" into the Name field, "R" – into the RefDes field and "47k " – into the Value field of the Pattern Properties panel. In the PCB Creator Pattern and Component Editors you need to define just a basic RefDes (not a number index). For example, when you place several resistors – R1, R2, R3, etc.,

designators will be assigned automatically. Use Windows Special Characters Map to find and copy/paste special symbols for Ohms, Farads etc.

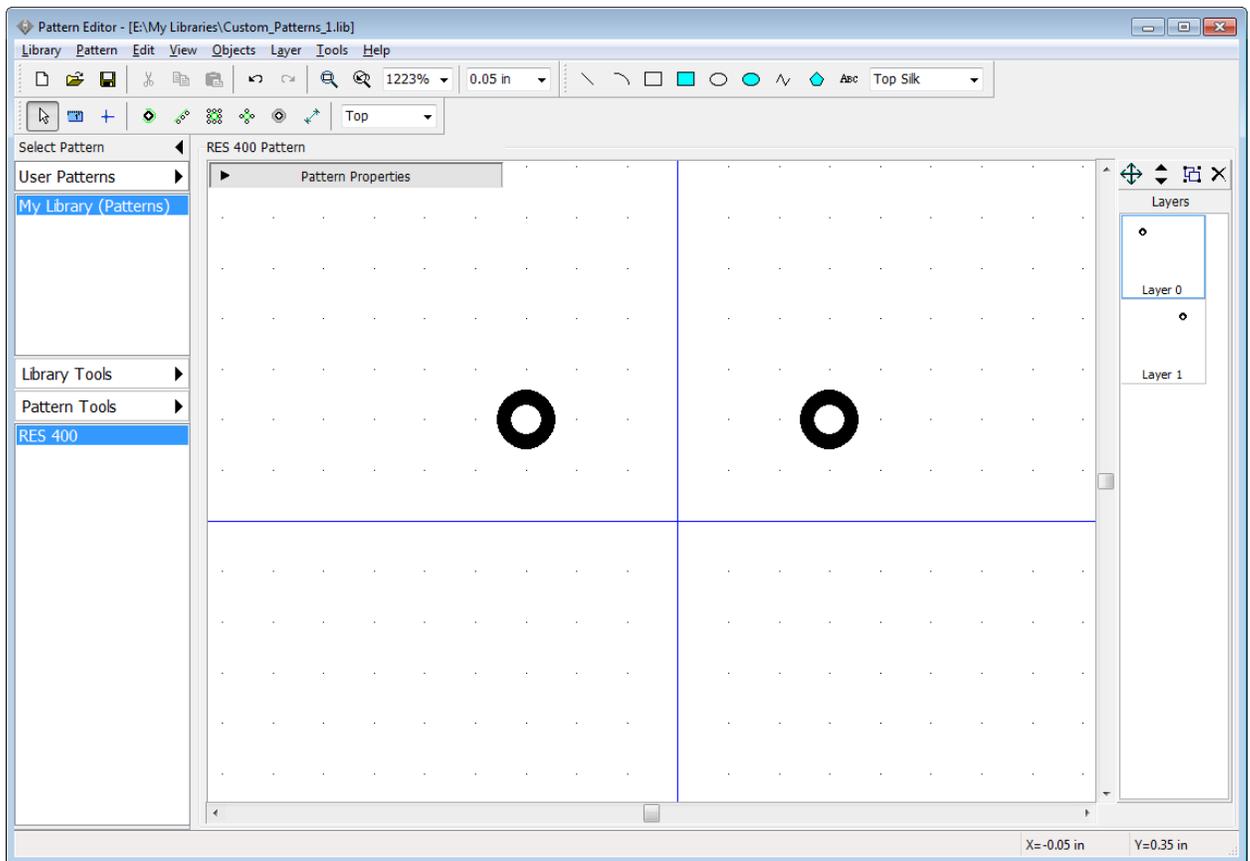


For this pattern we have used Free style, but it is faster to use Lines instead. You'll see how to do this in one of the subsequent topics.

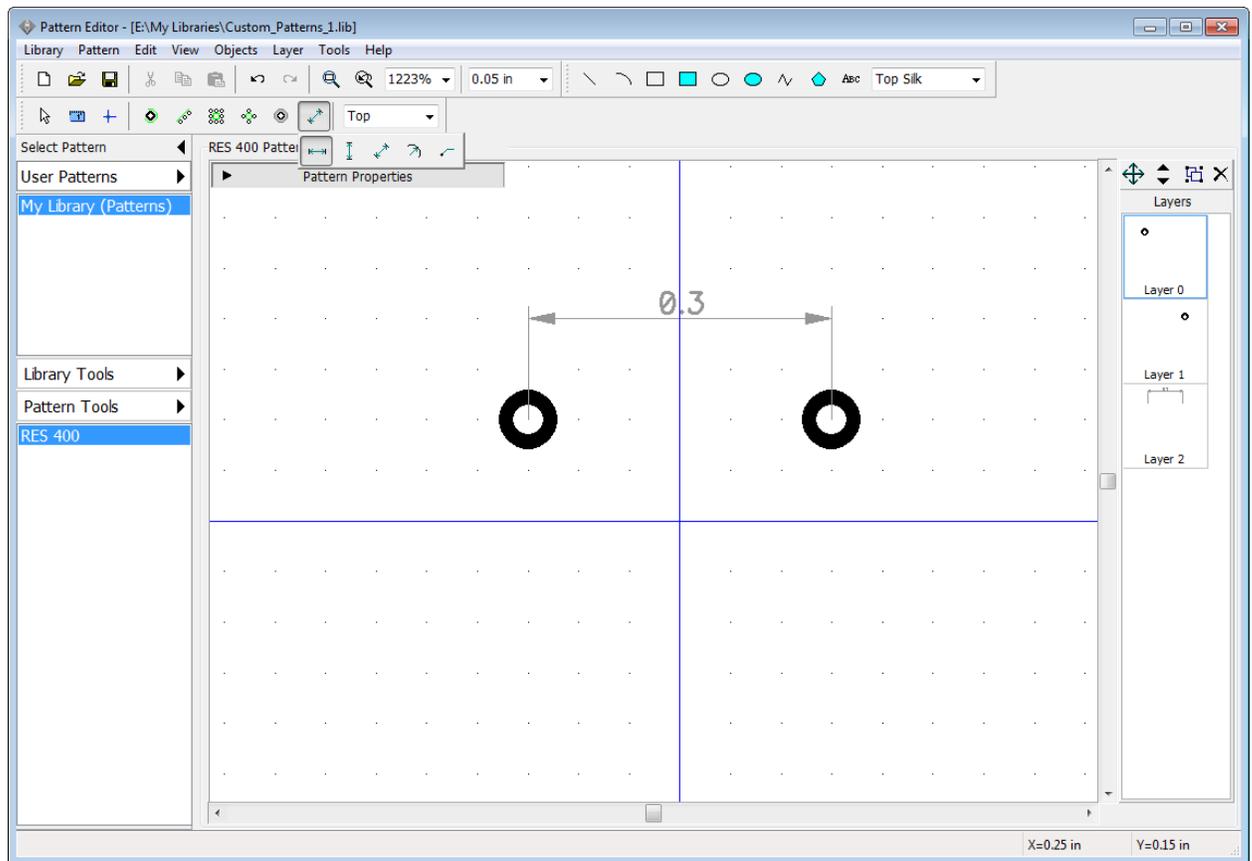
### Placing pads

Please make sure that "0.05 in" grid is selected (change measurement units in the "View \ Units"

main menu item) then minimize the Pattern Properties panel. Select the Place Pad tool (  button on the objects toolbar) and left click on the design area to place two pads like on the picture below. Right click to exit the placement mode.



Such placement is not an accurate method which requires manual correction of pad coordinates. Let's place a dimension line, this will make editing more simple and visual. Select "Objects \ Place Dimension \ Horizontal" from the main menu or Place Dimension \ Horizontal tool on the objects toolbar, left click in the center of the first pad, then in the center of the second pad, move the mouse pointer a bit upwards and left click one more time to place a dimension line. The key points of the object are highlighted when you hover over them with your mouse. Dimension pointer is attached to the key points and is recounted automatically when you move or resize objects.



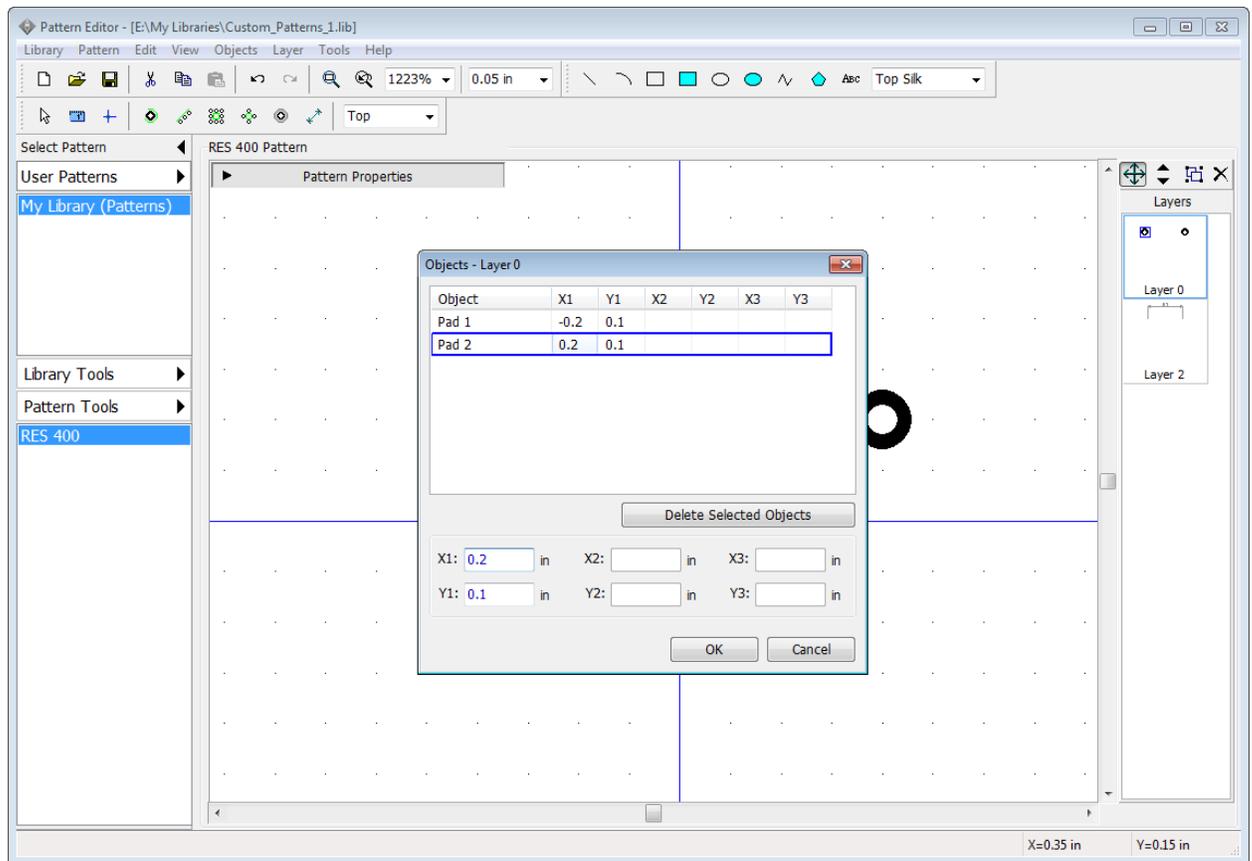
Right click on the dimension line and select Properties from the submenu to change Layer, Units, Arrow Size etc. Drag and drop the dimension line like a regular object if needed.

### Pattern layers

On the right side of the screen there is a layers list. One layer per object on the design area.

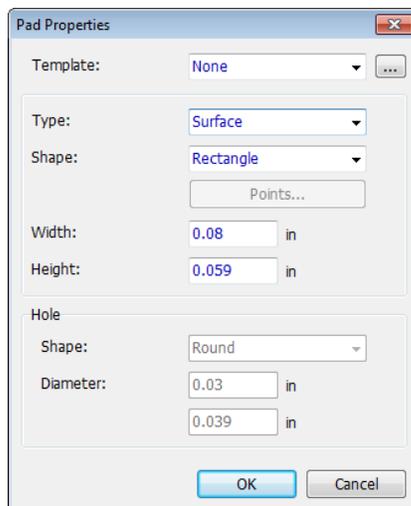
*These are the logical layers for editing (not signal or silk layers that you used to work with in the PCB Layout).*

Select layers 0 and 1 (hover over the "Layer 0", hold down the left mouse button, move the mouse cursor to "Layer 1", then release the mouse button). Now merge this layers by selecting "Layer \ Merge Layers" from the main menu or by pressing  button on the layers panel. You have made a single layer with two pads in it; double click it to open the Layer Objects dialog box. Select the pad with incorrect coordinates and change X coordinates to make 400 mils (0.4 inch) distance between the pads then click OK to close the dialog box.



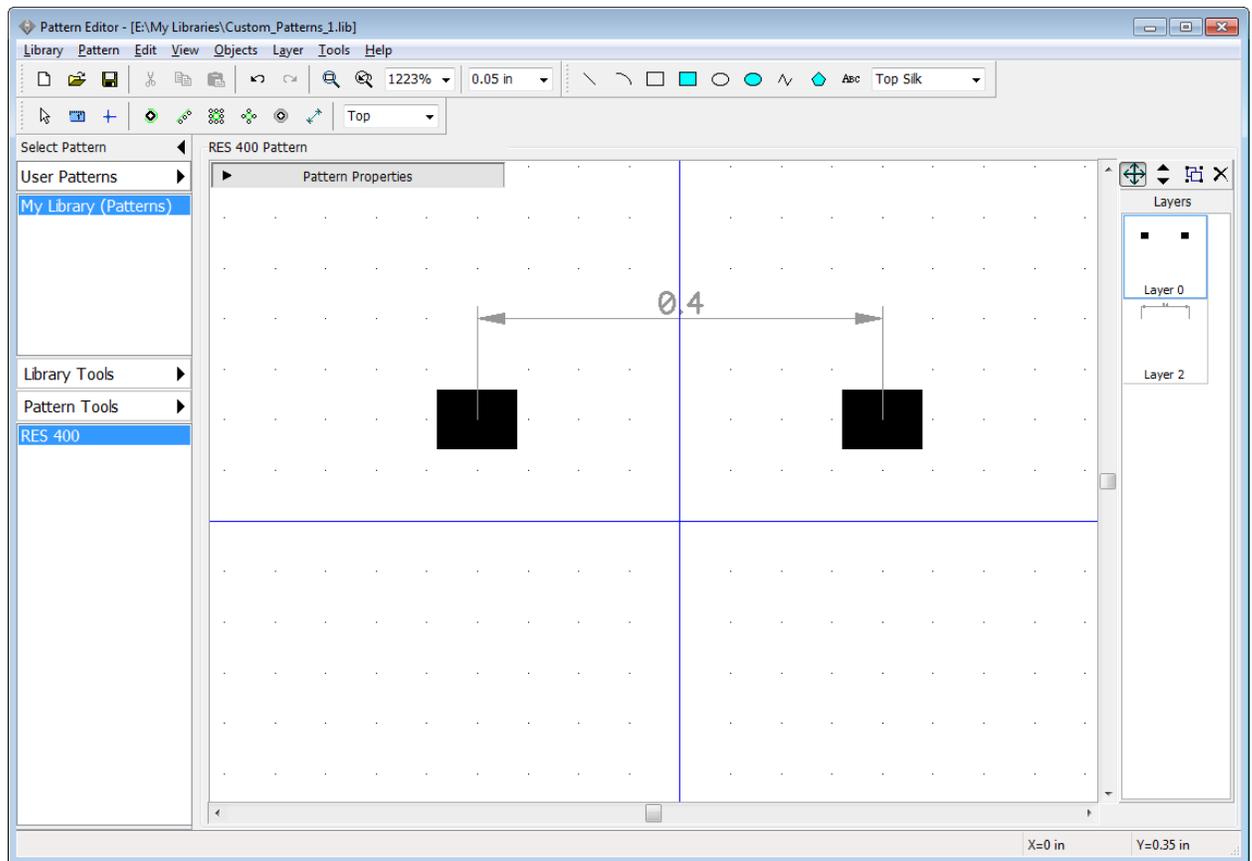
You can see that dimension line is recounted automatically and now displays correct distance value.

### Pad properties



Pads can have default or custom properties. Default ones are applied to all pads of the pattern, custom – only to selected pad/s. To change default pad settings select "Pattern \ Pad Properties" from the main menu or press the Pad Properties button on the Pattern Properties panel. In the pop-up dialog box change the pad shape: Ellipse, Oval, Rectangle or Polygon (click Points for polygonal pad customization). You can make round or oval holes and change the hole diameter (for Through pads only). Pad templates allow for quick changes in pad settings in different dialog boxes of the Pattern Editor and PCB Layout.

Change the pad shape to Rectangle, width to "0.08 in", height to "0.059 in" and Type to Surface, then click OK to apply changes. Measurement units can be changed (mil is 1/1000 inches).



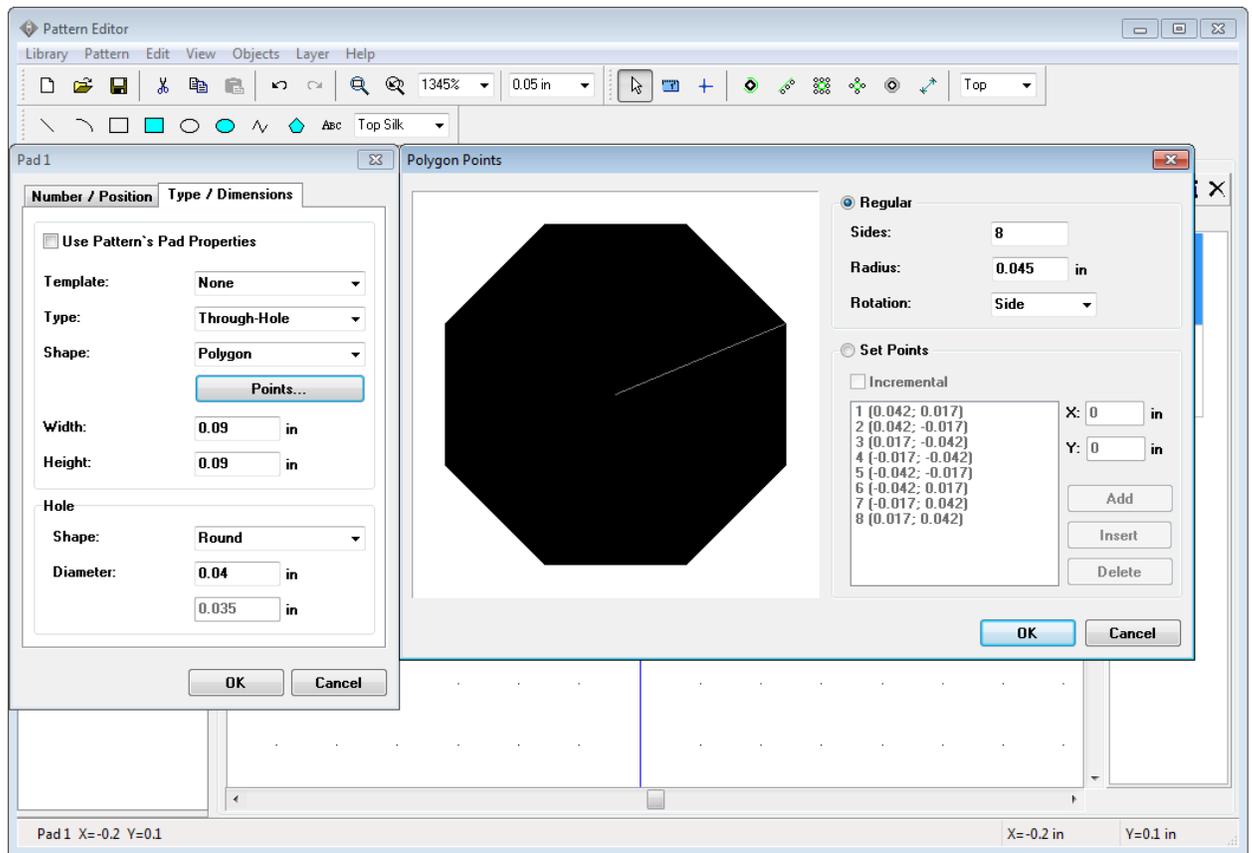
Notice that you can select the side for the surface pads, i.e. place them on the bottom side of the PCB. Select pad(s), right click on one of them and select Change Side from the submenu. Current side for placing new pads and shapes can be selected on the objects panel (drop-down box with "Top" text).

We need the first pad to have custom pad properties. Right click on the first pad and select Properties from the submenu (if the pad is not highlighted while hovering over it, you're not in the default mode – right click on any free spot or press  button on the objects toolbar). In the pop-up Pad Properties dialog box select the Type / Dimensions tab and uncheck Use Pattern's Pad Properties box to enable pad's custom settings.

### Polygon pads

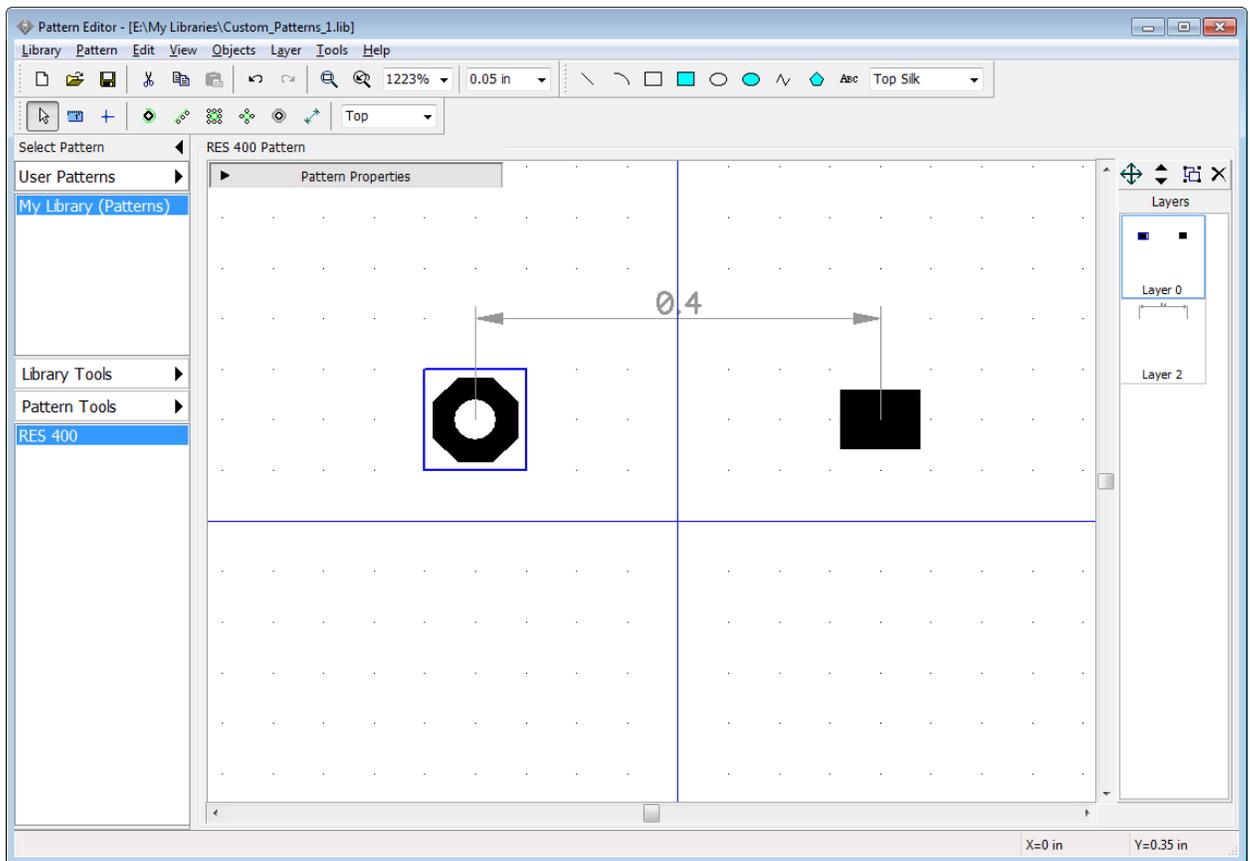
Let's create a hexagonal-shaped pad. The fastest way to achieve that would be selecting an appropriate ("Polygon1") template, but let's practice a bit. Do not assign any template, change type to Through-Hole, shape to Polygon, width and height to 0.09 in and pad the hole diameter to 0.04 in then press Points to open the Polygon Points dialog box. Here you can create regular or custom polygon shapes that are not available in the templates drop-down list.

Select: "Regular", "8 Sides" and specify 0.045 in radius. Custom shapes are edited with the table below or visually in the preview field (drag and drop the points).



Press OK to apply the polygon shape.

Pad coordinates and direction are changed in the Number / Position tab of the pad properties dialog box. Do not change anything in this tab. Press OK. Changes will be applied to all selected pads (not in our case).



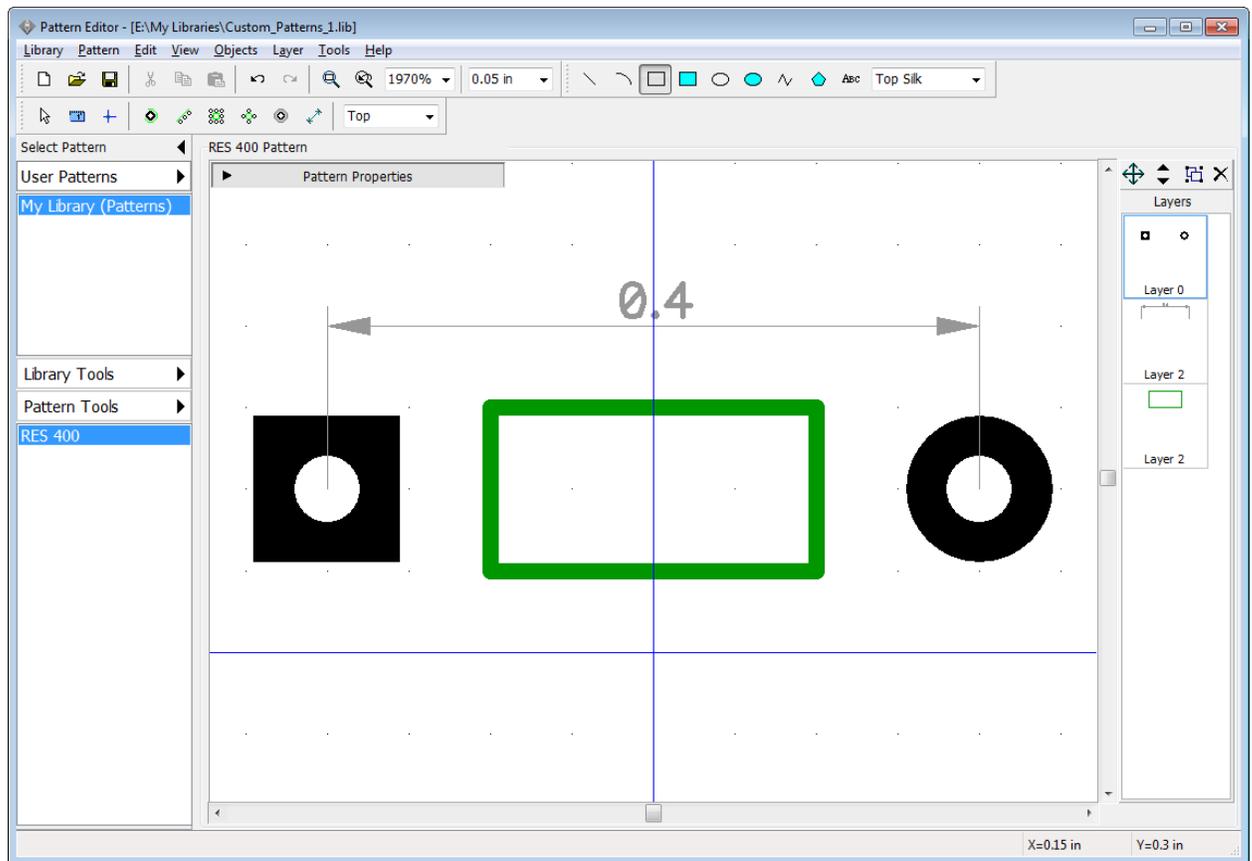
There is also another way to create polygon pads. Select the Polygon tool on the drawing toolbar and draw a shape directly on the design area. Make sure that shape is on the signal layer (select Signal from the drop-down list on the drawing toolbar prior to drawing or change the shape layer in the shape properties dialog box – right click directly on the shape on the design area). When the shape on the signal layer is ready, right click it and select Convert to pad from the submenu. This options is visible only for Polygon drawing tool.

Please define the following properties for the pads (in inches):

first pad – 0.09x0.09, Through-Hole, Rectangle, hole diameter – 0.04;

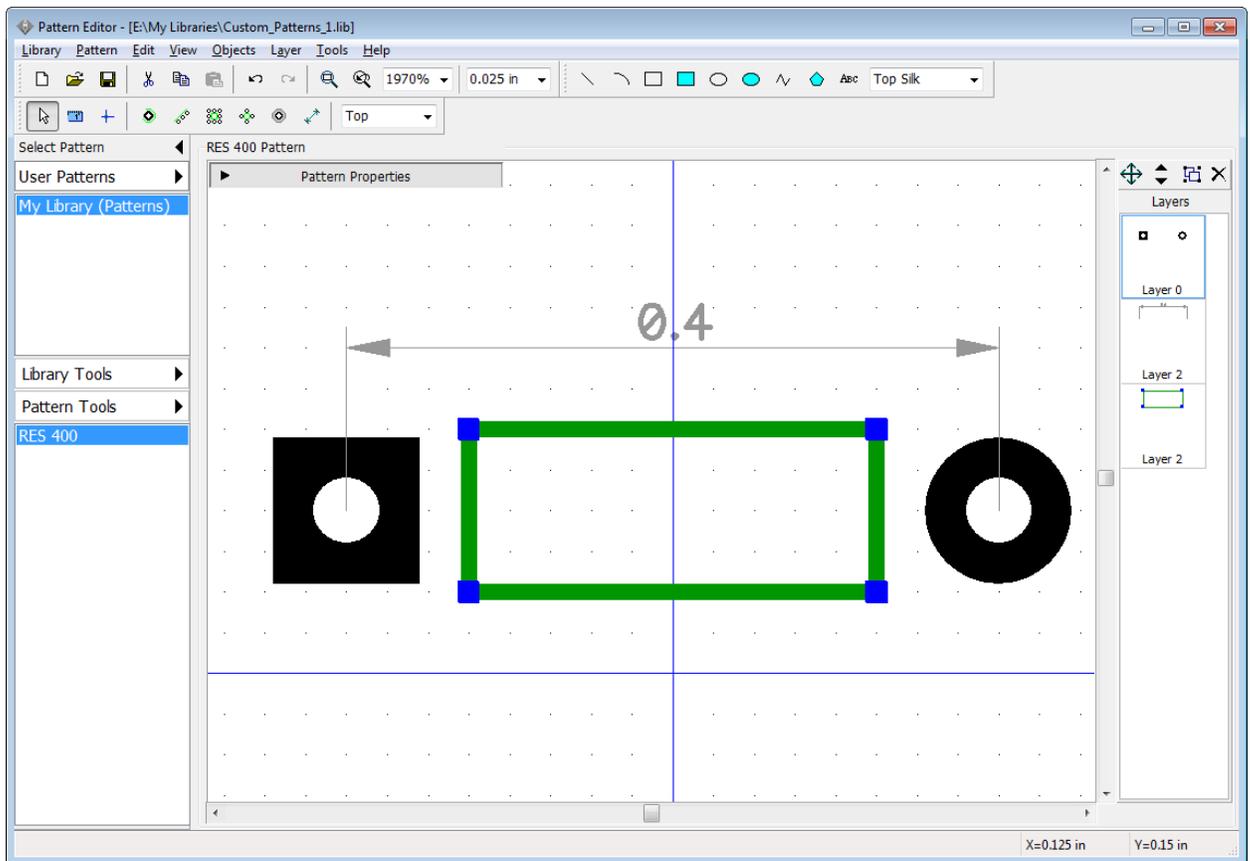
second pad – 0.09x0.09, Through-Hole, Ellipse, hole diameter – 0.04.

Now let's place a silkscreen for this resistor. Select the Rectangle button on the drawing panel, make sure "Top Silk" layer is selected. Place a rectangle by clicking on two key points on the design area.



Now disable rectangle placement mode (right click on the free area or press the Default Mode button).

Silk shape looks a bit small for this pattern. There are several ways to change it: 1) Select "Layer \ Objects" from the main menu and enter new coordinates; 2) Right click on the shape, select properties and change points in the pop-up dialog box; 3) Drag-and-drop shape's key points. We will use the last method, but at first we need to change the grid size to 0.025 inch with "Ctrl+minus sign" hotkeys or the grid box on the instruments toolbar. Then hover over rectangle's key points and resize the shape (mouse cursor shows possible directions).

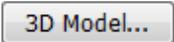


Center the pattern by selecting "Edit \ Center Pattern" from the main menu or "Ctrl+Alt+C". The resistor is ready.

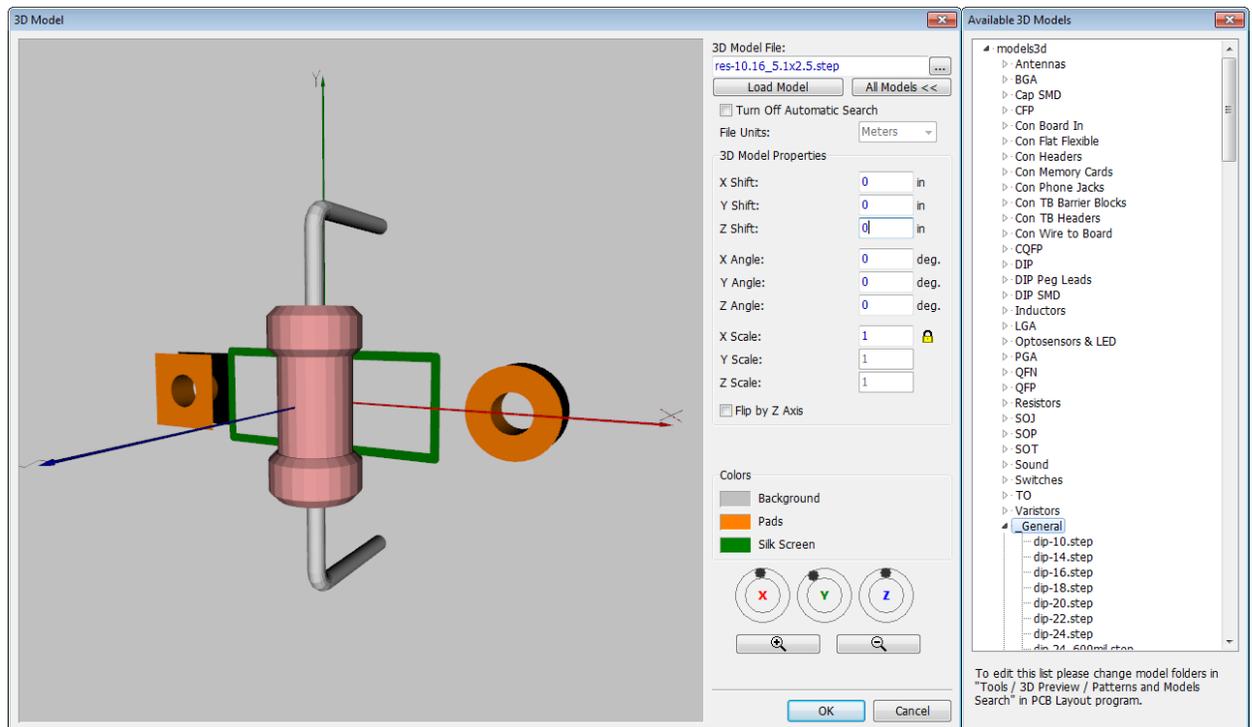
Try to rotate and mirror the first pattern of your library, select "Edit \ Rotate Pattern" to rotate and "Edit \ Vertical Flip", "Edit \ Horizontal Flip" to mirror it.  
Press Save button on the standard toolbar. We will attach a 3D model for this resistor in the next topic of this tutorial.

#### 4.1.4 Attaching a 3D Model

Please make sure that you have downloaded and installed a **free 3D Models package** from the [Bay Area Circuits website](#), which contains more than 6,500 3D models for various components.

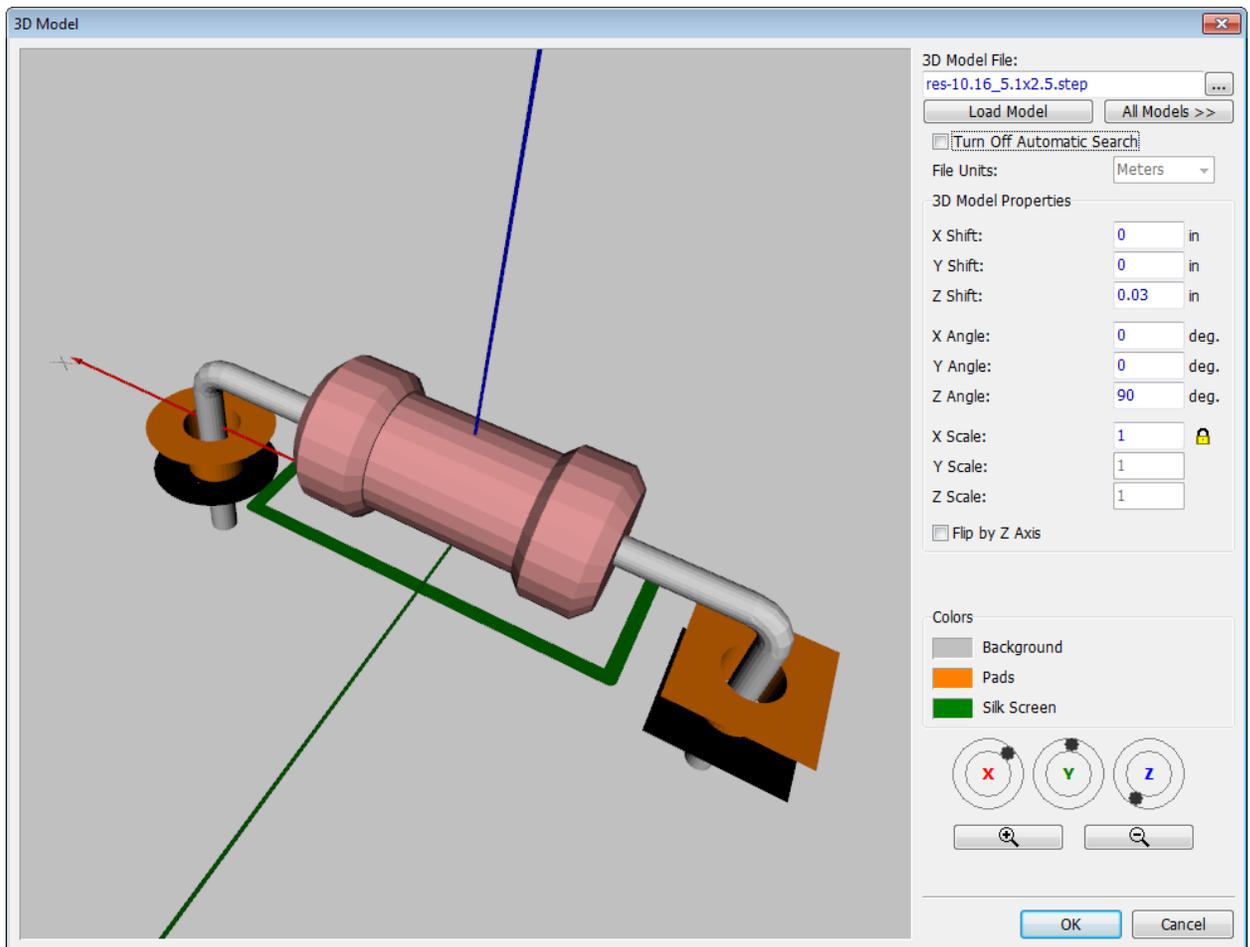
When the component footprint (pattern) is ready we can attach a 3D Model. Press  button on the Pattern Properties panel.

In the pop-up dialog box press the All Models button and the list of all available 3D models will pop up. 3D models are sorted by categories (libraries). We will probably find appropriate resistor model in the "\_General" library. Scroll it down to find "res-10.16\_5.1x2.5.step" model, click it and it will appear on the footprint. Sometimes you might need to map 3D model respectively to the footprint (like on the picture below).



### Mapping a 3D model

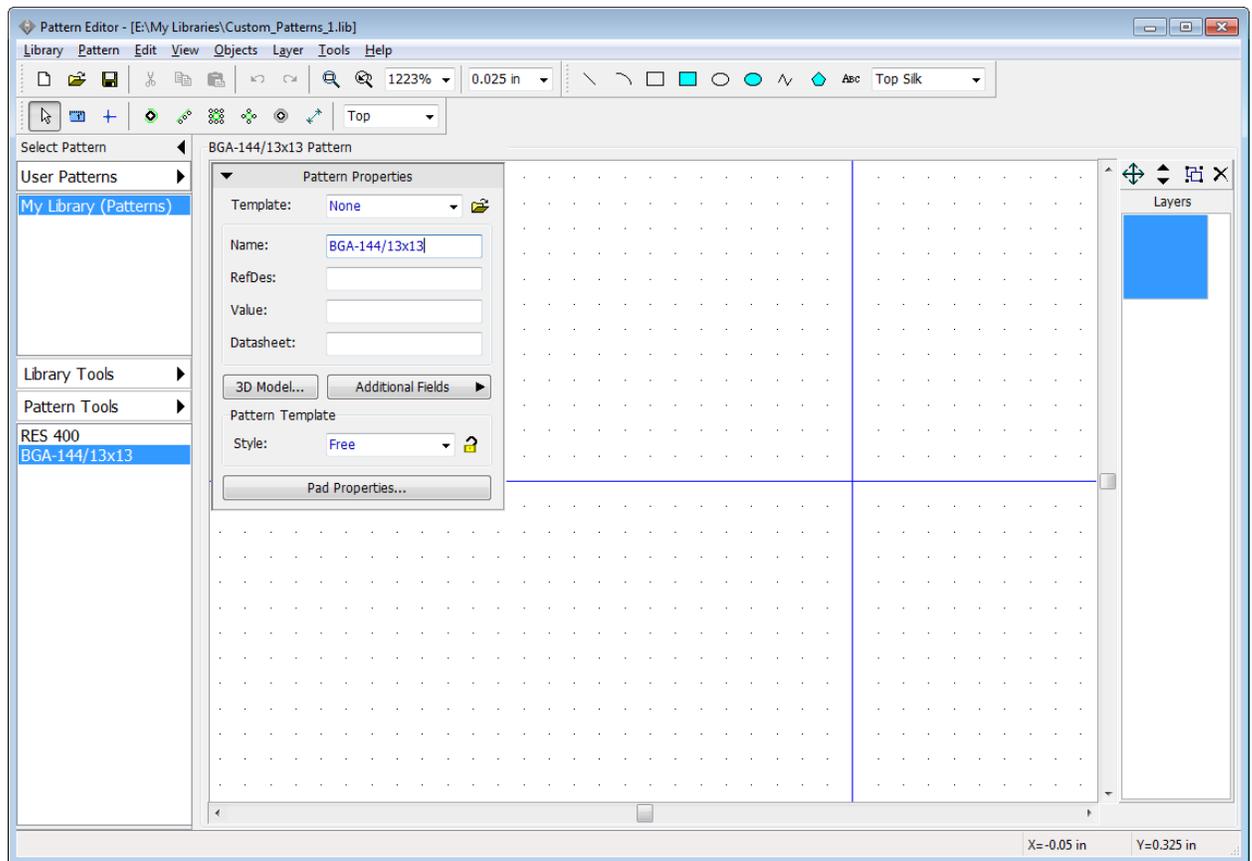
PCB Creator automatically places a 3D model to fit the pattern's drawing, however, sometimes you may need to adjust 3D model location or the scale. Just enter appropriate values into the corresponding fields of the 3D Model Properties section (shift, angle, and scale for each axis). In our case we see that we need to rotate the resistor 90 degrees and shift it up a bit. Specify 90-degree Z Angle and 0.03 inch Z Shift. You can rotate 3D model in three axes, zoom in and out and pan 3D Model preview with your mouse (left, right, and wheel clicks) or using X, Y, Z wheels at the bottom of the dialog box.



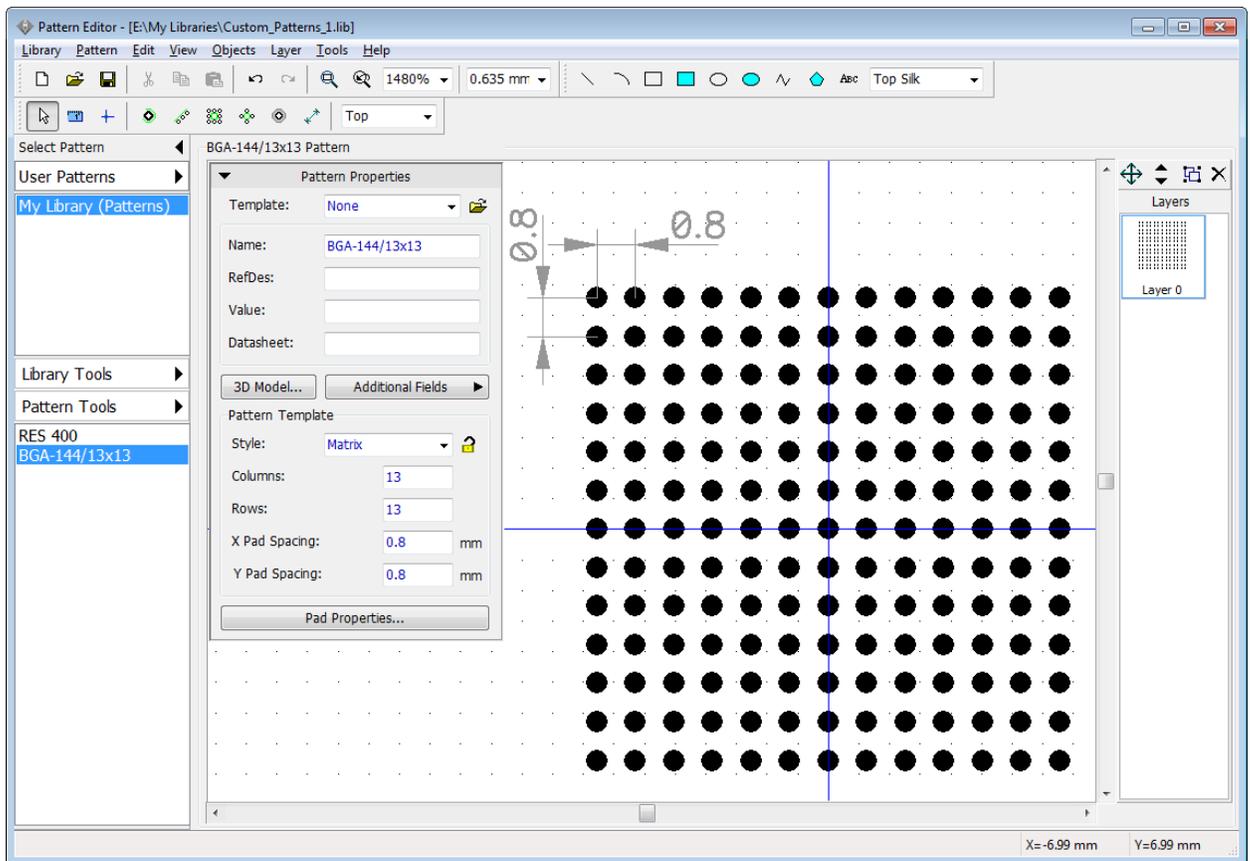
Press OK to attach a 3D model and then save the pattern library. More details about [PCB Creator 3D Module](#)<sup>[233]</sup> later in this tutorial and in the PCB Creator Help ("Help \ Pattern Editor Help" from the main menu).

#### 4.1.5 Designing BGA-144/13x13

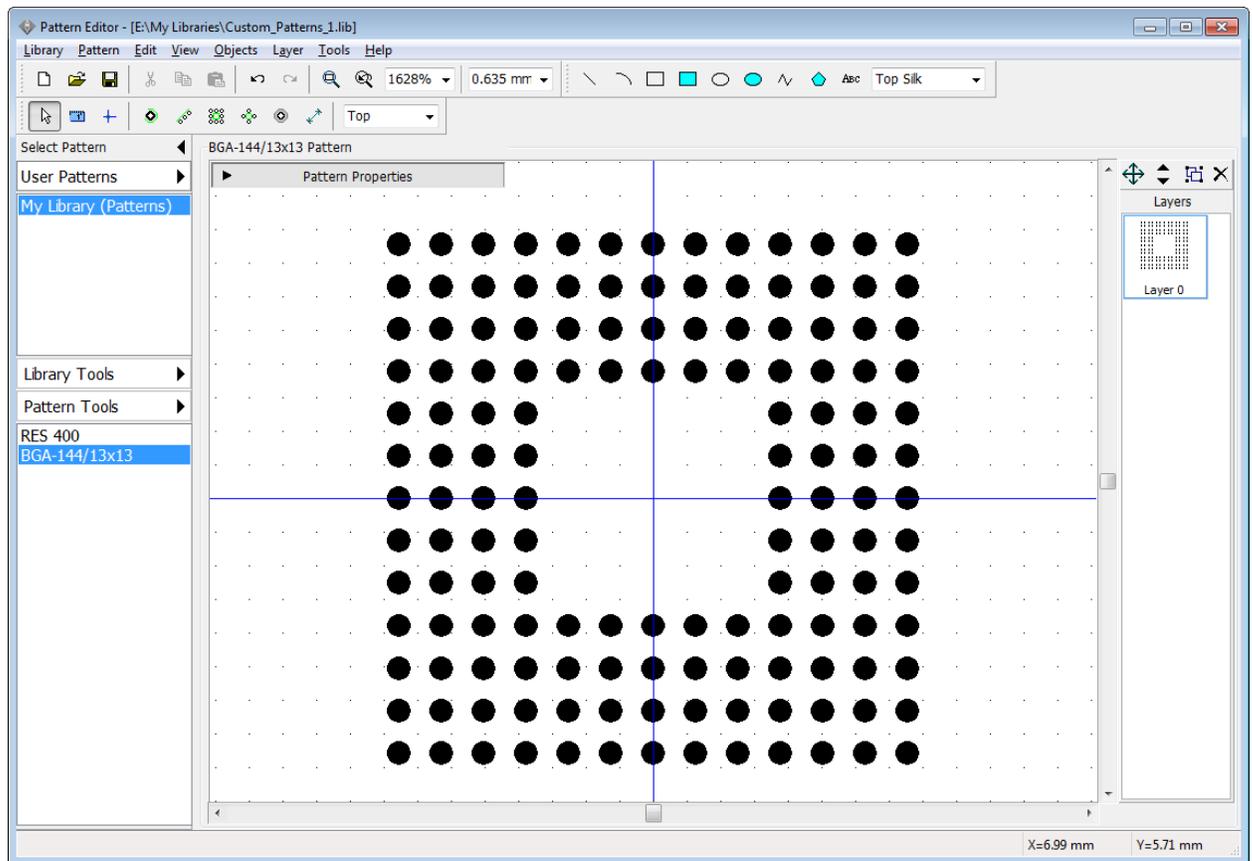
Press "Pattern Tools \ Add New Pattern to My Library (Patterns) Library" on the Libraries panel (to the left of the design area). This will add an empty pattern to the library. The pattern is selected. We will create BGA-144/13x13 (x0.8\_10x10) pattern using available pattern types and automatic pad numeration. Maximize pattern properties panel and type in the pattern's name.



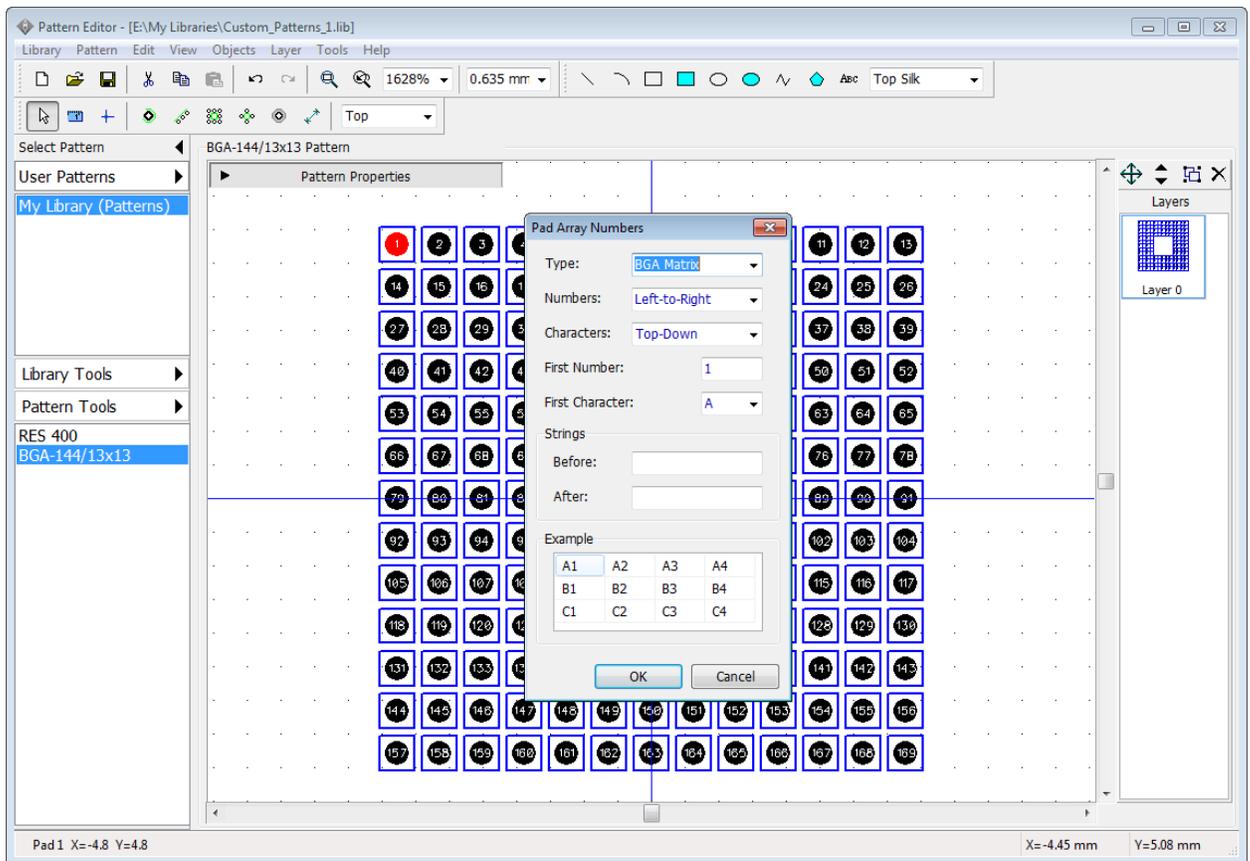
Change units to "mm", select "View \ Units \ mm" from the main menu. Then go to "Pattern \ Pad Properties" from the main menu and define: "Type: Surface", "Shape: Ellipse", "Width: 0.45 mm", "Height: 0.45 mm". Press OK to apply changes to default pad properties. On the Pattern Properties panel set: "Style: Matrix", "Columns: 13", "Rows: 13", "X Pad Spacing: 0.8 mm", "Y Pad Spacing: 0.8 mm". 13x13 pad matrix and pad spacing dimensions appears on the design area.



Click the Lock Properties button right next to the Style box on the Pattern Properties panel to prevent accidental changes. Minimize the Pattern Properties panel. Pan design area if necessary with the right mouse button or zoom with the mouse wheel. For BGA-144/13x13 pattern we should delete 5x5 rectangle of pads in the center, like on the picture below. Select these pads using the box selection (move the cursor to the upper-left corner, hold down the left mouse button, move to the bottom-right and release the button) then press "Delete" key on the keyboard to delete selected pads.

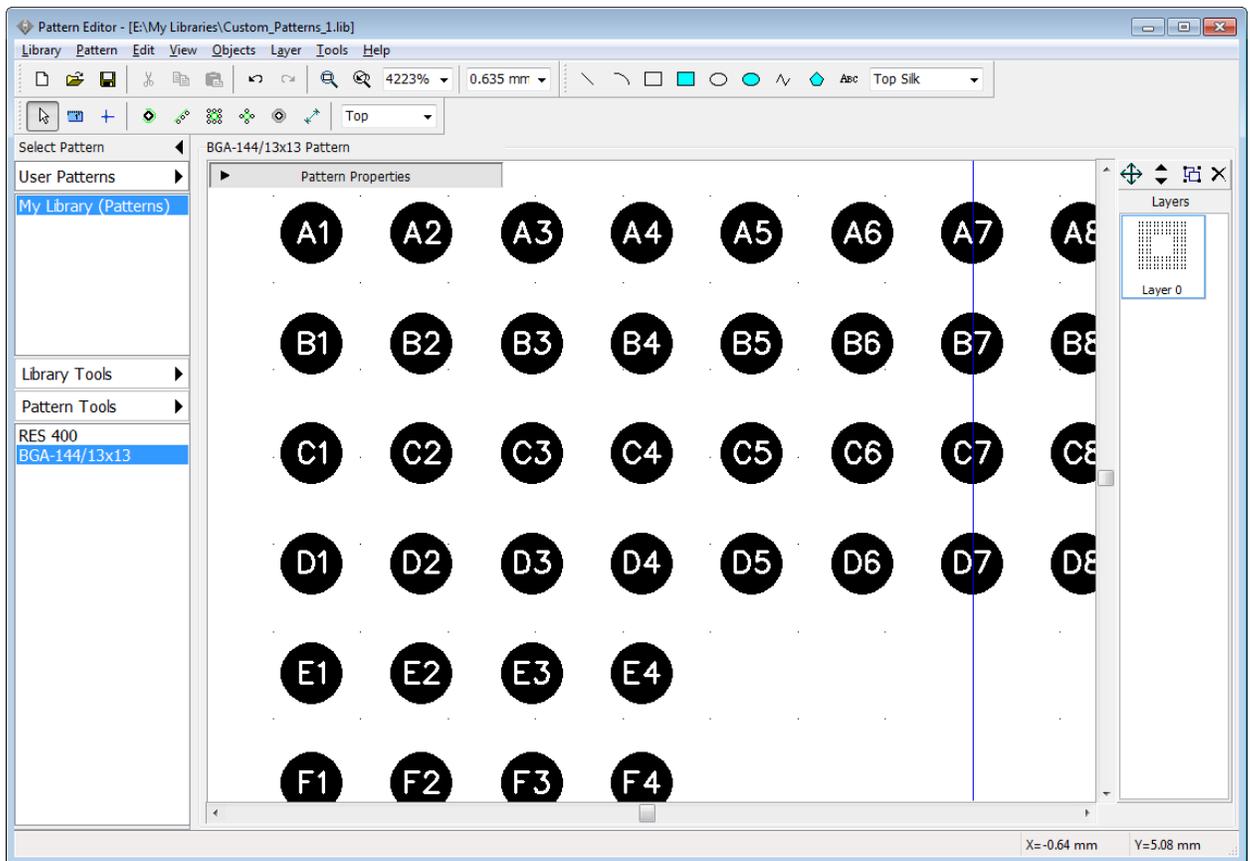


Select "View \ Pad Numbers \ Show" from the main menu to display pad numbers. Notice that our matrix has 1 – 169 numbers with some missing numbers in the center. This is not right. BGA pads should be A1,A2, A3 etc. Select all pads ("Ctrl+A" or a box selection), right click on one of the pads, and choose Pad Array Numbers from the submenu. In the pop-up dialog box select "Type: BGA Matrix", keep other settings and press OK.

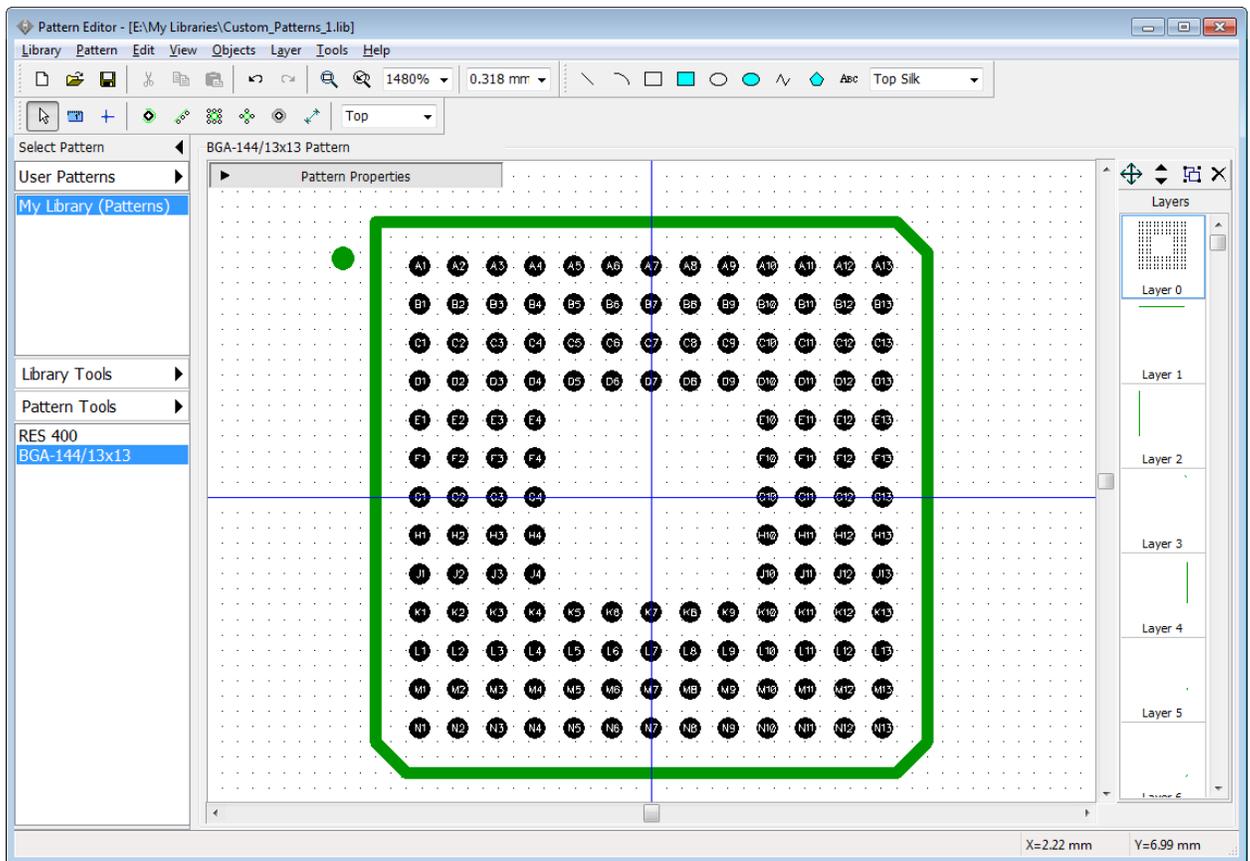


Now pad numeration is correct.

*Notice that for the Contour type of numeration first pad will be the one you right clicked on. This allows the user to numerate contours (QUAD patterns) starting from the upper-left, center, or any other pad of the pattern.*



Now draw the silkscreen for the pattern (like on the picture below), using the tools on the drawing toolbar. Grid can be changed with "Ctrl+plus sign", "Ctrl+minus sign" or turned OFF/ON with "F11" hotkey. Objects can be moved with drag and drop or by Move Layer feature (select layer in the list on the right side and drag-and-drop all objects of this layer directly on the design area).

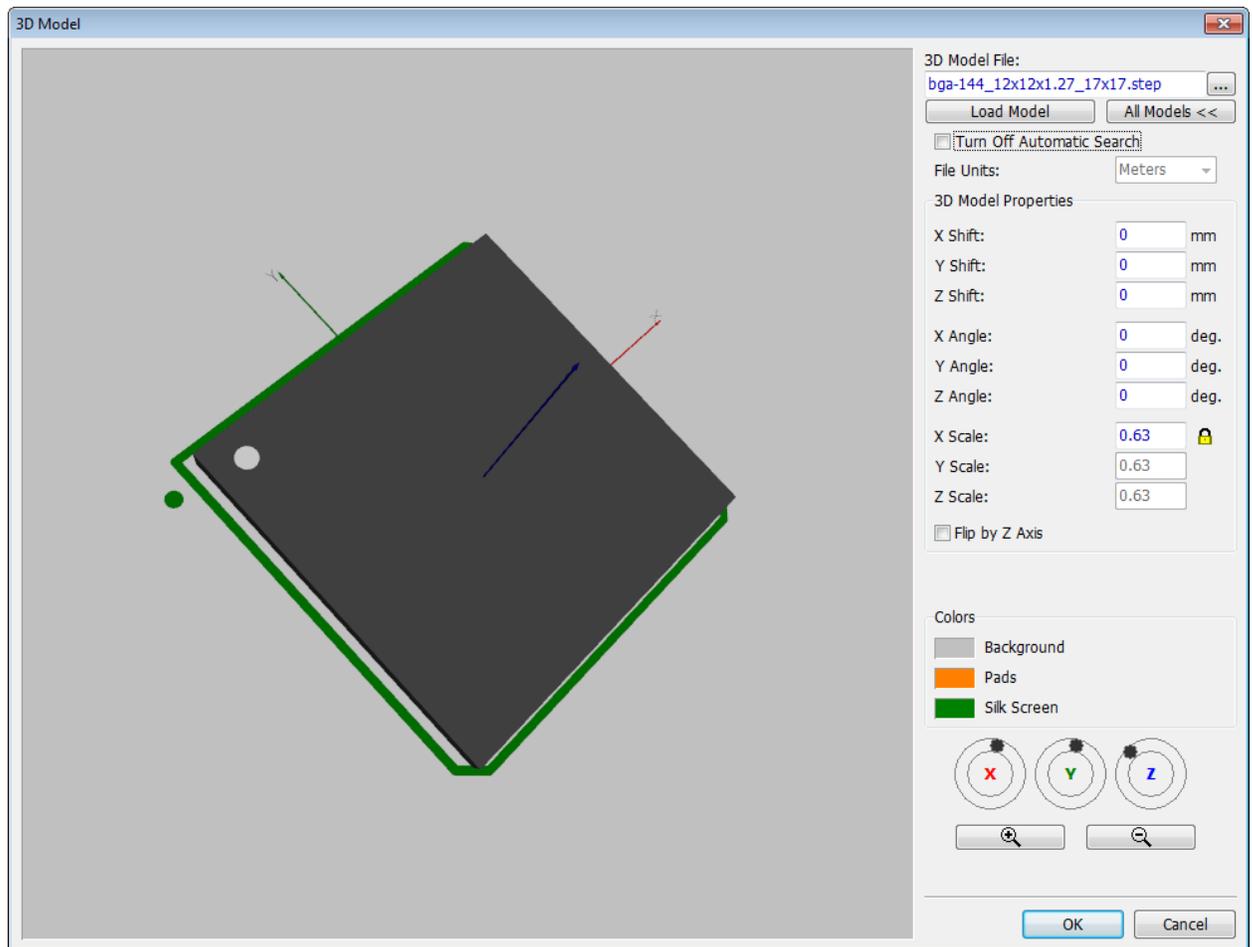


BGA pattern is ready. You can attach a 3D model now. Unfortunately, there is no BGA-144/13x13x0.8\_10x10 3D model in the 3D model libraries, but we don't need that precision for the 3D model. PCB Creator allows the user to attach any 3D models in \*.3ds, \*.wrl, \*.step, and \*.iges formats to any footprints. You can download models from the component manufacturer's websites or create them in any 3D CAD.

### 3D model properties

Press the 3D Model button on the Pattern Properties panel.

If you have appropriate model on your computer click "..." button, select a 3D model file on your HDD, and press Open. You can also enter 3D model's disk address and press the Load Model button – the 3D model will appear on the preview area. Change 3D Model Properties in case the model does not correspond to the footprint. For example, we have decided to take "bga-144\_12x12x1.27\_17x17.step" 3D model from the BGA category and scale it down a bit (see the picture below).



Try to enter different values into various fields and you will understand how to fix model's location easily. Press OK. Notice that **3D Model...** button glows green when the pattern has attached 3D model and glows red when it does not.

Save the pattern library ("Ctrl+S" or Save button on the standard toolbar).

#### 4.1.6 Real component Design. SOIC-28 pattern

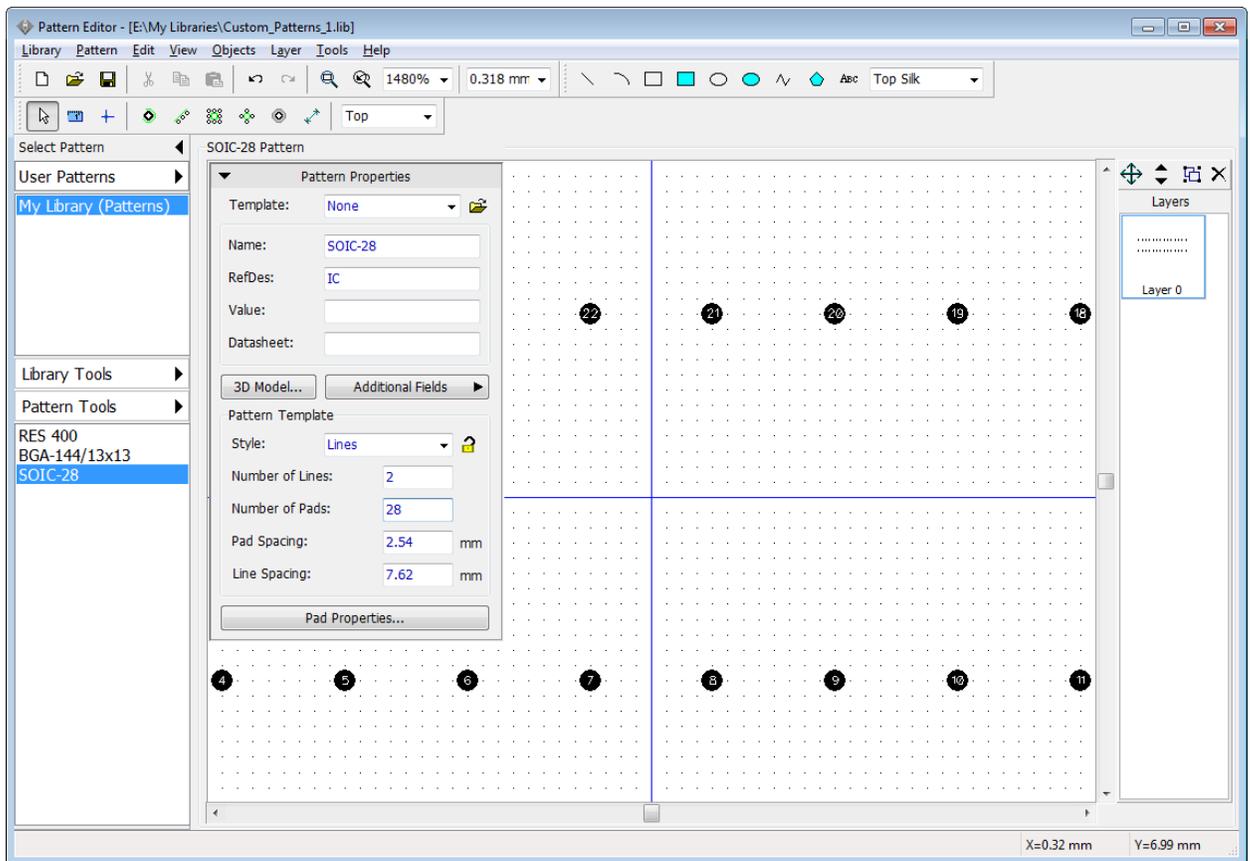
Now when you are familiar with the basics, we can try to create a real component according to the datasheet. It's gonna be the simple Microchip PIC18F24K20 component with SOIC-28 pattern.

*When you start creating the component from the scratch and you don't have an appropriate pattern, always start from the pattern creation and then proceed to component's symbol.*

We start from making a pattern.

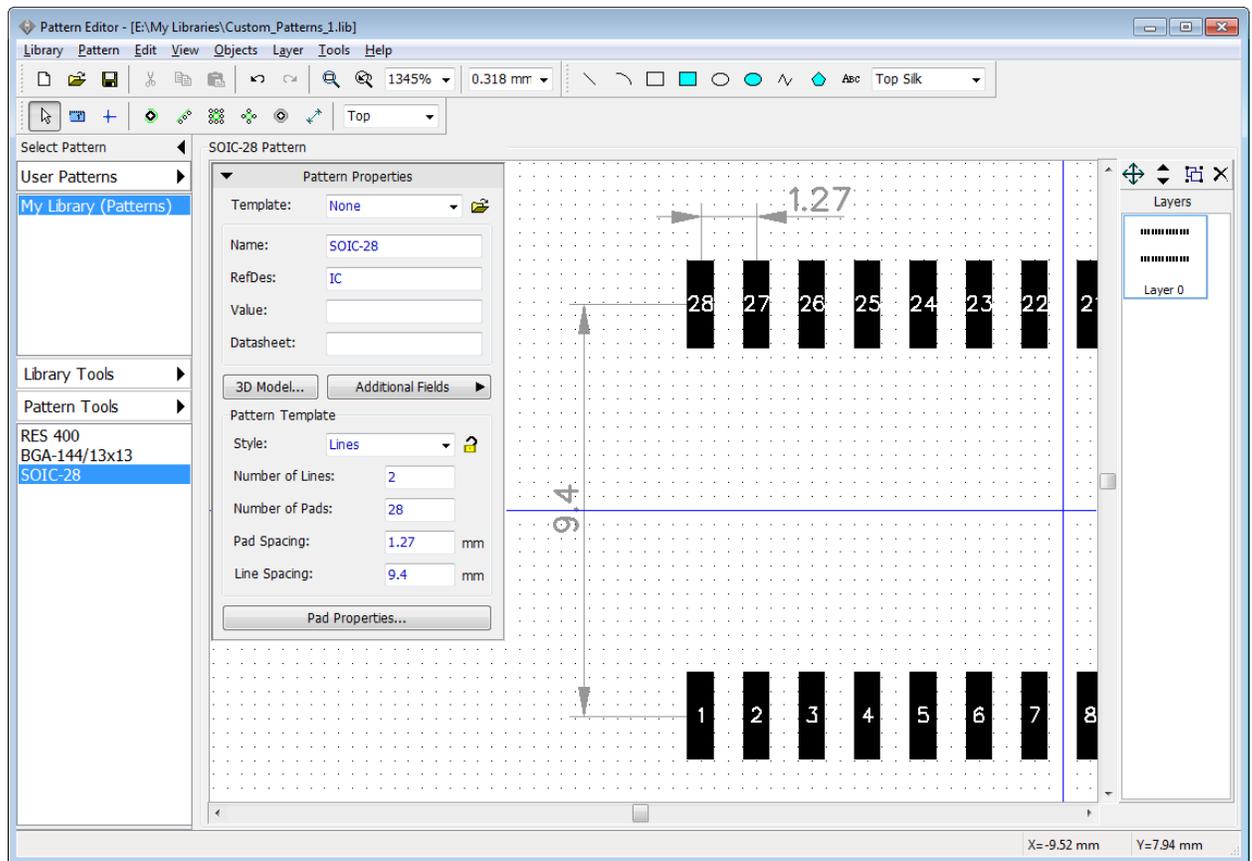
Add new pattern to the library ("Pattern \ Add New Pattern To My Library (Patterns) Library from main menu) then enter the name "SOIC-28" and RefDes "IC".

Select "Style: Lines" on the Pattern Properties panel and set "Number of Pads: 28". In our case pads are way to small for this pattern.

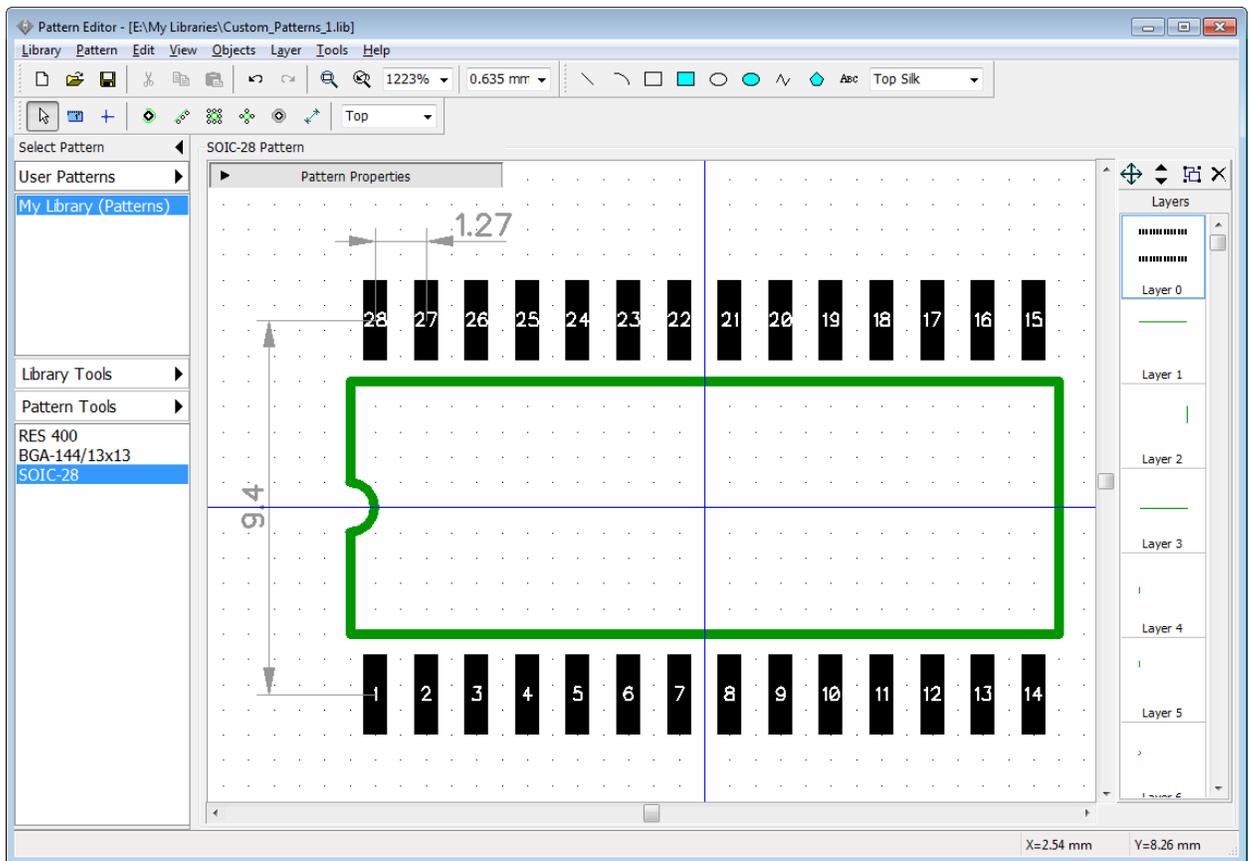


We should define correct pad spacing, line spacing and pad settings. SOIC-28 (7.50 mm) footprint dimensions can be found in the [Microchip package specifications PDF](#) at Microchip website (page 196 in the latest revision at the moment of writing this tutorial) or you can take SOIC-28 pattern from the standard PCB Creator libraries as an example.

Define default pad settings (press the Pad Properties button to open the dialog): "Type: Surface", "Shape: Rectangle", "Width: 0.6 mm", "Height: 2 mm". Press OK. Then specify "Line Spacing: 9.4 mm" and "Pad Spacing: 1.27 mm" on the Pattern Properties panel.



Pad Numbers are correct, we don't need to renumber them. Lock pattern properties to avoid accidental changes. Draw the silkscreen (like on the picture below), using the line / polyline and arc tools found on the drawing toolbar (turn ON/OFF the grid, change grid size and hide Pattern Properties panel if you need).



Datasheet requires the pattern to be rotated 90 degrees - select "Edit \ Rotate Pattern" or "Ctrl+Alt+R". Attach "soic-28\_300mil.step" 3D model from the "\_General" 3D Models category. We will attach this pattern to PIC18F24K20 component which we will create in the [Component Editor](#)<sup>[141]</sup> later in this tutorial.

Save this library and close the Pattern Editor.

## 4.2 Designing a component library

Open PCB Creator Component Editor, i.e. go to "Start \ All Programs \ PCB Creator \ Component Editor" in Windows.

Component Editor allows the user to create / edit and manage components and libraries in PCB Creator. Component Editor allows attaching component's pattern to the symbol, but it doesn't allow pattern editing – [Pattern Editor](#)<sup>[91]</sup> should be used.

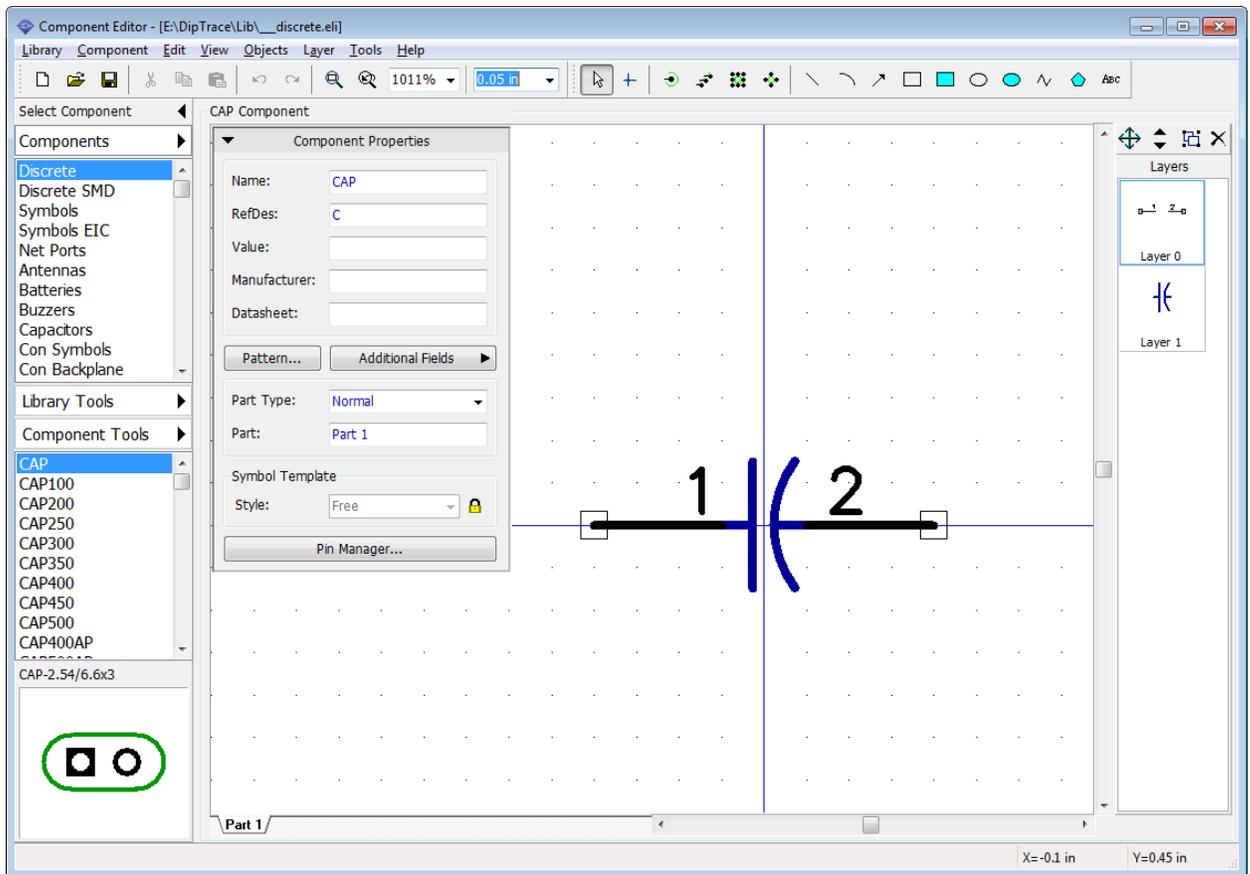
### 4.2.1 Customizing the Component Editor

Customizing the Component Editor is almost the same as customizing Pattern Editor. Select "View \ Display Origin" from the main menu to show a zero point and X, Y axes (or press "F1") if it is not displayed yet. Component Properties panel in the upper-left side of design area can be minimized or hidden using the buttons on the panel and in the main menu.

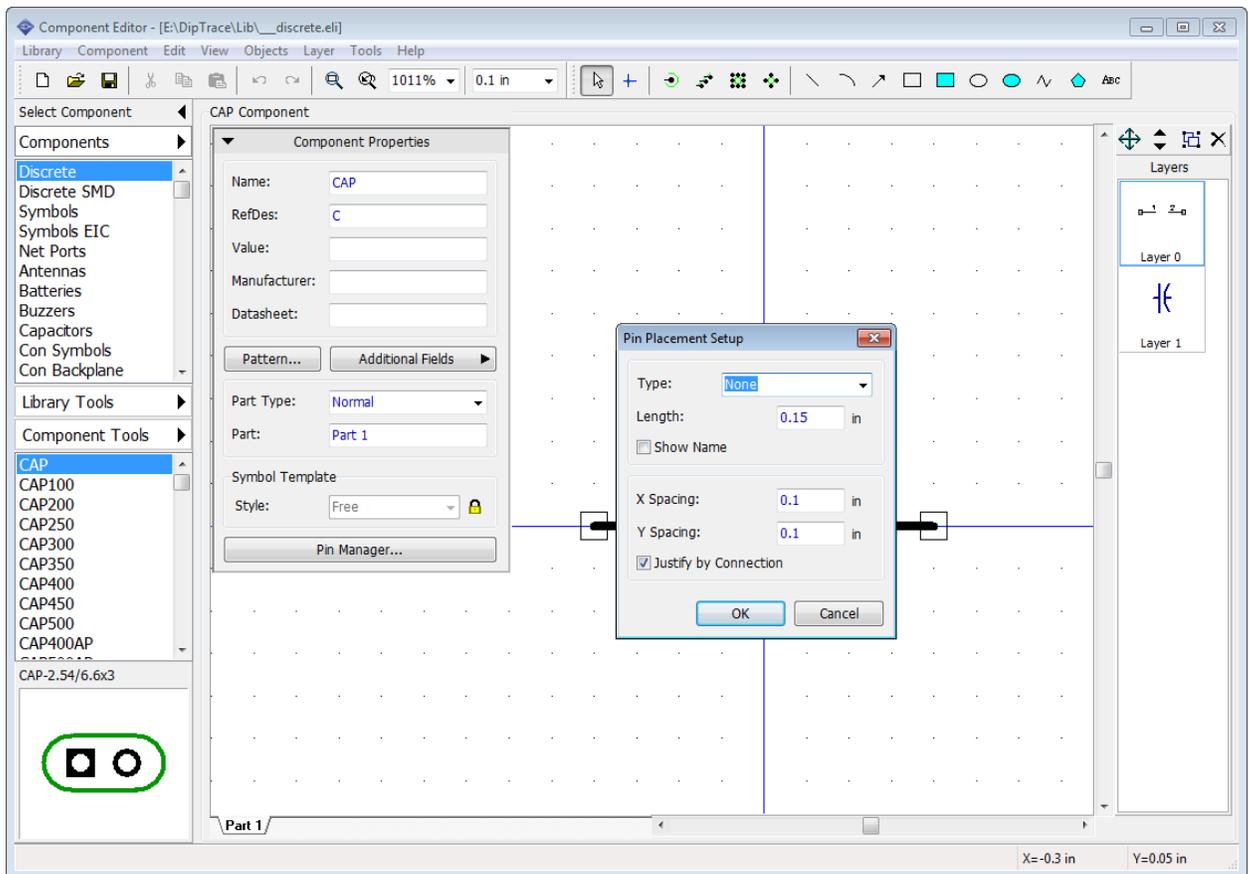
In the Component Properties panel you can define the symbol style (4 styles available): Free

(without any specific properties), 2 sides, IC-2 sides, IC-4 sides. The only difference between "2 sides" and "IC-2 sides" is a rectangle shape (IC Symbol) for the last one.

Part Type can be Normal, Power and GND, or Net Port. Component can contain only one Power and GND part (if you prefer to hide all power nets of your schematic then all power pins should be placed into this part). Net Port is a single-part component. It is used to connect wires together without visual connections, usually applied to Ground or Power nets and schematics with flexible structure (we will design such component later). Part field indicates current part of multi-part component.



If you need to define pin settings before creating a component select "Objects \ Pin Placement Setup" from main menu. We will not change these properties now, but notice that the length and X,Y spacing should be **divisible** by the grid step to create all key points on the grid points. We recommend to use 0.1 inch grid.

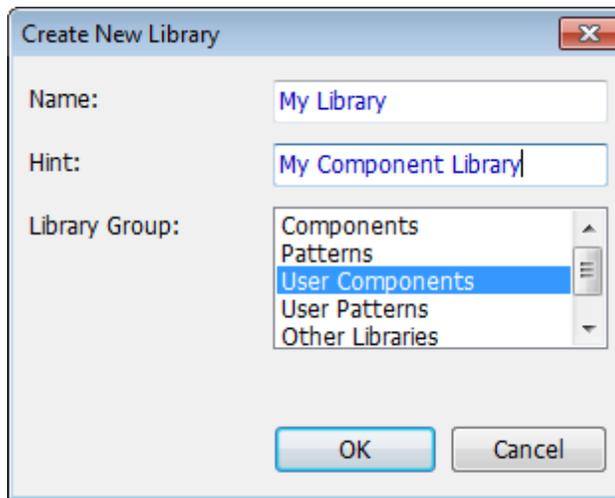


#### 4.2.2 Designing a Resistor (component)

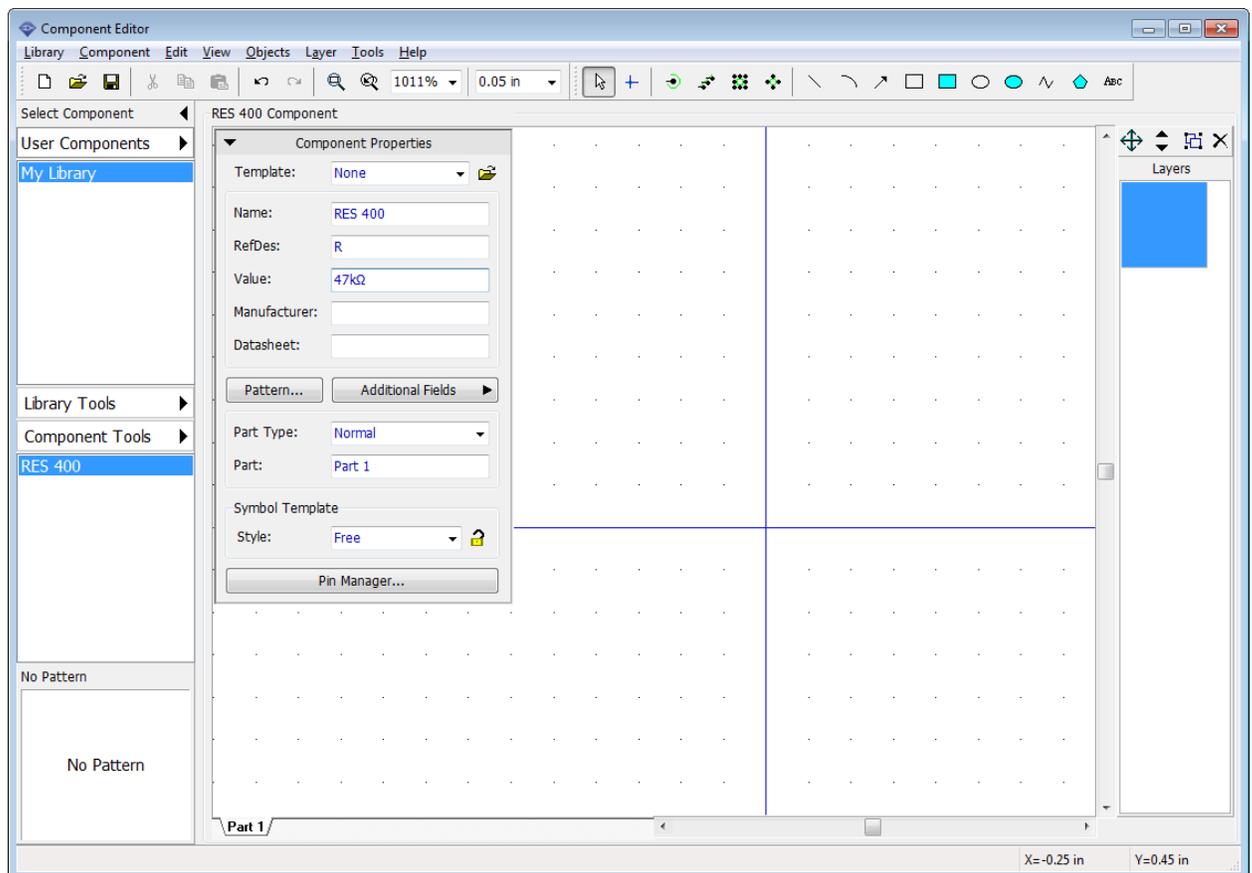
Like in the Pattern Editor, we need to create a new library first, because PCB Creator won't let you add new components to standard libraries.

##### Create a library

Press "Library Tools \ New Library " on the Libraries panel. In the pop-up dialog box enter the library name, hint and select library group. We recommend to save this library in User Components library group, offered by default. Press OK.



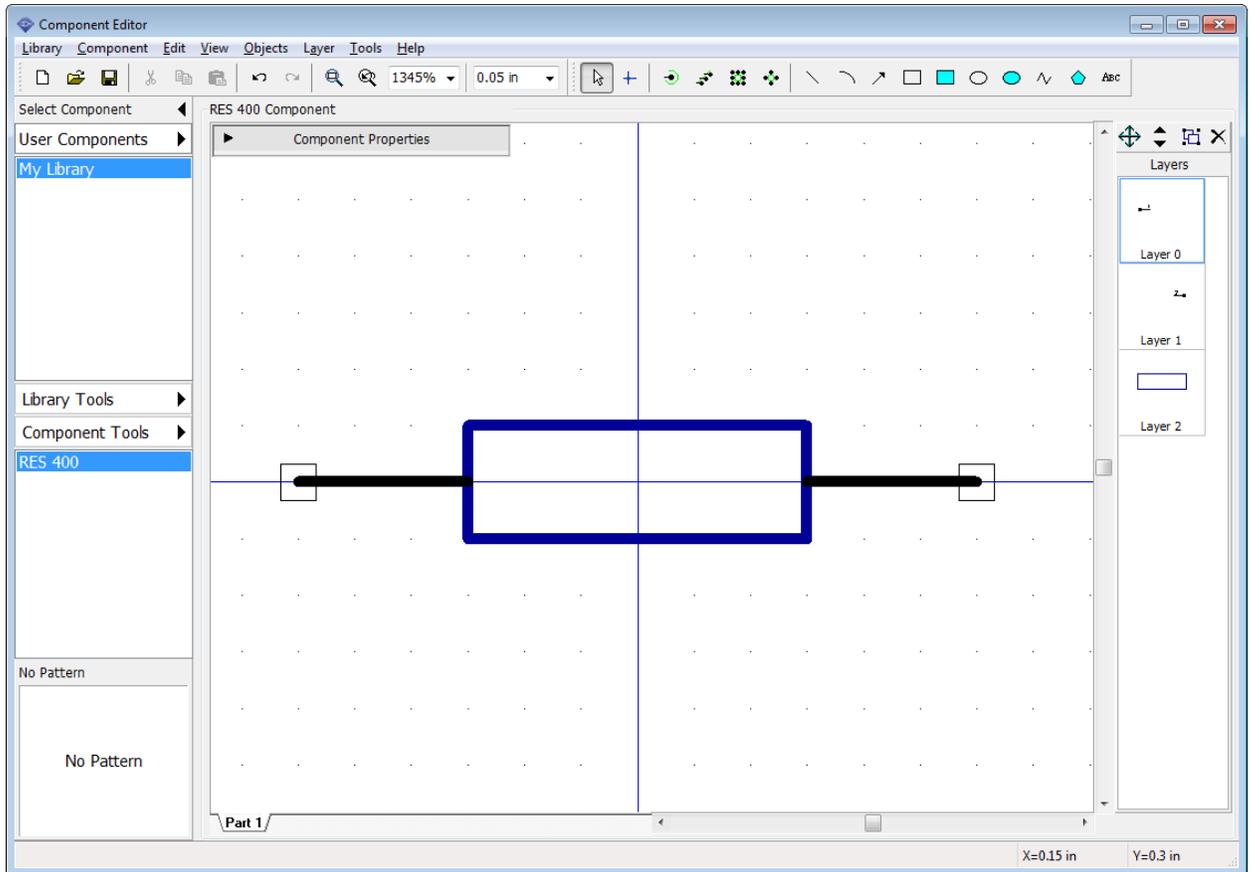
We will design resistor using the Free style and visual pin placement. Define component name, RefDes and value "47k ", use the corresponding fields on the component properties panel. After specifying these attributes please minimize the panel, using arrow in its upper-left corner.



### Place pins

Select Place Pin tool on the objects toolbar then move mouse arrow to the design area and place two pins with left clicks. Rotate one pin 180 degrees, (select it and press "Ctrl+R" twice).

Then select the rectangle tool and place graphics for the resistor. Pins should be placed by 0.1 grid and rectangle shape by 0.05 grid (use "Ctrl+plus sign", "Ctrl+minus sign" to quickly change the grid).



*Notice that you can move pin(s) using drag-and-drop method. If you want to move or rotate several pins, select them first.*

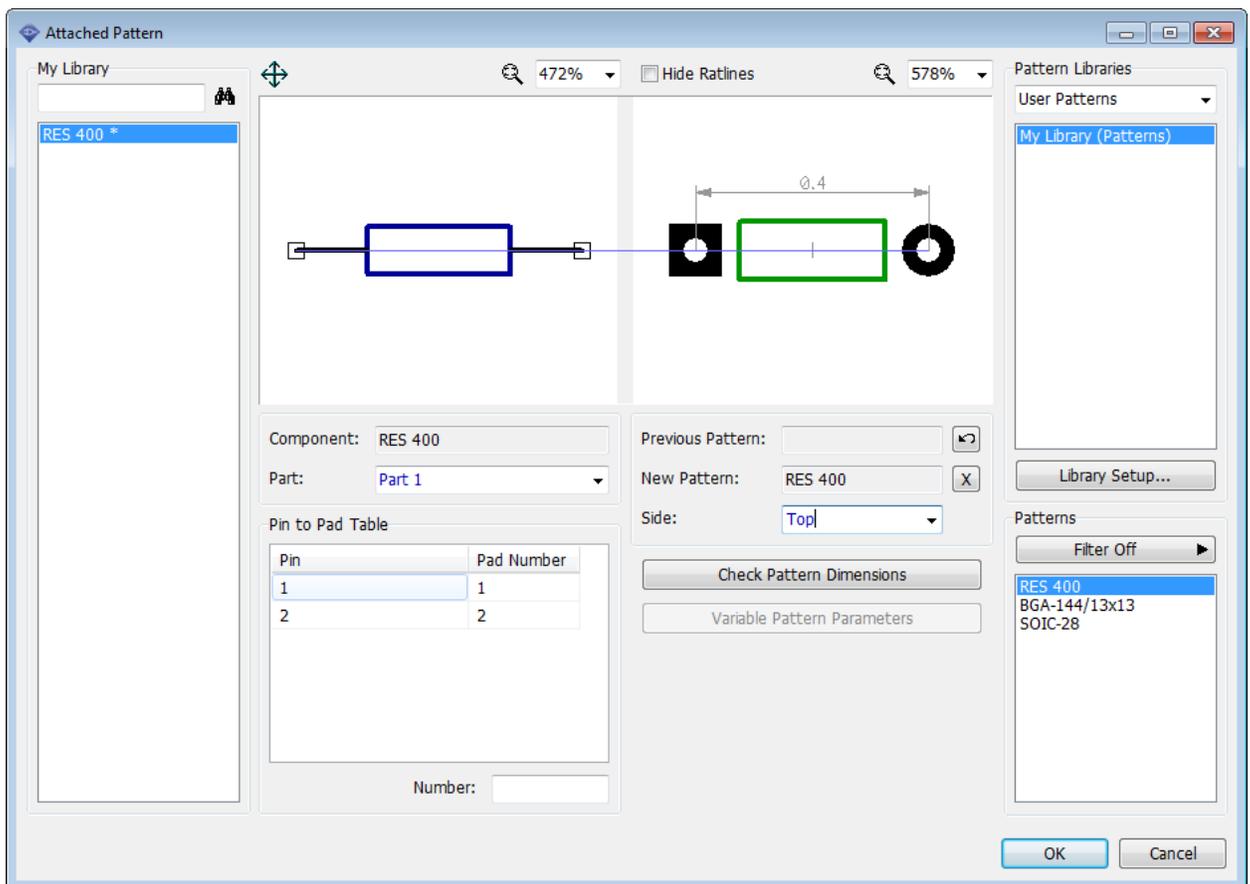
### Attach a pattern

The symbol of the resistor is ready, but the component is not ready yet. We should attach a pattern to this symbol, otherwise we will not be able to generate PCB Layout from the Schematic with this resistor. Select "Component \ Attached Pattern" from the main menu or press the Pattern button on the Component Properties panel. We need to connect this symbol drawing with a pattern drawing created before in the Pattern Editor. Select User Patterns library group (because, as you remember, we've saved the pattern library there (see "[Designing Resistor](#)"<sup>93</sup>) topic of this tutorial). There is only one library in that group. Select it and then select "RES 400" pattern that we've created earlier.

*Notice that PCB Creator automatically creates pin to pad connections by numbers. You can review and reassign them if necessary. Connections should be like on the picture below.*

To create or redefine pin to pad connections hover over the pin, left click it then move mouse to the corresponding pad and left click it to connect them. To delete connection, right click on the pin or pad and select Disconnect pin from pad item from the submenu. When you move cursor

over one of connected pins/pads, both are highlighted. If component is more complex - use Pin to Pad table (select pin and type in corresponding pad number into the Number field below).



Pin numbers (therefore component's pin to pad connections) can be changed with the Pin Manager (select "Component \ Pin Manager" from the main menu) or in the pin properties dialog box.

If current pattern is wrong, you can undo to the previous one or delete it by pressing corresponding buttons (Previous Pattern, New Pattern). Change pattern side with corresponding drop-down list.

All components of the library are in the left part of the dialog box, this allows the user to attach patterns to several components at a time. However, we don't need this now. Our Library has only one component.

Everything looks good. Press OK to close the Attached Pattern dialog box. Resistor is ready and contains both schematic part and PCB pattern with 3D Model.

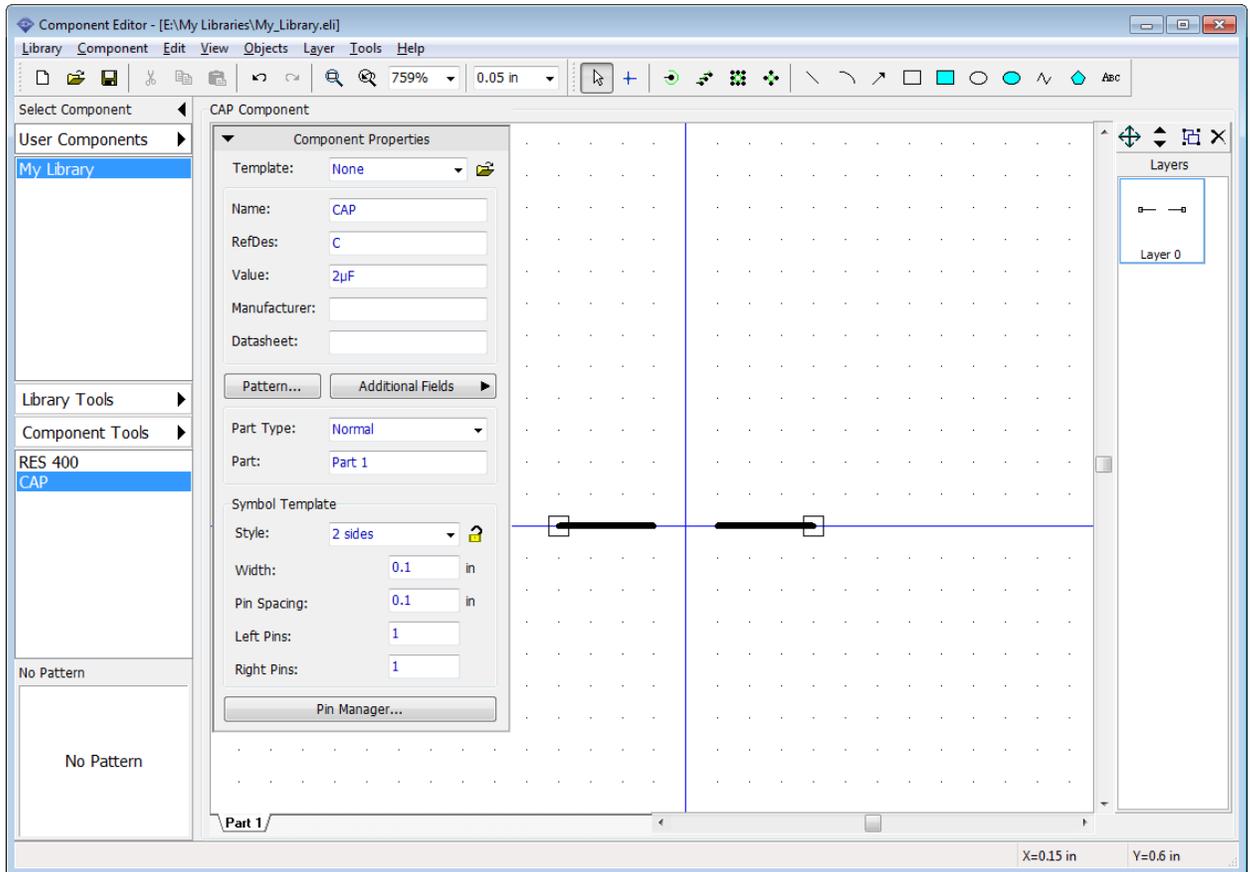
Save component library. Press Save button on the standard toolbar, select location (not in the folder with standard libraries), enter the file name and press Save.

Notice that this is the file with ".eli" extension, this means that this is component library file.

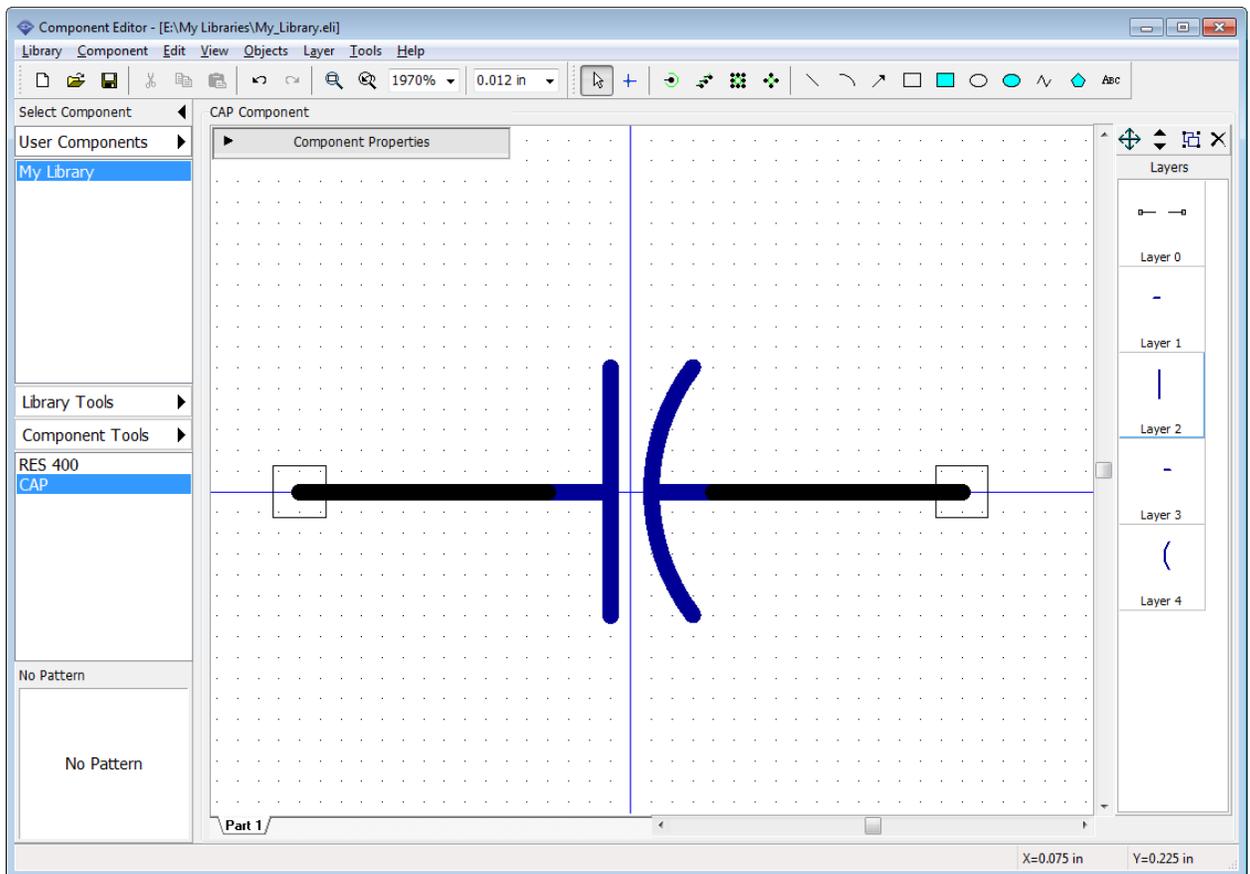
### 4.2.3 Designing a Capacitor

Select "Component \ Add New to "My Library" from the main menu to add new component to the library.

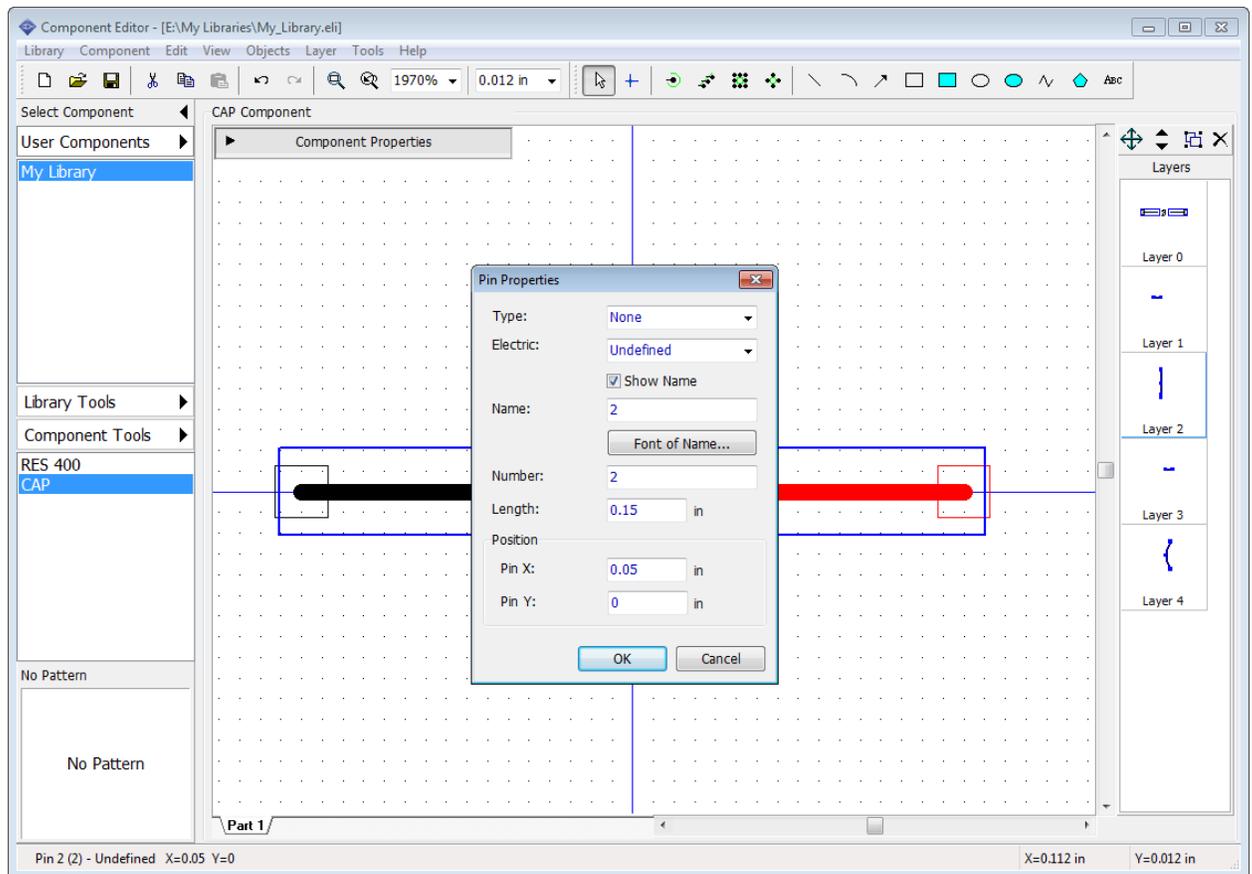
We will design capacitor using "2 sides" component style (Style box on the component properties panel); type in the component's name "CAP", RefDes – C, Value – 2 $\mu$ F. Change component width and pin spacing to "0.1 in", left and right pins to "1".



Now please minimize the component properties panel, change the grid size to 0.012 in and draw the capacitor's graphics, using three lines and one arc.

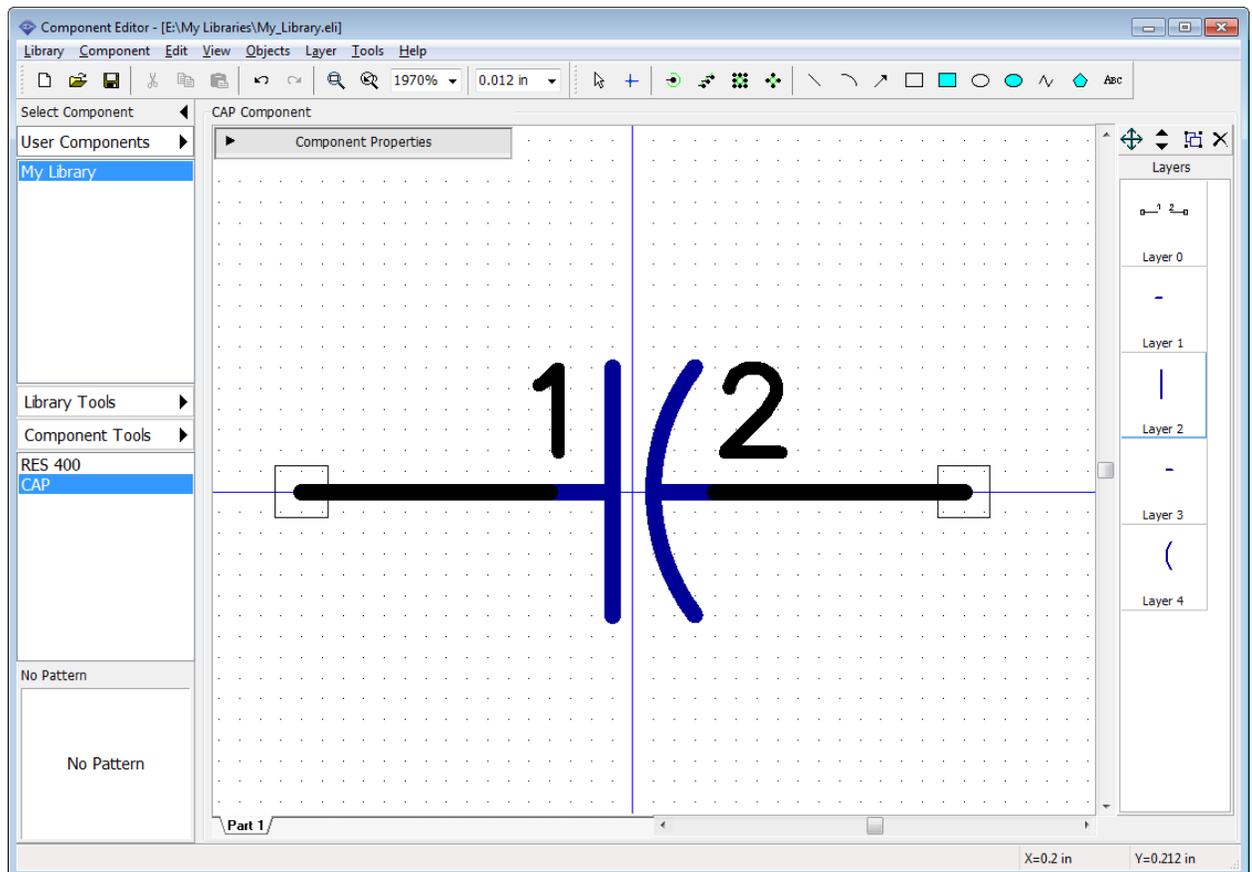


Show pin names for the component symbol, select pins (or select all using "Ctrl+A"), right click on one of them and choose Pin Properties from the submenu. In the pin properties dialog box check Show Name and press OK.

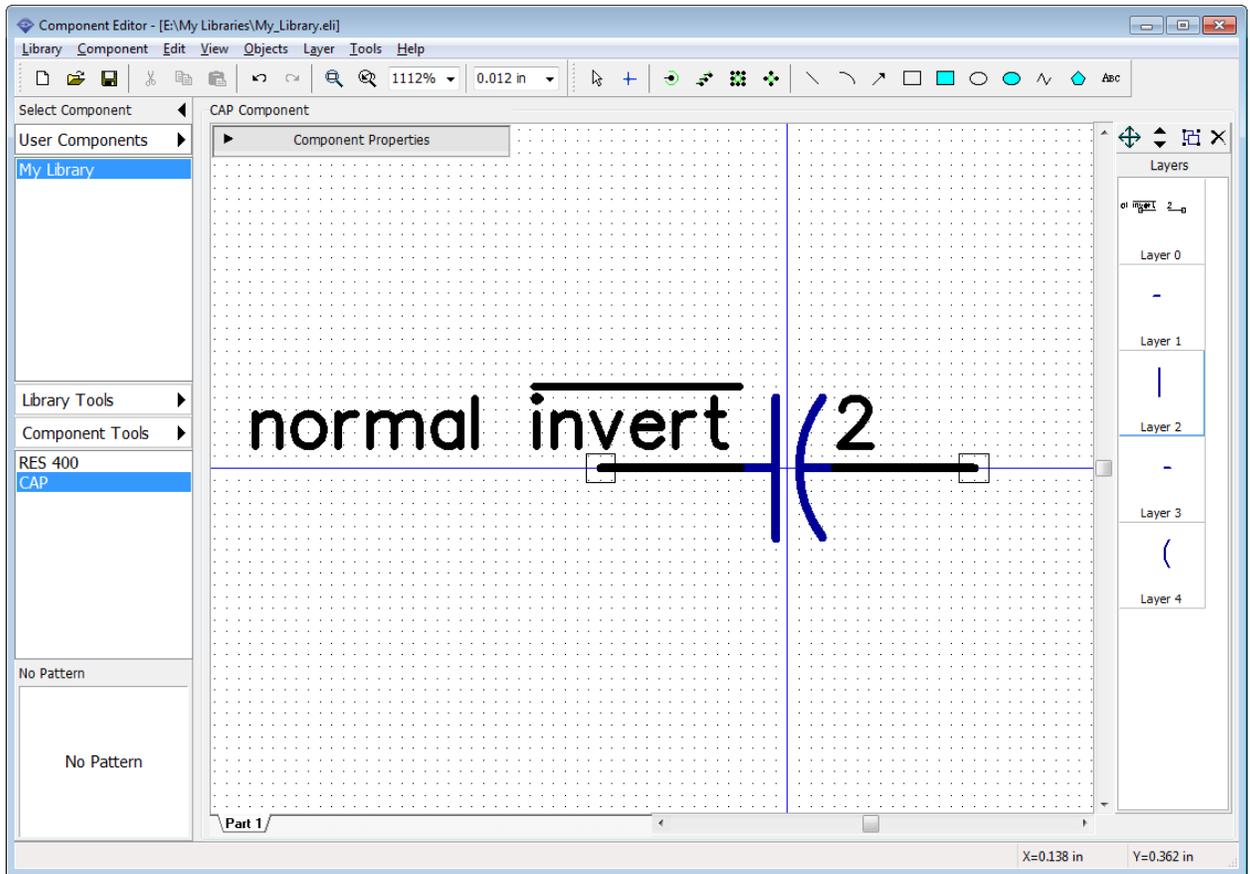


Names are shown, but they are in wrong positions (probably overlaying each other) and you need to move them. Select "View \ Move Tool" from the main menu or simply press "F10" then drag and drop pin names to their new positions. When done, right click to return to default mode.

*Notice that you can use such method to move pin names, numbers and part attributes in the Schematic.*



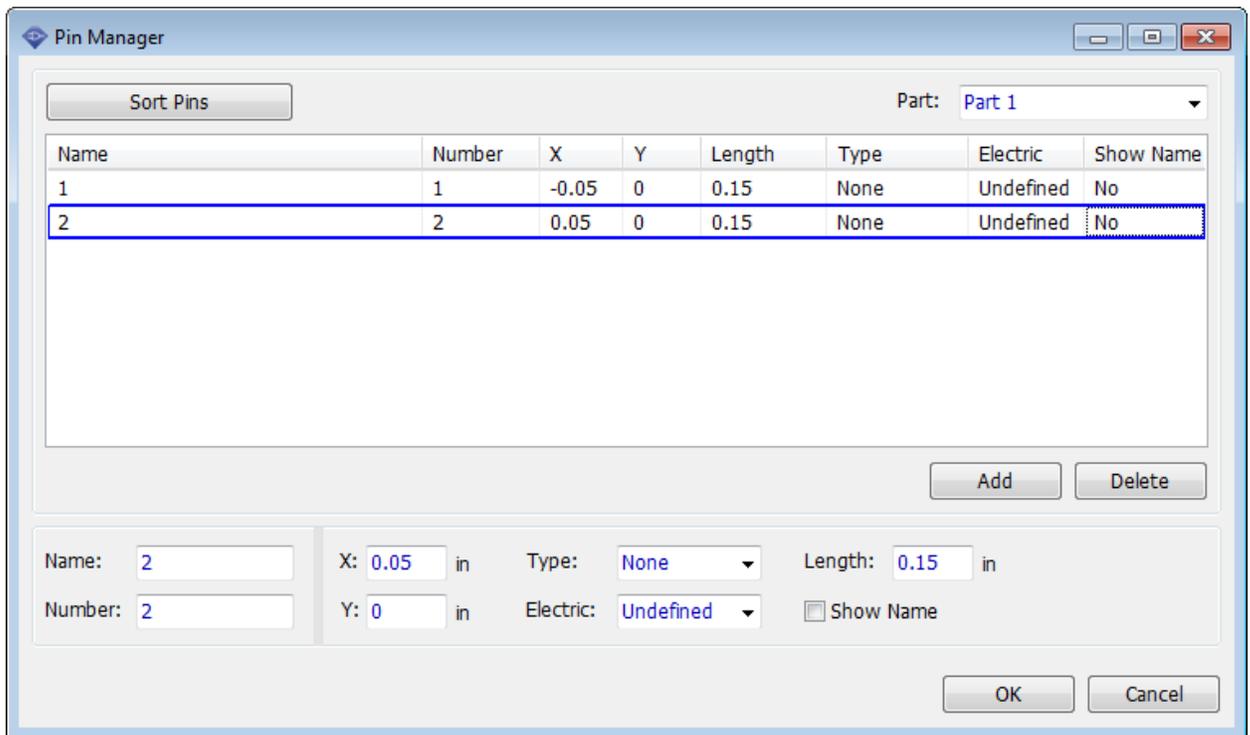
You can show inversion line in the pin name, hover over the pin with your mouse, right click, and select the Pin Name item from the submenu, type in "normal ~invert" text and press OK, then move the pin's name using the move tool ("F10" hotkey). Tilde sign "~" in the pin name starts and ends the inversion, so using it you can define the inversion for separate parts (signals) in the pin's name.



You probably don't need to see the pin names for simple passive components like capacitors.

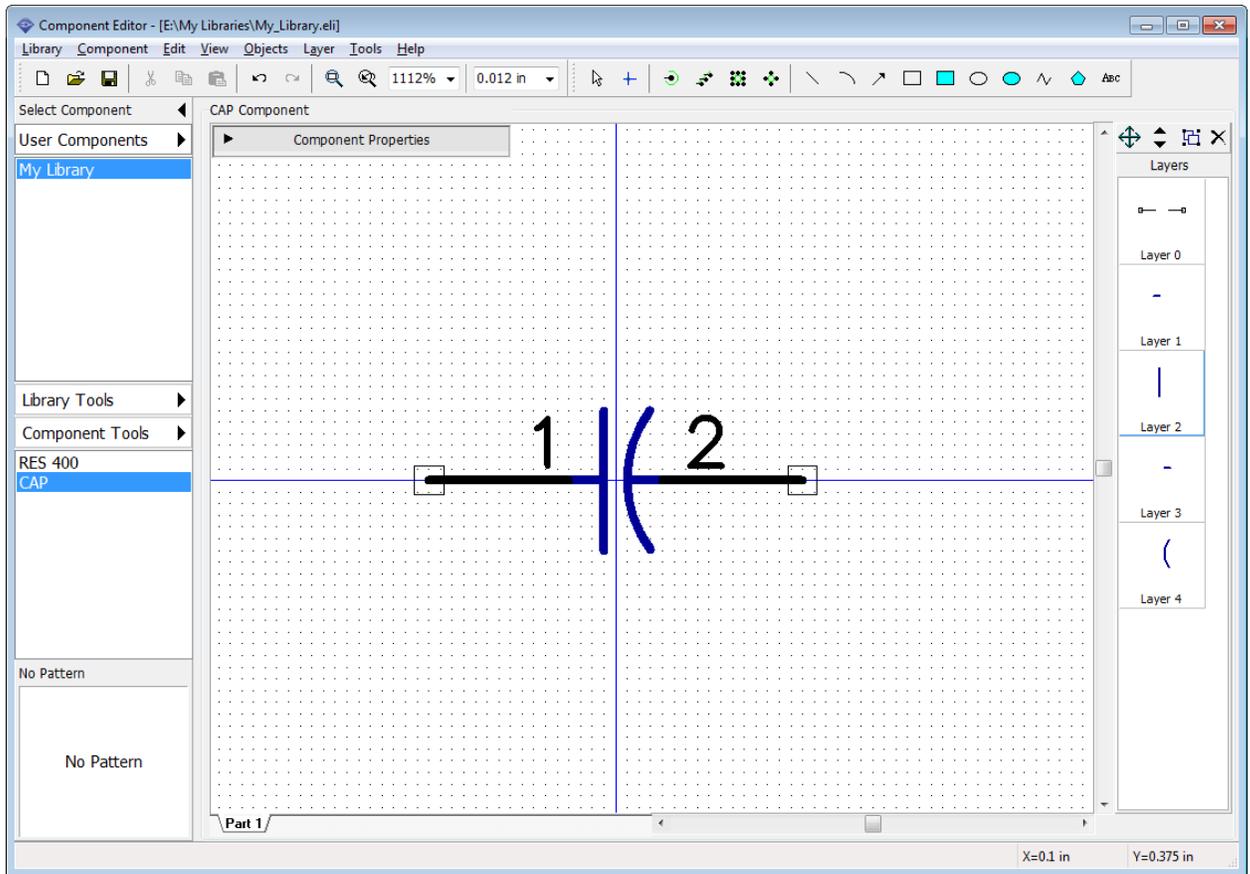
Select "Component \ Pin Manager" from main menu to open the Pin Manager dialog box and change "normal ~invert" pin name back to "1". Then hide pin names for both pins: select pin row in the table and uncheck Show Name box at the bottom of the dialog box. Close the Pin Manager.

*Notice that you can change the pin numbers (i.e. related pads), coordinates, length, type, and electric type of pins in the Pin Manager dialog box.*



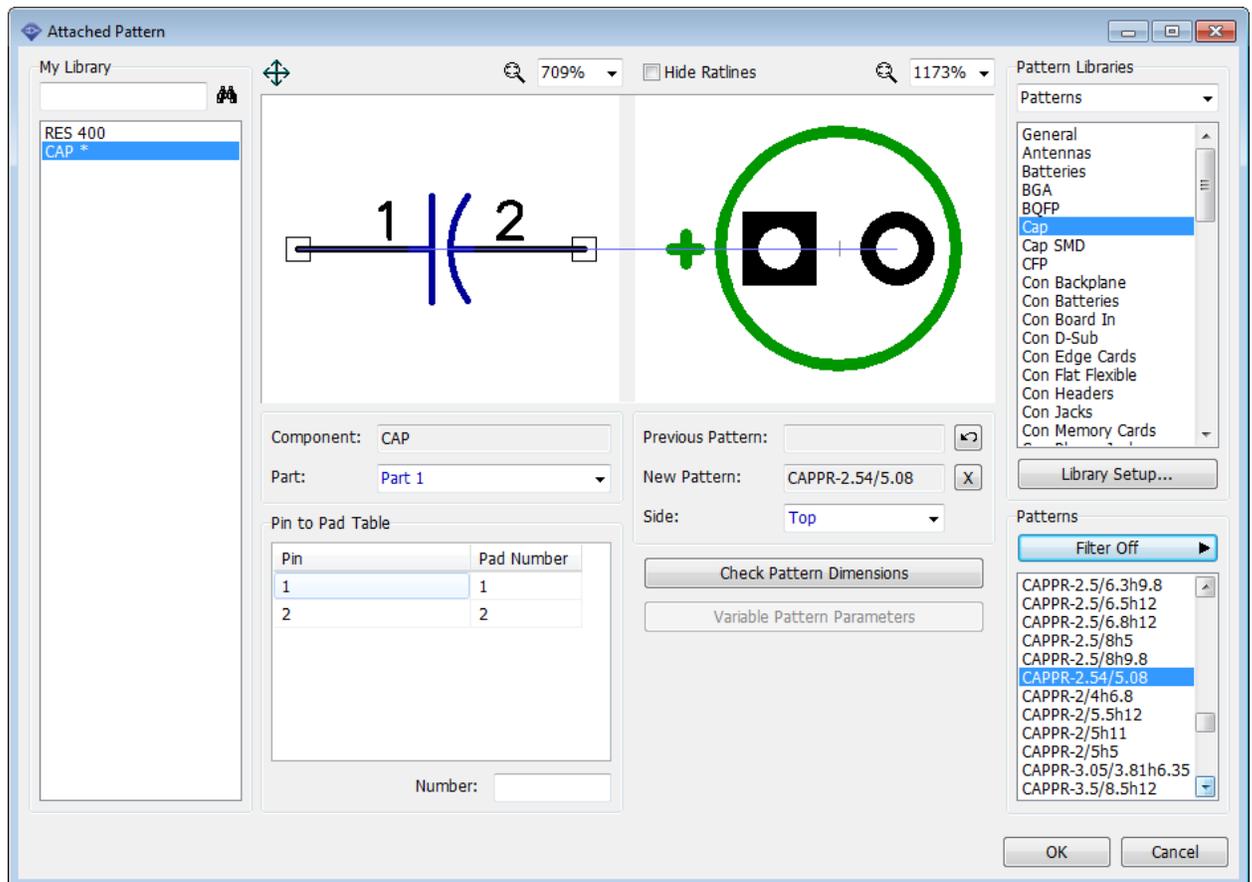
In Component Editor you can set individual show/hide pin numbers settings for current component ("Component \ Pin Numbers" from the main menu) and common program settings (the same as in Schematic) in "View \ Pin Numbers \ Show" from main menu.

Let's show capacitor's pin numbers. If you need to move pin numbers use move tool ("F10").



The next step is attaching pattern to the capacitor. Open the Component Properties panel and press the Pattern button. We did not create a pattern drawing for this component, but it is available in the standard PCB Creator libraries. Select Patterns library group then select CAP library below and CAPPR-2.54/5.08 from the list of patterns. You can use the search filters.

Pin to pad connections assigned automatically are good.



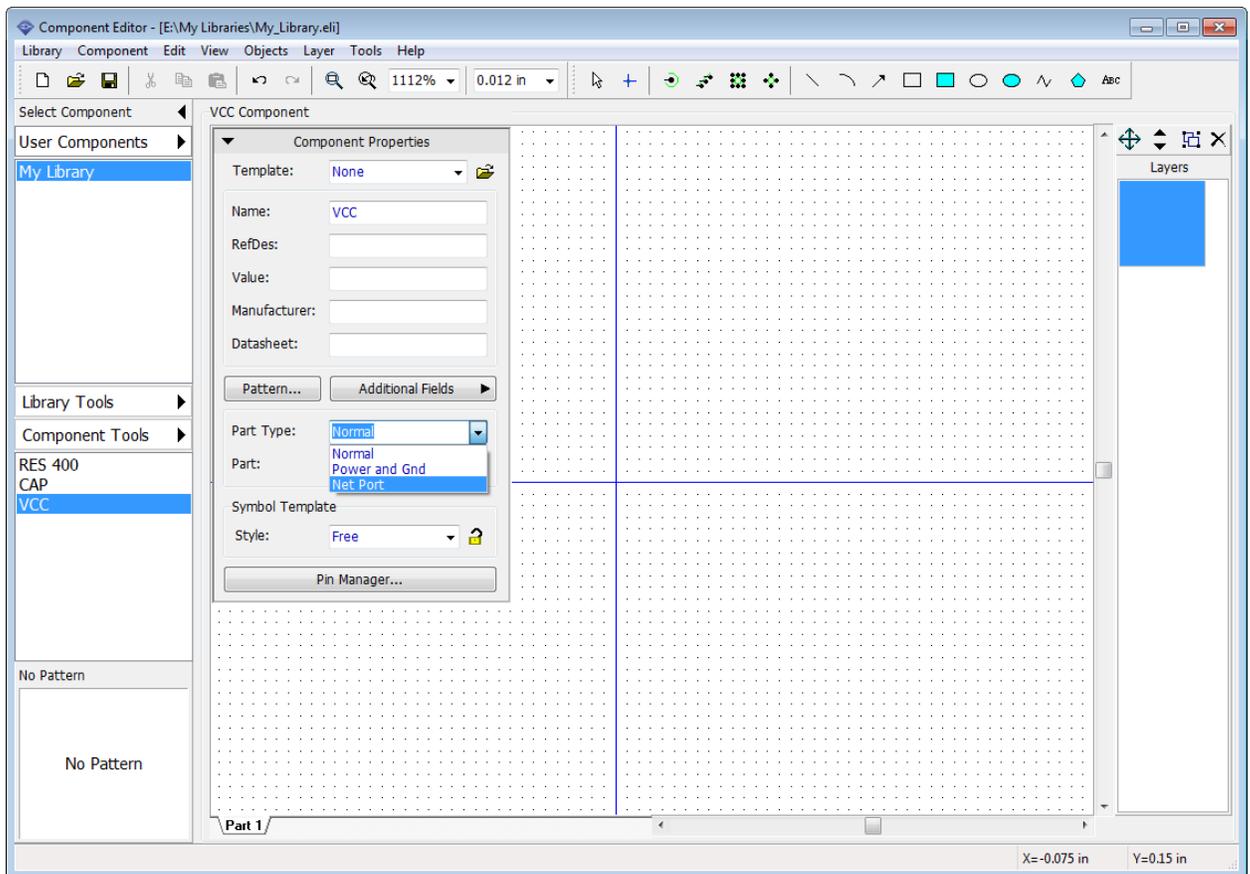
Press OK. The capacitor is ready. Save changes.

#### 4.2.4 Designing VCC and GND symbols

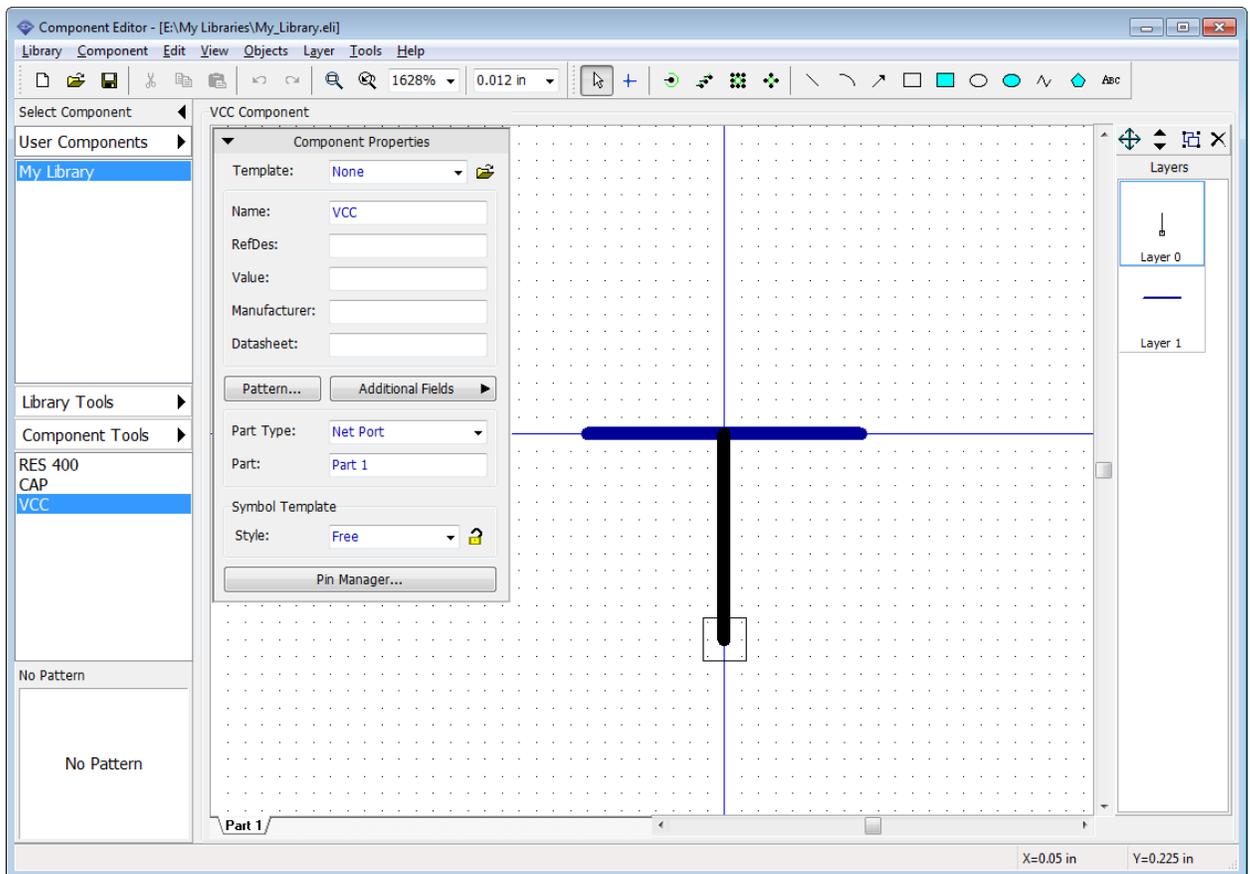
In this lesson we will practice in creating **net ports** by designing VCC and GND symbols.

##### VCC

Press "Component Tools \ Add New Component to My Library on the Libraries panel. Type in "VCC" in the component name field on the Component Properties panel and select Net Port in the Part Type drop-down list.



Minimize the Component Properties panel, then press the Place Pin  button on the objects toolbar and place a single pin, rotate it vertically (select it and press "R" hotkey). Select the line tool on the drawing toolbar and draw a line like on the picture below, use 0.05 inch grid.

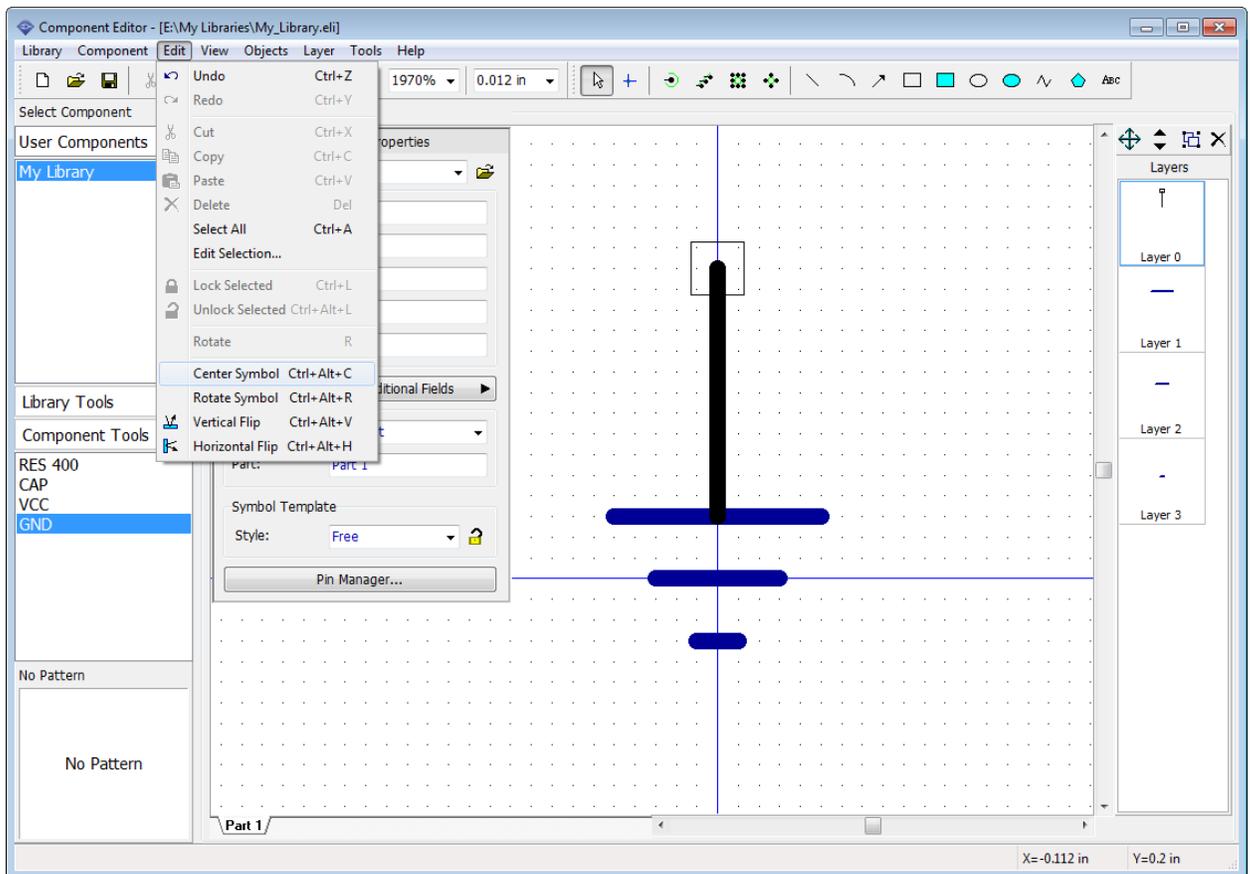


Hide the pin number if it is visible, select "Component \ Pin Numbers \ Hide" from the main menu. Pin number for the single-pin component looks kind of weird. VCC symbol is ready.

### GND

Now add one more component ("Ctrl+Ins" hotkeys) and create a GND symbol the same way like the VCC.

Select "Edit \ Center Symbol" or press "Ctrl+Alt+C" for GND because in our case its origin is not in the center, so you have to center it to make component's origin hidden by default in the Schematic. Use 0.012 inch grid to draw a GND symbol's graphics.



*Notice that net ports do not need patterns. This special type of components is used only in Schematic to connect wires without visual connection and they do not exist on the circuit board.*

Save the library file.

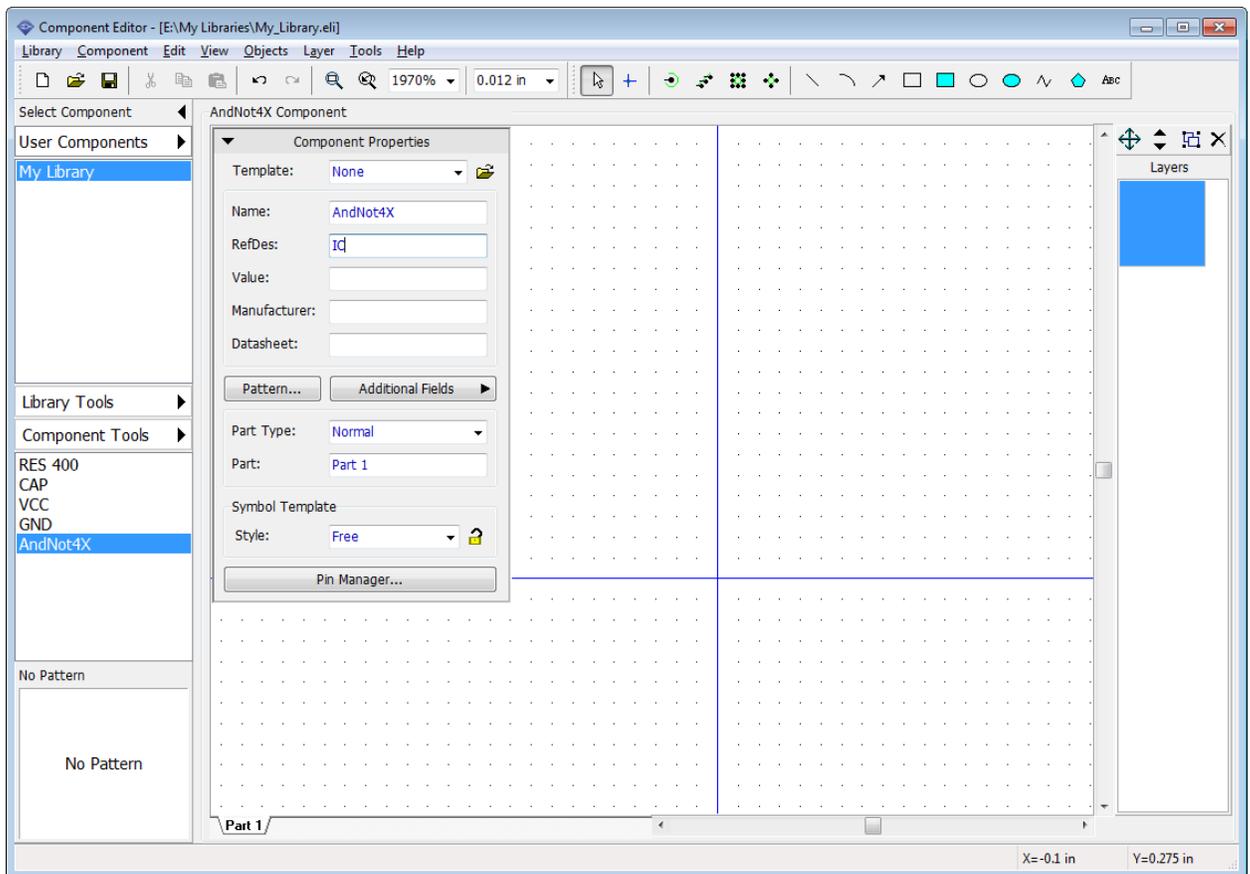
#### 4.2.5 Designing a multi-part component

Despite being represented as a single physical package on the circuit board, some electronic components has a multi-part (often called "multi-segment") schematic symbols. Building a multi-part schematic symbols involves a bit different technique.

##### Creating a symbol

We will design a simple multi-part component with four "And-Not" symbols, a power part and attach DIP-14 pattern, which is available in the standard libraries.

Add new component to the library, i.e. select "Component / Add New Component To Library" from the main menu. Enter the name and Reference Designator.

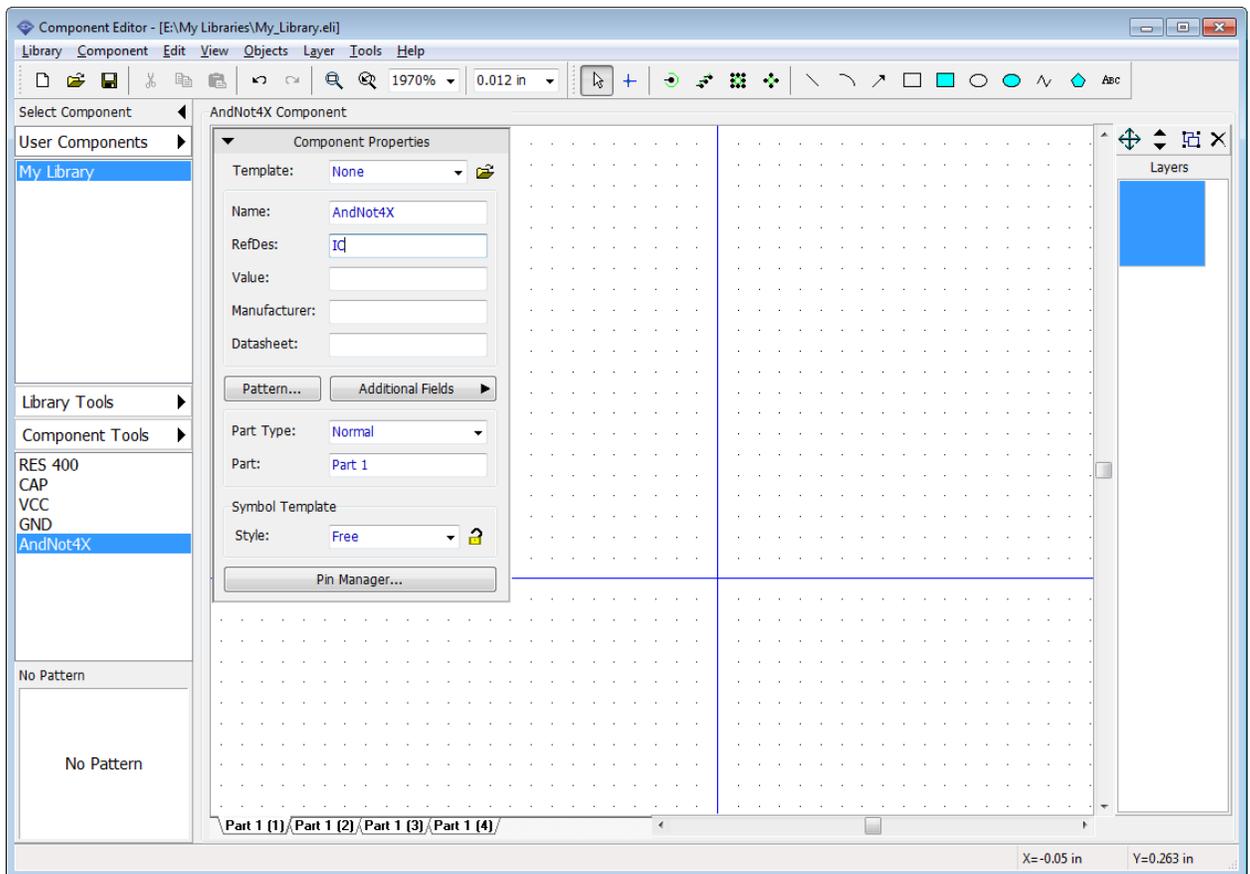


### Creating parts

PCB Creator allows for designing separate parts and group similar parts of the multi-part component in the part groups. All parts in the part group have the same pin layout and drawing, except the pin numbers (i.e. related pads). Each part of the multi-part component can be Normal, Power and Ground, or Net Port. Power parts and power nets can be hidden in the schematic capture. Only one power part per component is allowed.

We will design a component with 4 similar AndNot gates and one power part. Select "Component \ Create Similar Parts" from the main menu, type "4" in the dialog box, and press OK to apply. Tabs with the part names has appeared in the bottom-left of the design area (look like tabs).

*Similar parts are created based on currently selected part.*

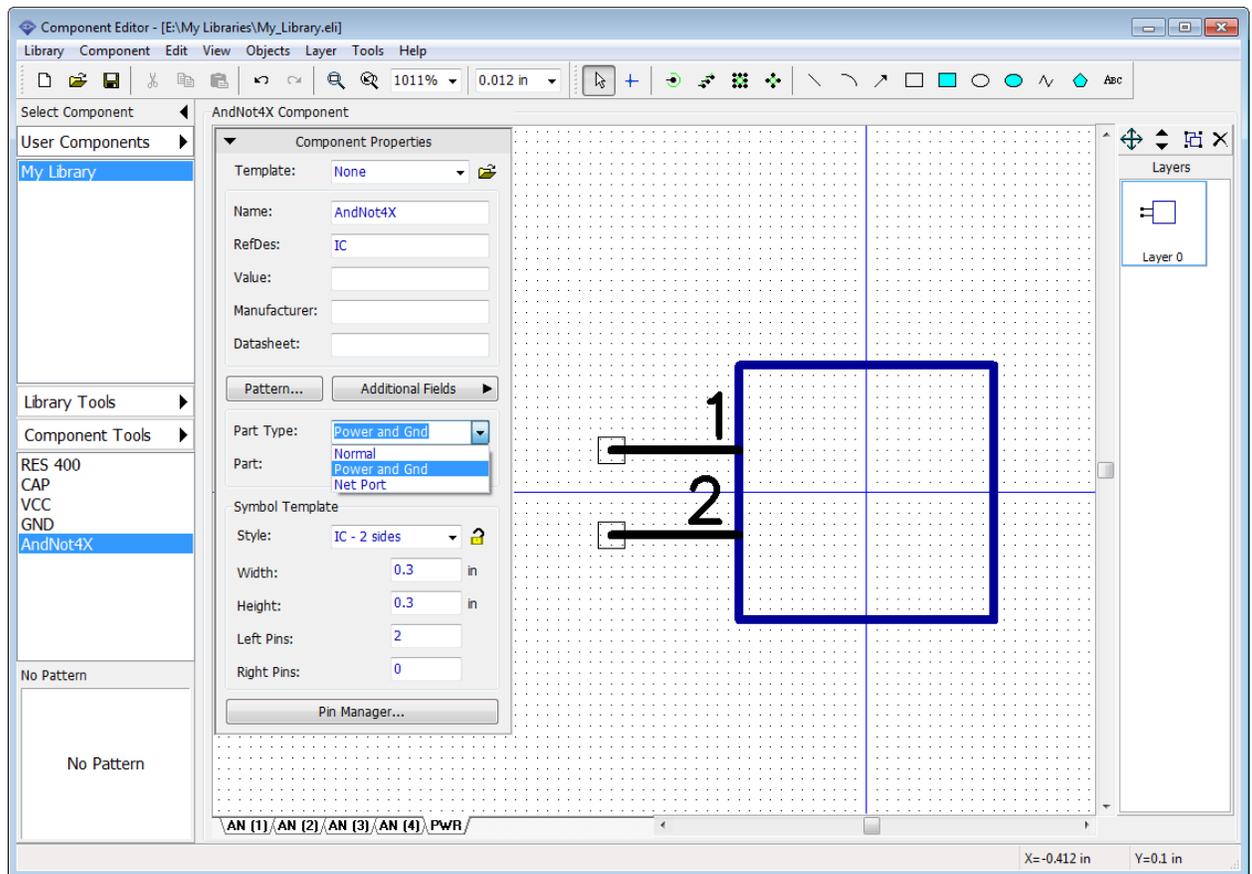


Now you can see the following 4 parts: Part 1 (1), Part 1 (2), Part 1 (3), and Part 1 (4). All similar parts have the same part name. Change it quickly (in Part field on the Component Properties panel) for example to "AN".

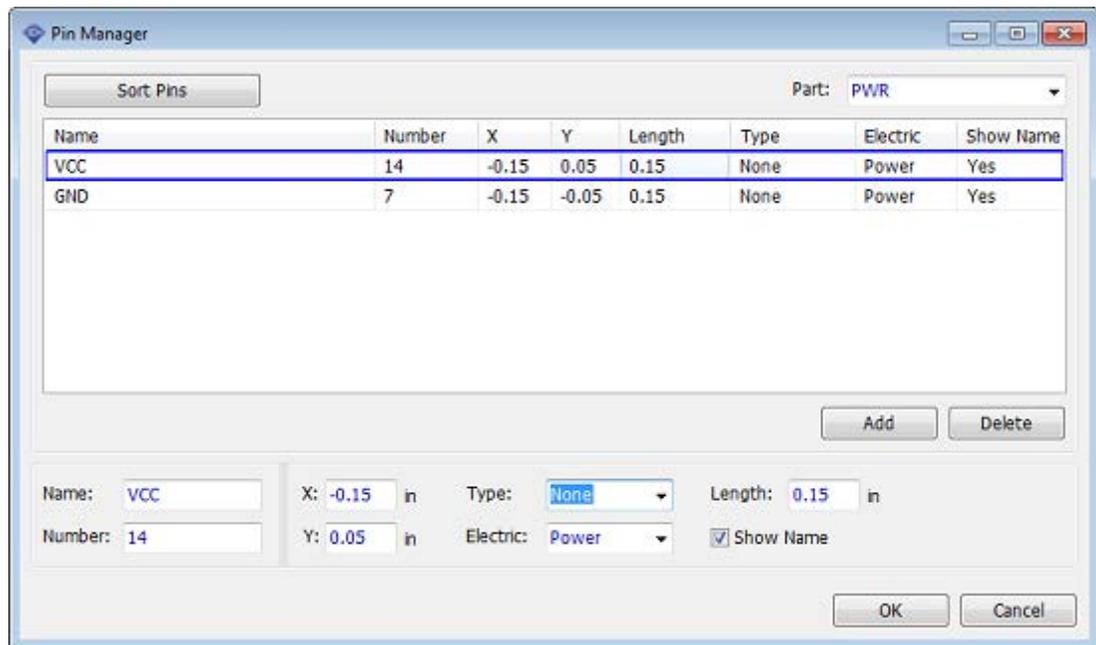
The power part for the component is missing. Select "Component \ Add New Part" from the main menu to add a single part to the component, select new part tab (Part 1) in the bottom-left of the design area and rename it to "PWR".

*Notice that new part is a separate part and does not belong to AN group.*

Start designing a component with the power part. On the Component Properties panel specify: "Style: IC - 2 sides", "Width: 0.3 in", "Height: 0.25 in", "Left Pins: 2", "Right Pins: 0". Then select Power and Gnd from the Part Type drop-down list. Make general component pin numbers visible ("View \ Pin Numbers \ Show" from the main menu), if they are not visible.

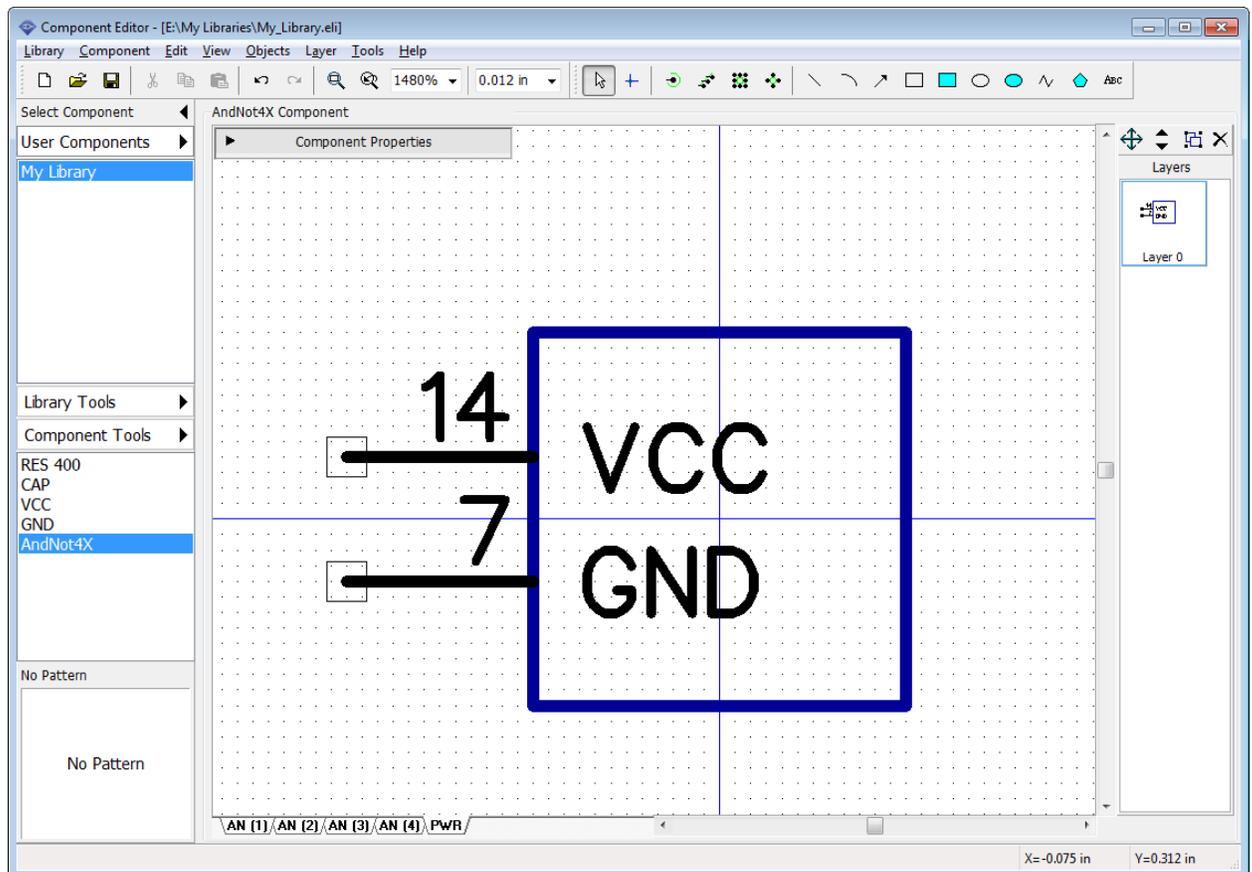


Press the Pin Manager button on the Component Properties panel and change pin names to "VCC" and "GND", pin numbers to "14" and "7", electric type to Power, Show Name box should be checked for both pins. Notice that you can change Type, Show Name and Length parameters for multiple pins.



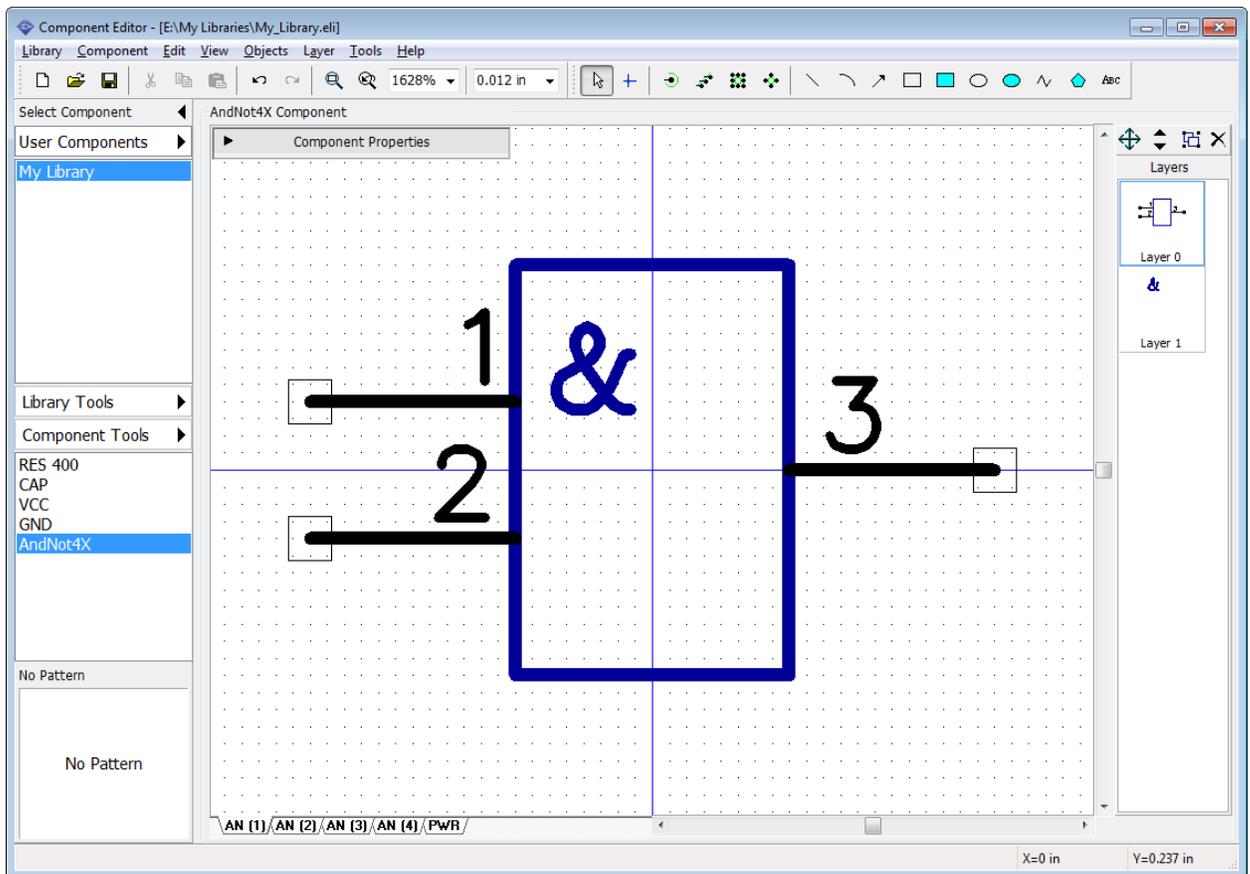
Pin Manager dialog box itself and the width of the rows are resizable. These settings are saved when you close the program.

Now press OK to close the Pin Manager dialog box. Then minimize the Component Properties panel and see the first ready part of the component.



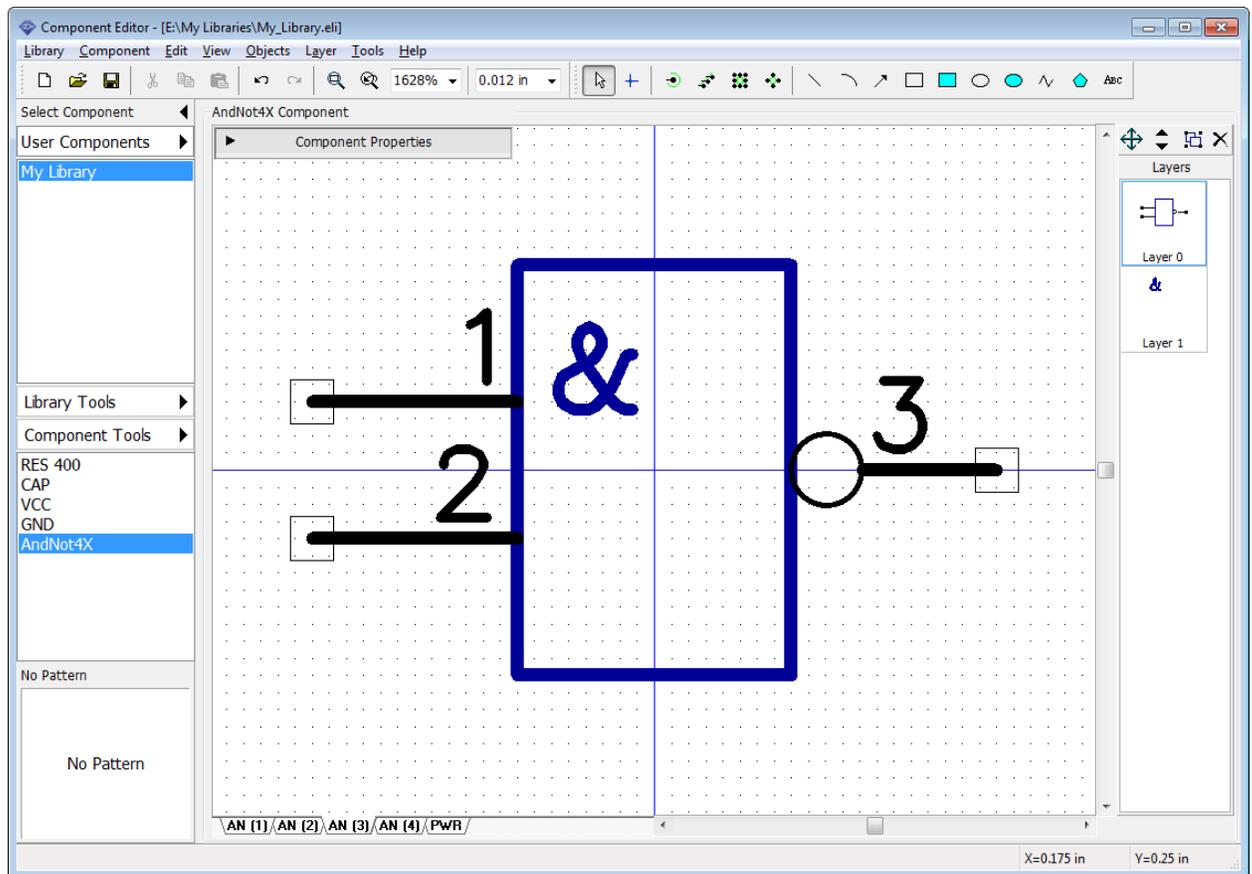
Design other parts of the multi-part component: select one of the AN parts, maximize the Component Properties panel and define the following parameters: "Style – IC-2 sides", "Width: 0.2 in", "Height: 0.25 in", "Left Pins: 2", "Right pins: 1". Then minimize the Component Properties panel again.

Select the text tool on the drawing toolbar (  button), hover over the symbol, left click and type "&" character, then press Enter or left click to place the text (see the picture below). Text can be moved.



Right pin of a typical "And - Not" (Not And) part should be inverted or Dot type. Right click on the third pin, select Pin Properties from the submenu, and specify "Type: Dot" in the pop-up dialog box. Click OK to apply changes and close the dialog box.

*Notice that you don't need to draw another AN parts of the component if they were created as a group, they will be automatically created, based on the first part.*



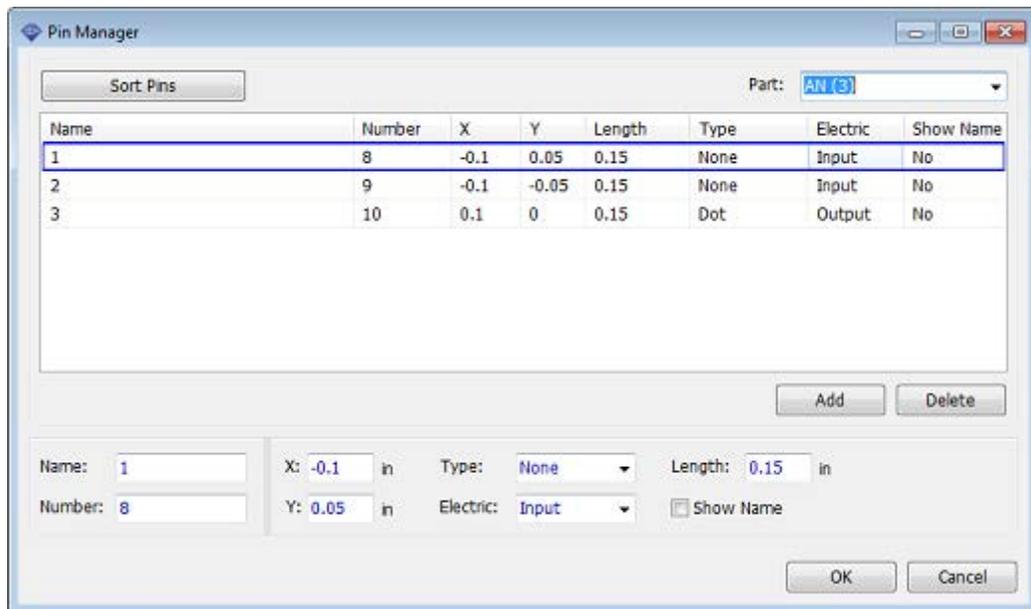
Select AN (3) or AN (4) part just to make sure that segments (parts) are the same. All parts in the group are absolutely identical, but with different pin numbers.

### Pin Manager

Select "Component \ Pin Manager" from the main menu. In the Pin Manager dialog box select part (using the drop-down list in the upper-right), define pin numbers then select the next part and so on until you define pin numbers for all AN parts. Use Down arrow key or Enter to quickly switch to the next pin when you're typing in the Number or Name fields.

Don't forget that pin number "7" is used in the GND part, therefore you should miss it while renumbering pins of the functional parts, going from pin #6 straight to the pin #8.

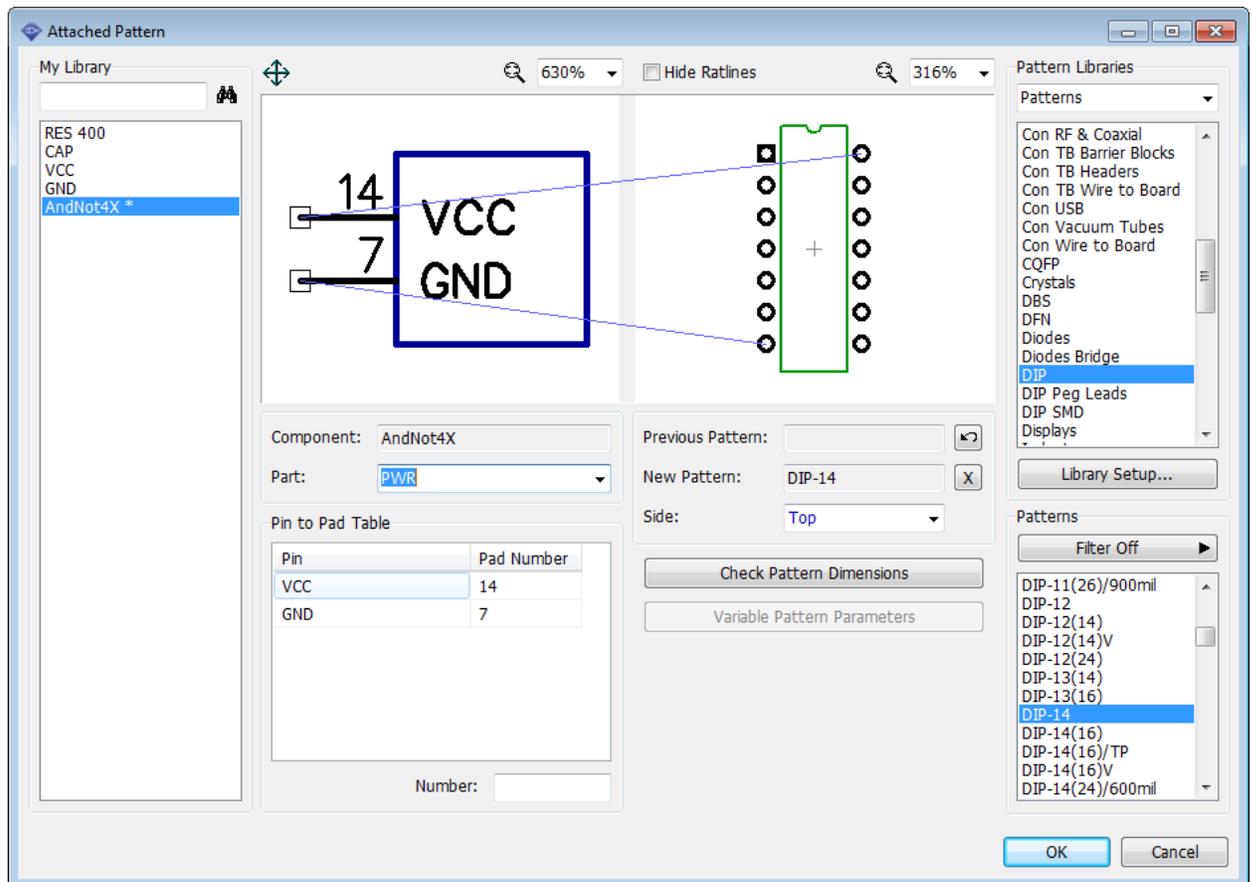
Please set correct Electric type (2 input pins and one output pin) for one of the parts and the other parts will be changed automatically. Click OK.



The next step is attaching related pattern to the multi-part component. Press the Pattern button on the Component Properties panel. In the Attached Pattern dialog box select Patterns library group (library group with all standard pattern libraries) then select DIP library and DIP-14 pattern.

*Notice that you don't need to specify pin-to-pad connections, they have been assigned automatically and should be correct, because we've specified correct pin numbers in the Pin Manager (this is why the pin numbers array was not straight).*

Select different parts (drop-down list below the preview field) and visually check connections to ensure that they are all right. Press OK to attach pattern and close this dialog box.



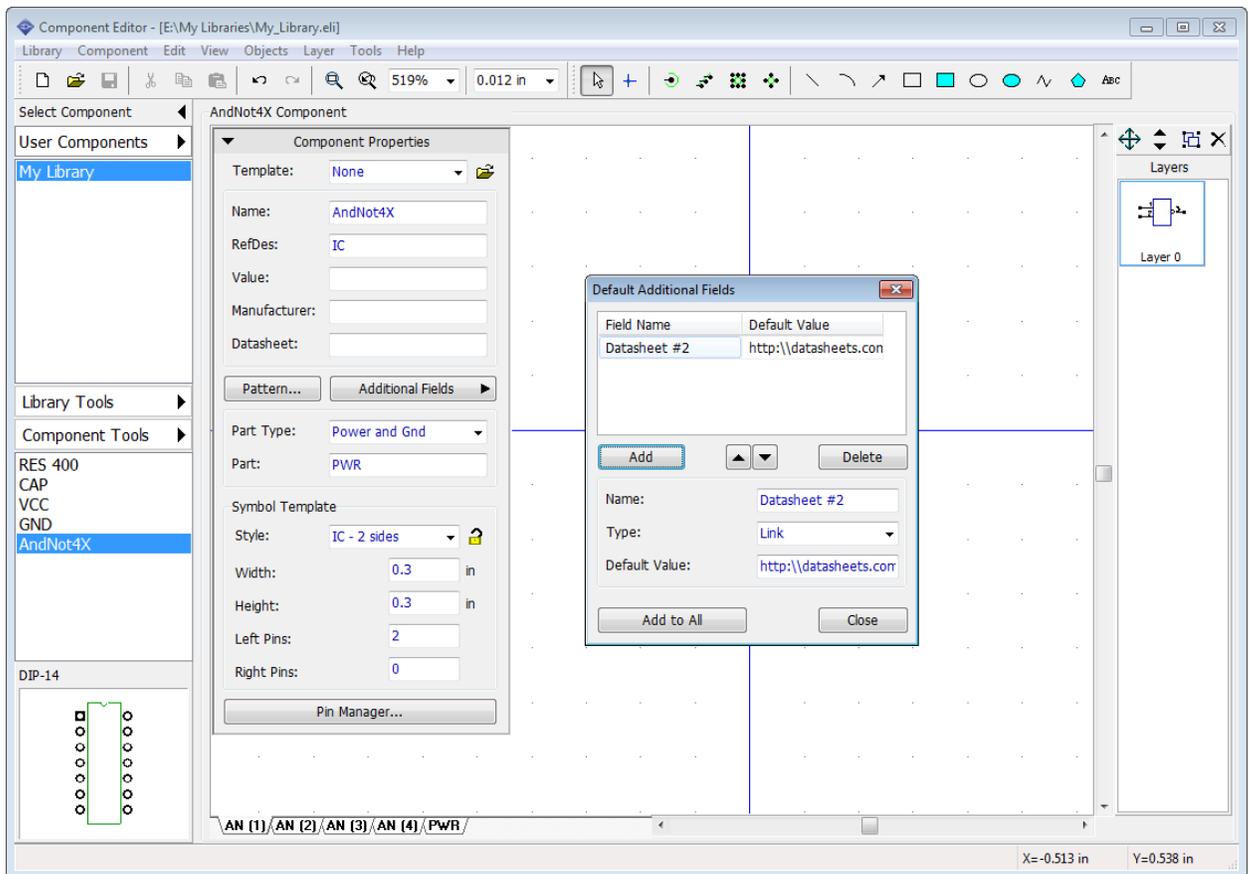
Multi-part component is ready. Save the library file.

#### 4.2.6 Using additional fields

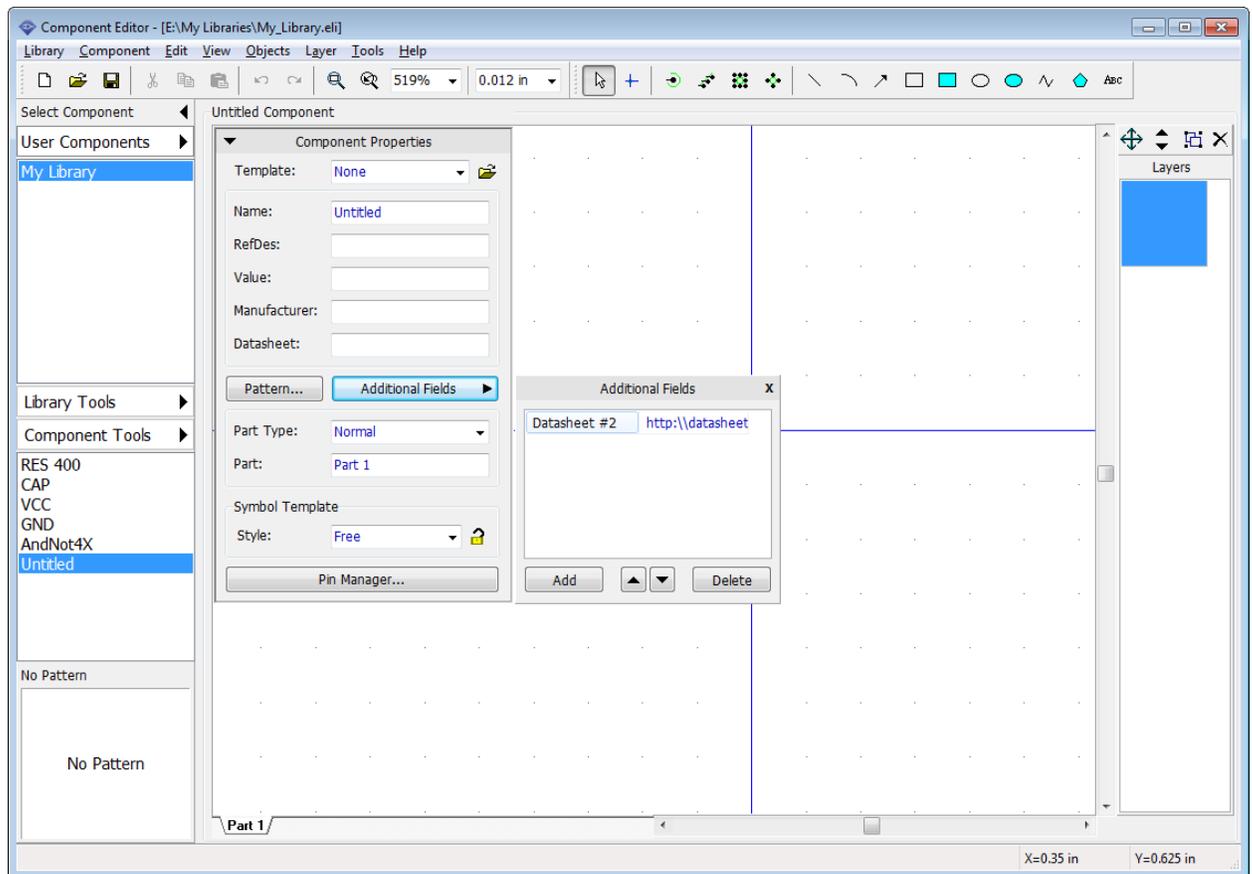
Name, RefDes, Value, Manufacturer, and Datasheet are default component fields in the PCB Creator, usually these are what most users will ever need, but sometimes additional description or other information is necessary. In this case you can use additional fields.

Select "Component \ Default Additional Fields" from the main menu. This dialog box allows you to specify default additional fields and their values, these fields will be added to **all new** components in the library.

For example, we need to have the link for one more datasheet online: type in "Datasheet #2" into field name box, specify "Type: Link", enter the web address and click Add button. Then press Add to all.



From now on all components in this library will have this additional field with the datasheet link. Close the Default Additional Fields dialog box. Select "Component \ Add New Component to My Library or press "Ctrl+Ins". Maximize the Component Properties panel using arrow in its left side (if the panel is minimized) and click Additional fields button to see the list of additional fields for the component.



Select Untitled component on the Library Manager panel, right click it and select Delete Components or simply press "Ctrl+Del" to delete it. You can also select several components and delete them at a time.

If you want to assign custom additional fields to the component, press Additional Fields button on the Component Properties panel.

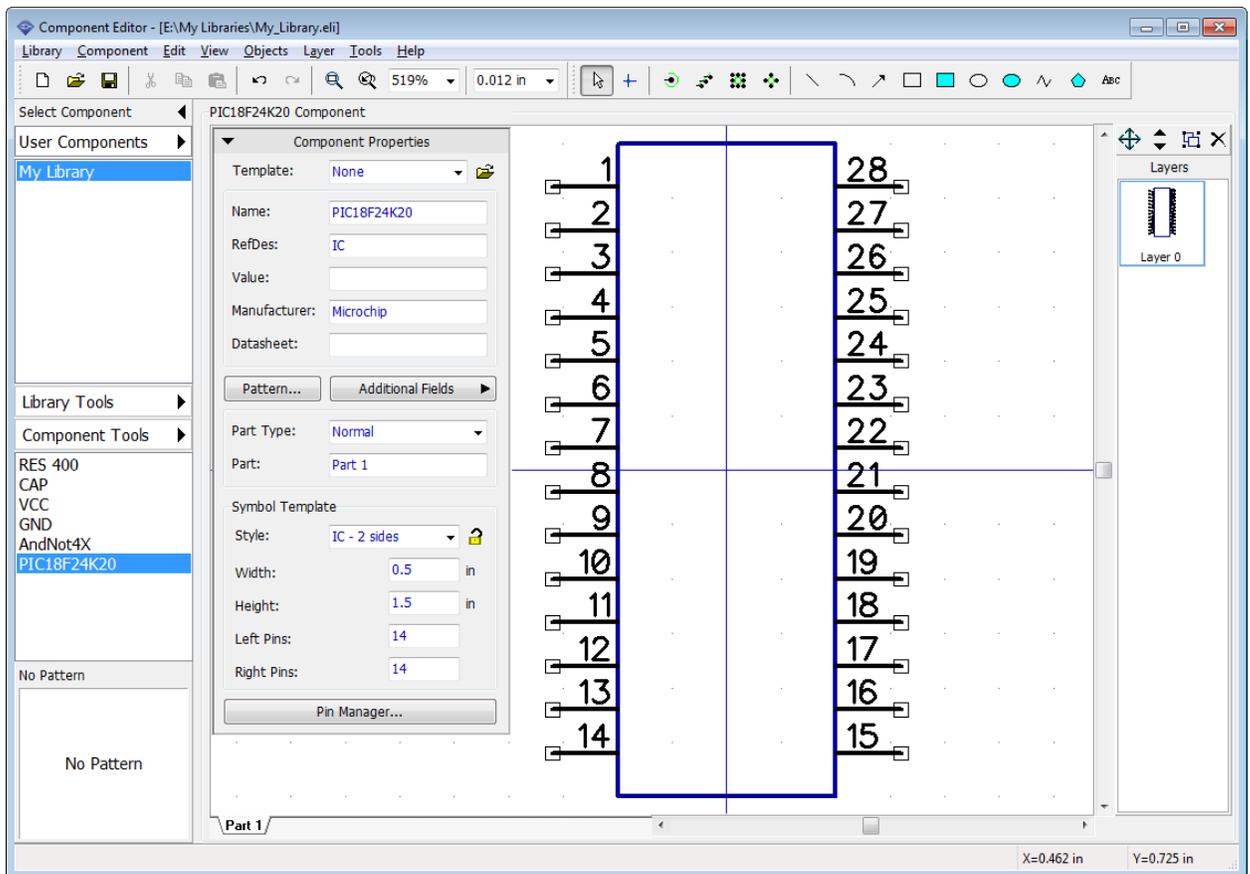
**Additional Fields in the Schematic.** Right click on the component with links to the datasheets on the design area of the PCB Creator Schematic and select Links from the right-click submenu – the web browser will pop up automatically. Additional fields can be assigned as component markings for all or selected components in the PCB Creator Schematic.

#### 4.2.7 Designing PIC18F24K20

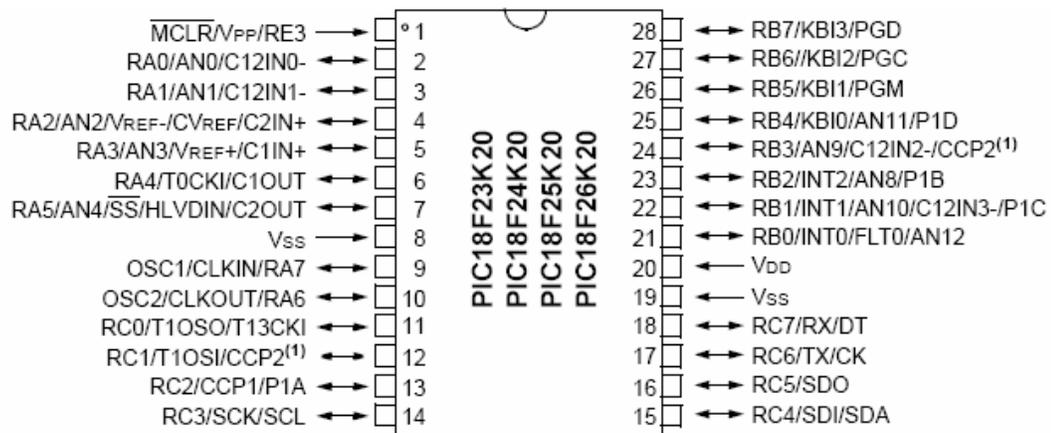
In this topic we will create a real-life PIC18F24K20 component according to the datasheet and attach SOIC-28 pattern that was created [before](#)<sup>[111]</sup>.

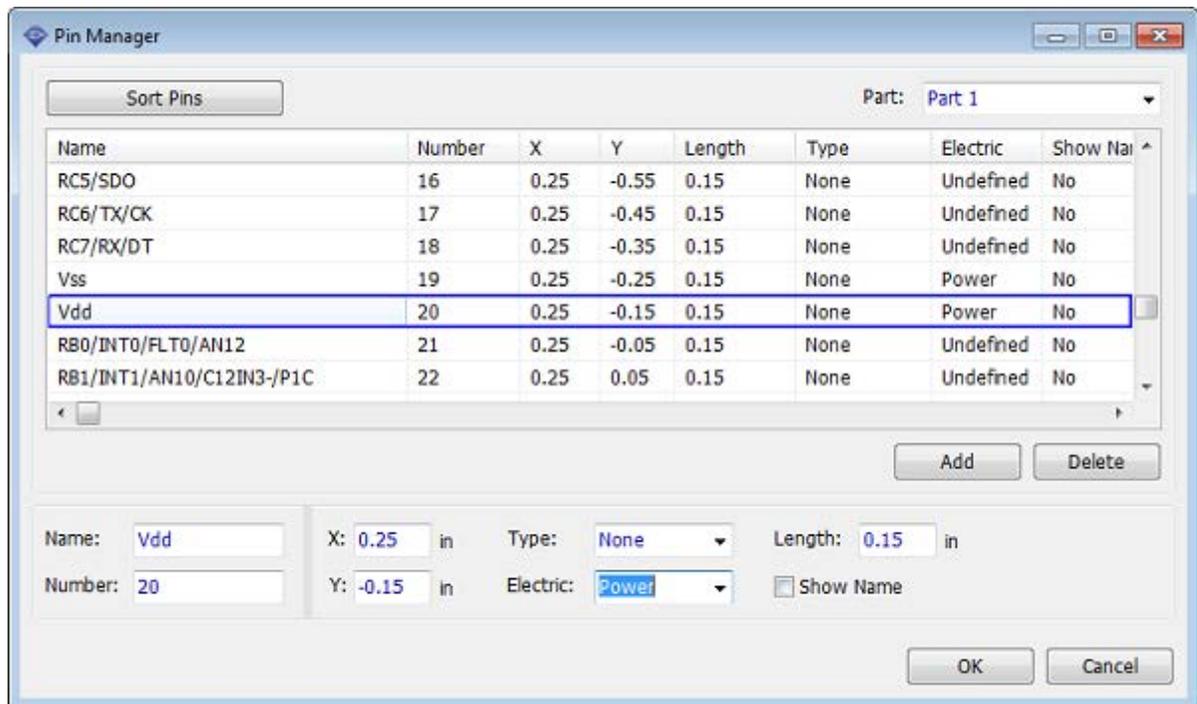
Go to the [Microchip website](#) and search PIC18F24K20, then select Datasheets on the left. Or use this direct [link](#) (however we don't guarantee that it works at the moment you read this tutorial). Go to Pin Diagrams, the first diagram is what we need.

Switch to the PCB Creator Component Editor and add a new component ("Ctrl+Insert"), type in the name "PIC18F24K20" then specify: "Style: IC - 2 sides", "Left Pins: 14", "Right Pins: 14", RefDes, and manufacturer.



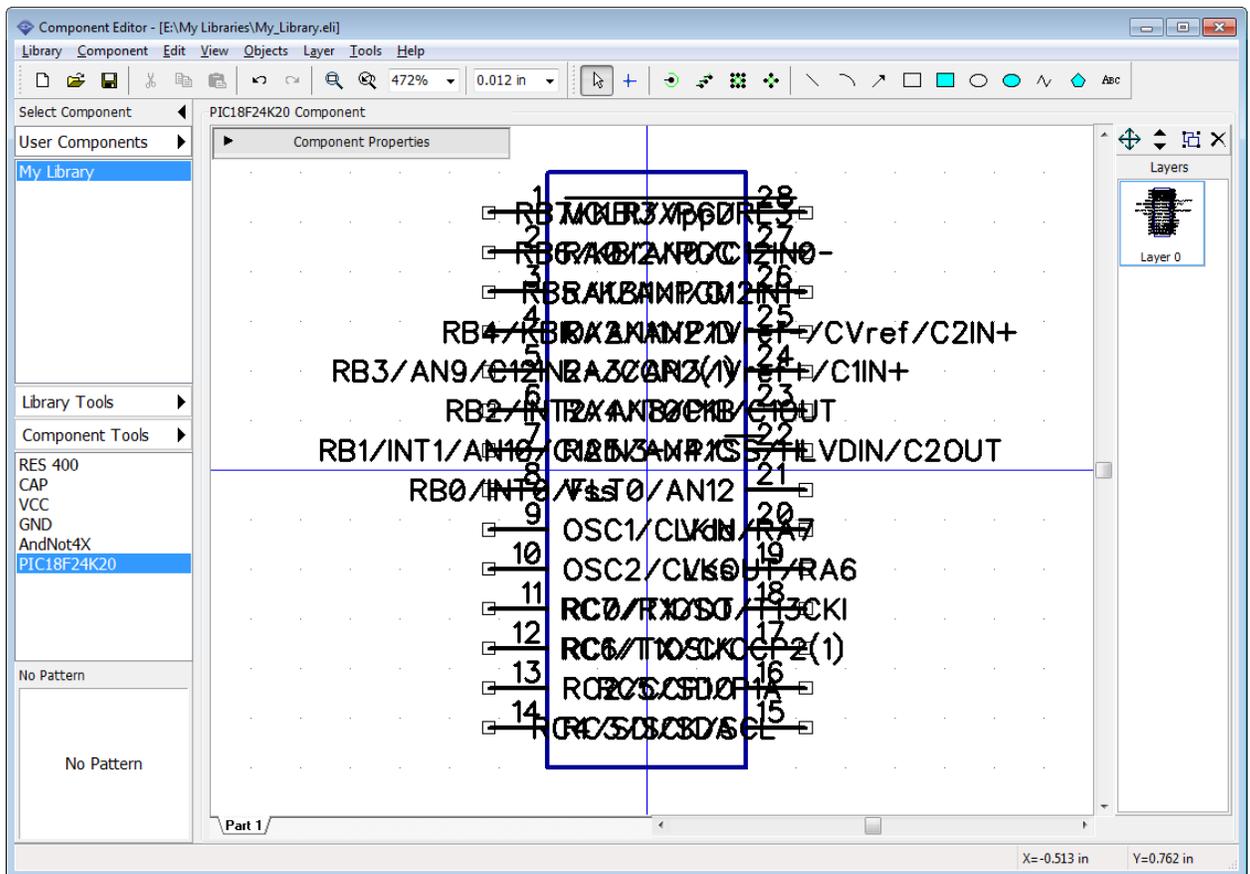
PCB Creator allows the user to enter pin names manually or import them from external BSDM file (select "Library \ Import \ Add BSDM Pinlist" from the main menu). We will do this manually. Press the Pin Manager button on the Component Properties panel and enter pin names from the pin diagram in the datasheet found online. Notice that you can resize the pin manager panel and change the width of the columns (we made the Name column wider to see the full pin names). Also when you entered the pin name, just press Enter to switch to the next pin name.



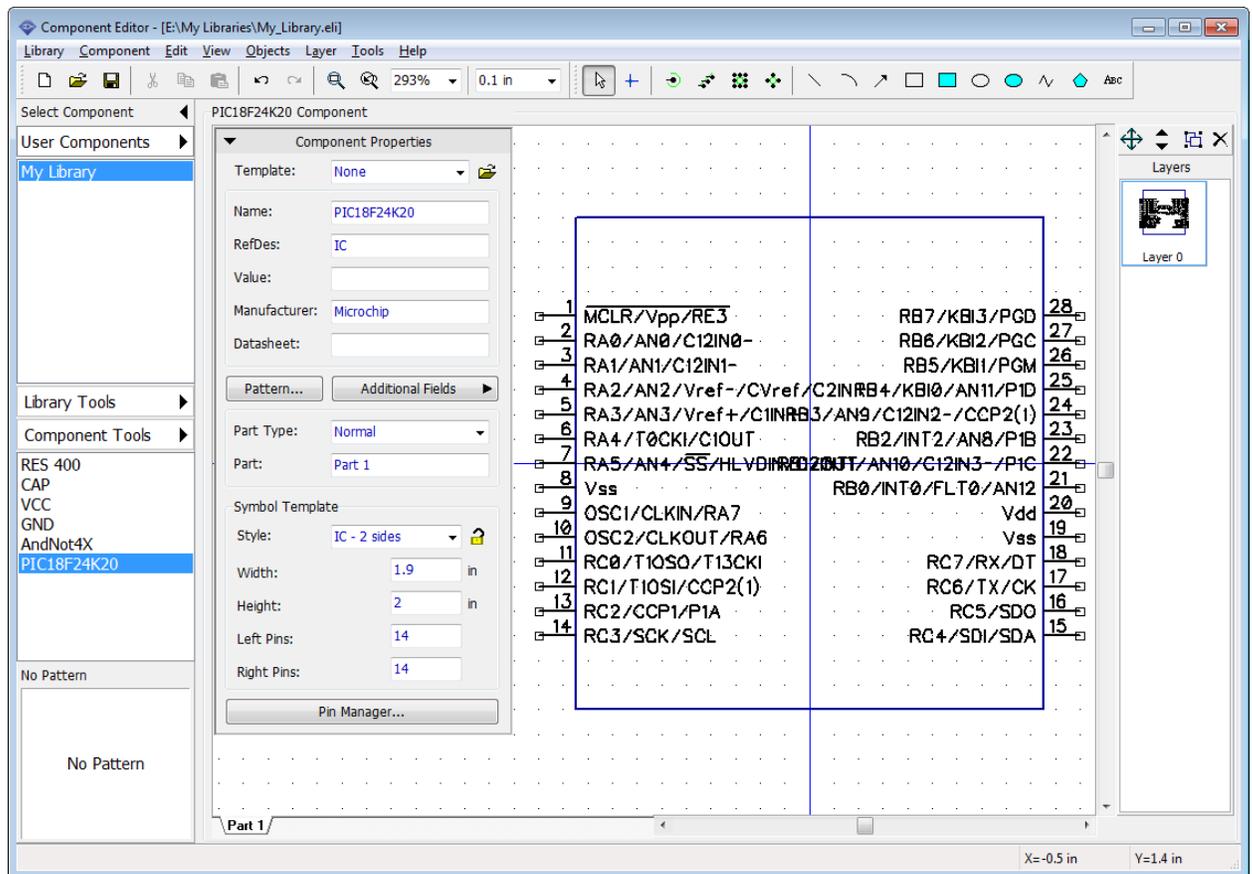


After entering all pin names please specify electric type for pins and check Show Name box for all pins of the component. Notice that you can select as many rows as you need and change certain properties at a time. Press OK.

Our symbol has inappropriate look, width is too small, the pin names overlay.



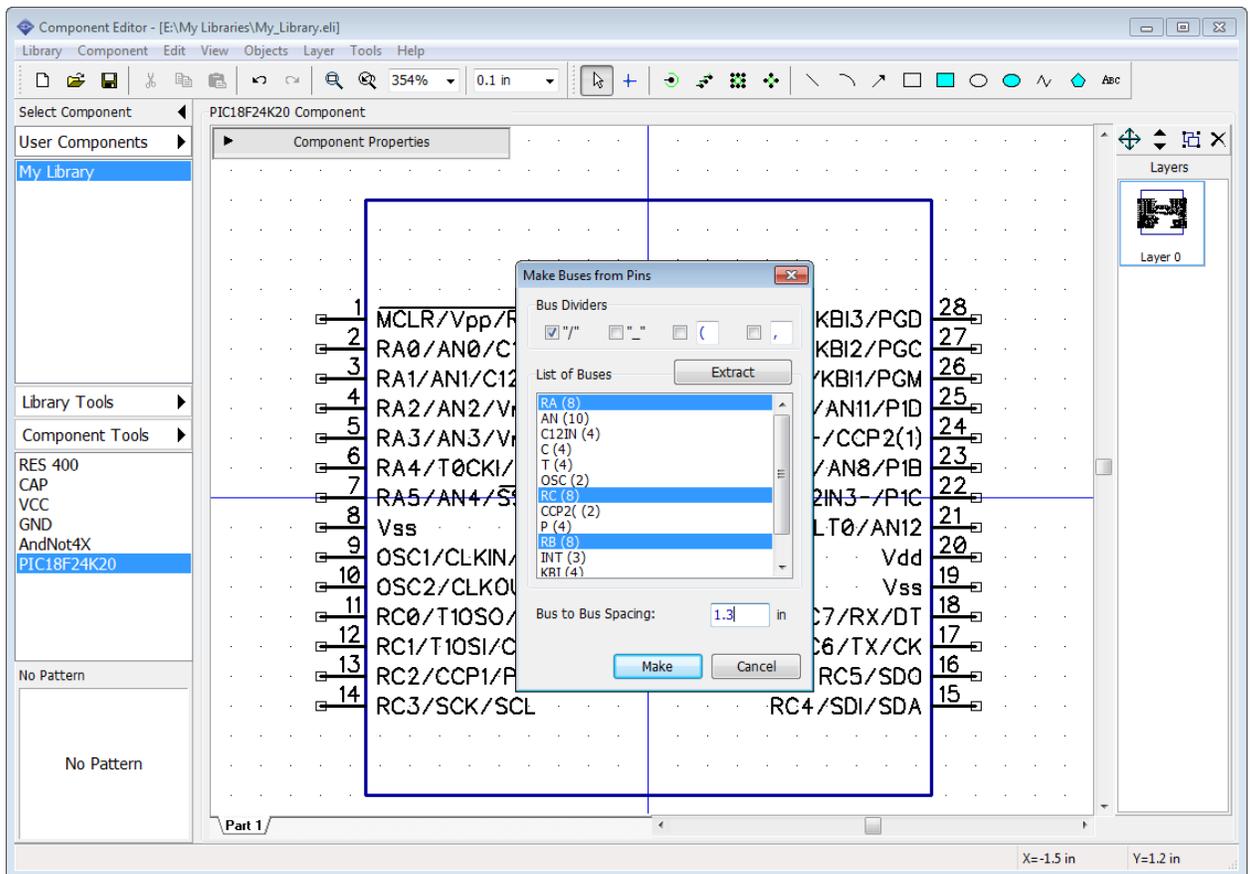
On the Component Properties panel change width to "1.9 in" and height to "2 in". Pin names still overlay a bit, but we will make pin regrouping which will probably fix this. We made an IC a bit bigger, because it will make it easier to do the pin regrouping. Change the grid to 0.1 inch and place pins by this grid (select all pins, right click, and select Snap to Grid).



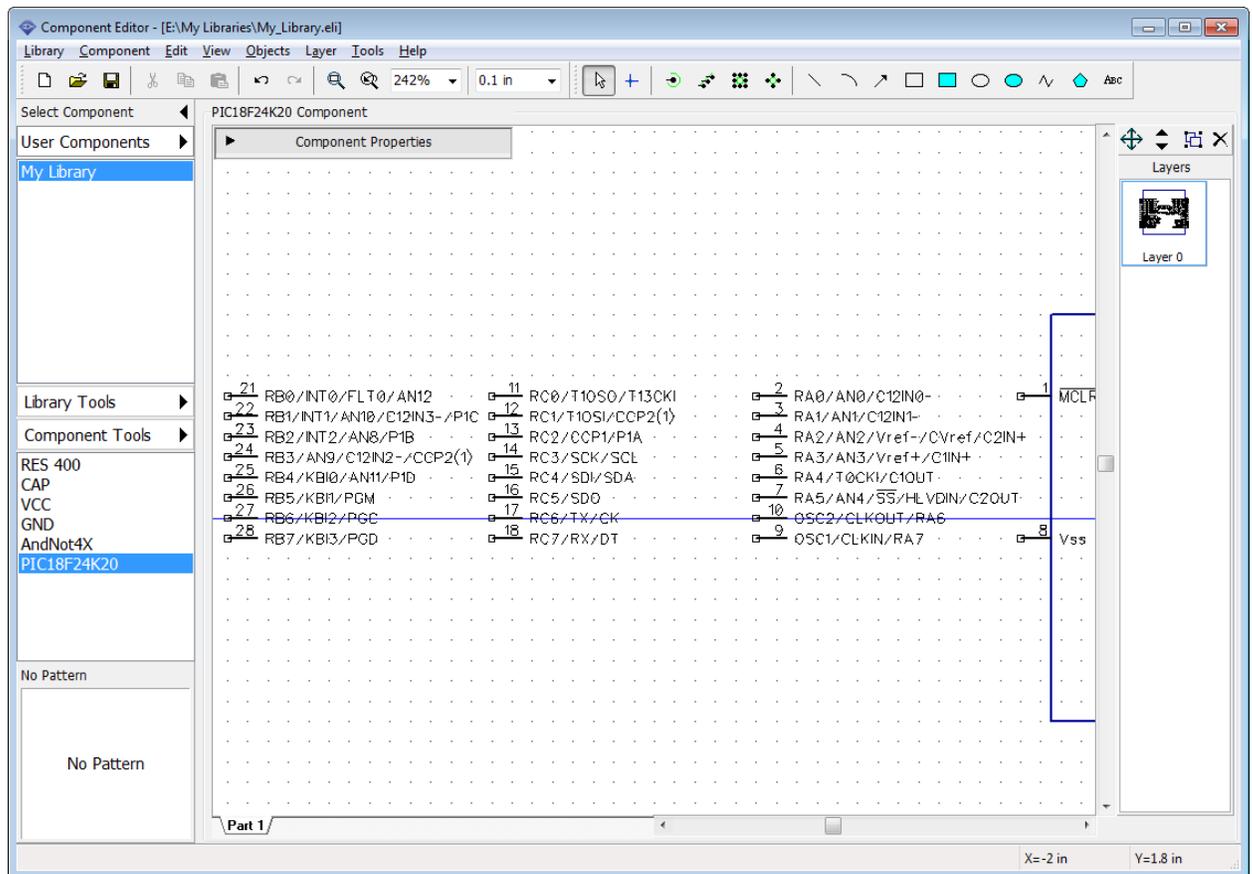
### Make buses (Group pins)

We need to group pins logically. First we will make the busses: select "Component \ Make Buses from Pins" from the main menu. This feature allows the user to extract busses based on the pin names and group pins by busses. In the pop-up dialog box you can define possible bus dividers. By default only "/" is selected and it is OK for current component, however, some manufacturers use different dividers.

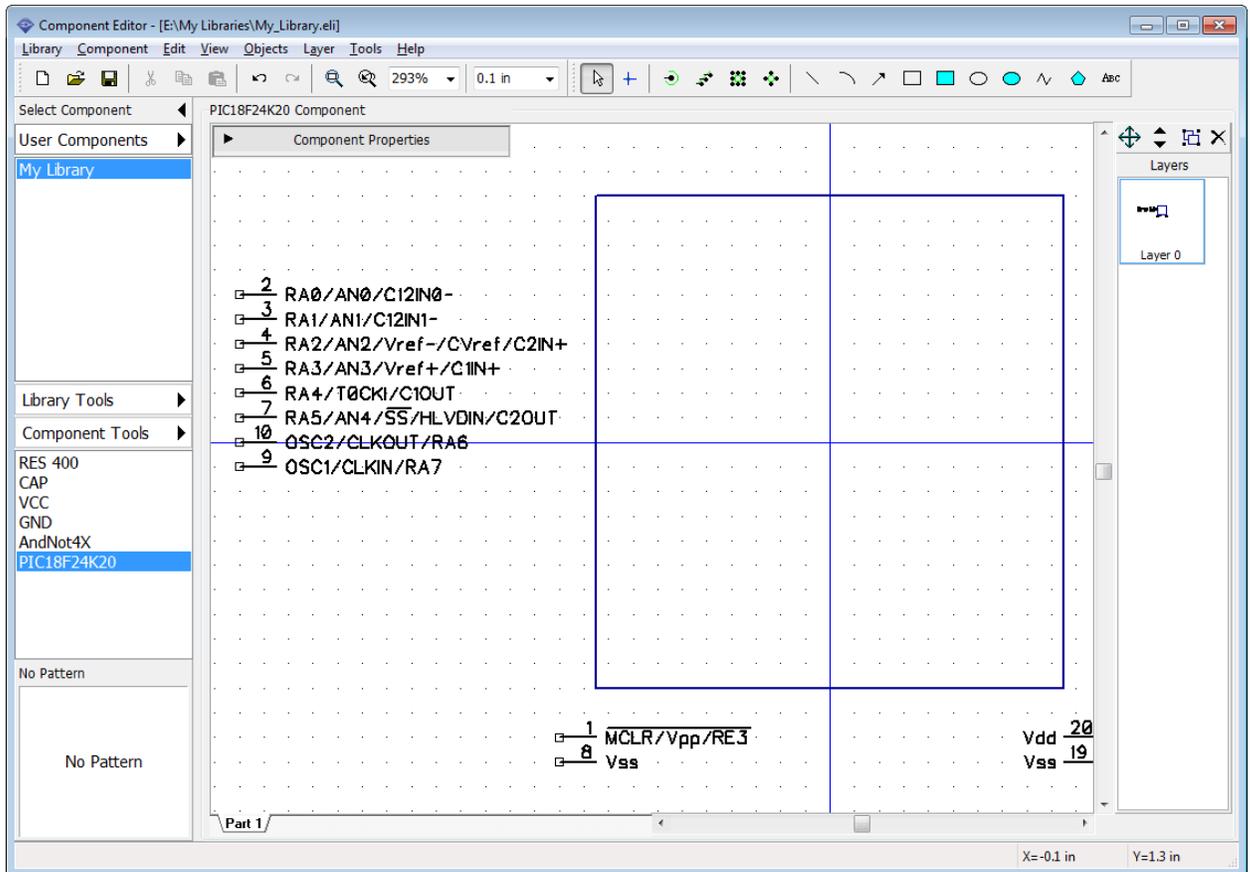
Press Extract button and you will see available busses and number of pins for each of them. Select RA, RB and RC busses using "Ctrl" key. Change Bus to Bus spacing to 1.3 inches, because pin names are quite long.



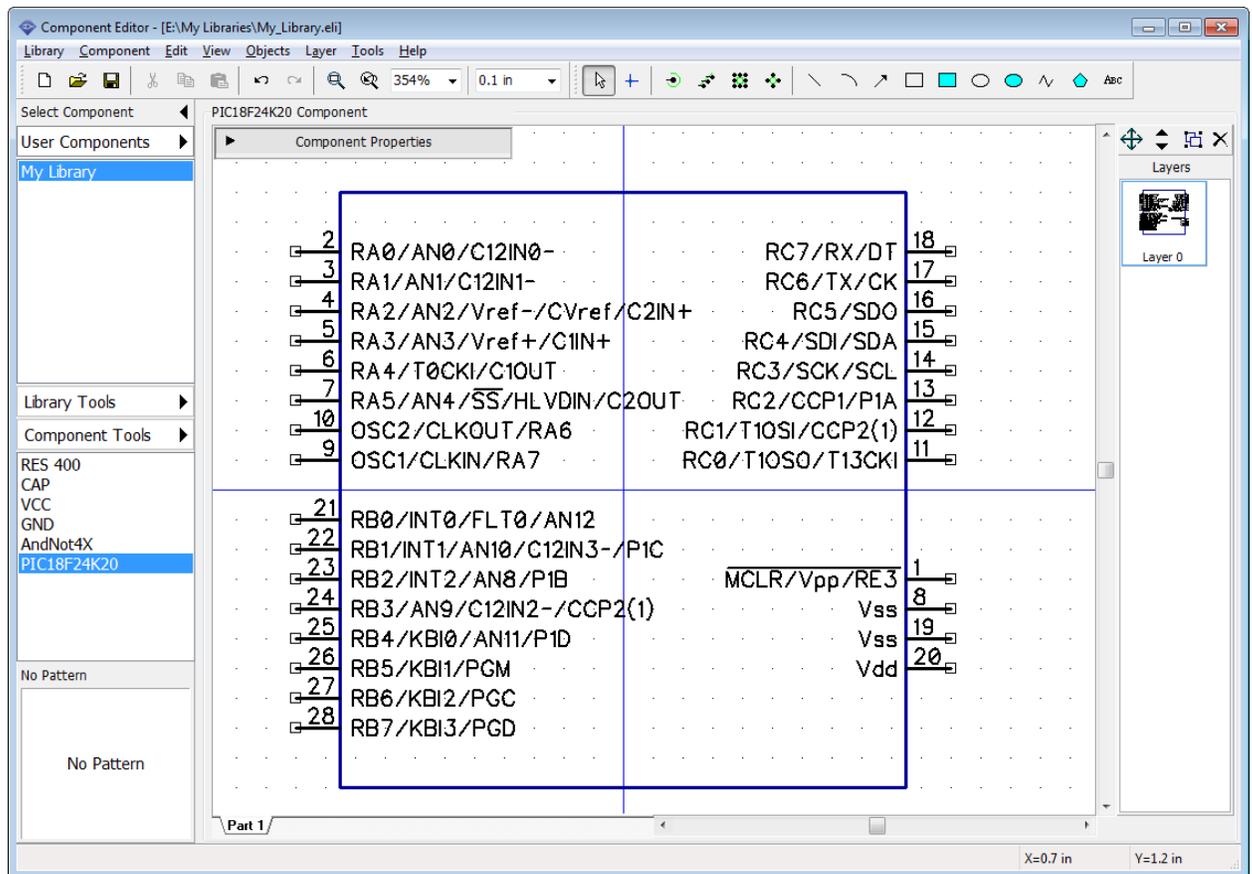
Press Make button to make the busses and close the dialog box. Sorted wires that belong to the buses will be placed to the left from the IC symbol.



There are some pins that do not belong to selected busses (4 pins that left on the symbol). Select them, use "Ctrl" and box selection, then move pins away from the symbol, for example, to the bottom, because we need to place the busses first.



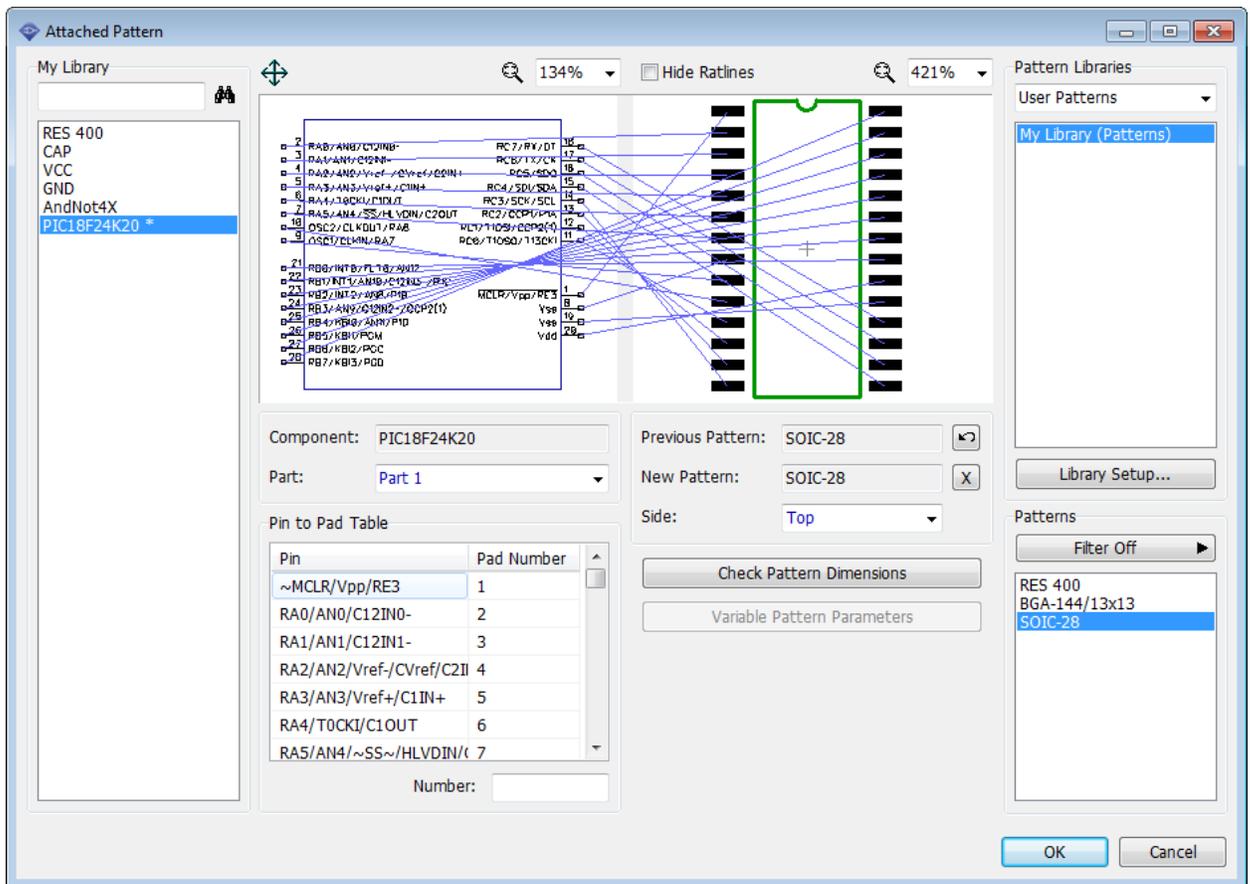
Place buses to the IC rectangle like on the picture below. Use box selection to select a bus, then drag it. "Shift+R" can be used to rotate a bus and "Shift+F" – to flip pins, these commands can be selected from the pin submenu as well (right click on one of bus pins). Then move the rest of pins to the IC rectangle ("R" can be used to rotate selected object/pin).



Sometimes you need to place pins by electric type, select "View \ Pin Colors by EType" and pins of various electric types will have different colors, making it an easy task.

Lock properties on the Component Properties panel (lock button is to the right from the Style drop down).

Final step is attaching SOIC-28 pattern to the component. Press the Pattern button on the Component Properties panel and select SOIC-28 pattern from My library (we've created it [before](#) [111]). Select "User Patterns" library group, then select My Library (Patterns), and SOIC-28 from the list below. All pin names and pin numbers are already there. You can check pin to pad connections in the table. We don't need to change anything. Just press OK.



PIC18F24K20 component is ready! Save the library.

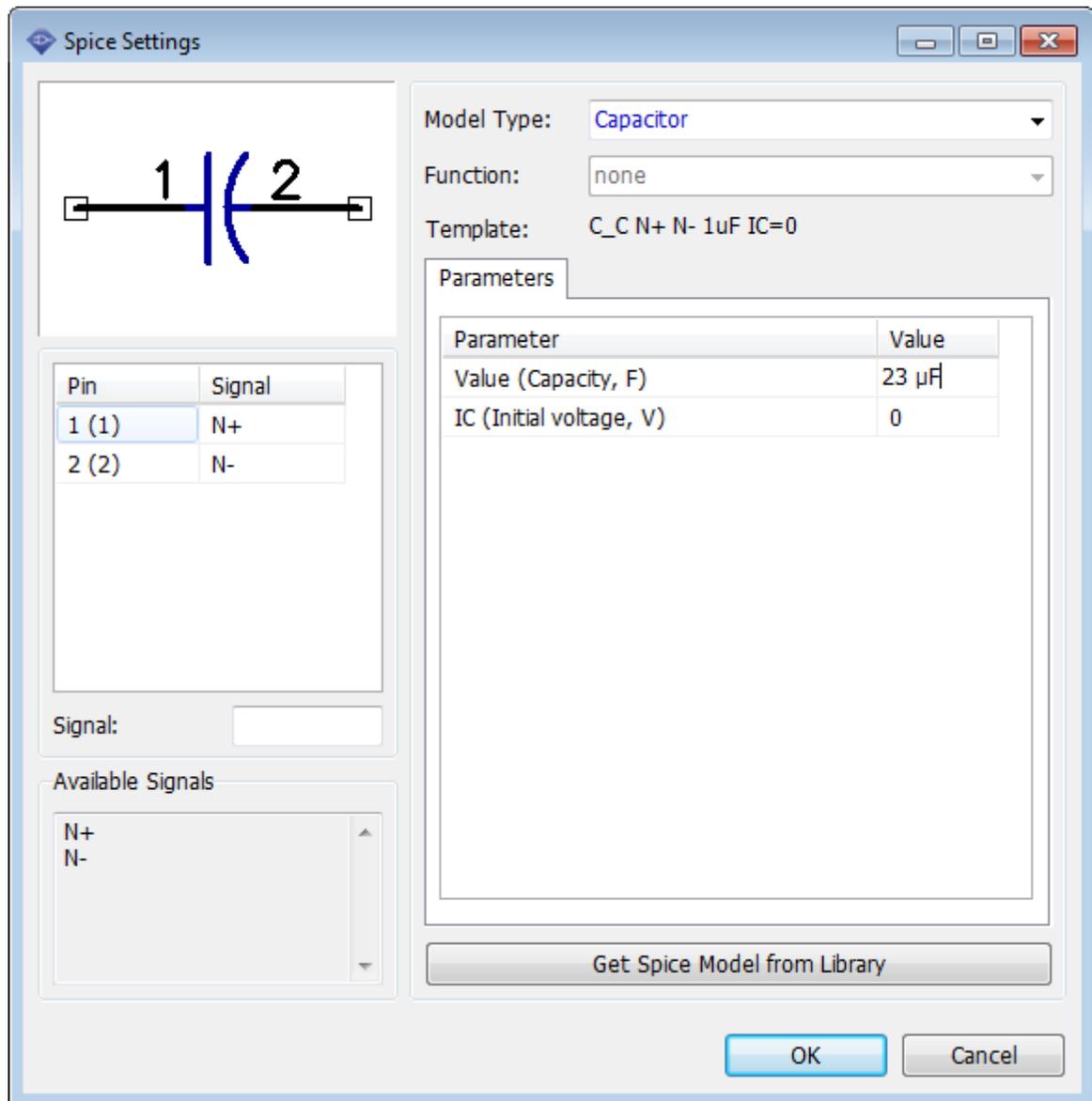
### Connect one pin to several pads

In PCB Creator you can connect one pin of the schematic symbol to several pads of the footprint. Just left click on the corresponding pads in the pattern preview of the Attached pattern dialog box.

## 4.2.8 Spice settings

With PCB Creator you can export schematics into LT Spice to simulate it and see how the circuit works. We will review this step-by-step later. You need to define correct spice settings for the component, otherwise Spice simulation for this component will not work. Now we will specify that our CAP component is a capacitor with some value and it can be added to the Spice netlist. Select CAP in the components table. Then select "Component \ Spice Settings" from the main menu. Select "Model Type: Capacitor", then double click in the "Parameters : Value" (cell with "1uF" text) and enter a new value – "23  $\mu$ F" (PCB Creator supports Unicode characters). Press Enter or just move focus to another field.

In the Template field you can see how this part looks in the Spice netlist language. In our case pin-to-signal map is correct, however, if you need to edit it, enter signal names into the table on the left side of the Spice Settings dialog box. The list of available signals is right below that table.



Capacitor is a very simple part, we don't need specific text file model to let the simulation software know how this component works (model type and capacity is just fine). However, for transistors you can load models from external files (usually Spice models are available from the manufacturers' websites) or enter model text manually if you know how to do that (see Spice Language documentation for details). Also there is a SubSkt model type, which allows the user to enter/load model of almost any part as a program.

Press Get Spice Model from Library to load existing Spice settings from another component.

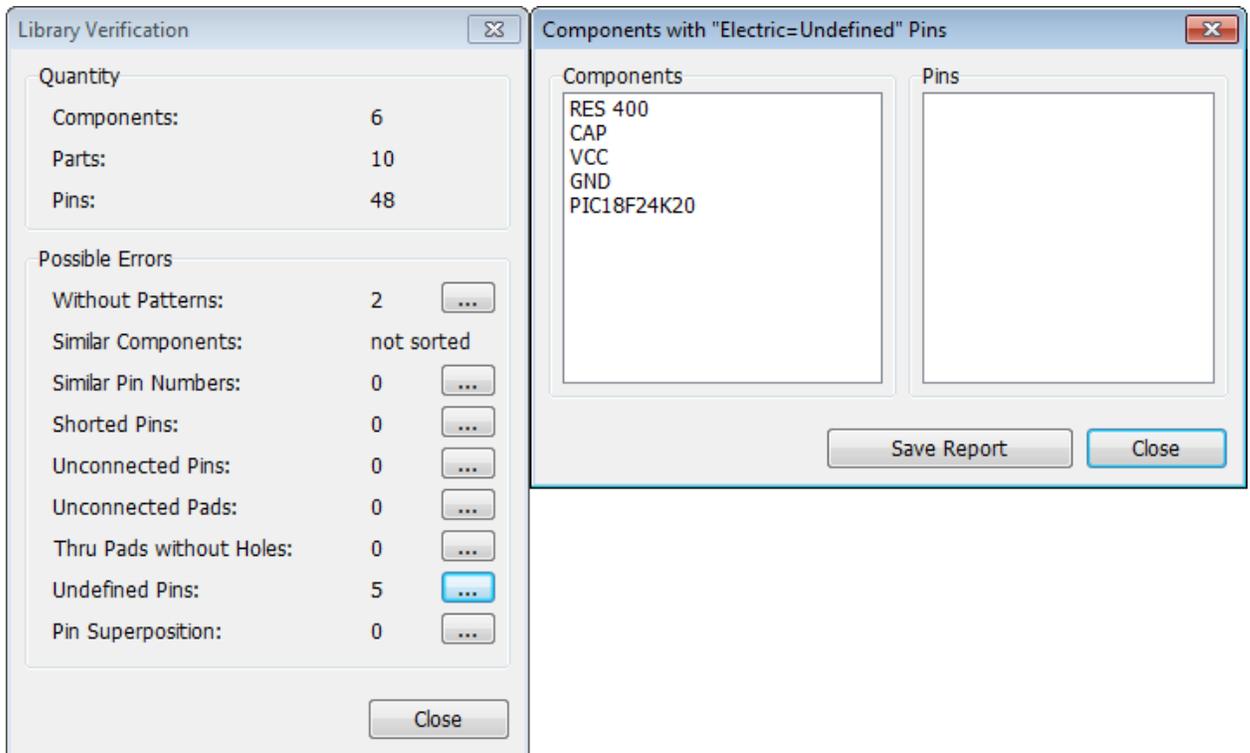
Notice that this dialog box is also available in the Schematic Capture, you can define Spice settings after completing or while drawing the schematic.

Library design is ready. Click OK to apply changes and close the Spice settings. Save the library file.

## 4.2.9 Library Verification

It is very important to verify library for the most common types of errors. We've investigated the work of our library designers and added possible automatic error verifications to the Component Editor.

In the Component Editor select "Library \ Check "My Library" Library" from the main menu. In this dialog box you can see the total number of components / parts / pins in your library and all possible errors.



The following errors can be found automatically:

1. Components without patterns – keep in mind that some components may intentionally have only schematic symbols.
2. Similar components – components with similar names. Notice that library should be sorted ("Library \ Sort Components in My Library) to allow for correct verification with this feature.
3. Similar pin numbers – two or more pins with the same numbers (connected to the same pad). In 99% this is a mistake in the component, press "." button and check pin numbers for listed components.
4. Shorted pins – pins shorted by internal pad-to-pad connections.
5. Unconnected pins – pins without a corresponding pattern pads. Not always a mistake.
6. Unconnected pads – unused pads of the pattern (no corresponding pins). Not always a mistake.

7. Through pads without holes – in majority of cases this is a mistake in SMD pattern, please check if pads have surface type selected.
8. Undefined pins – pins with Undefined electric type.
9. Pin superposition – pins superposed on the symbol, in majority of cases this is a design mistake.

To see details (list of components and pins) press "..." button next to the corresponding mistake. List of errors can be saved as a text file.

Save changes and close the Component Editor.

## 4.2.10 Placing parts

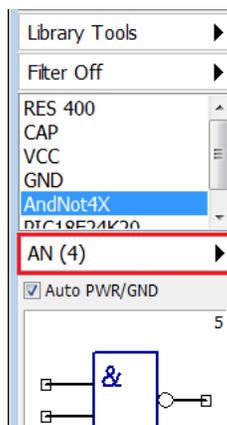
### Schematic

Open the Schematic module, i.e. go to "Start \ All Programs \ PCB Creator \ Schematic" in the Windows OS. Select My Library from the User Libraries library group. Place a couple of resistors and capacitors to the design area. Just left click on the component on the component placement panel (Library Manager) and left click on the design area. If the design origin is shown, press "F1" to hide it.

*Notice that you can place components using "Objects \ Place Component" panel or with the corresponding button on the objects toolbar.*

*Notice that colors of components, selections, etc., depend on the color template and user preferences.*

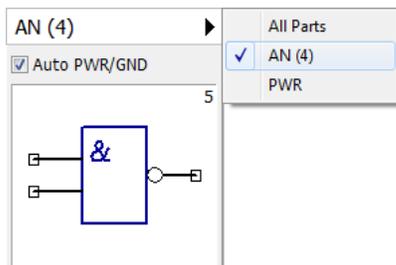
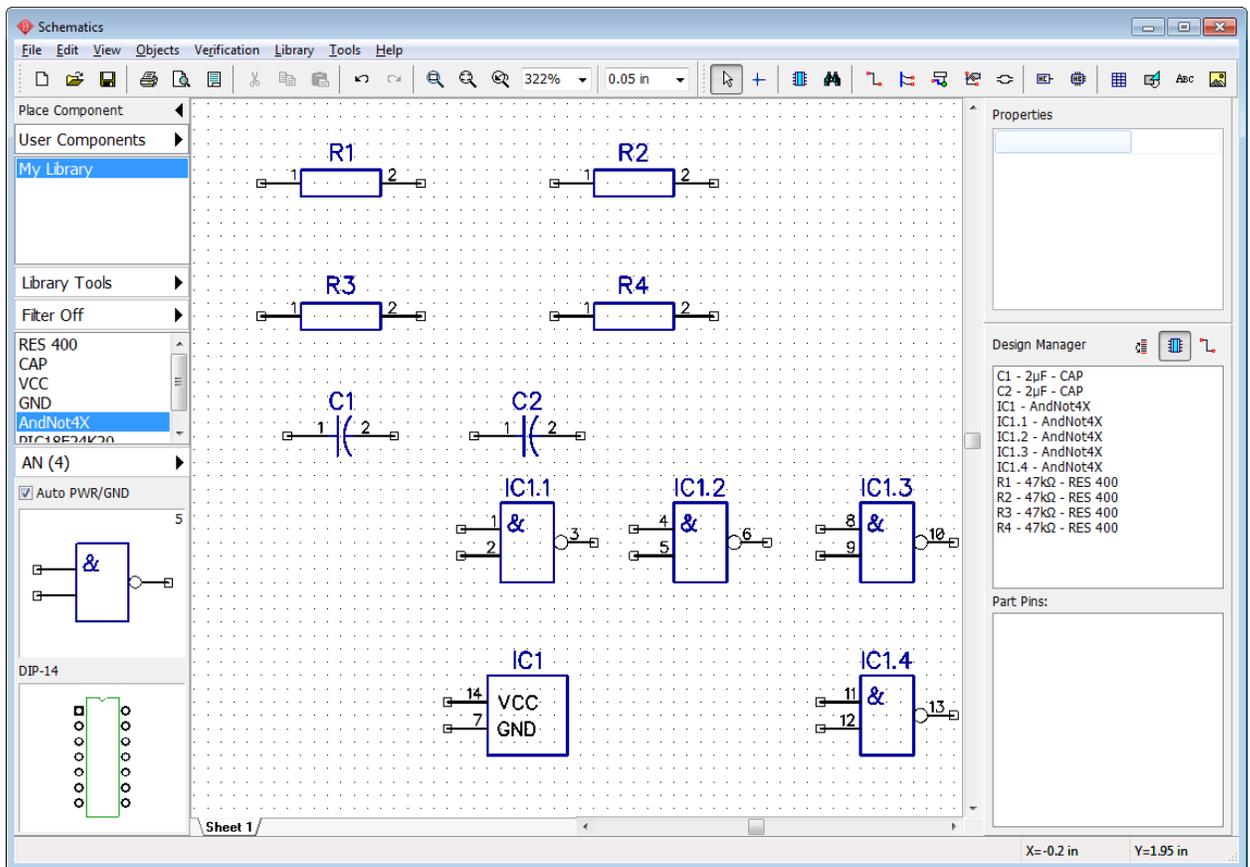
### Multi-part component placement



Select multipart component (ANdNot4X). The PCB Creator always shows that current component is multi-part or net port in the symbol preview field on the component placement panel (<number of parts> or "Net Port" text, see the picture). As you remember, we have created "AndNot 4X" with four similar parts and a power part in the previous paragraphs of this tutorial. All parts can be placed at a time or each part separately, use part selection button (highlighted in red on the picture). Switch to "AN(4)" to place each part one-by-one, or select "PWR" to place only the power part. If Auto PWR/GND item is checked component's power part will be automatically placed on the design area together with the first logic part.

Select "AN(4)", leave Auto PWR/GND checked and place AndNot4X component part by part. The PCB Creator automatically selects the next part from the part group and places the power symbol for the component.

*Notice that program will automatically switch to the next part, when all parts had been placed. Make sure you place parts only with RefDes from 1.1 to 1.4, like on the picture below.*

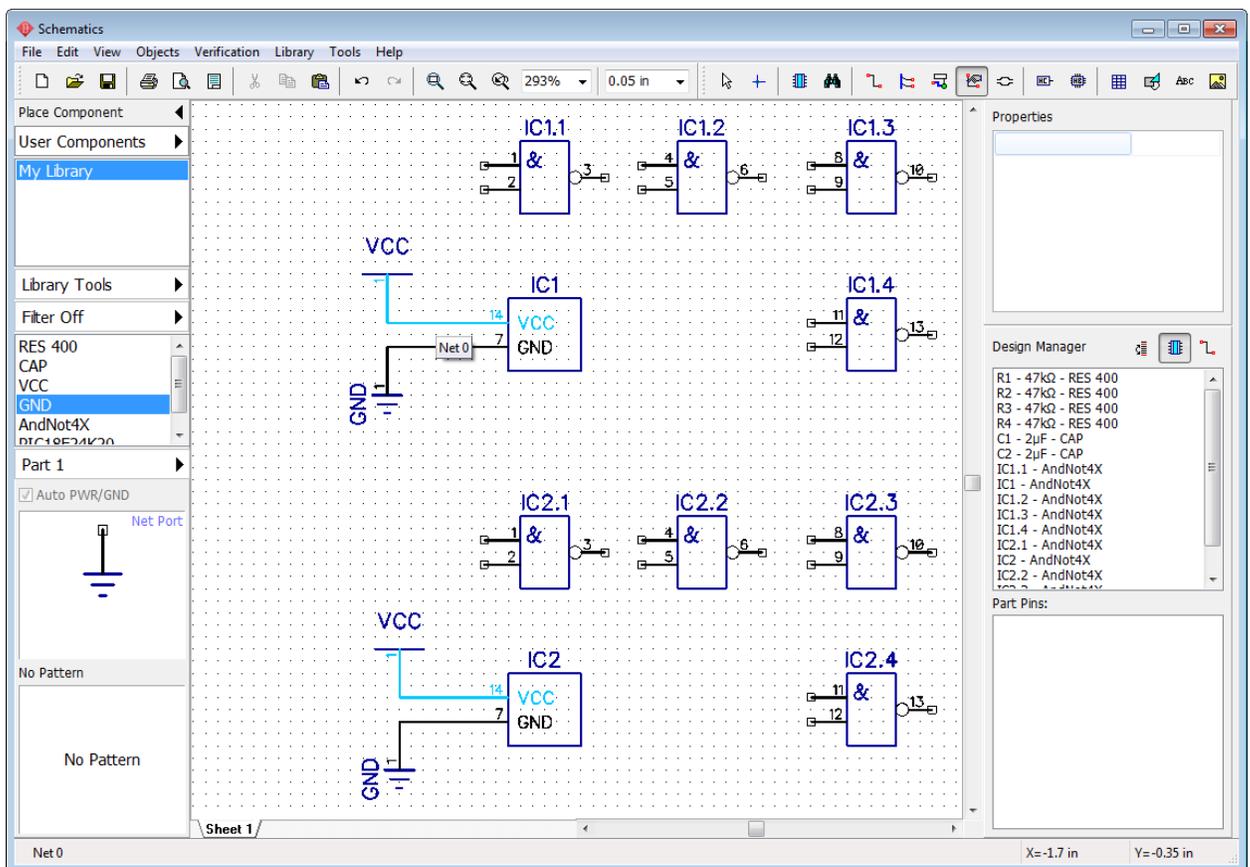


As you can see, the similar parts of the multipart component are already grouped inside the part selection drop-down list. Select "View \ Group Similar Parts" from the main menu to ungroup parts. If it is checked, logic parts of the component ("AN(4)" in our case) will be grouped inside All Parts pop-up menu. If unchecked, you will be able to select and place each AN part separately.

### Connection via net ports

Make sure that you have two AndNot components on the design area (IC1 and IC2) with two power symbols. Now we'll try to use the net ports. Select VCC and GND net port symbols from the library and place two of each to the schematic. Connect pins like on the picture below.

*Notice that two wires connected to the same pins of same-type net port are connected to a single net automatically.*



To rename the net which connects VCC pins, right click on the wire and select first item from the submenu or right click on the pin and select Net Name.

*Notice that you can change part names directly in Schematic (in the Component Properties dialog box).*

Schematic allows the user to:

- 1) Connect pins to nets without wires (right click on the pin, select Add to Net, then select net, check Connect without wire box, and press OK);
- 2) Merge nets by name (check Connect Nets by Name box in the net properties dialog window);
- 3) Connect pins to the net with similar name automatically (check Connect Net to Pins by Name box in the net properties dialog window). The last method is the fastest way to connect VCC, GND (if you plan to hide power nets and parts), CLK nets, etc. More information [later](#)<sup>[165]</sup> in this tutorial. Close the Schematic. Do not save the changes.

### PCB Layout

Open the PCB Layout module, i.e., go to "Start \ All Programs \ PCB Creator \ PCB Layout" in Windows.

*As you already know, correct component always includes at least a schematic symbol (for Schematic) and attached pattern (for PCB Layout). Schematic works only with symbols, while PCB Layout allows the user to select component libraries and place component's patterns on*

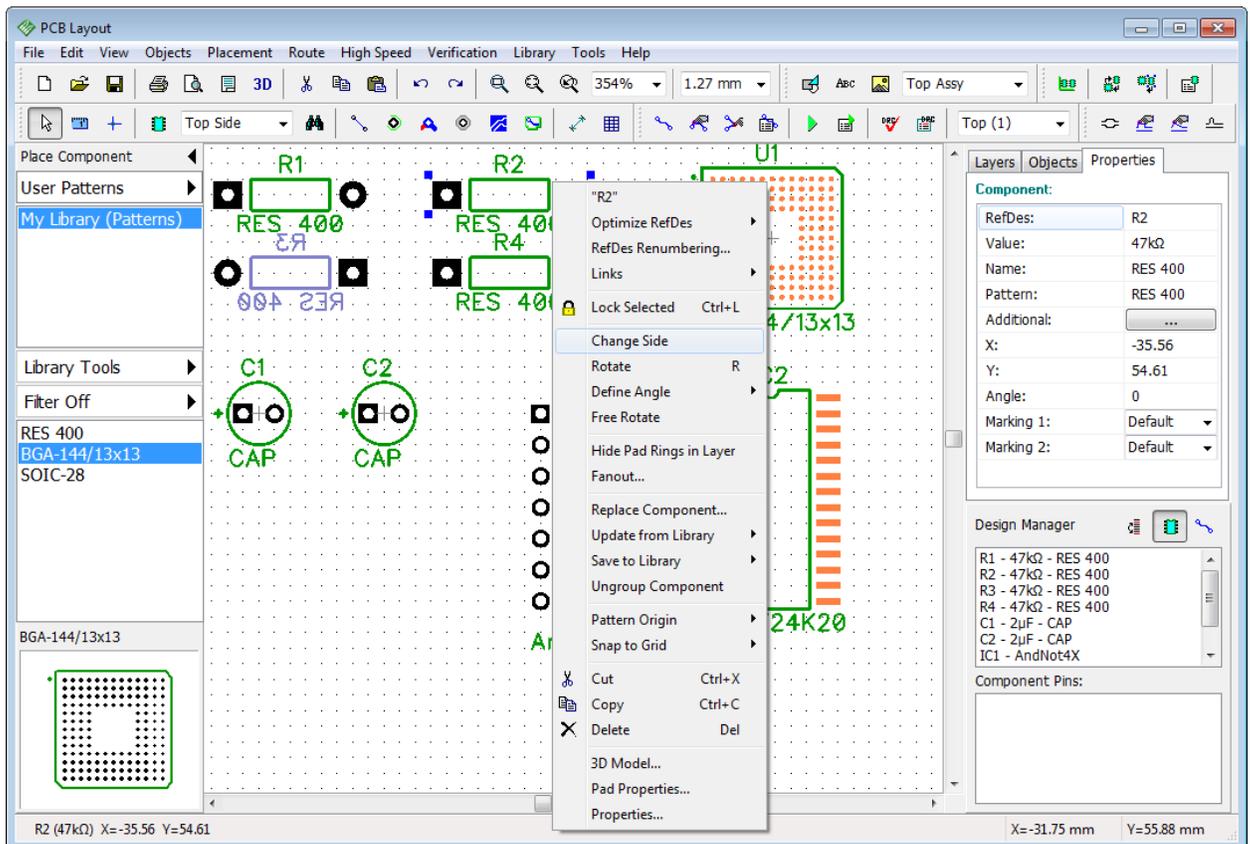
the board. If you have selected component library and there are components without attached patterns, you can not place them on the board. Notice that there is Patterns library group, which allows the engineer to place patterns as separate entities (no schematic symbols attached).

Select User Patterns library Group and My Libraries (Patterns). You will see that all three patterns that we have created during the lessons of this tutorial are available here. Now select My Library from the User Components library group. As you remember, we did not attach patterns only to net ports, therefore we can not place them on the board in the PCB Layout. All other components are visible, because they have attached pattern.

Place all components of the library to the design area, excluding "VCC" and "GND", then return to My Library (Patterns) from the User Patterns library group and place BGA-144... pattern, which we did not attach to any of the symbols, therefore it is not available in the component library.

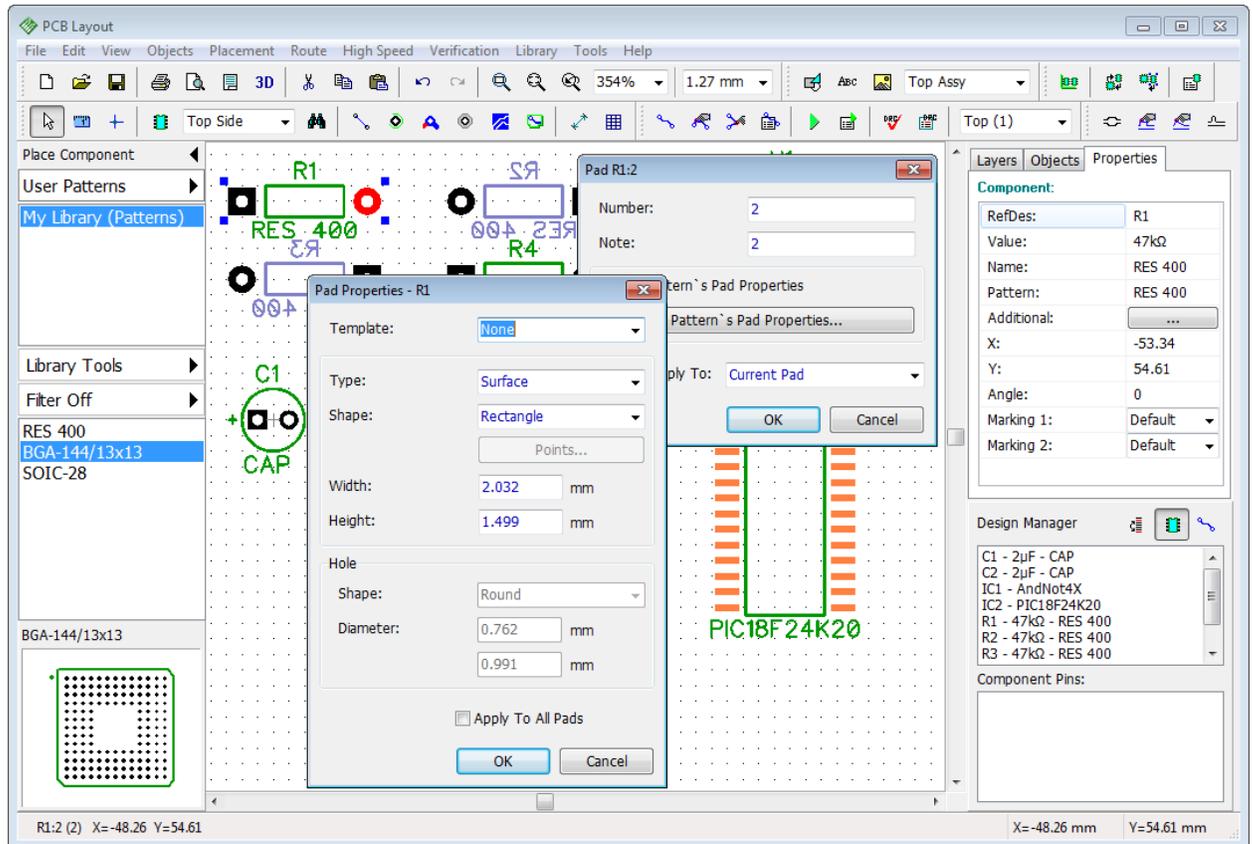
Change common marking settings to show RefDes and Name ("View \ Component Markings \ Main \ RefDes" and "View \ Component Markings \ Additional \ Name" from the main menu). For individual customizations right click on the component \ Properties \ Markings. You might need to justify texts in "View \ Component Markings \ Main (Additional)\ Justify \ Auto".

Select the Bottom Side in the drop-down list on the Objects toolbar if you want to place components on the opposite side of the board. For existing components you can change the side with the right-click submenu. For example, R4 resistor is on the bottom side (see the picture below).



Pad properties for the separate pad or entire component's pattern can be changed directly in the

PCB Layout. Let's change one of the resistor pads. Hover over the pad you want to change (it should be highlighted), right click it and select Pad Properties from the submenu. In the pop-up dialog uncheck Use Pattern's Pad Properties box for custom pad settings or press the Pattern's Pad Properties button to change default pad settings for the pattern. To edit pattern pad properties right click on the pattern (not the pad) and select Pad Properties from the submenu.



*Notice that if some pattern's origin is different from the pattern's center position it will be shown while placing that pattern.*

Pattern origin can be shown / hidden for all selected components: right click one of them and select Pattern Origin from the submenu. Try to rotate different components and you will see that pattern origin is actually the center of rotation. When hovering over the pattern you will see the coordinates of the pattern, which are actually the coordinates of the pattern's origin.

## Part 5: Other features and tools

This part of the tutorial includes the description of important PCB Creator features not reviewed above. We consider that the reader already knows how to accomplish basic tasks, therefore we can move on to more complex stuff.

## 5.1 Connecting

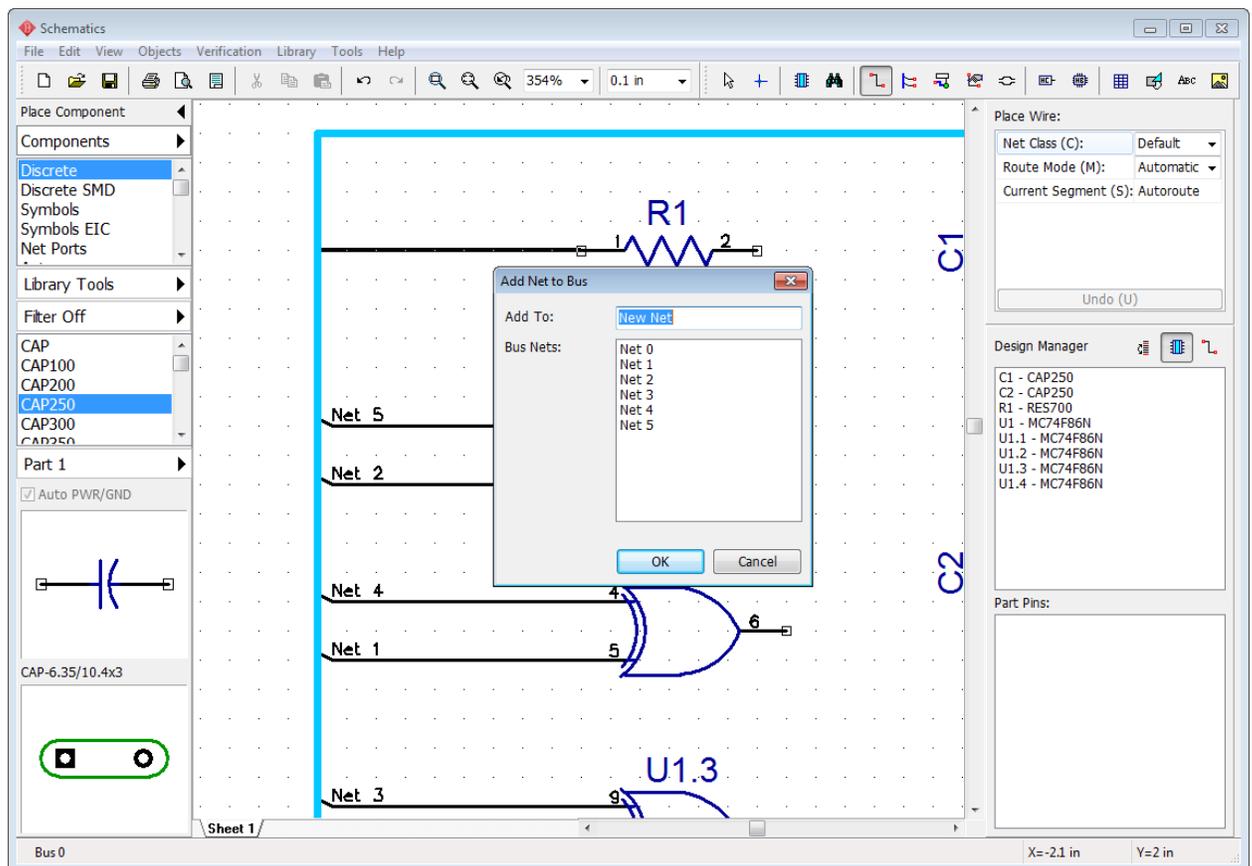
### 5.1.1 Working with Buses and Bus Connectors

Now we will learn how to use buses and connect sheets with bus connectors in the Schematic. You can work with the circuit from the previous subsection of this tutorial or create a new schematic with random components for practicing.

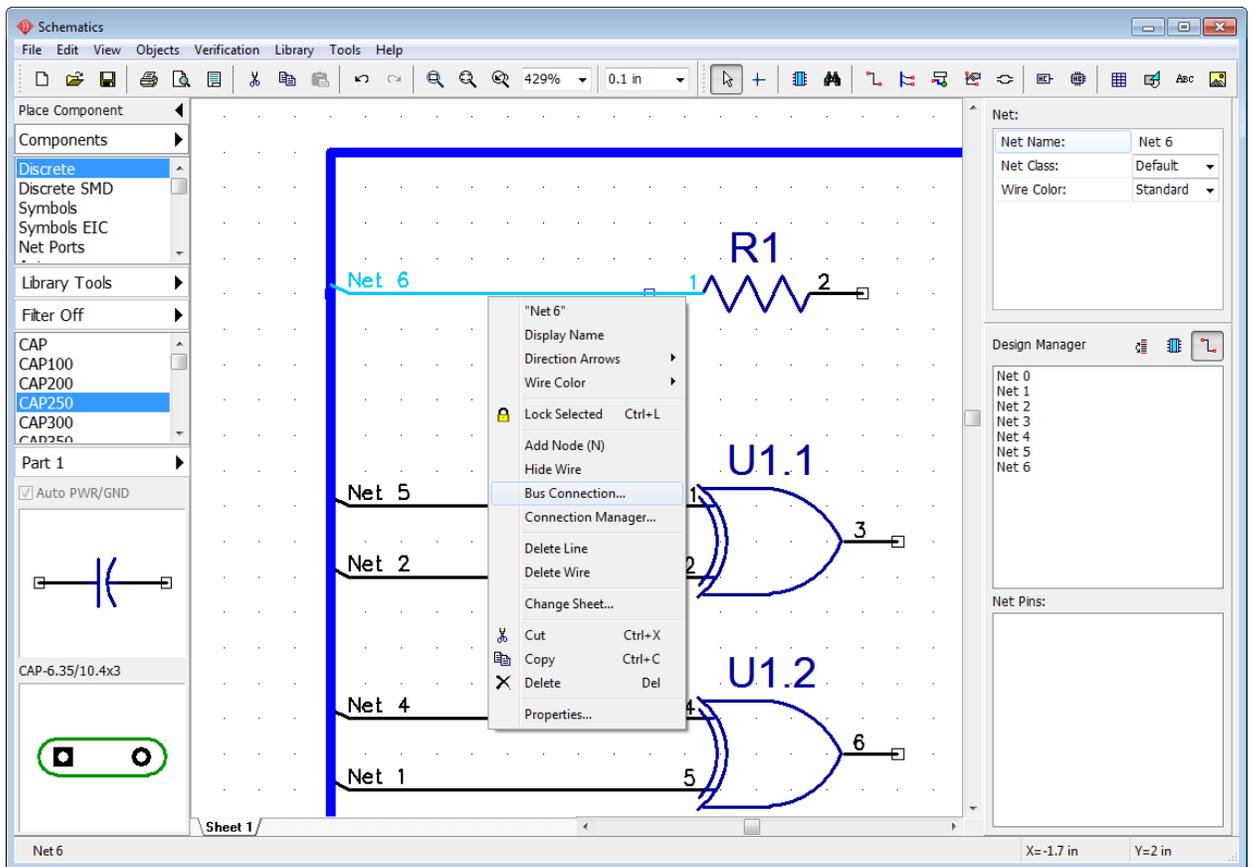
#### Create a bus

Select "Objects \ Circuit \ Place Bus" from the main menu or press  button on the objects toolbar, then draw a bus line on the design area by defining its key points. Right click and select Enter to finish bus placement. Right click on a free space to switch to the default mode. Hover over some pin, left click it, and move mouse arrow to the bus and left click again – connection wire will appear.

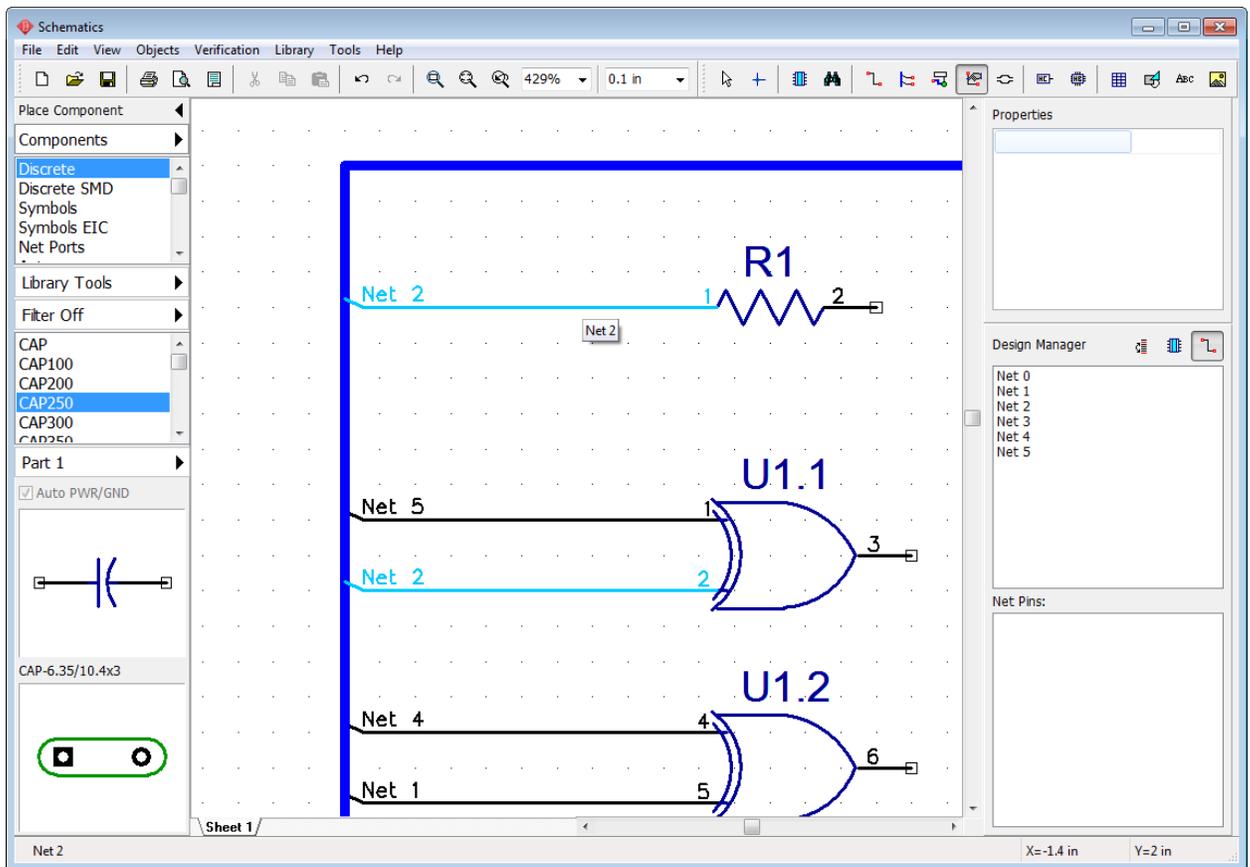
In the pop-up dialog box you can define the name of a new net or connect wire to one of existing nets (which are already connected to that bus).



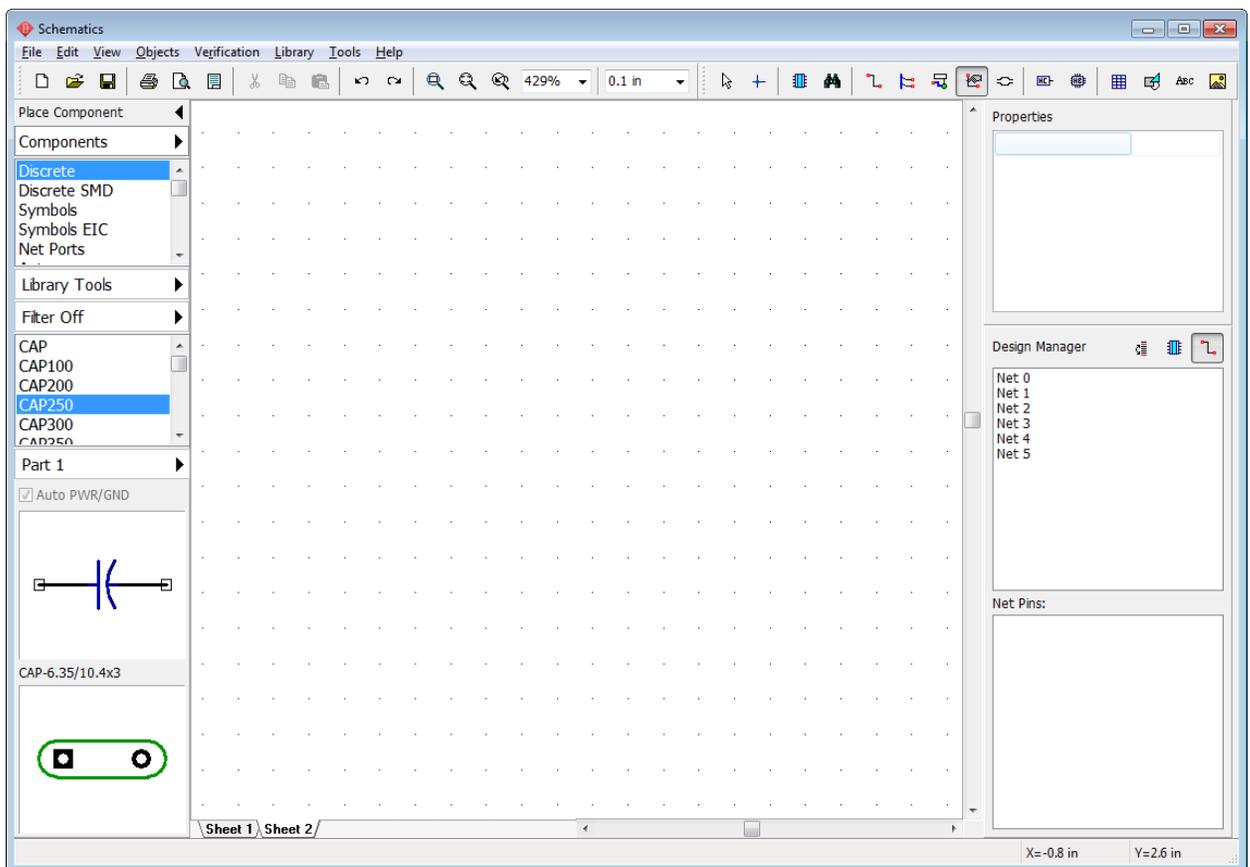
We didn't connect any wires to existing nets, this is why we have seven separate nets (Net 0 – Net 6) not connected to each other via bus. Fortunately, you can change wire-to-bus connections at any given moment – move mouse to the wire segment connected to the bus, right click, and select Bus Connection from the submenu.



In the pop-up dialog box connect Net 6 to the Net 2 (select Net 2 from the list of the bus wires). Now there is no Net 6 anymore, we only have a single net connected via the bus – Net 2.



Add new sheet to the schematic (select "Edit \ Add Sheet" from the main menu or press "Ctrl+Ins"). You can see the list of all sheets as tabs at the bottom-left corner of the design area. Select Sheet 2. [Multi-sheet and hierarchical structure](#)<sup>223</sup> will be described later in the corresponding section of this tutorial.

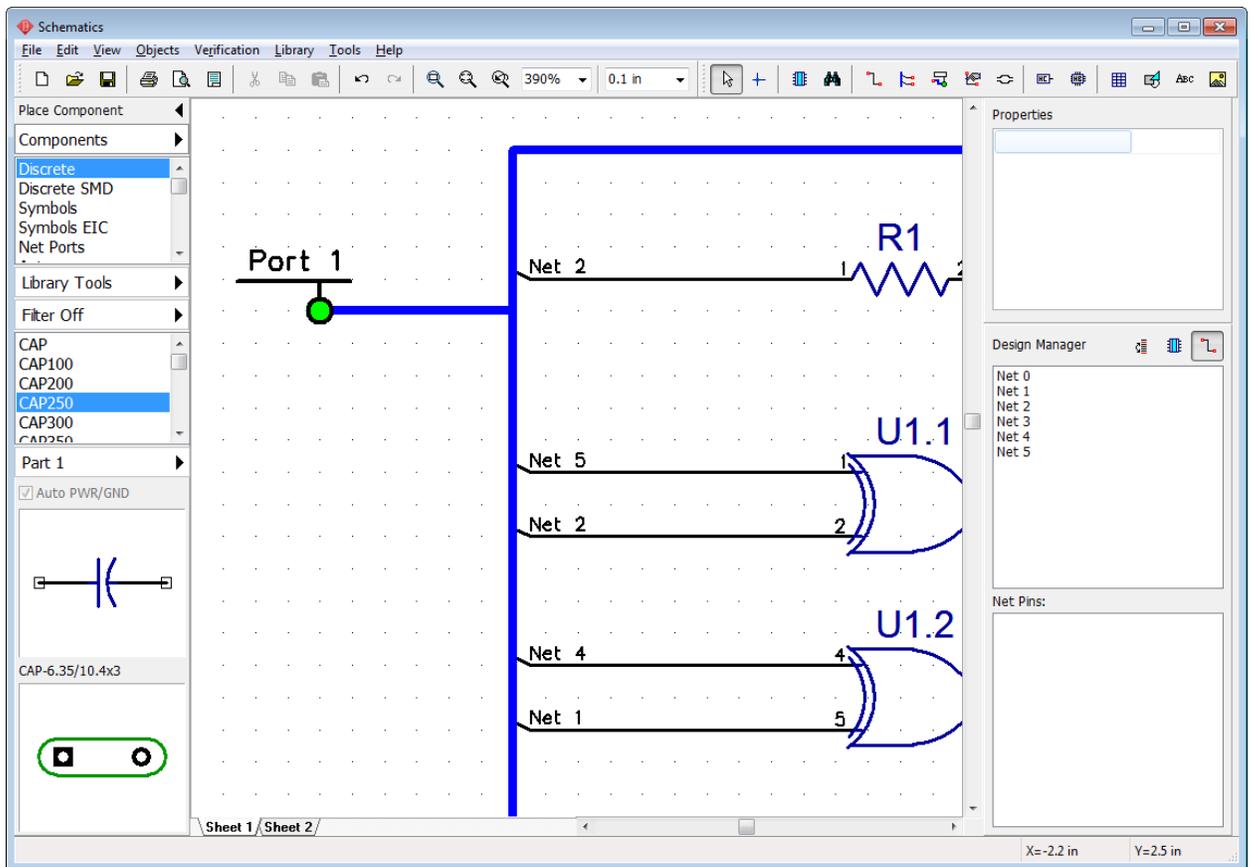


You can rename, move, delete, or insert new schematic sheets, right click on the tab in bottom-left of the design area and select appropriate item from the submenu.

### Bus connector

Press  button on the objects toolbar or go to "Objects \ Circuit \ Place Bus Connector" from the main menu and place it to the Sheet 2 (it should have "Port 0" name) then select Sheet 1 and place one more bus connector there (it becomes "Port 1" automatically). Then connect existing bus to the Port 1 connector: select the bus tool, left click on the bus and draw the line to the bus connector (blue circle in the center of the connector), and left click it to finalize.

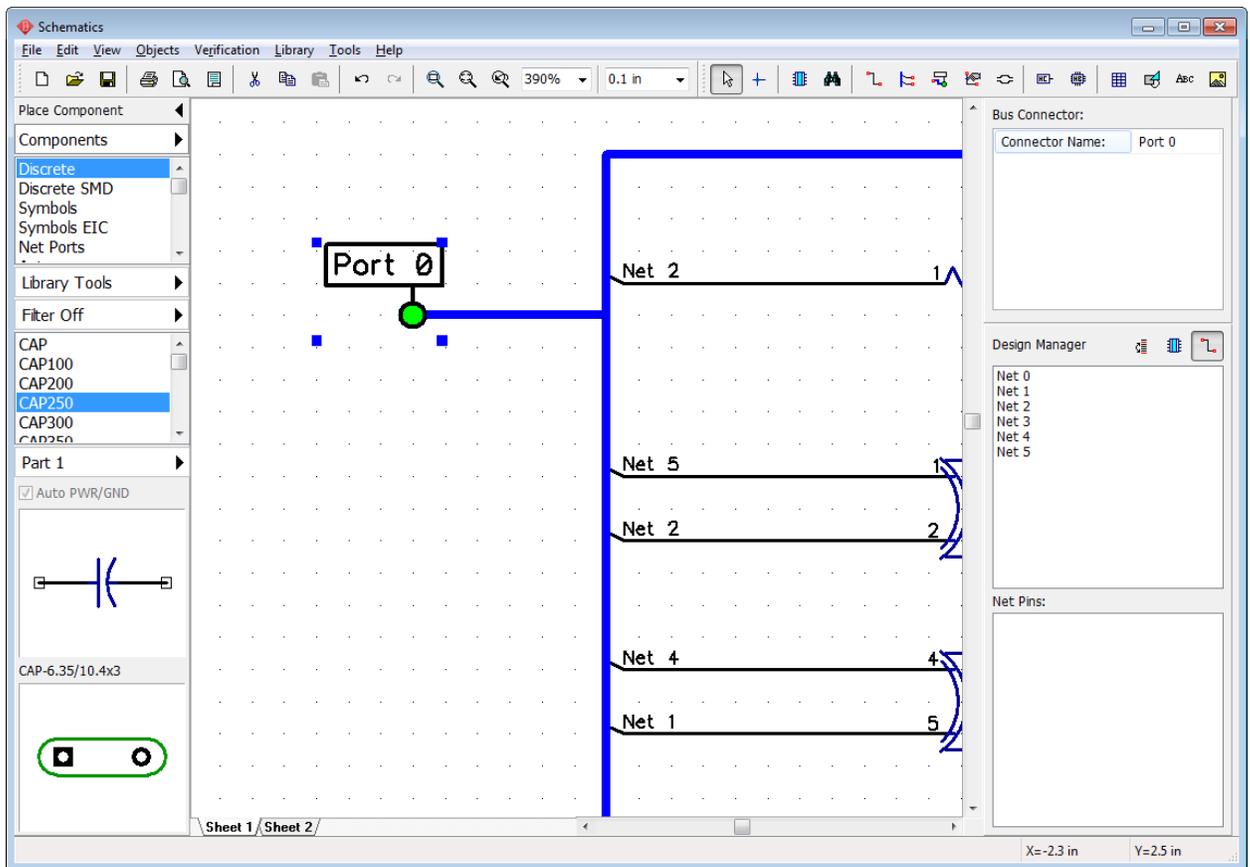
*Notice that bus connector glows **green** if the bus is properly connected to it, blue circle means that there is no connection.*



*Notice that two bus connectors on different sheets are still unconnected (they should have the same name in order to be connected).*

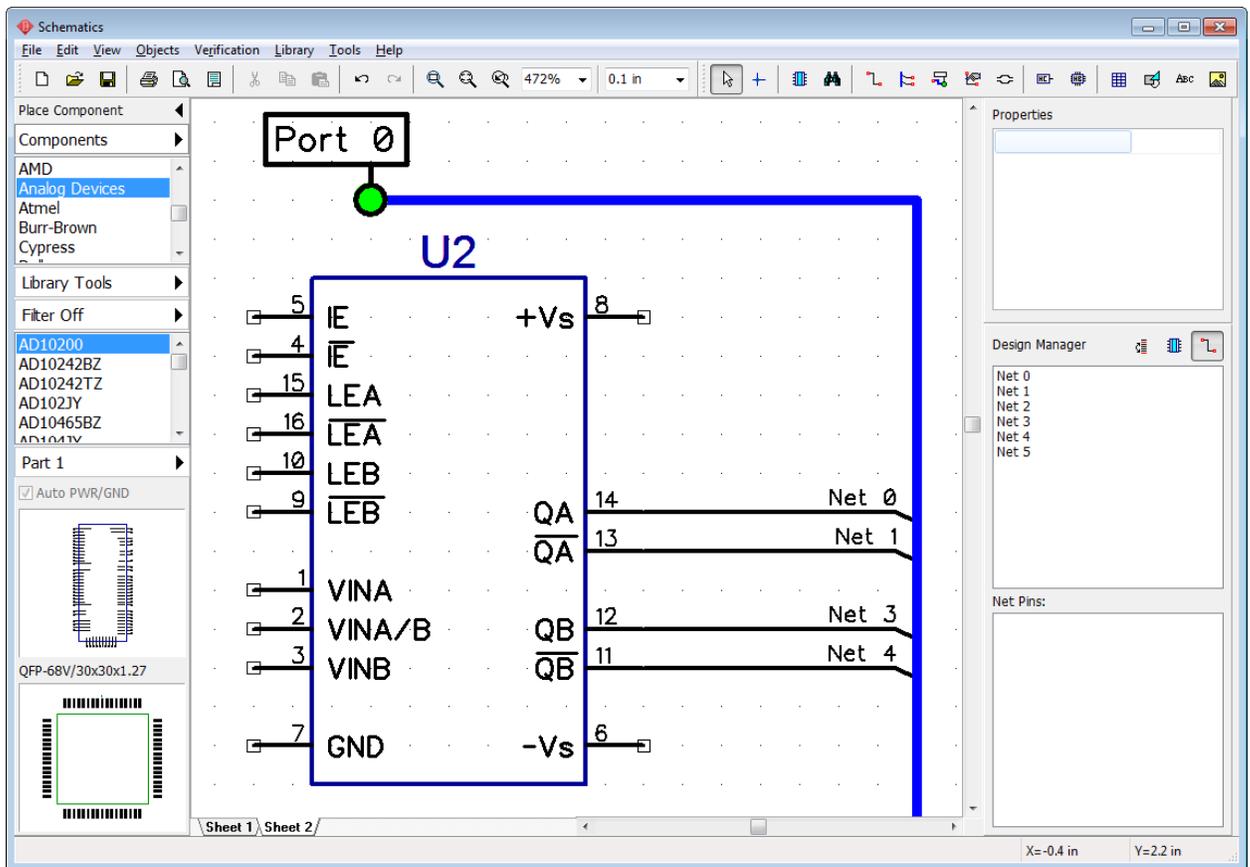
Rename Port 1 – hover over it with your mouse cursor, right click it, select the first item from the submenu and rename the bus connector to "Port 0" (as you remember we've placed Port 0 on the Sheet 2). Press OK. You can see that the box has appeared around the port's name. This means that current bus connector is connected to another bus connector. In our case connector from Sheet 1 is connected to the connector on the Sheet 2.

*Notice that designer can connect more than two bus connectors by defining the same names to all of them.*



Select Sheet 2 and create a bus connected to the Port 0.

Notice that the name of the bus on this sheet is the same as on the first sheet, i.e. this is a common bus. Now you can place random electronic components on the second sheet (for example AD1317 from Analog Devices library) and connect their pins to the nets on the first sheet via the bus.

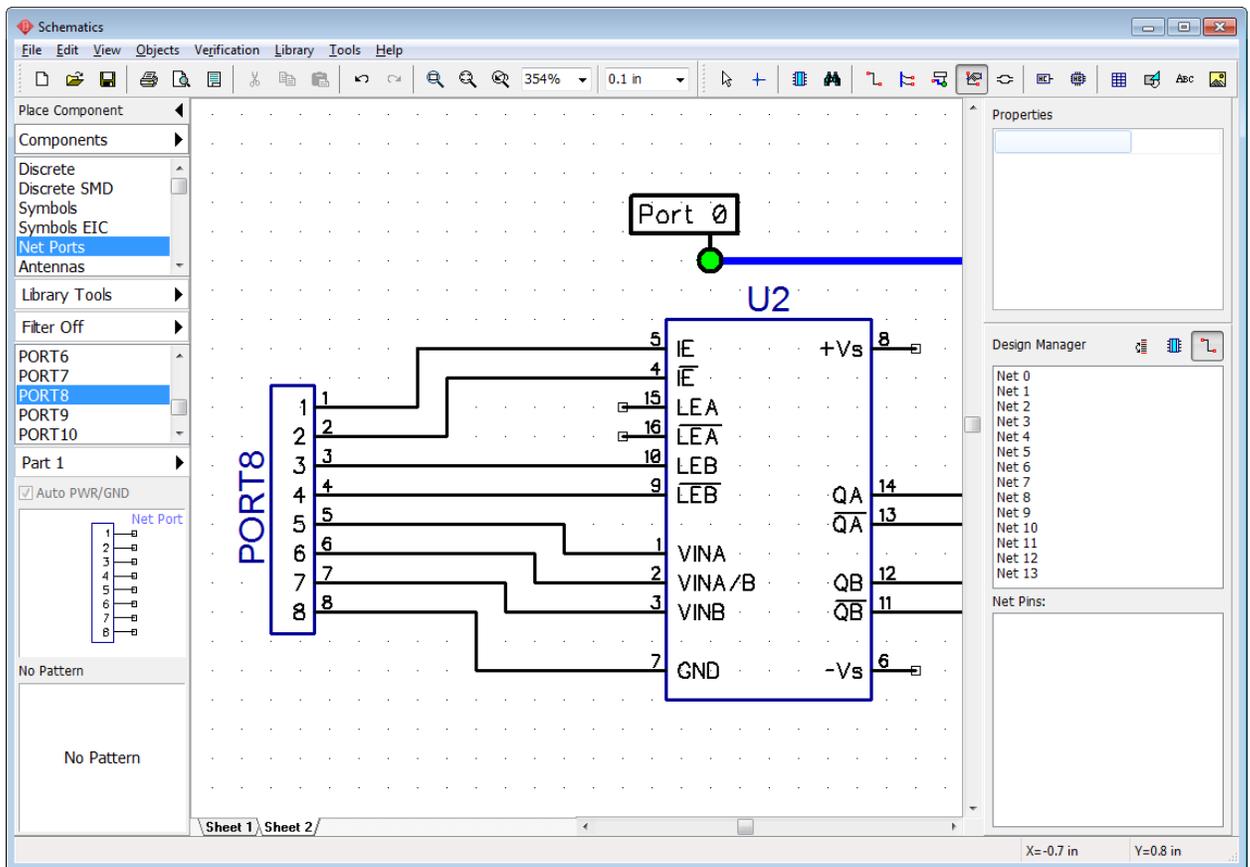


### 5.1.2 Working with Net Ports

We tried to use single-pin net ports [before](#) <sup>[153]</sup> in order to create VCC and GND connections. This is the most common application of such components in real-life. But sometimes multi-pin net ports is a more comfortable solution.

Make sure there are unconnected pins on the Sheet 2. Select Net Ports library from the Components library group on the component placement panel, find Port 8 component and place it to the design area.

Make connections from the component pins to pins of the Port 8, then place one more Port 8 component to Sheet 1 and connect it with some nets. Notice that net names connected to the same pins of Port 8 on Sheet 1 and Sheet 2 are the same, i.e. all wires connected to pin 1 of the Port 8 are connected to a single net, the same with other pins. You can connect or disconnect ports (i.e. change schematic structure) by renaming net ports.

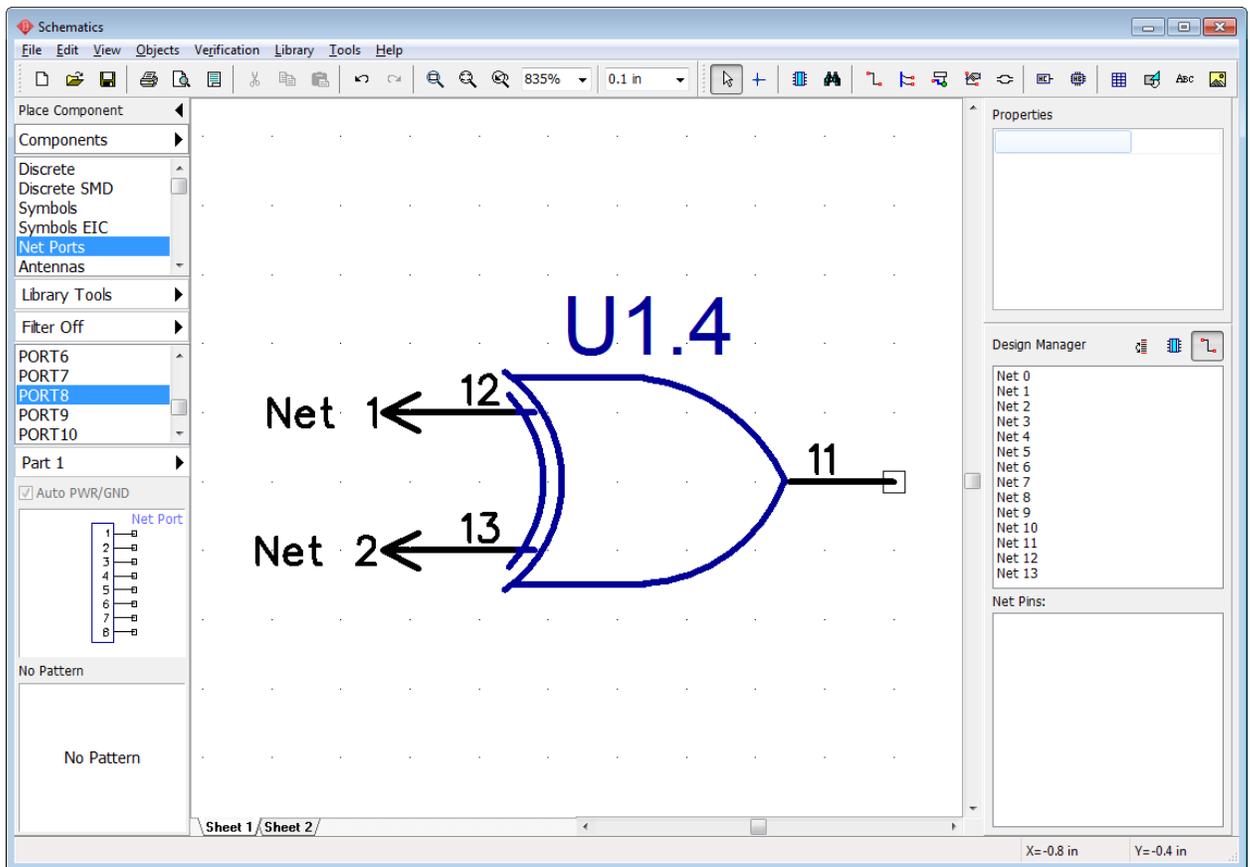


### 5.1.3 Connecting without wires

As you already know PCB Creator allows the user to connect pins visually (wires, buses) and logically (without wires, by name, with net ports).

#### Connecting pins without wires

Pins can be connected logically without wires. In this case they don't depend on the sheet or part location. Hover over selected unconnected pin with your mouse, right click it, and select Add to Net. In the pop-up dialog box select net from the drop-down list and check Connect without Wire box, then press OK. On the picture below you can see two pins connected without wires.

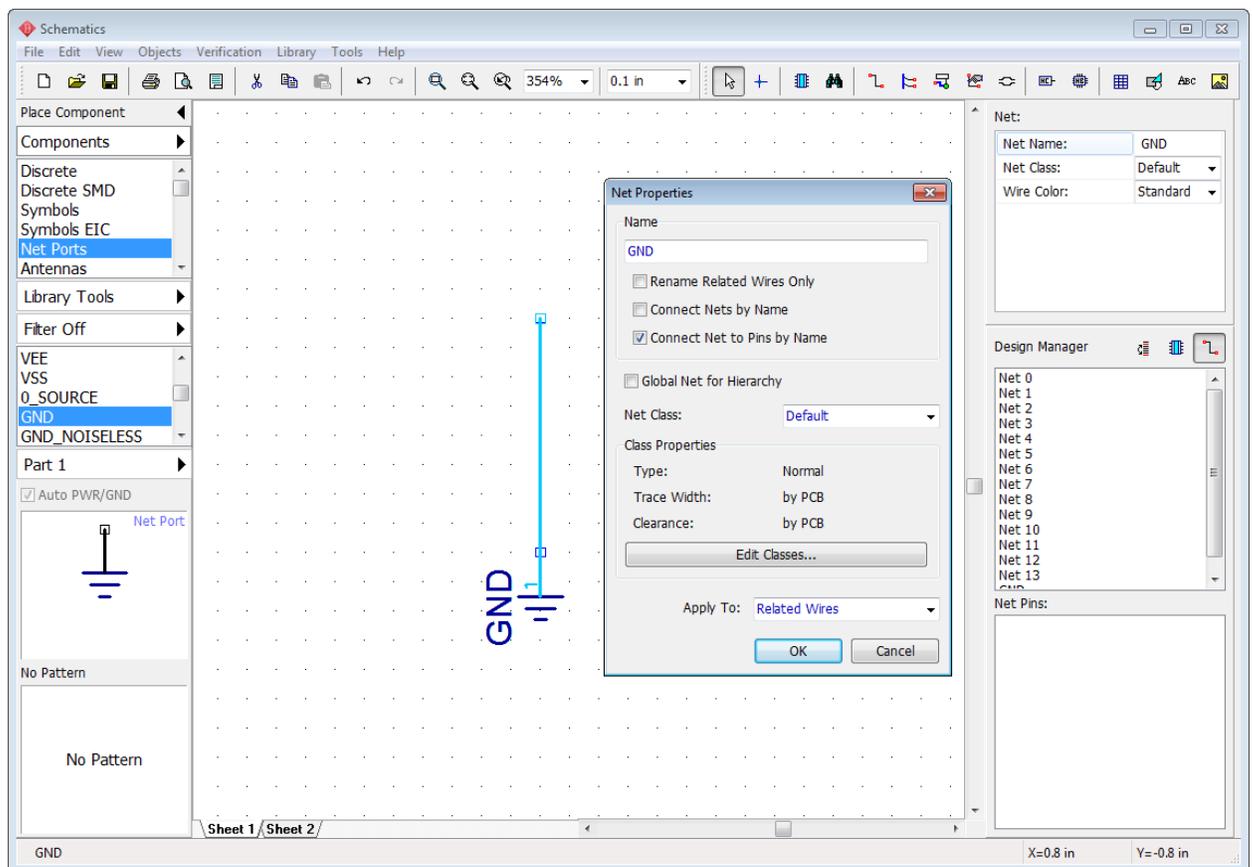


### Connecting pins by name

Now find a blank spot on the design area where we will try to connect pins to the net by name. Place a single GND symbol from the Net Ports library and create a small wire from the GND's pin. Move mouse a bit upwards, and press Enter, like on the picture below. Right click on the wire segment connected to the GND net port, and select Properties from the submenu.

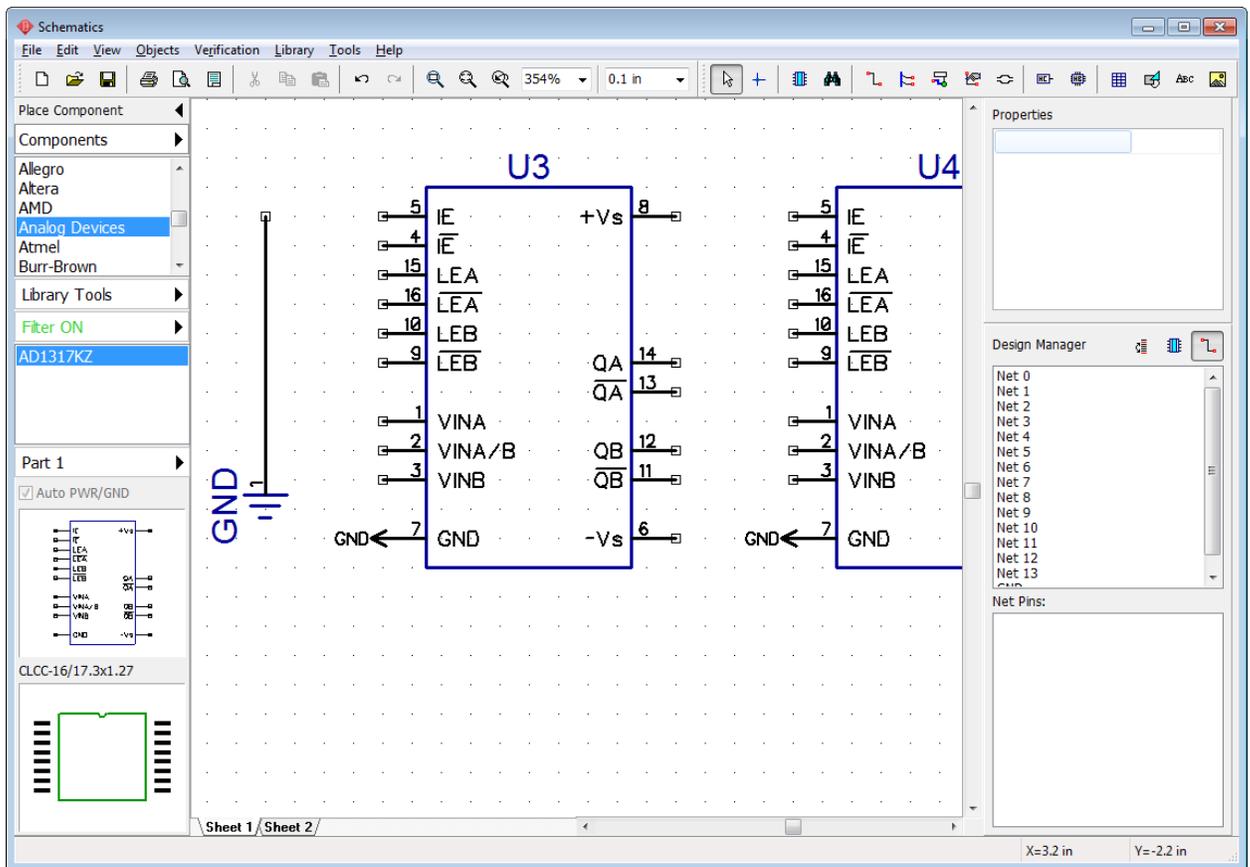
In the Net Properties dialog box rename net to "GND" and check Connect Net to Pins by Name box. Press OK to apply changes.

*PCB Creator will automatically connect all unconnected pins with corresponding name to this net.*



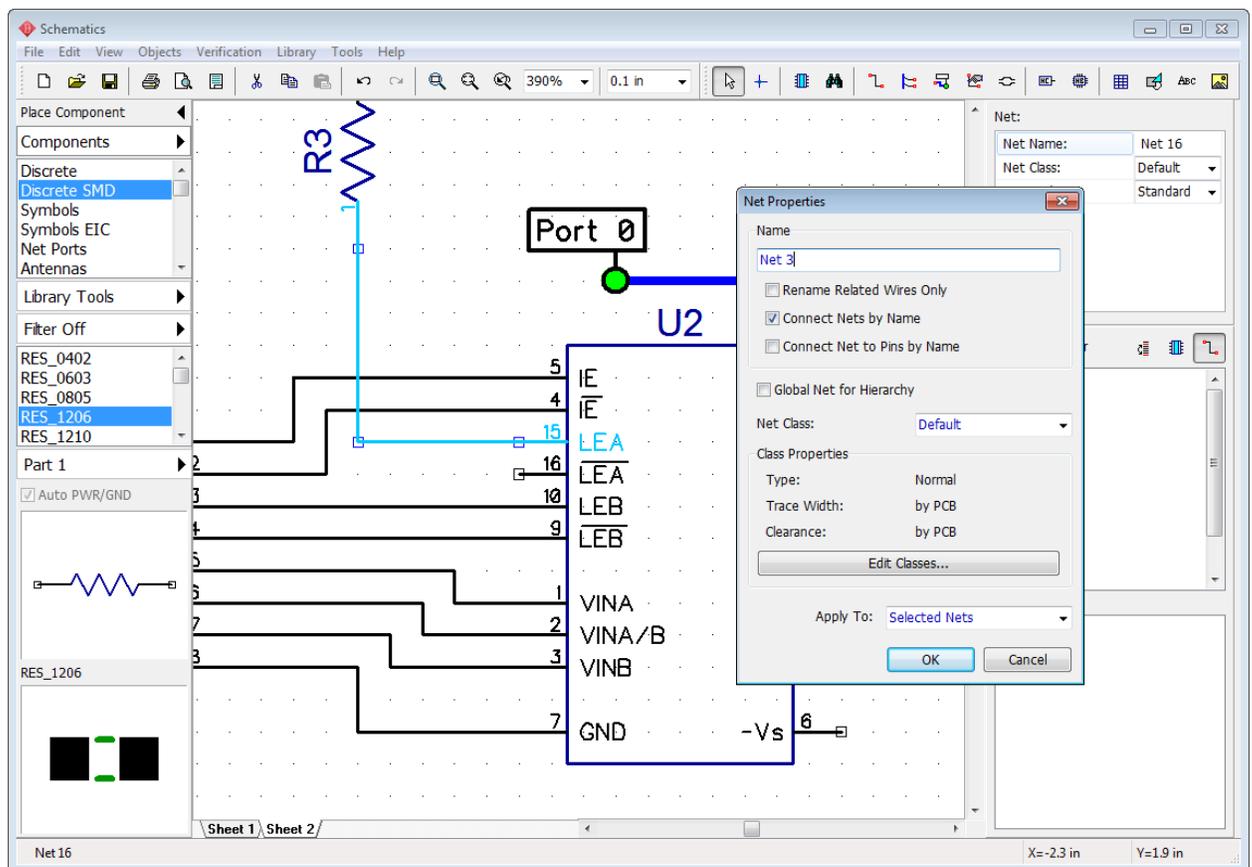
Select Analog Devices library and [find](#)<sup>193</sup> AD1317KZ component (press Filter ON/OFF button, type "AD1317" into the name field and press Apply Filters). Place several AD1317 components on the design area or copy them if you already have one.

Notice that all GND pins of placed component are automatically connected to the GND net without wires due to active Connect Net to Pins option in the Net Properties dialog box. This feature is the easiest way to connect pins with the same names for entire schematic. Usually applied to POWER, CLK pins or even data buses.



### Connect nets by name

PCB Creator allows the user to connect nets on different sheets without net ports or buses. Remember the name of some net that you have on the first sheet (for example, Net 3), then go to Sheet 2 and right click on the net you want to connect to Net 3. Then select Properties, type in "Net 3", and check Connect Nets by Name item. Press OK.



This feature works like a regular net renaming. When you enter the net name that already exists anywhere on the schematic, PCB Creator will ask if you want to connect these nets by name.

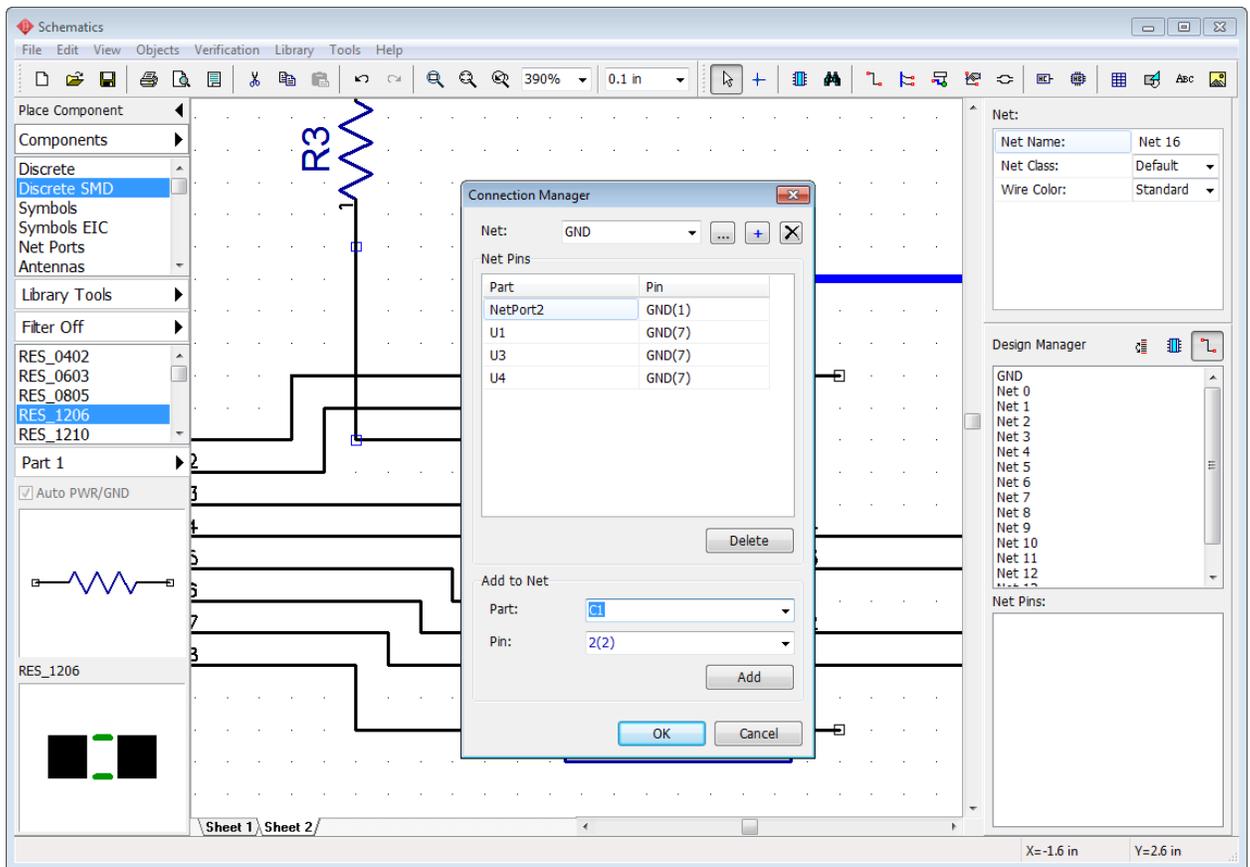
*Notice that only **global nets** can be connected by name on different levels of hierarchy.*

We will learn how to use global nets later in the [Hierarchical Schematic](#)<sup>[223]</sup> paragraph of this tutorial.

#### 5.1.4 Connection Manager in Schematic and PCB Layout

Connection Manager is another tool which allows the user to create / edit / delete connections in Schematic and PCB Layout. Select "Objects \ Connection Manager" from the main menu in Schematic or "Route \ Connection Manager" in the PCB Layout.

Open connection manager in Schematic. Select some net from the drop-down list and you will see all pins of selected net. You can easily add / delete pins to / from the net. To add new pin to the net, select the component and its pin with corresponding drop-downs and press Add button. Notice that only unconnected pins are available in these drop-down menus, so if you can't find some pin that you need, it is probably already connected to another net.



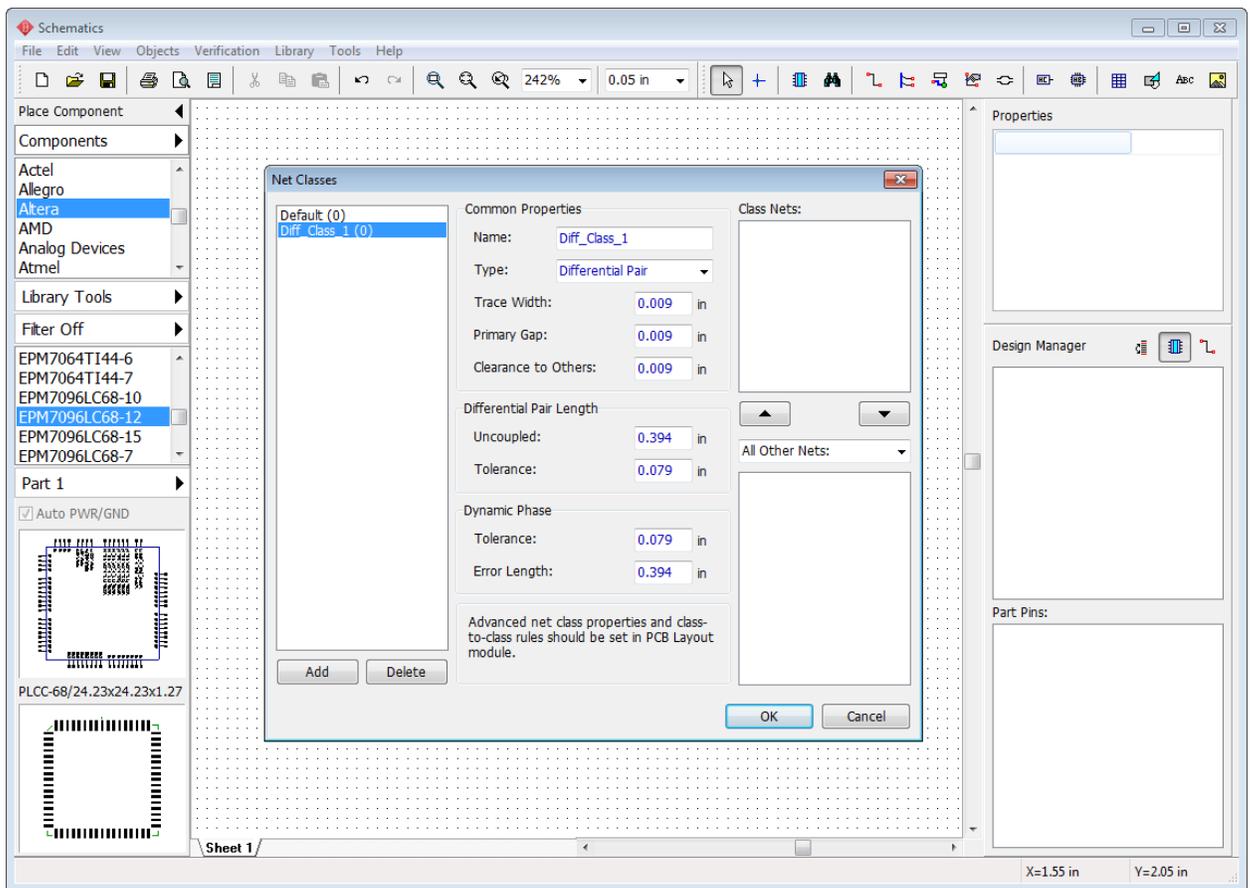
Press OK to apply changes and close the Connection Manager or press Cancel to close it and recover the old net structure.

## 5.2 High-Speed differential connections

### 5.2.1 Create a differential pair

Differential pair signaling is the method of transmitting high-frequency and usually low-voltage signals using two tightly coupled tracks on the circuit board. One track carries the signal and the other one carries an equal but opposite image of the same signal. Differential pairs are EMI immune, they generate less noise than single-track connections, and, basically, are the only acceptable way of transmitting high-frequency pulses.

Open PCB Creator Schematic and go to "Objects \ Net Classes" from the main menu. We need to create a new special-type net class for differential pairs. Press Add button in the pop-up dialog box, then type in the name of the net class (for example, "Diff\_Class\_1"), and change Type to Differential Pair. Now you can enter specific differential pair routing parameters.

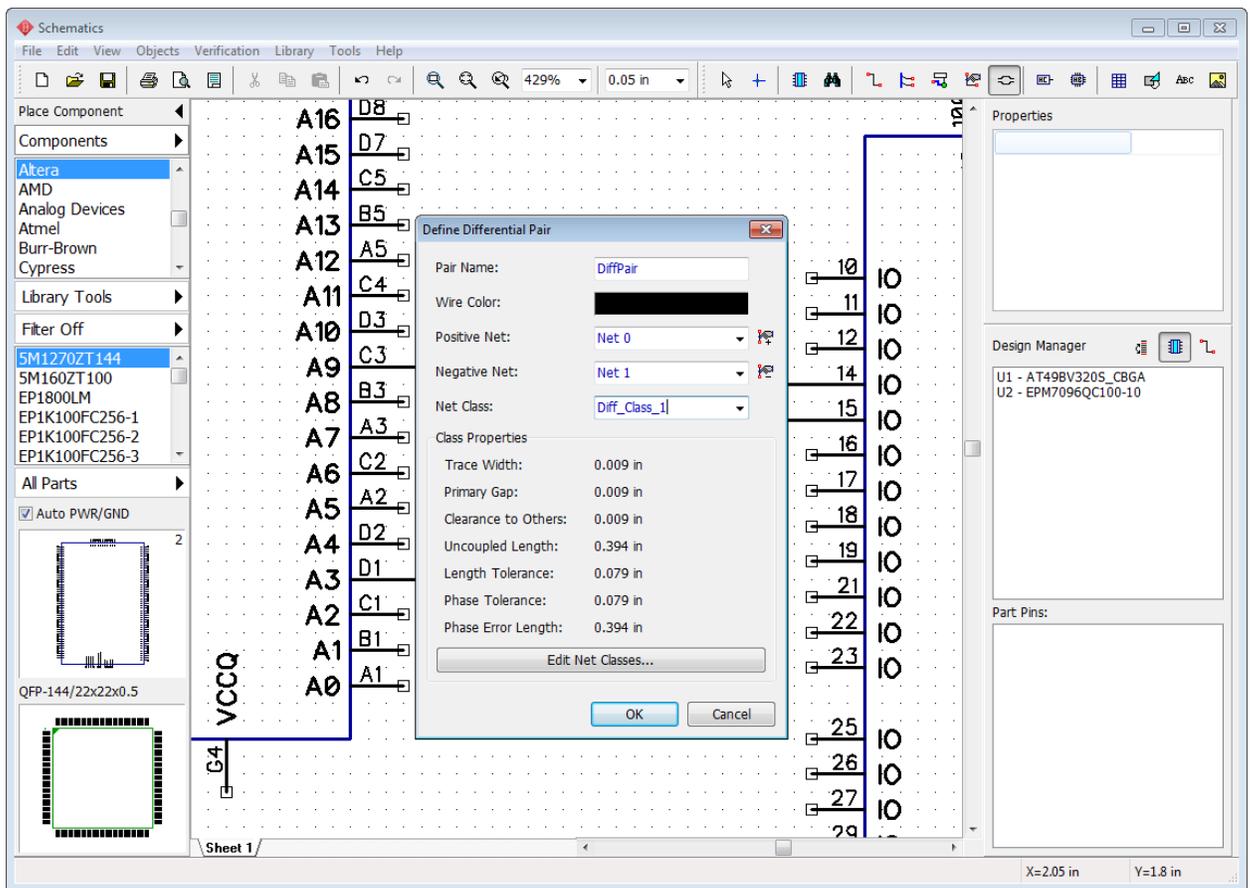


In PCB Creator all differential pair parameters are governed by net classes and the program does not allow the user to create differential pairs outside of a net class. Please enter trace width for tracks of the pair, primary gap between the tracks, and clearance to other objects on the PCB. You can leave default values for now and change them later, when on the PCB. Notice that the net class is empty (because the design is empty itself).

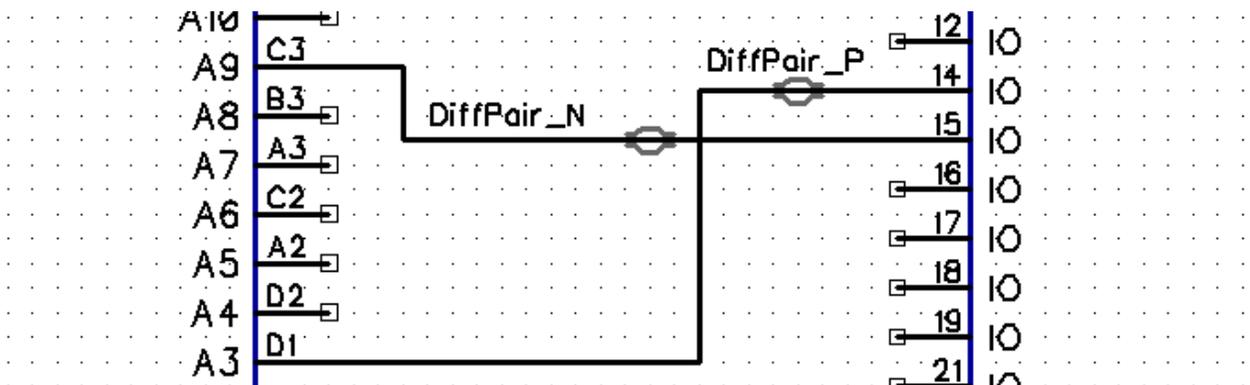
Press OK to apply changes.

Now we need to populate our schematic with some components of your choice for practicing with differential pairs. We have Atmel AT49BV320S flash-memory module and a bit obsolete Altera EPM7096QC100-10 CPLD chip on the design area, but differential pair declaration can be assigned to any nets between any components. Create a couple nets which will eventually become a differential pair. For example, we have Net 0 connecting A3 to pin#14 and Net 1 connecting A9 to pin#15.





Press OK. Net 0 and Net 1 are a differential pair now, these nets are renamed to DiffPair\_P and DiffPair\_N respectively and marked with a special symbols on the design area. You can hover with the mouse over one net and the other net of the pair will be highlighted. You can display the net names on the design area if you right click on each net and select Display Name from the drop-down submenu.



Now let's switch this design to the PCB Layout stage and practice more in customizing, routing, and verifying differential pairs.

## 5.2.2 Differential pair routing / editing

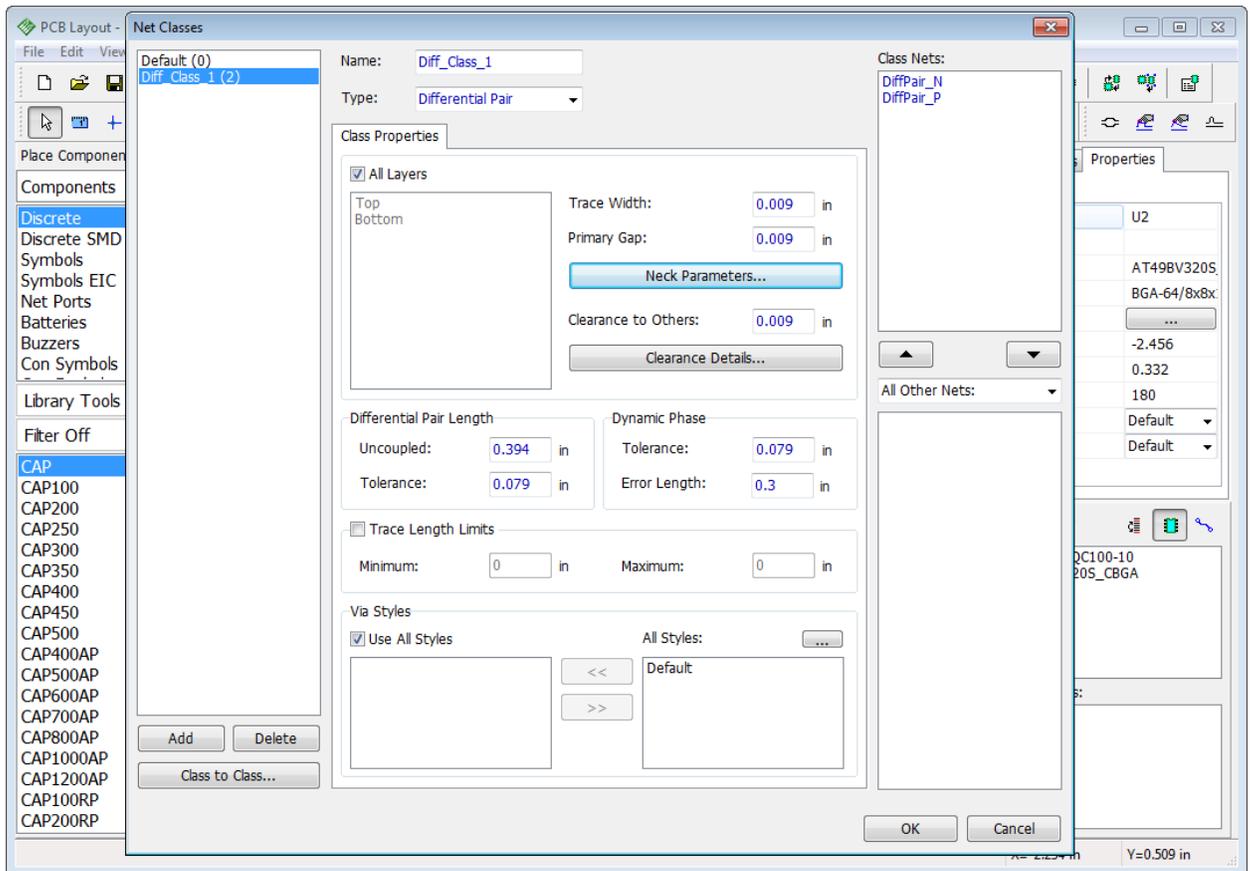
*Notice that built-in PCB Creator autorouter does not support differential pairs. We recommend to route these nets manually or you can use external autorouters via DSN / SES interface.*

Select "File \ Convert to PCB" from the main menu in the PCB Creator Schematic and proceed with the schematic rules. In the PCB Layout move components closer together. You can see that differential pair is also marked with a special symbol, like on the schematic. You can see that ratlines are twisted, it's possible to route the differential pair like that, but it would be much easier to rotate U2 pattern to eliminate the twist. Select the component and press "R" hotkey two times to rotate the component 180 degrees. Now we're almost ready to start the routing. But first we need to go to Net Classes dialog box to check if we have acceptable routing parameters. Select "Route \ Net Classes" from the main menu, then select "Diff\_Class\_1" net class.

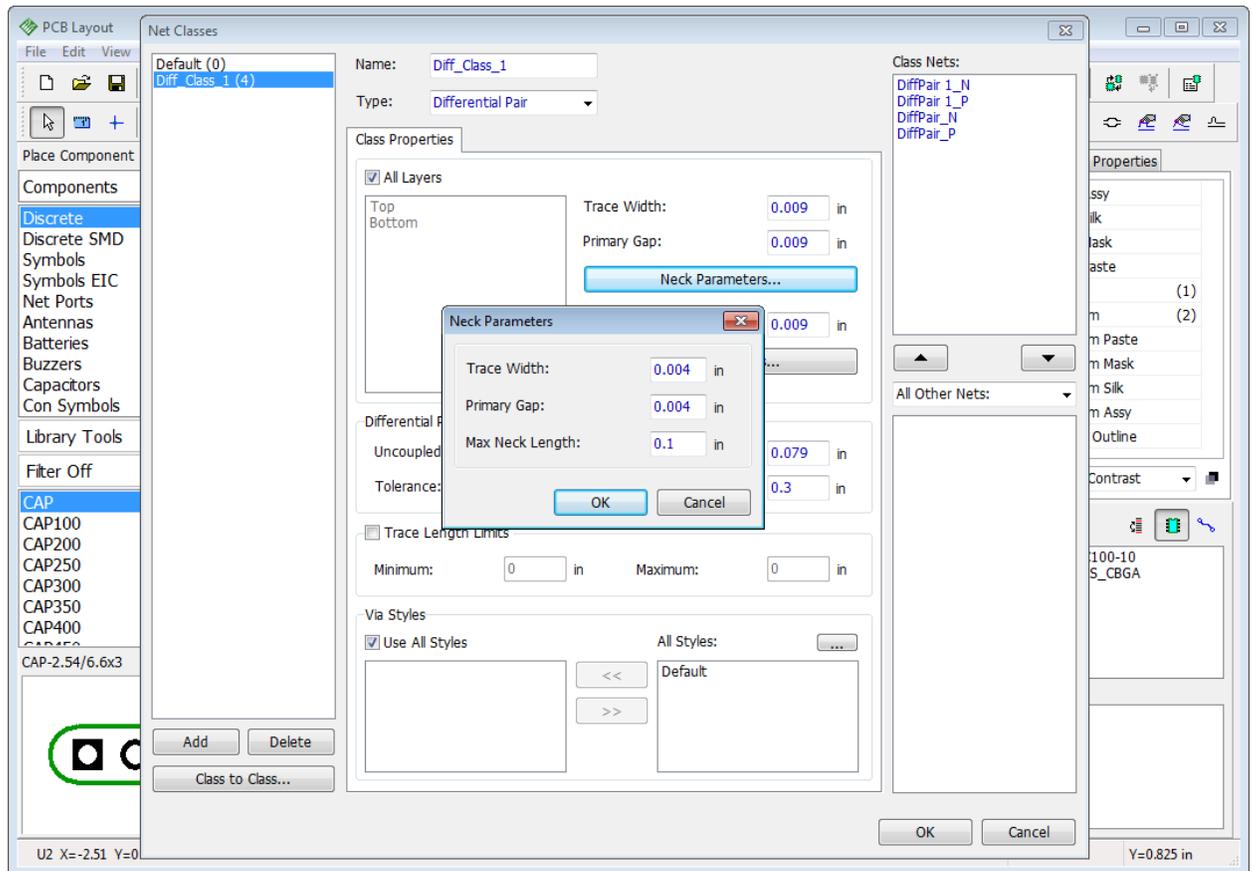
Differential Pair Routing Parameters:

- Maximum uncoupled length for two traces of the differential pair.
- Length tolerance between the traces.
- Dynamic phase tolerance (length difference between the corresponding segments of each track which shifts the phase of the signal).
- Dynamic phase error length, which means that DRC will report any phase shift as an error only if it occurs on the track segment longer than the Error Length value.

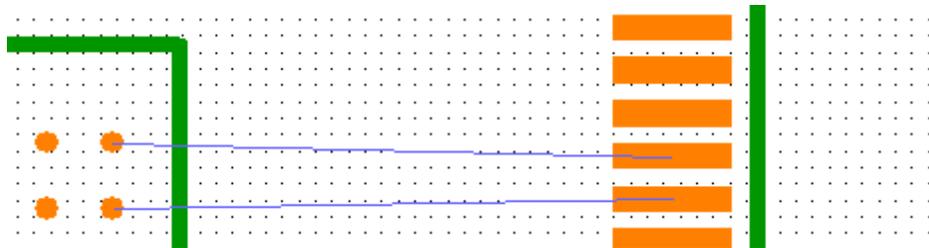
Let's change Dynamic Phase Error Length to 0.3 inch. Leave default values in other fields. DRC takes these values as routing constraints when verifying the PCB.



Since we have some small-pitch BGA footprints on the board we can estimate that our differential pair with 0.009 inch wide tracks, gap, and clearance is too big to route. PCB Creator allows the user to define neck parameters, which can be easily applied to differential pairs while routing in tight spaces. Press the Neck Parameters button. In the small pop-up dialog enter 0.004 inch width and gap, and 0.1 as maximum neck length. This means that necked segment longer than 0.1 inch will be reported as an error by DRC. Press OK to apply neck parameters and then press OK to close the Net Classes dialog box.



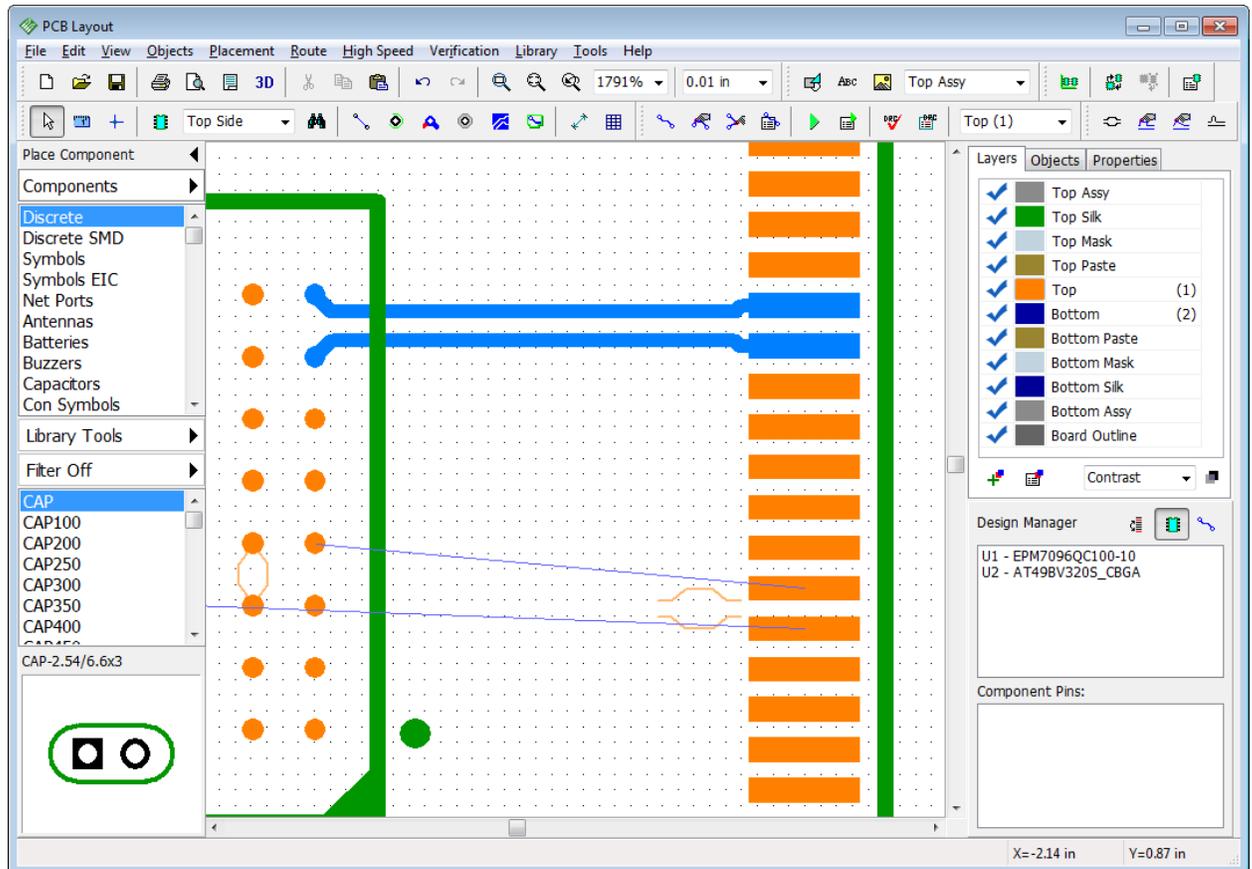
Let's create one more differential pair directly in the PCB Layout. First we need to create a couple more nets with Place Ratline tool . For example, like on the picture below.



Then press Define Differential pair button  on the high-speed toolbar. Define Differential Pair dialog box is the same as in the Schematic module, so that will be easy to pin point on the positive and negative net, change color for convenience (blue in our case) and apply the same Diff\_Class\_1 net class to this differential pair. Press OK and the second differential pair with custom color will appear on the design area. We have created a custom grid with 0.001 in size.

Now press the Route Manual button on the routing toolbar, or go to "Route \ Manual Routing \ Add Trace" from the main menu, and left click on any pad of the differential pair, two tracks will appear simultaneously. Routing differential pair is very similar to [routing a single net](#)<sup>[53]</sup>. Ratlines show the direction where to lead the traces. Continue from the U1 component to the U2, and left click on the corresponding U2 component pad. The second trace of the pair will be automatically

connected to another pad. You can change routing layer, route mode, current trace segment and other routing parameters on the Manual Routing panel (Design Manager) with the corresponding hotkeys (noted in the brackets on the Manual Routing panel), or with right-click submenu on the go.

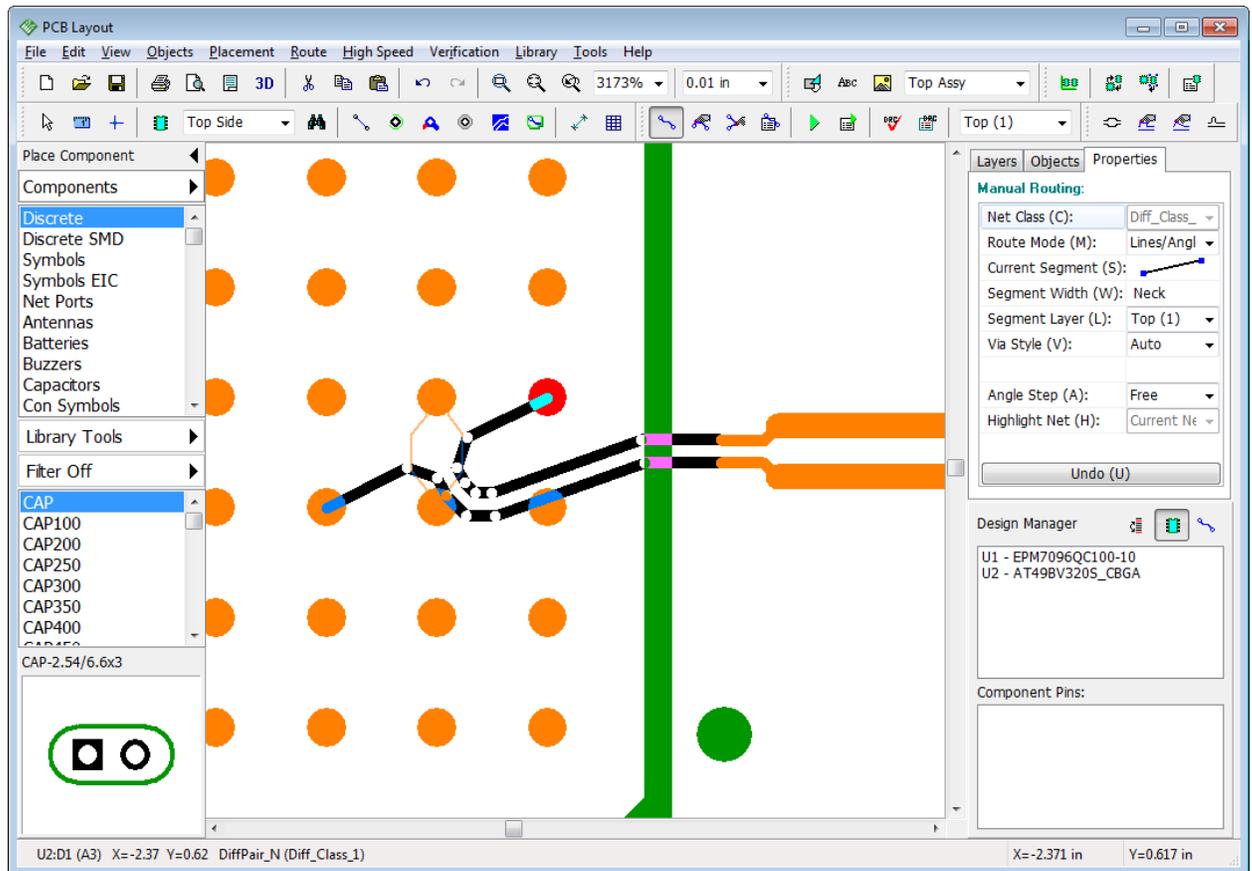


If you want to change the active trace of the pair that you route, select Change Control Trace from the right-click submenu while routing.

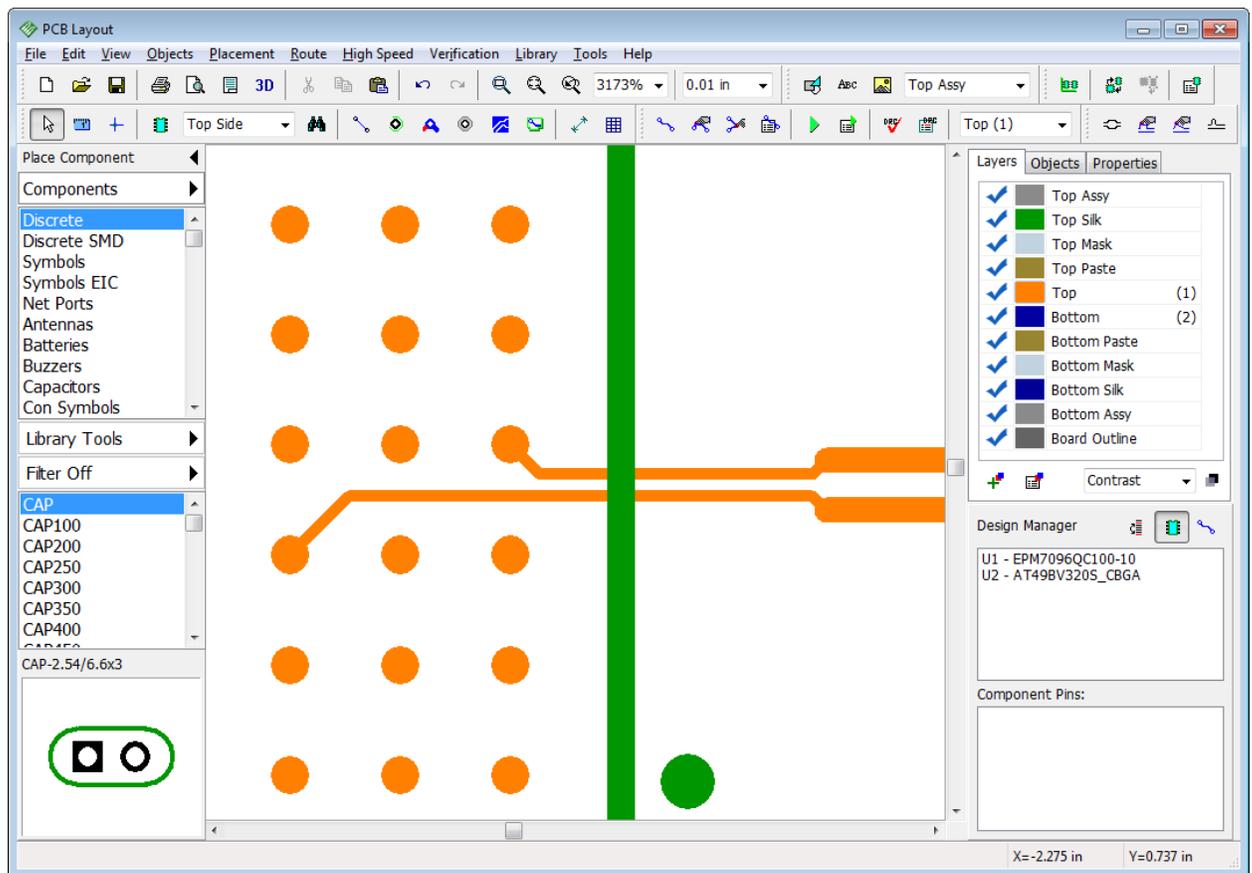
### Single-track routing

Press the Route Manual button on the routing toolbar, or go to "Route \ Manual Routing \ Add Trace" from the main menu, then change Current Segment to Free Line on the Manual Routing panel (press "S" hotkey two times), and start to route the differential pair from U1 to U2. You will notice that BGA pitch is very small and now is the time to apply the trace necking defined earlier in the Net Class properties. When approaching the U2 left click to place trace segments and then right click and select Neck-down Segment from the submenu or press "W" hotkey. The width of two tracks and the gap between them has changed. Left click again closer to the U2 component to create a small narrow segment (orange traces are routed). Now the tracks are small enough to fit the BGA's pitch, but since the pads are in different rows, you are not able to finish routing the pair automatically, hover over the terminal pad to see that the layout offered by the program is unacceptable.

\



Routing each trace of the differential pair will allow you to succeed in this situation. Right click and select Route Single Trace from the submenu or press "Z" hotkey. Then change current segment to regular lines with 45-degree angle on the Manual Routing panel, or just press "S" hotkey, then left click on the terminal pad of the trace. PCB Creator will switch to the second trace of the pair for routing. If you want to change the trace of the pair that you route, select Change Control Trace from the right-click submenu. Left click on the second pad to complete the differential pair.



### Controlling the trace length

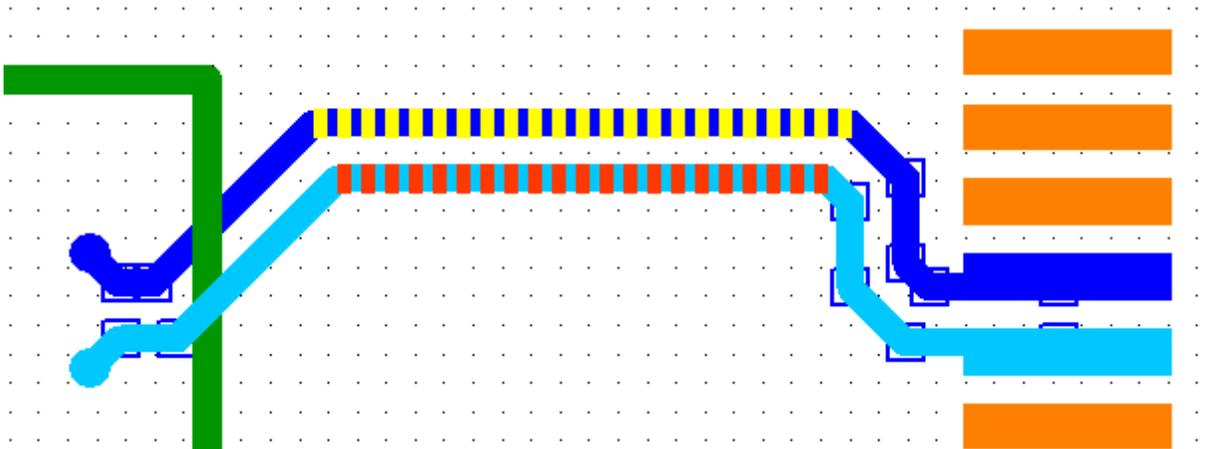
Controlling the total length of each trace is very important for successful high-speed routing. Right click on any trace of the differential pair and select Show Trace Length from the submenu, the total length of each track of the differential pair will appear right next to the pads of the differential pair in current measurement units. In order to hide the trace length, select the same item from the same right-click submenu again. In our case trace length is hidden to keep design more empty and easy to understand.

### Edit differential pairs

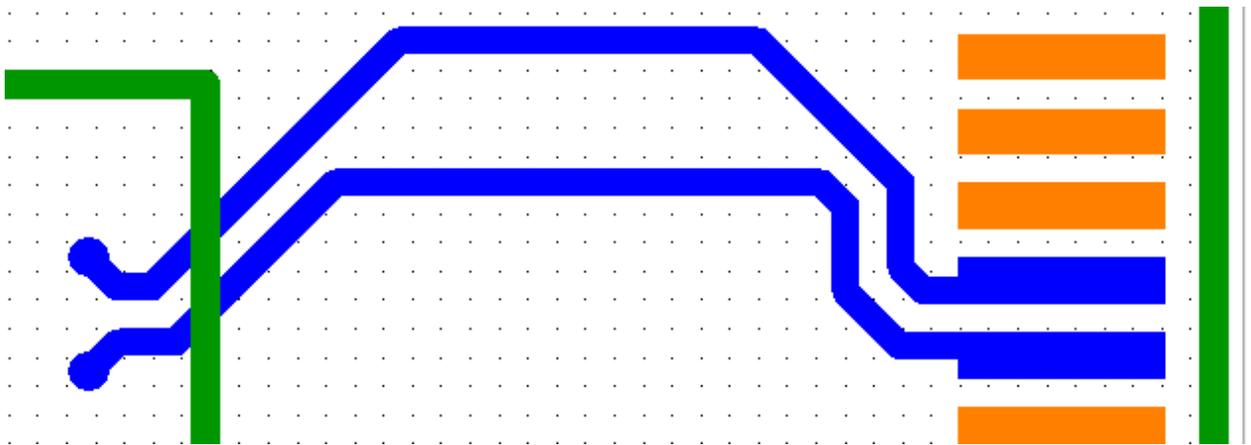
Right click on the differential pair and select required action from the submenu. You can unroute tracks or separate segments, change color of the differential pair, layer of the traces or segments, delete differential pair declaration from the nets, remove necking, and much more. There are two distinct editing modes applied to differential pairs: regular editing and single track

editing. Each one can be in a regular or free style. For example, press  button on the routing toolbar, then drag and drop differential pair's traces to another location. Notice that two tracks

move with respect to the differential pair gap. Regular Free edit tool (  button) can be also applied to differential pairs.



The other mode is similar to single-track routing, because it allows the designer to edit each trace of the differential pair separately from the other. Press  button on the high-speed toolbar and move one track of the pair further away.



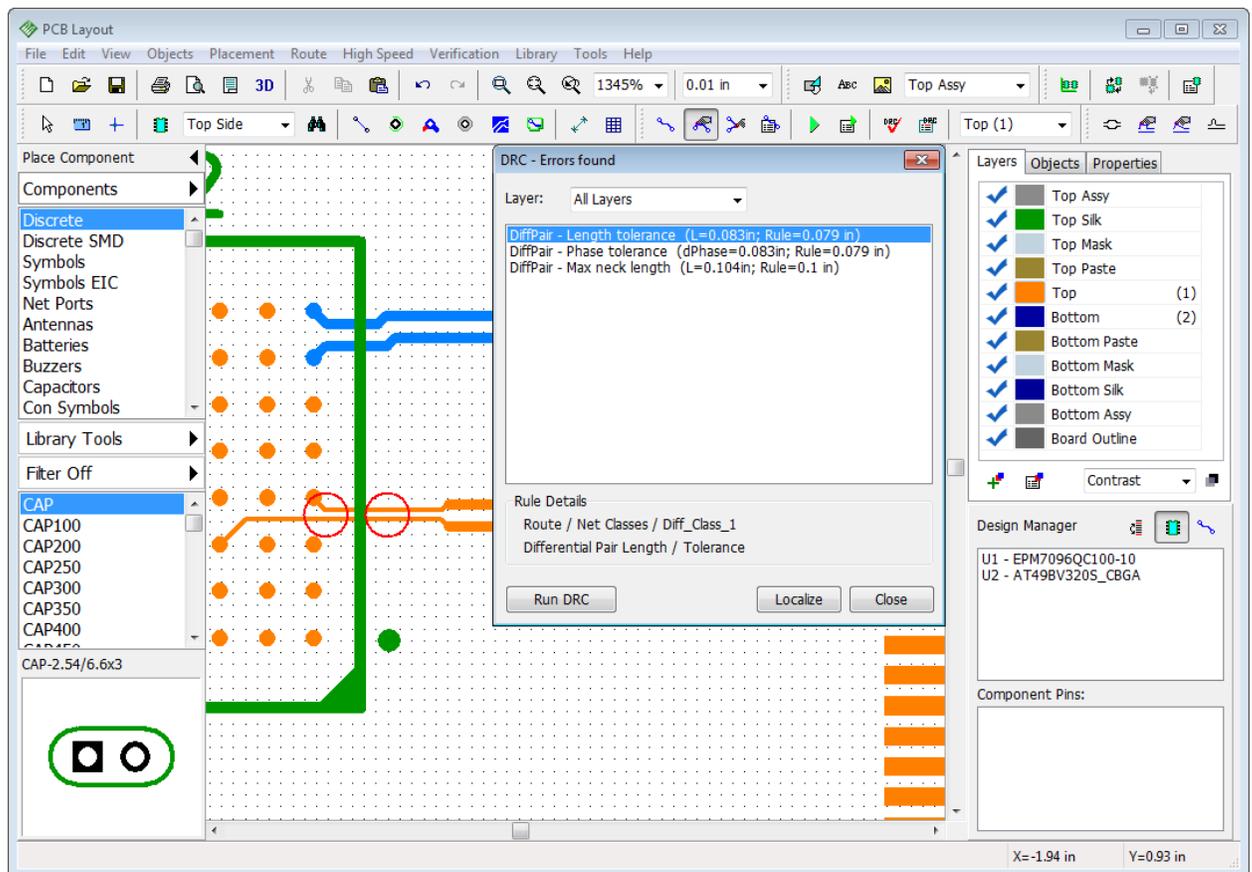
Single-track free edit mode (  button on the high-speed toolbar) behaves in the similar way to the regular Free edit mode, but applies only to one trace of the pair. You can practice more with differential pairs and change the layer of the trace segment. Right click on the longest differential pair's segment and select "Segment Layer \ Bottom". Two traces will move to the bottom layer. Vias will appear automatically.

Please undo the last changes to return blue differential pair to its initial state, then proceed to the topic below for Differential Pair verification.

### 5.2.3 Verification and phase tuning

In our case we have Real-time DRC OFF, this is why we do not see any errors on the design area. [Turn it ON, and run DRC](#)<sup>[78]</sup> in the regular mode to see the error-report dialog box. DRC checks all clearances and sizes against the target values defined in "Verification \ Design Rules"

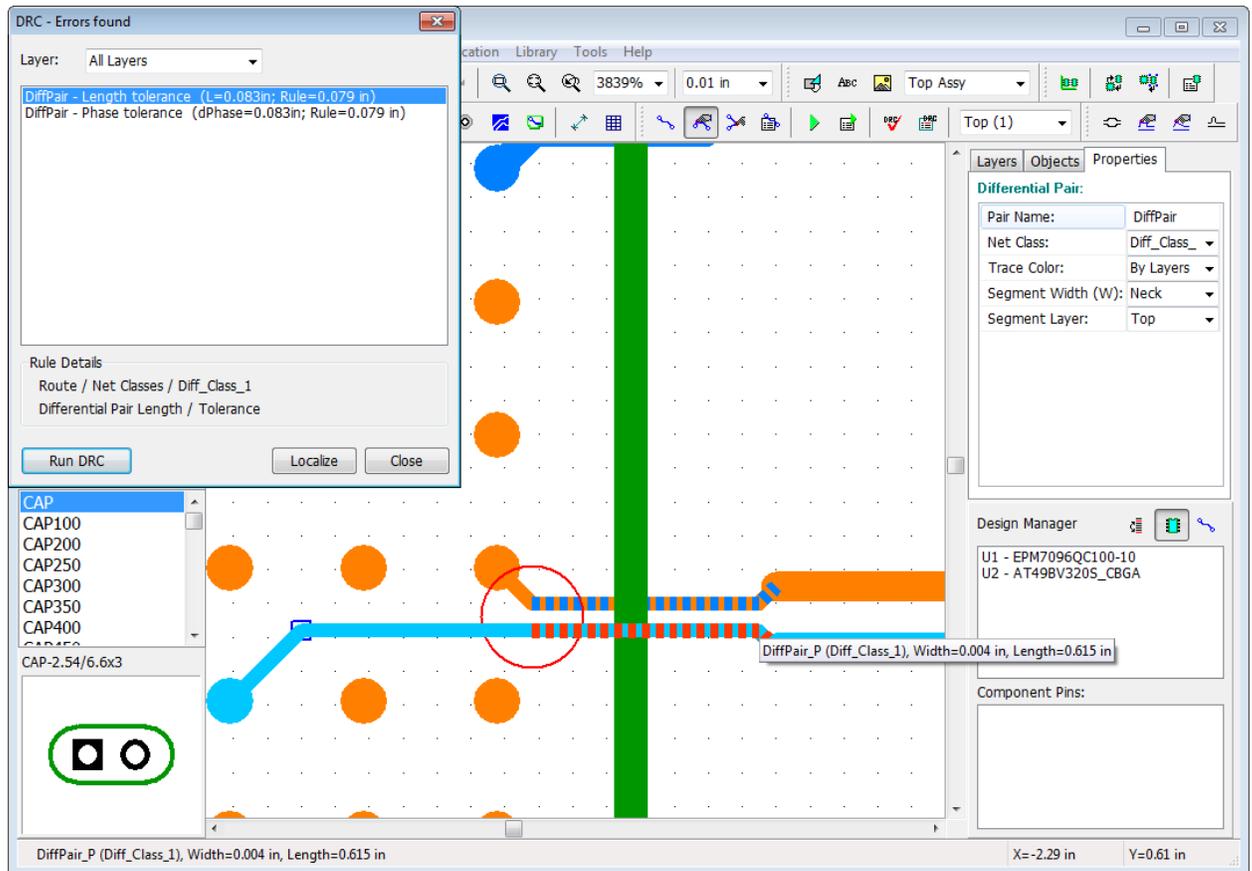
main menu item and takes differential pair properties from the Net Classes dialog box as design constraints. In our case we have three errors related to the differential pair. When you click on the error in the list you see the description of each violation, including current and target values. Rule Details section right below the list shows where you can change the target values.



Move the DRC error-report dialog box, so it will not obstruct the view. First of all let's make the necked differential pair segment a bit shorter, as we can see only 0.1 inch is allowed by the

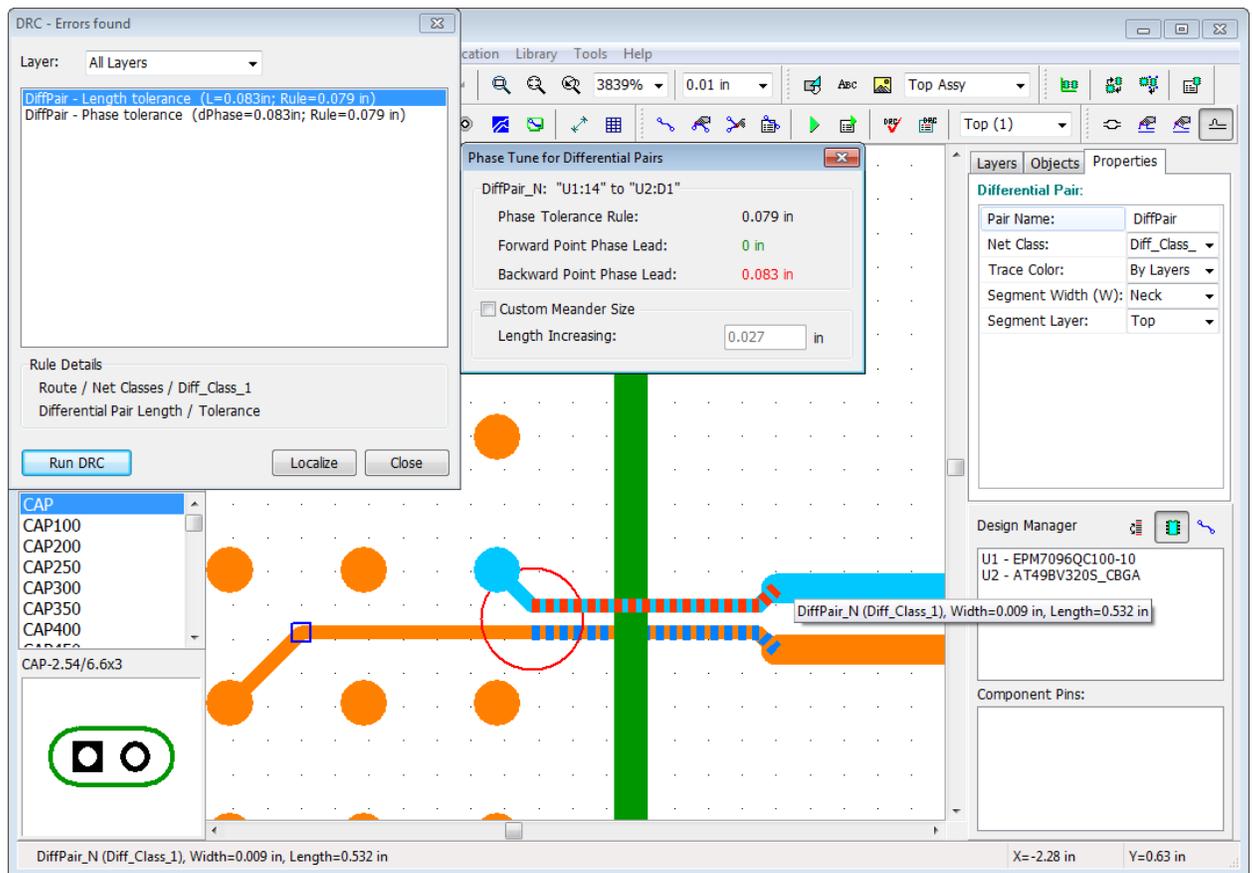
necking constraint in the differential pair net class. Press  button on the routing toolbar then hover over the place where traces become narrow, the mouse cursor should appear as double-sided arrow **parallel** to the traces. Then drag and drop wide traces closer to the terminal pads to make necked traces shorter, like on the picture below. You can apply single-trace editing tools

(for example ) from the high-speed toolbar, if you can't succeed without an error while moving two traces simultaneously.

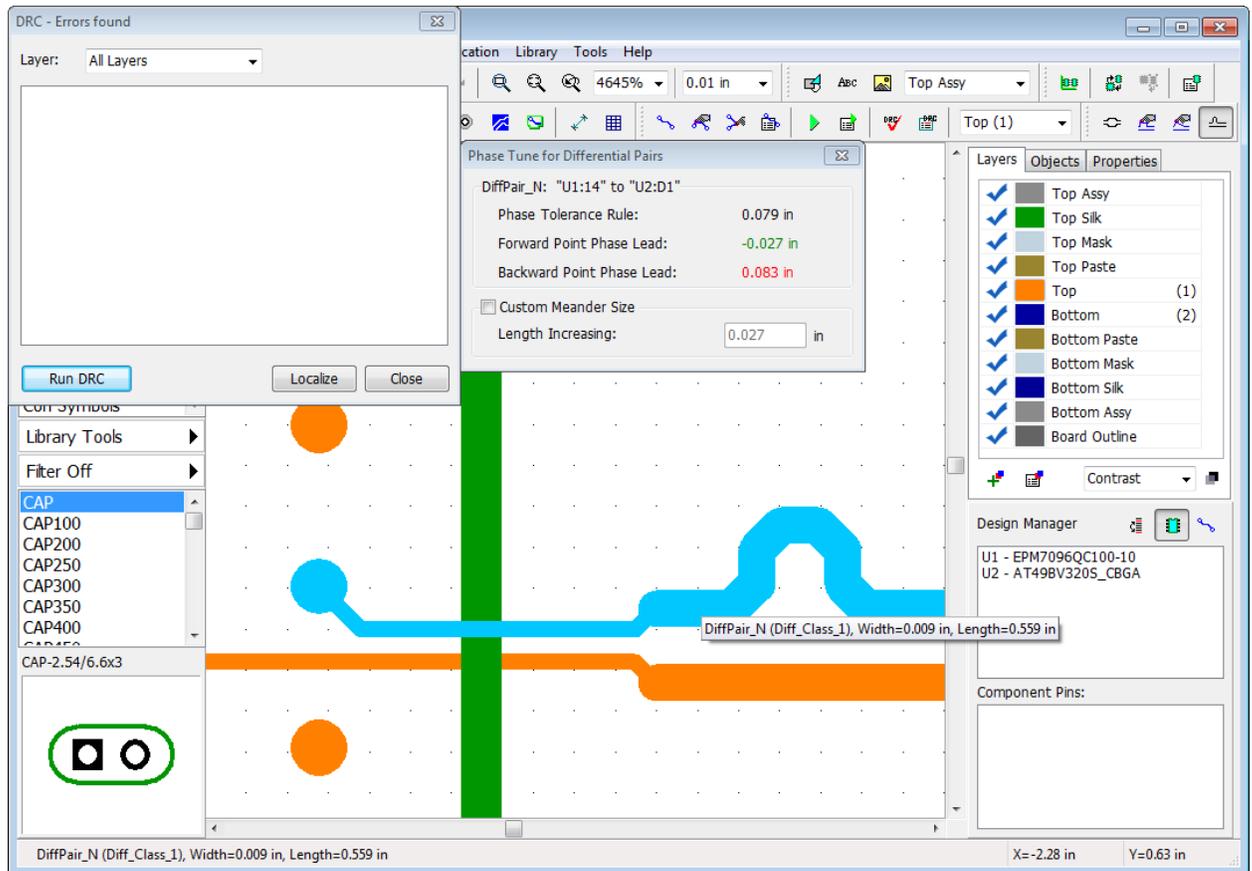


Now only two errors left. The first error says that one trace is much longer than the other, the second error reports that we have phase tolerance violation and it happened on the segment that is longer than Dynamic Phase Error Length value in the Net Class parameters. We need to make the shorter trace a bit longer and fix the phase shift. Since our differential pair is almost the straight line, there is a chance that we can fix both errors at the same time with the Phase Tune

tool. Press  button on the high-speed toolbar or go to "High-Speed \ Differential Pair Tools \ Phase Tune" from the main menu. In the pop-up dialog box you can check which track needs phase tuning. Hover over the bottom track of the differential pair with your mouse cursor and review Point Phase Lead values that will appear in the Phase Tune dialog box. Then hover over the upper trace – violation is highlighted in red.

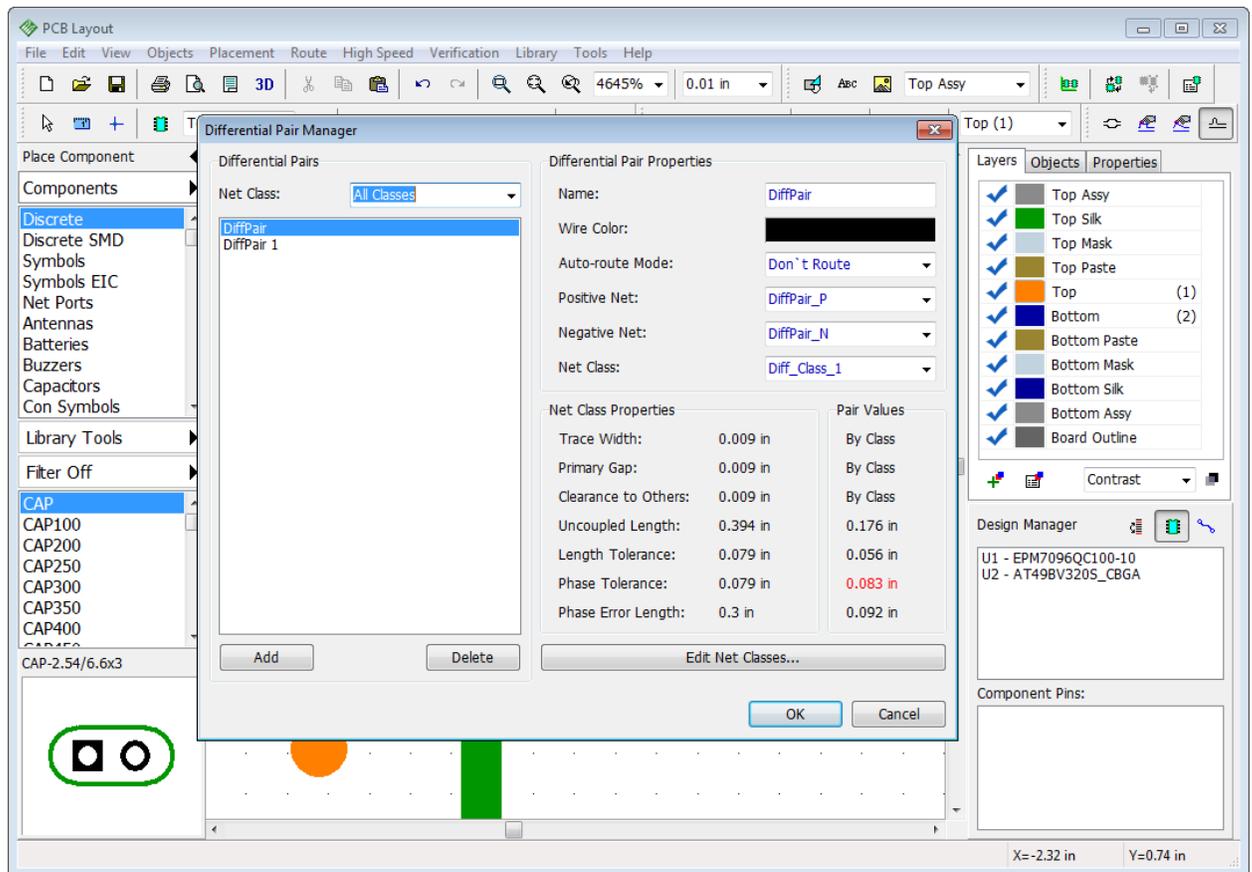


In our case we see that the signal in the upper trace is a bit faster than in the lower. Left click on the upper trace to create a meander, which will make the trace a bit longer. By default meander is approximately 3X the trace width. Check Custom Meander Size box and type in the new value into Length Increasing field if you need bigger meanders. In our case default meander works fine and the error-report dialog is now empty. You can hover over the trace segments and find out that while the segment closer to the U1 is OK, the segment right next to the U2 component still has a backward point phase lead violation, but PCB Creator does not report it as an error, because the segment is shorter than Dynamic Phase Error Length value in the Net Class parameters. This means that we have a phase shift but it occurs on the relatively short trace segment, which doesn't matter for our project. Now you can close the DRC error-report box and switch to the default mode.



### 5.2.4 Differential Pair Manager

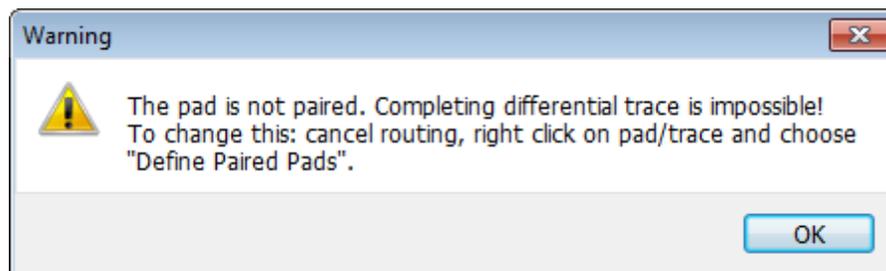
If you have complex design with a lot of differential pairs, it becomes hard to manage them directly on the design area. PCB Creator has Differential Pair Manager which allows for easy managing / editing / deleting differential pairs. Select "High-Speed \ Differential Pair Manager" from the main menu. In the pop-up dialog box specify the exact net class in the Net Class drop-down to display only the pairs of the net class, or select All Classes to show all pairs in the list right below. However, this does not make any difference in our case, because we have only two differential pairs of the same net class.



Press Add to open Define Differential Pair dialog box and create a new differential pair, or select existing one from the list and modify its name, color, nets, net class etc. Right below you can see the Class Properties and Pair values if this differential pair is routed. Red-highlighted value is the violation. In our case Phase shift on the selected differential pair is bigger than the tolerance, but it is not considered an error unless there is a Phase Error Length violation.

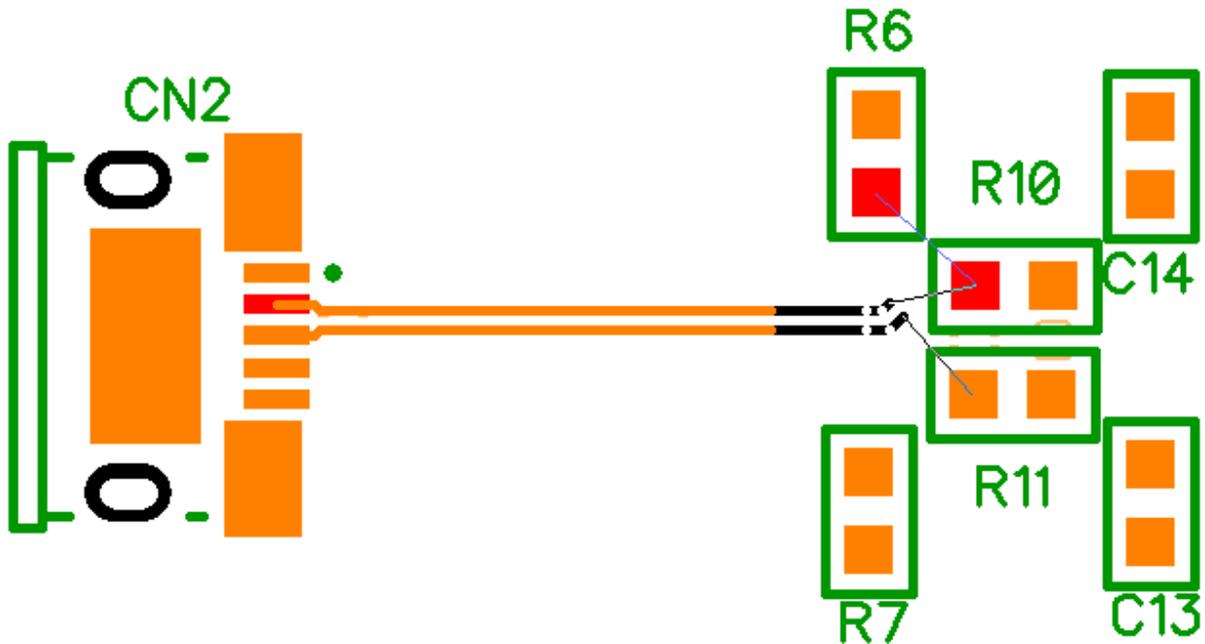
### 5.2.5 Define Paired Pads

On very rare occasions for the circuit boards with lots of various differential pair pads very close to each other, PCB Creator might not be able to draw two traces and finish routing. In this case you will get a warning message that offers to use Define Paired Pads feature.

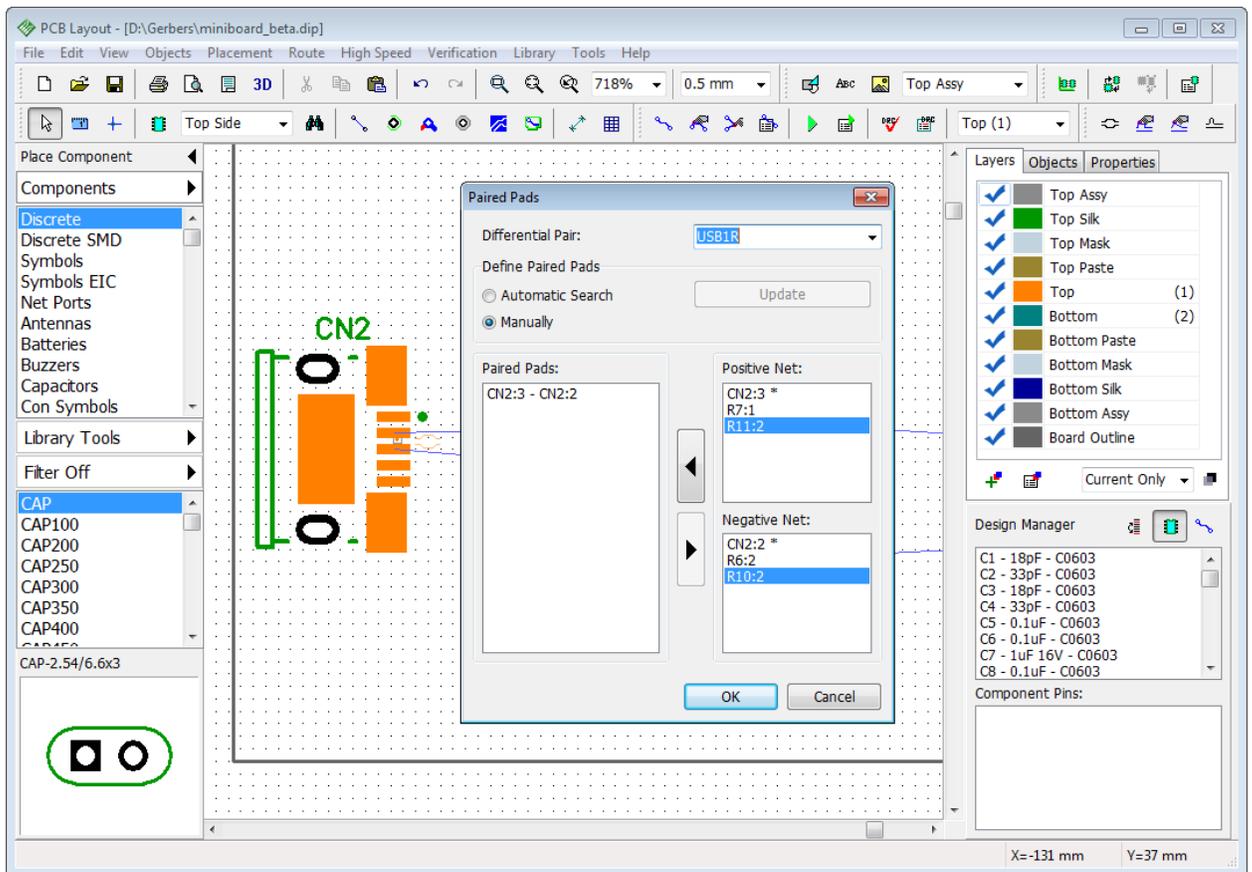


Here is a typical situation. We have R6 and R10 pads that belong to the negative net and R11, R7 pads from the positive net of the differential pair. It is possible that the software in such case

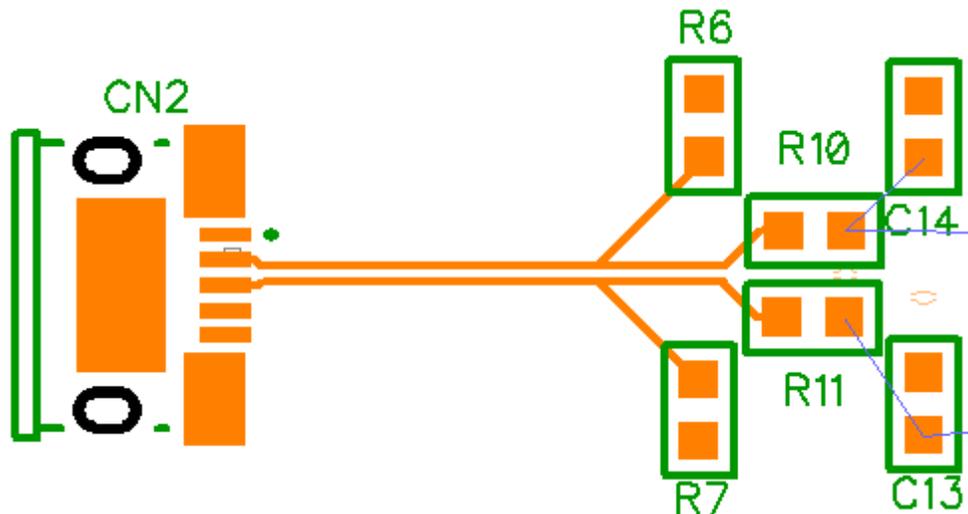
can terminate the second trace on the R7 pad (not on the R11), or either drop the warning message.



To fix this issue, you need to group the pads. Go to "High-Speed \ Define Paired Pads" from the main menu. In the pop-up dialog box use the Differential Pair drop-down list to select the pair. You can press Update button to let the program find pads automatically, or check Manually box and define the pair of pads using the lists right below. We have one pair of pads on the side of the CN2 connector (CN2:3 and CN2:2). Select R11:2 from the Positive Net list and R10:2 from the Negative Net list, because we want the differential pair to terminate on these pads. Then press the left arrow button to add selected pads to the Paired Pads list and press OK to apply changes.



Now route the differential pair. Everything works fine because the software clearly knows where you want to connect the differential pair nets. Finally, draw two traces from the R6 and R7 pads to their respective nets.



## 5.3 Reference Designators

Now we will work with schematic examples located in the Documents \ PCB Creator \ Examples folder. Open Schematic\_2.dch file from the Examples folder.

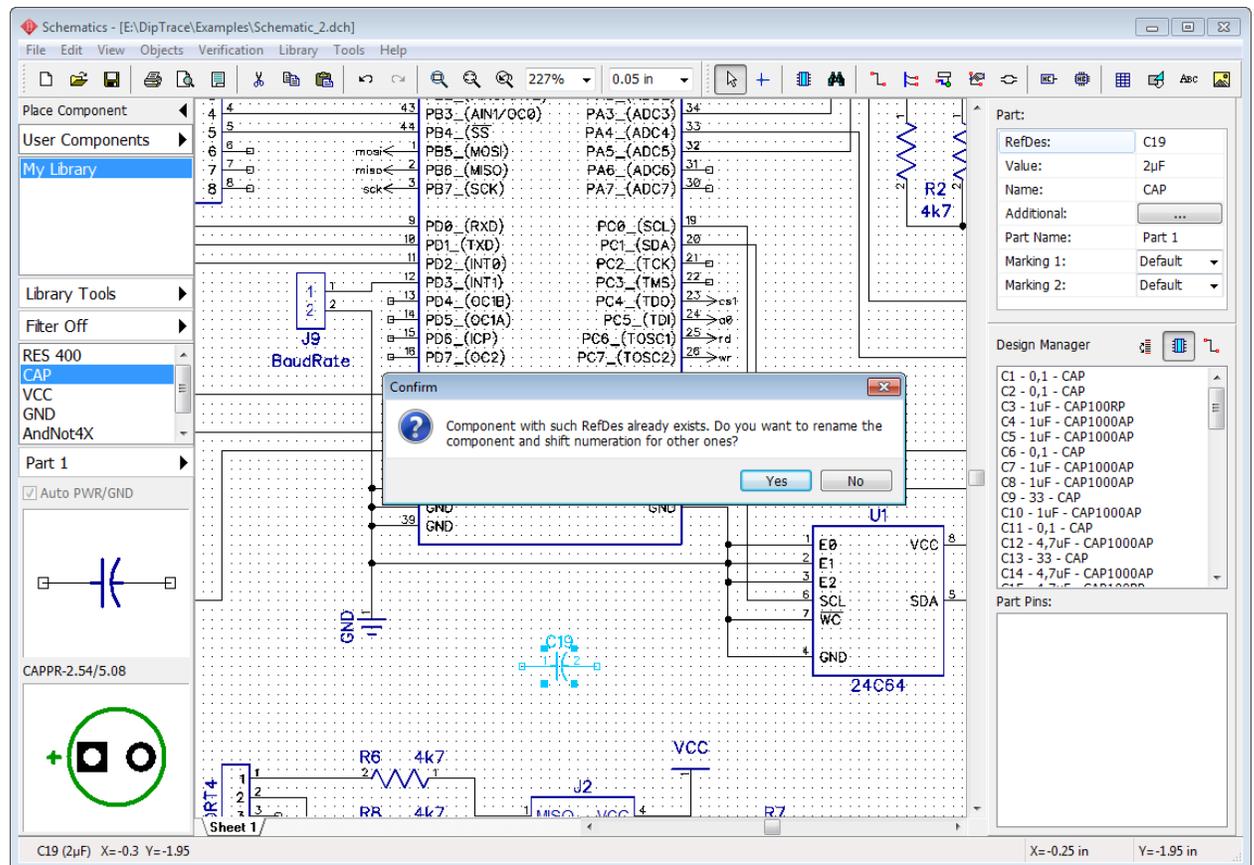
This project demonstrates applications of various PCB Creator features. However, we will experiment with some basic principles of working with Reference Designators in the Schematic.

Current Schematic contains 23 capacitors from C1 to C24 (C19 is missing).

### Optimize RefDes

While editing any circuit you will probably need to insert and delete components from time to time. Let's add, for example, C5 capacitor to the schematic. Pick up some capacitor from the library you've created recently (My Library) or Discrete SMD Library and place it to the design area. We want "C5" RefDes for this component, but it has automatically become "C19" (because C19 was missing and C5 already exists on the schematic).

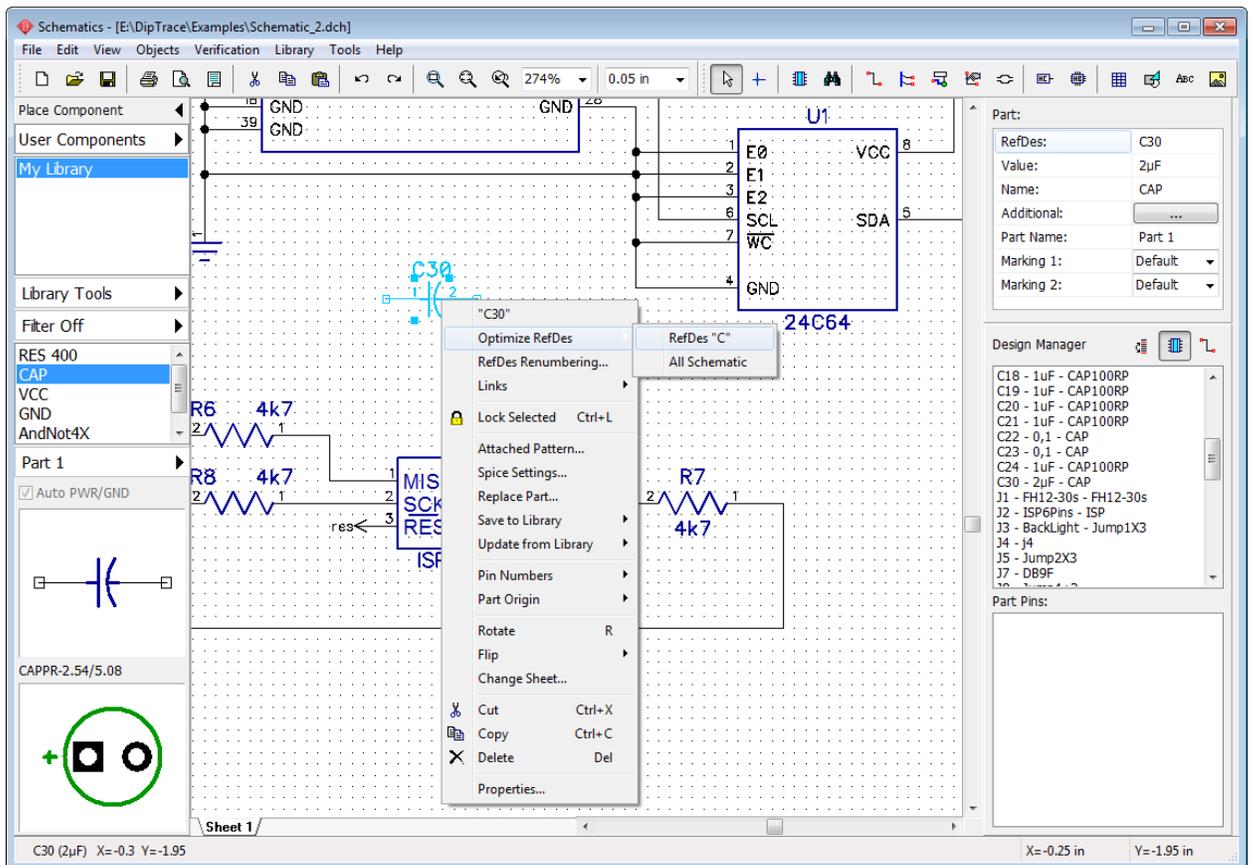
Right click on this capacitor and select first item from the submenu, enter "C5" text and press OK. The software will show the warning message and suggest to rename the component to shift the RefDes numeration. Press Yes.



C19 capacitor was renamed to C5 and an old C5 became C6, and so on till C18 capacitor which is C19 now. See the design manager panel to the right to find out that C19 designator is not missing anymore, because you've inserted C5 and C5–C18 RefDes indexes were shifted ( use

 button to sort the components in the list). We have a correct array of RefDes indexes for all capacitors.

Now please rename C5 capacitor to "C30" then check the list of capacitor designators on the design manager ("Ctrl+2" to show/hide the design manager, press  button to sort the components in the list) – C5 and capacitors with C25 till C29 indexes are missing. To correct this issue, right click on any capacitor and select "Optimize RefDes \ RefDes C" – C30 becomes "C24". The reason is simple – while optimizing RefDes, PCB Creator removes all empty places in the designator index array, therefore C6–C24 become C5–C23 and C30 becomes C24.

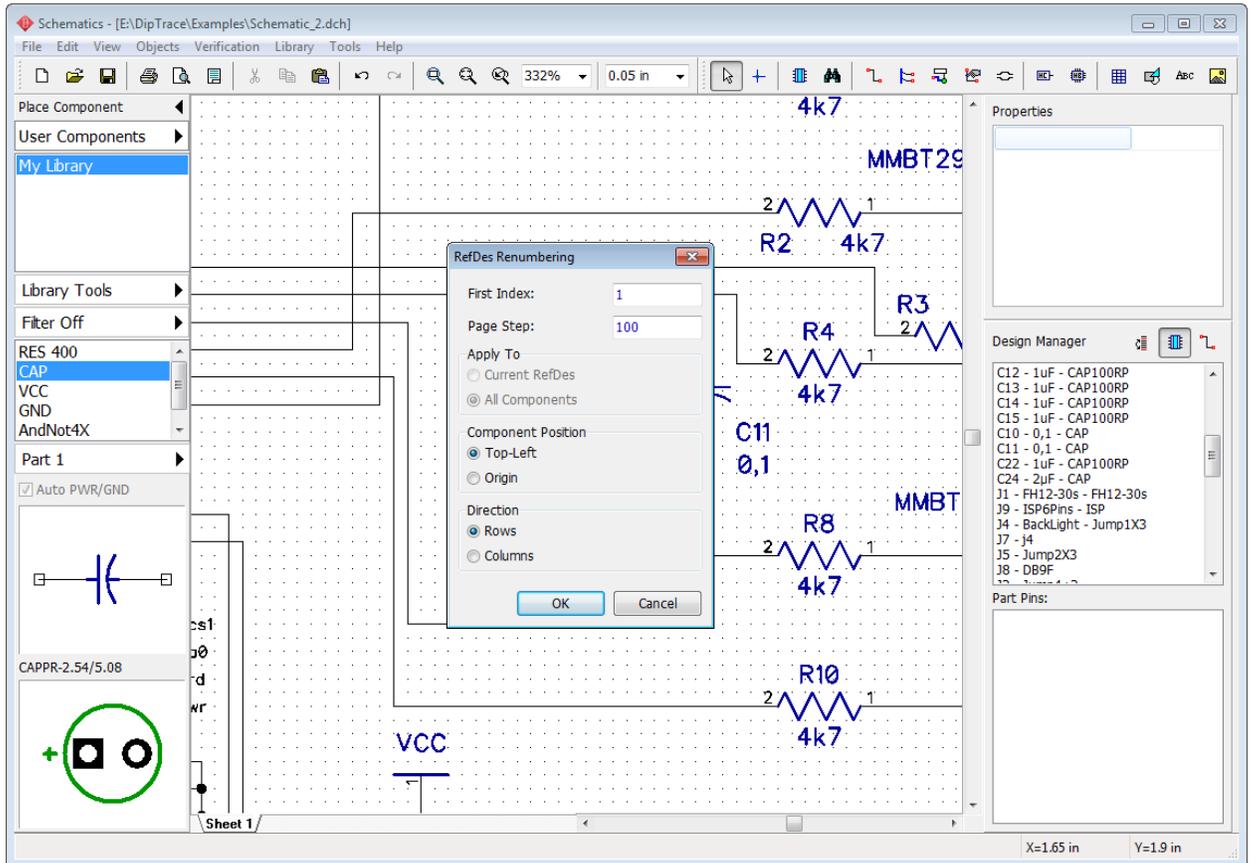


## RefDes renumbering

But what if we need to renumber Reference Designators in a very easy way that makes it simple to navigate through the design? Select "Tools \ RefDes Renumbering..." from the main menu. Do not change the First Index (starting point of renumbering) and Page Step (if page step = 100, designators on the second page will be R101, R102, IC101, etc.). Then specify renumbering direction: in rows or columns, and choose how PCB Creator is going to count components while renumbering. There are components of different sizes and shapes. If we choose "Top-left" in the Component Position section, the PCB Creator will renumber components based on the position of the top-left corner of each component. If you choose Origin, it will use component origins.

*Notice that renumbering always goes from left to right and from top to bottom of the circuit.*

Press OK and all components will be renumbered.



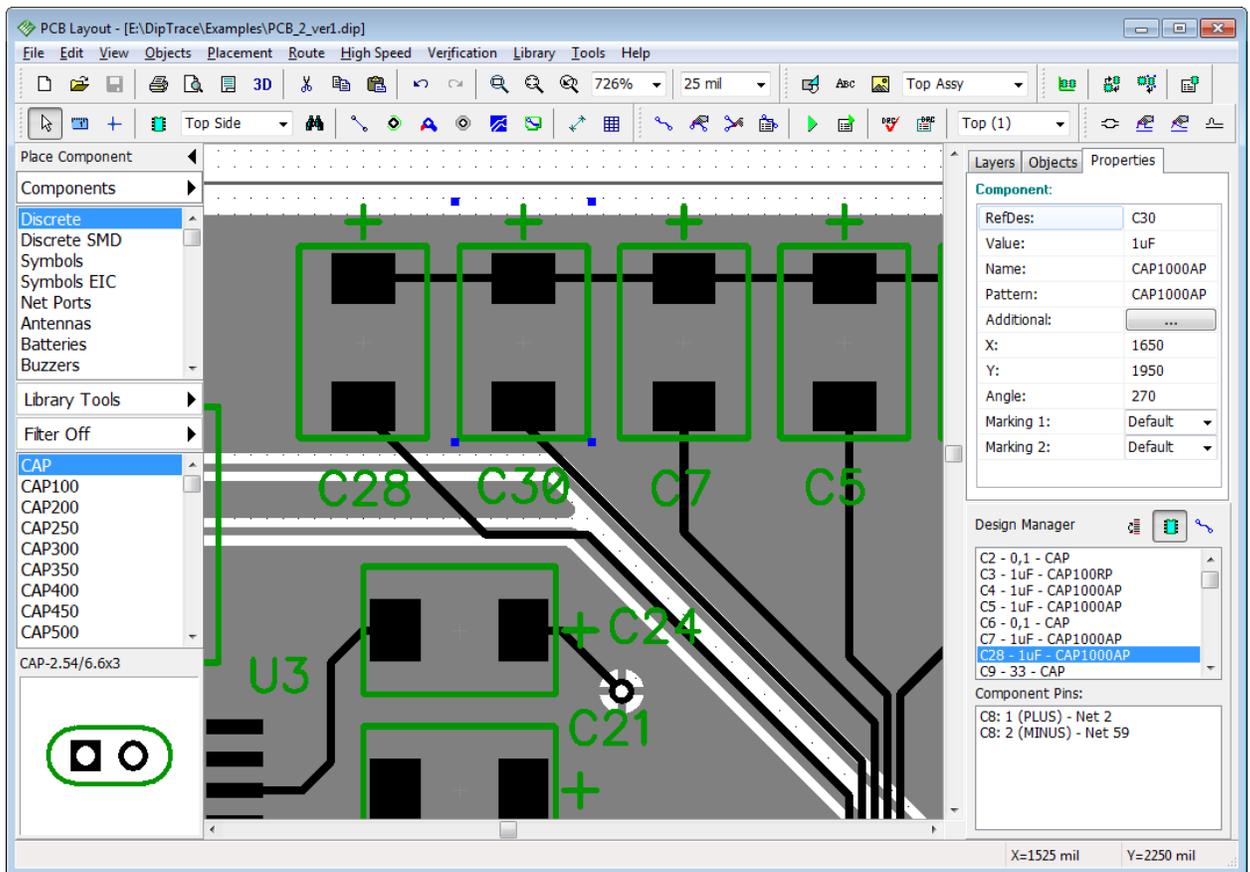
If you need to renumber only designators of the components of selected type, right click on one of the components and select RefDes Renumbering from the submenu. You will see the typical RefDes Renumbering dialog box, but you'll be able to apply renumbering to current RefDes or to all components of the schematic.

*RefDes Renumbering works the same in the PCB Layout.*

Please close the PCB Creator Schematic **without** saving progress and launch the PCB Layout module.

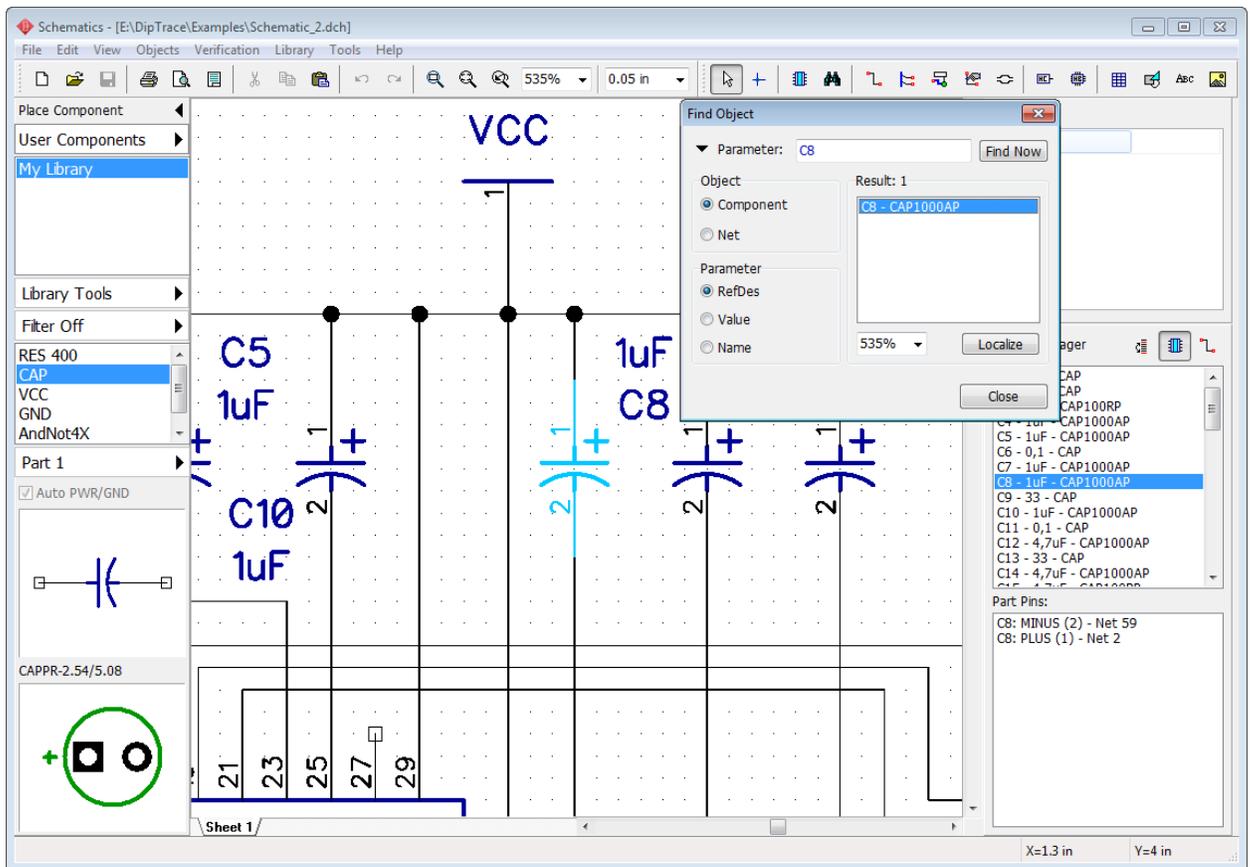
## 5.4 Back Annotate

Launch the PCB Creator PCB Layout module. Then open "PCB\_2" file from the Examples folder. Use design manager to find C8 and C10 capacitors – double click on the component name in the list to localize and pan design area to selected component. Change the reference designators of C8 and C10 capacitors to C28 and C30 (right click on the component and select the first item). Select "File \ Save As" and save changed PCB in another file, for example "PCB\_2\_ver1".



Close the PCB Layout and open Schematic (notice that you can open it directly from the PCB Layout by selecting "Tools \ Schematic" from the main menu).

Open Schematic\_2.dch file and find C8 and C10 components. You can use the design manager or press "Ctrl+F" (or select "Edit \ Find Object" from the main menu), type "C8", and press Enter. C8 capacitor will appear highlighted in the center of the screen.

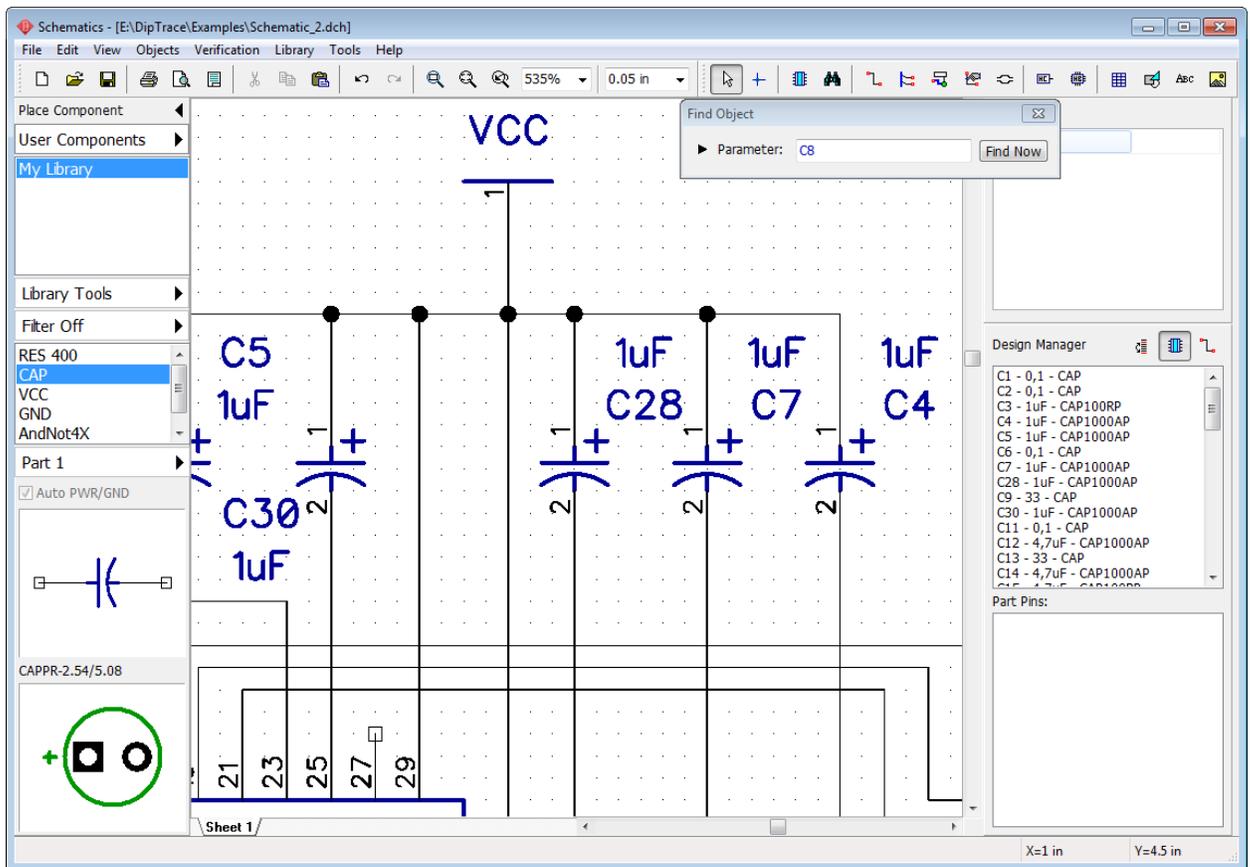


Notice that you can minimize Find Object panel by clicking the arrow in its upper-left and use it while editing your design without showing all parameters.

Zoom in the schematic to see C8 and C10 better. PCB\_2 is the circuit board related to Schematic\_2, but as you remember we have renamed C8 and C10 capacitors on the board. We can rename them on the schematic manually (applicable only to small-scale projects) or use Back Annotate feature.

Go to "File \ Back Annotate" from the main menu and select the PCB file where we saved modified copy of the PCB\_2 board ("PCB\_2\_ver1" in this case). Press Open. Now you can see that reference designators in the Schematic (in our case C28 and C30) has changed according to the PCB.

*Notice that net names and net classes are also back annotated from the PCB, but this does not apply to new nets and components added directly on the circuit board.*

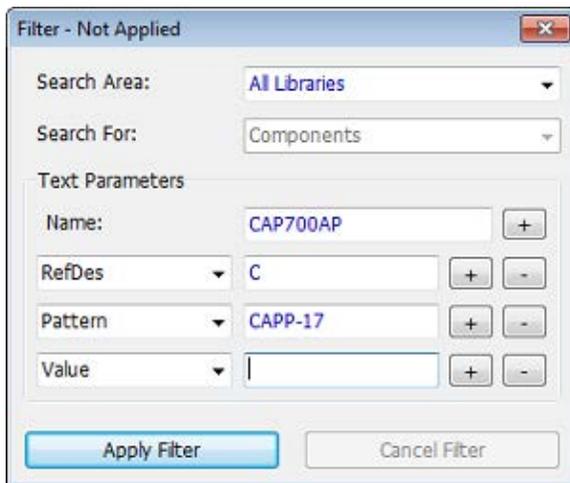


## 5.5 How to find components in the libraries

PCB Creator 3.0 includes 130,000+ components in standard libraries. Components are sorted by manufacturer and by type. Cross-module Library Management system with custom library groups and search filters allows to seek through the libraries and quickly find what you need.

### Search filters

Go to "Objects \ Find Component" from main menu in Schematic or press Filter ON/OFF button on the component placement panel to customize the search filters.



In the pop-up dialog box specify the search area: all libraries, current library group or current library and type in the name or part of the name of the component. PCB Creator allows to filter components by RefDes, Value, Pattern, Manufacturer, Datasheet or Additional Fields. Press "+" or "-" buttons next to the corresponding search filter to add or delete this search filter respectively. Use the drop-down list to select the search parameter.

As you can see on the picture, we search for CAP700AP capacitor with "C" RefDes and CAPP-17 pattern.

Press Apply Filter and close the dialog box. You will see that only filtered components, which correspond to the search filters, are visible on the component placement panel.

Search filters state is always displayed on the component placement panel (Library manager) "Filter ON/OFF". To disable the search filters open the search filters dialog box and press Cancel Filter.

### Place component

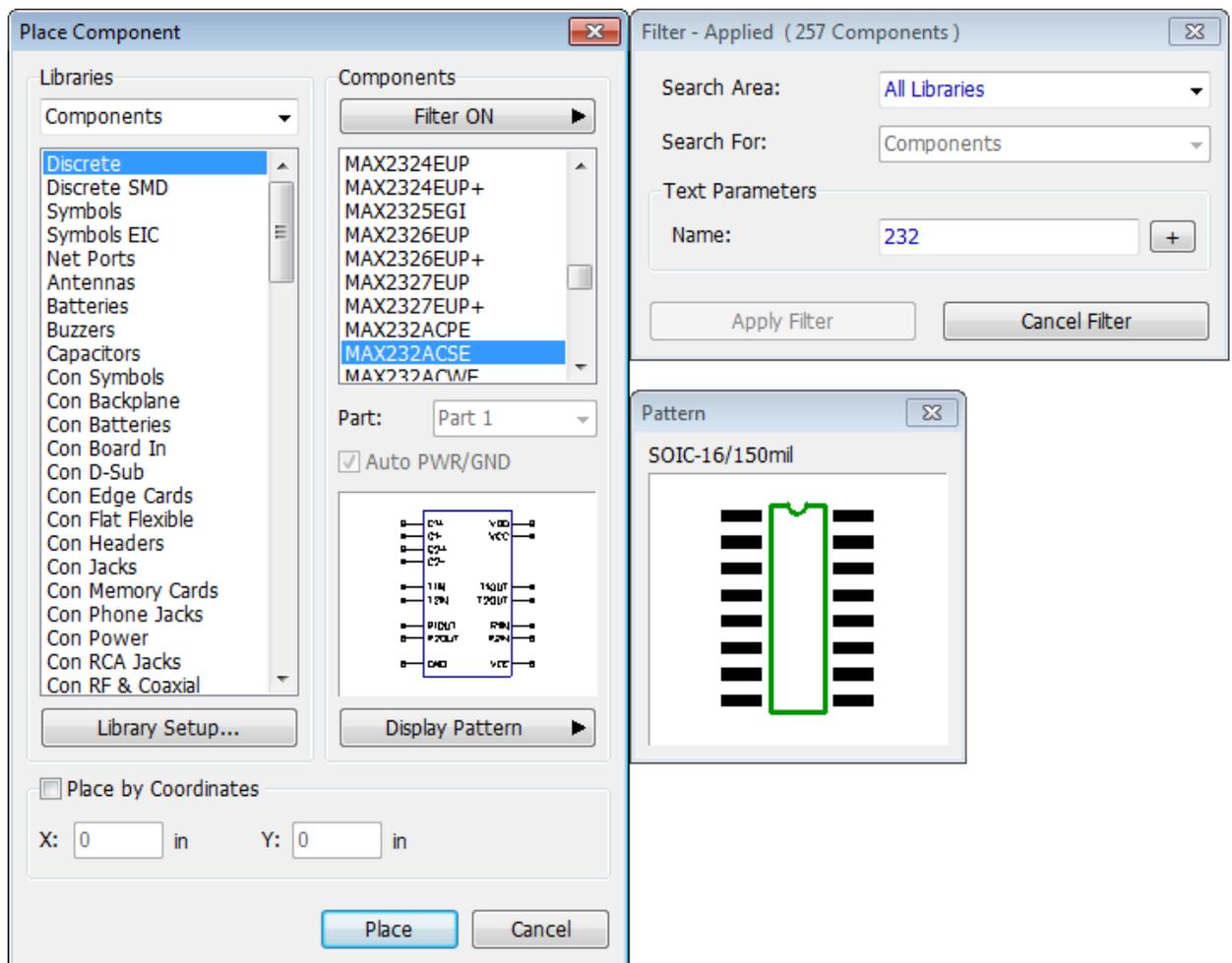
You can find and place components with special Place Component dialog box ("Objects \ Place Component" from the main menu in the Schematic). There are two sections in the pop-up dialog box: Libraries and Components. Select library group from the drop-down list, then select library of this group and, finally, component from the corresponding list. Press Filter ON/OFF button to customize and apply search filters.

*"Objects \ Place Component" dialog box is similar in PCB Layout.*

For example, we need a component that contains "232" in its name, but we don't remember the other characters, letters or even library that it belongs to. You can try to find it manually, select the Components library group which contains all standard components or press Filter ON/OFF button, select "Search area: All Libraries", type in "232" in the Name field and press Apply Filter. PCB Creator has found 230 components with 232 text in their name. Components are in the list right below. Select component and you will see it's schematic symbol or press Display Pattern to see how it looks on the PCB.

Add more search filters if you have any further details.

Press Place button and left click on the design area to place selected component or check Place by Coordinates box and enter exact coordinates where component will be placed automatically. This dialog box has all necessary tools to work with multi-part components as well.



## 5.6 Electrical Rule Check

Electrical Rule Check (ERC) feature is one of the main verification features in the PCB Creator. ERC checks circuit for pin type conflicts, unconnected and superimposing pins, as well as one-pin nets, and short circuits.

Open Schematic\_2.dch from the Examples folder in the PCB Creator Schematic. Define electrical rules, select "Verification \ Electrical Rule Setup" from the main menu. In the pop-up dialog box specify incompatible pin-to-pin connections and program reaction to them (error, no error or warning, depending on the color) by clicking in the grid cells with green, yellow, and red squares.

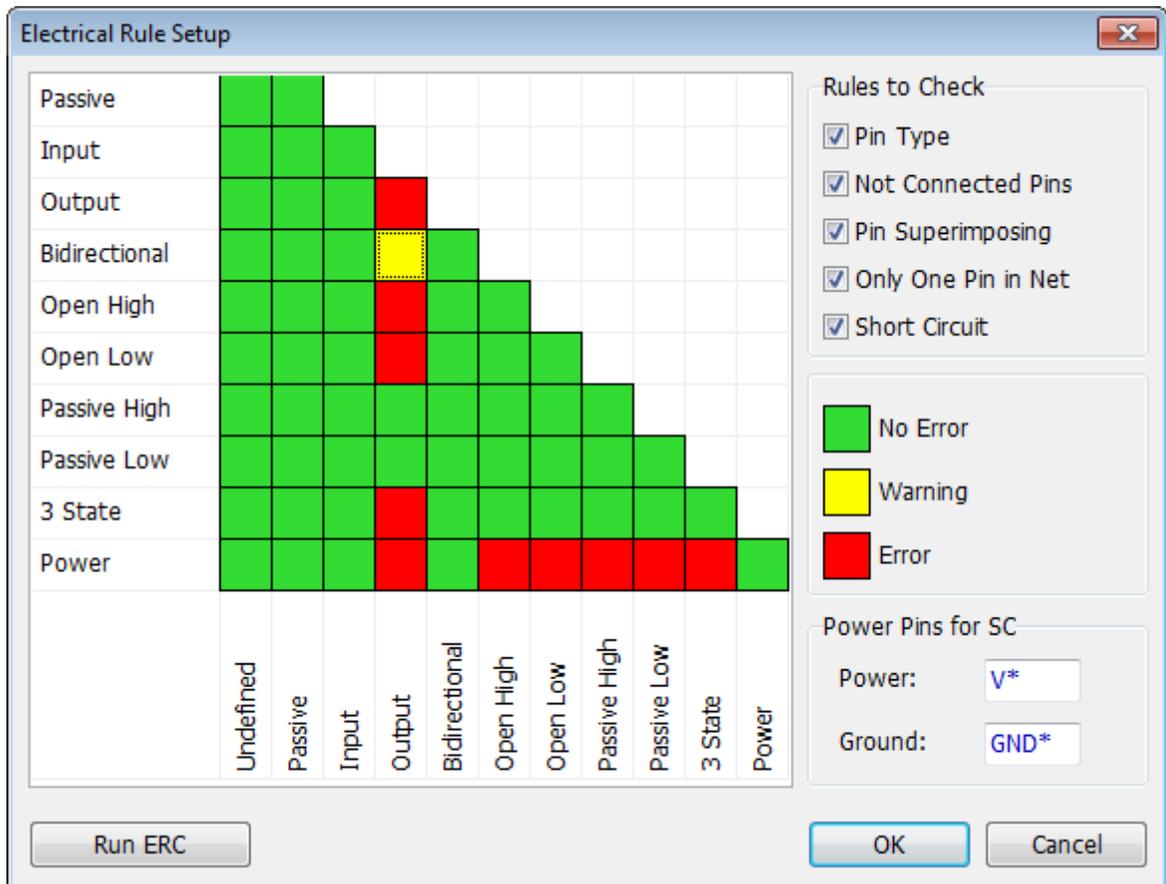
Pin Type check box means checking pin-to-pin connections defined in the grid;

Not Connected Pins – the program searches for unconnected pins;

Pin Superimposing – the program searches for pins overlaying each other;

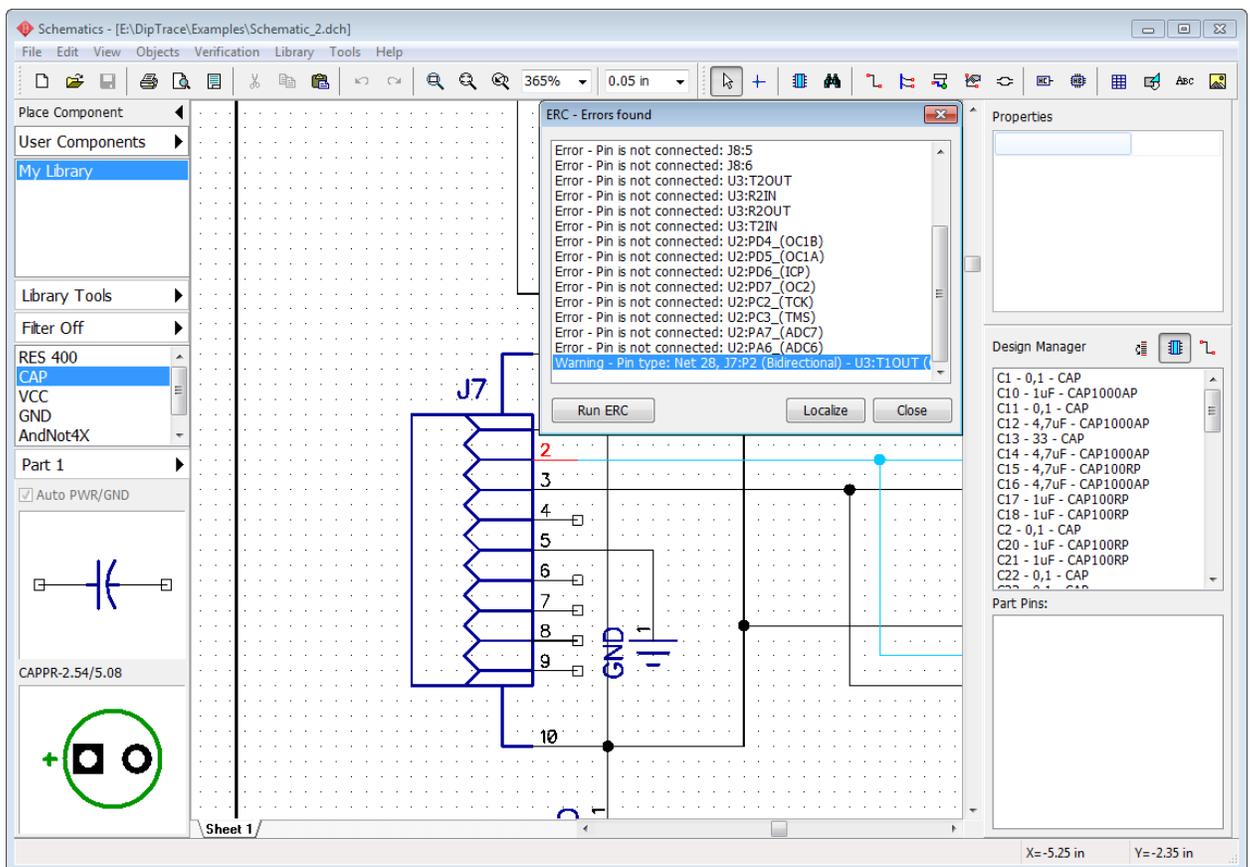
Only One Pin in Net – program reports nets with only one pin, i.e. net that makes no sense. It can be potential error in the net structure;

Short Circuit – program reports Power to GND connections. Define names for PWR and GND nets of the project in "Power Pins for SC section below.



Press OK to close the dialog box.

Now select "Verification \ Electrical Rule Check (ERC)" from the main menu. If you verify "Schematic\_2" file, according to the rules from the picture above you should get a lot of error reports in the list: one warning for Bidirectional to Output connection and number of Not Connected pin errors. To localize error on the schematic, double click it in the list (or press Localize). Errors can be corrected without closing the ERC report dialog. Press Run ERC button to start verification again.



Notice that "Not connected" pins error often is not an error at all. ERC will not report errors for pins, which are unconnected intentionally. Right click on one of these pins and select "Not Connected" from the submenu. Or you can uncheck corresponding item in ERC settings dialog box, but then possibility of an error increases, because all unconnected pins are now OK for the software.

## 5.7 Bill of Materials (BOM)

PCB Creator Schematic module has BOM feature that allows the user to customize columns and rows, add tables or pages to existing project, export files to Excel CSV format or save Bill of Materials as a text file with appropriate table formatting.

Select "Objects \ Bill of Materials" from the main menu. There are two main sections in the pop-up dialog box: Table Rows and Columns. Specify "Group Rows by: Components" and adjust Row Height if you need.

Include Power/GND parts option allows the user to include corresponding parts to the BOM file.

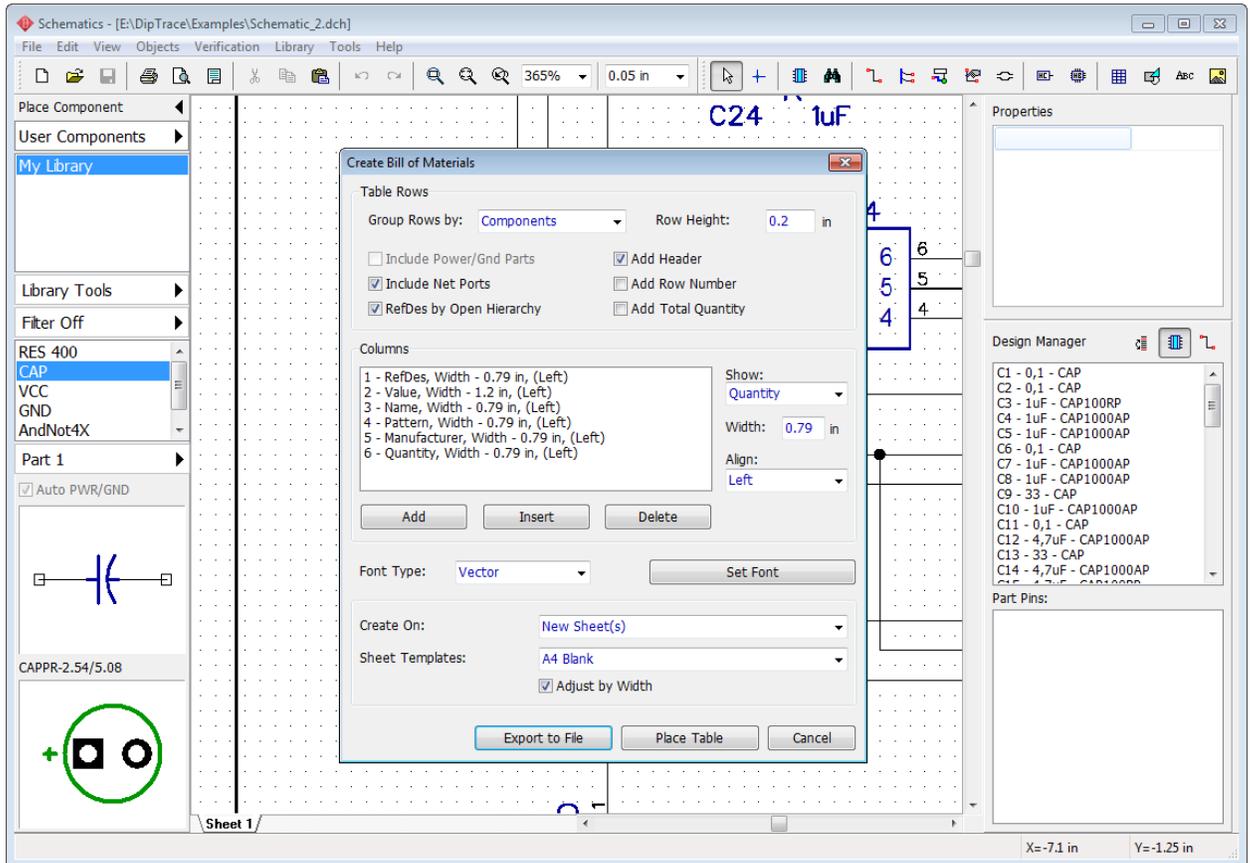
Include Net Ports checkbox allows for net ports in the BOM table.

RefDes by Open Hierarchy allows the user to show RefDes with hierarchy block prefixes, other check boxes are here to add header, row number, and total quantity. Check them if you need.

Now add columns to the BOM file with the settings like on the picture below. Select the

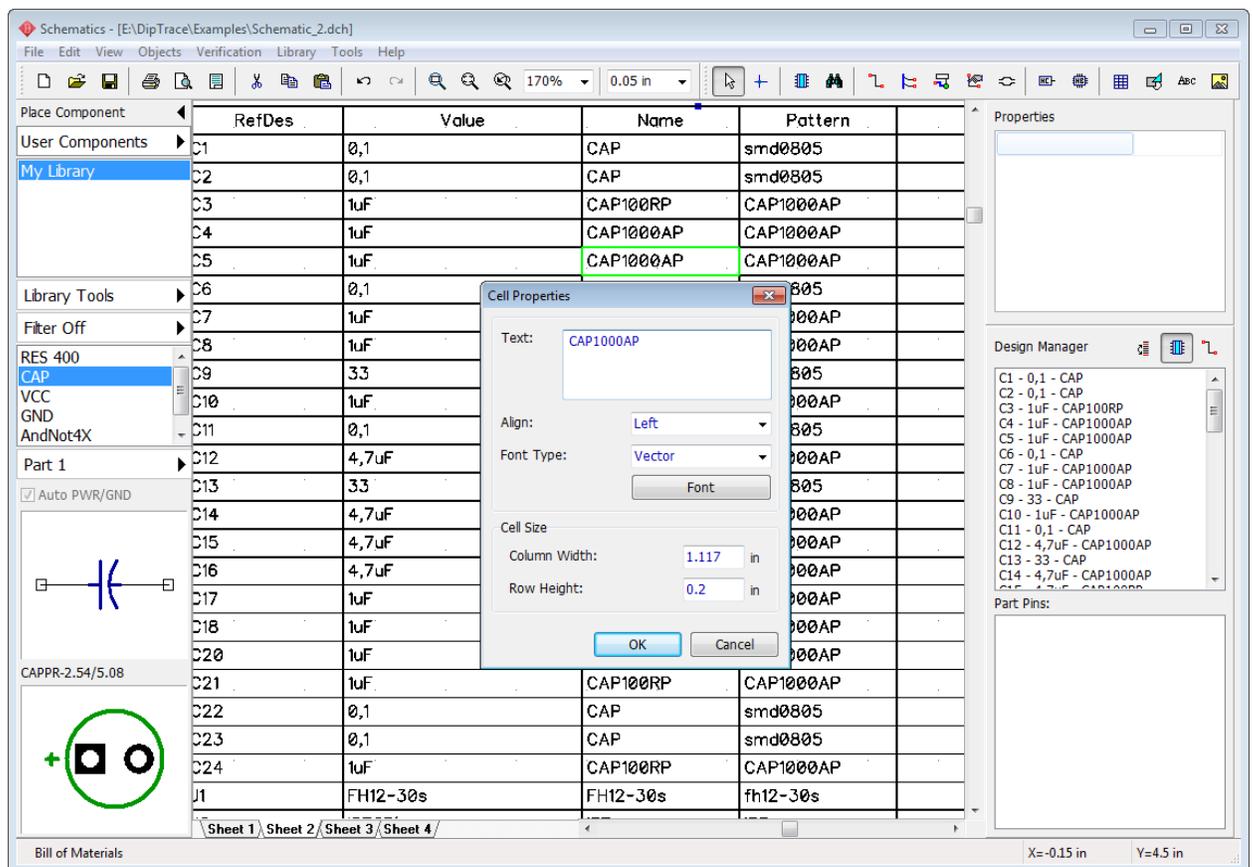
corresponding item (Quantity) from the Show drop-down list and press Add button. Rows can be deleted and edited. Press Set Font in order to customize the font settings. Notice that TrueType font should be used for Unicode characters.

Select "Create On: New Sheet(s)" and A4 Blank in the sheet templates box. Check Adjust by Width to stretch the BOM table according to the page width.



Press Place Table button to add new A4 sheet(s) with ISO title and BOM table to the current project (several sheets for big projects or detailed BOMs). Do not forget to check Display Titles and Display Sheet in the View main menu item if you need.

Select Sheet 2 or Sheet 3. Notice that you can edit row height, number of lines for cells, alignment, font, etc., just left click the cell and make necessary customizations in the pop-up dialog box.



Notice that you can place BOM table to the same sheet with the circuit: select "Create On: Current Sheet", press Place Table, and choose table location after closing the dialog box (left click on the design area). If you have multi-sheet schematic with many components, separate BOM tables for each schematic sheet are possible.

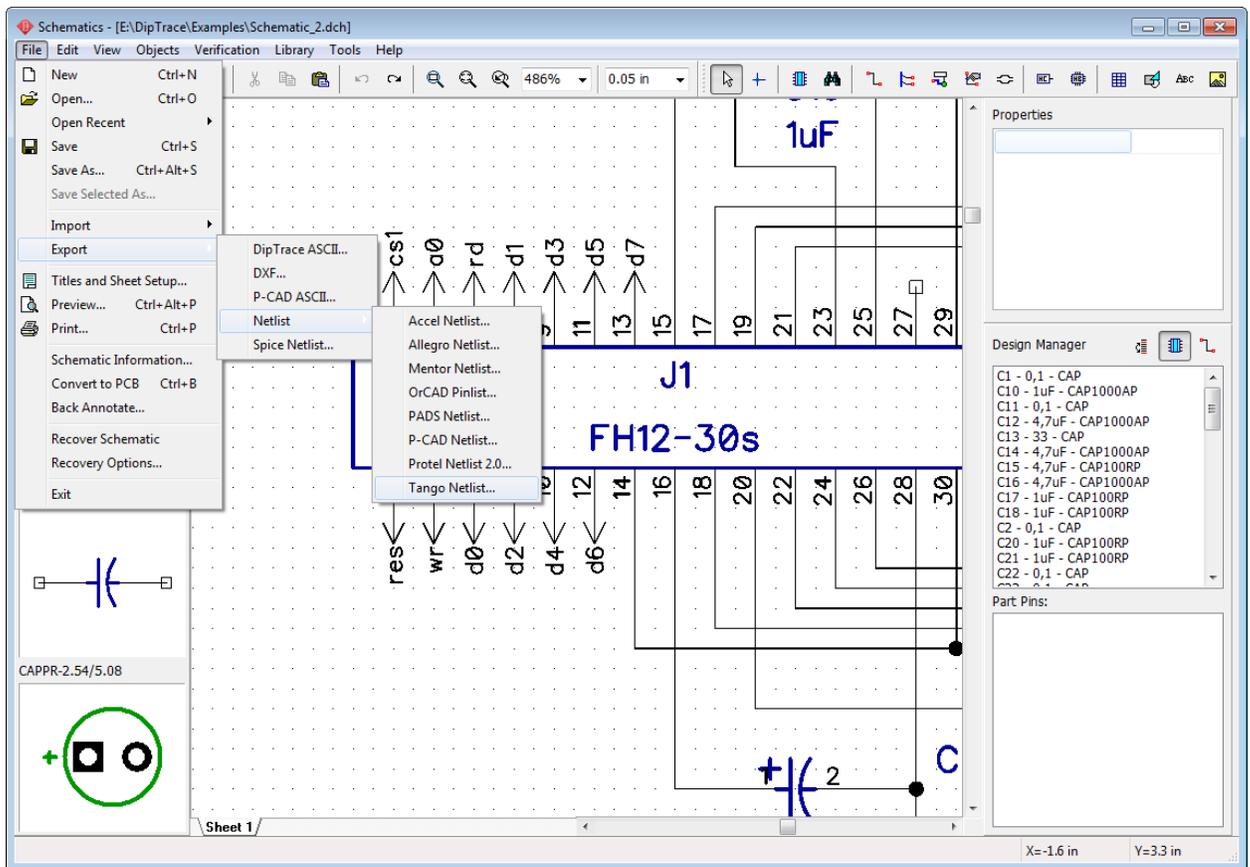
If you need some columns that can not be inserted directly in the PCB Creator BOM dialog box. You can save the Bill of Materials into CSV-file or text-formatted table, just right click on the existing table and select Save to File from the submenu. And then open this CSV file in any spreadsheet editor.

## 5.8 Importing/Exporting netlists

PCB Creator allows the user to import and export netlist files of various formats (Accel, Allegro, Mentor, OrCAD, PADS, P-CAD, Protel, Tango). Netlist export may be used for schematic net structure review or board design in other software.

### Export netlist

To export netlist from the PCB Creator Schematic, select "File \ Export \ Netlist" from the main menu and select the netlist format. Specify location and file name, and press OK to save the file.



### Import netlist

PCB Layout allows for importing netlists created in other software. We will import Tango netlist as an example. Launch the PCB Layout, create a new project and select "File \ Import \ Netlist \ Tango" from the main menu then select "tango\_1.net" file from the C:\Users\

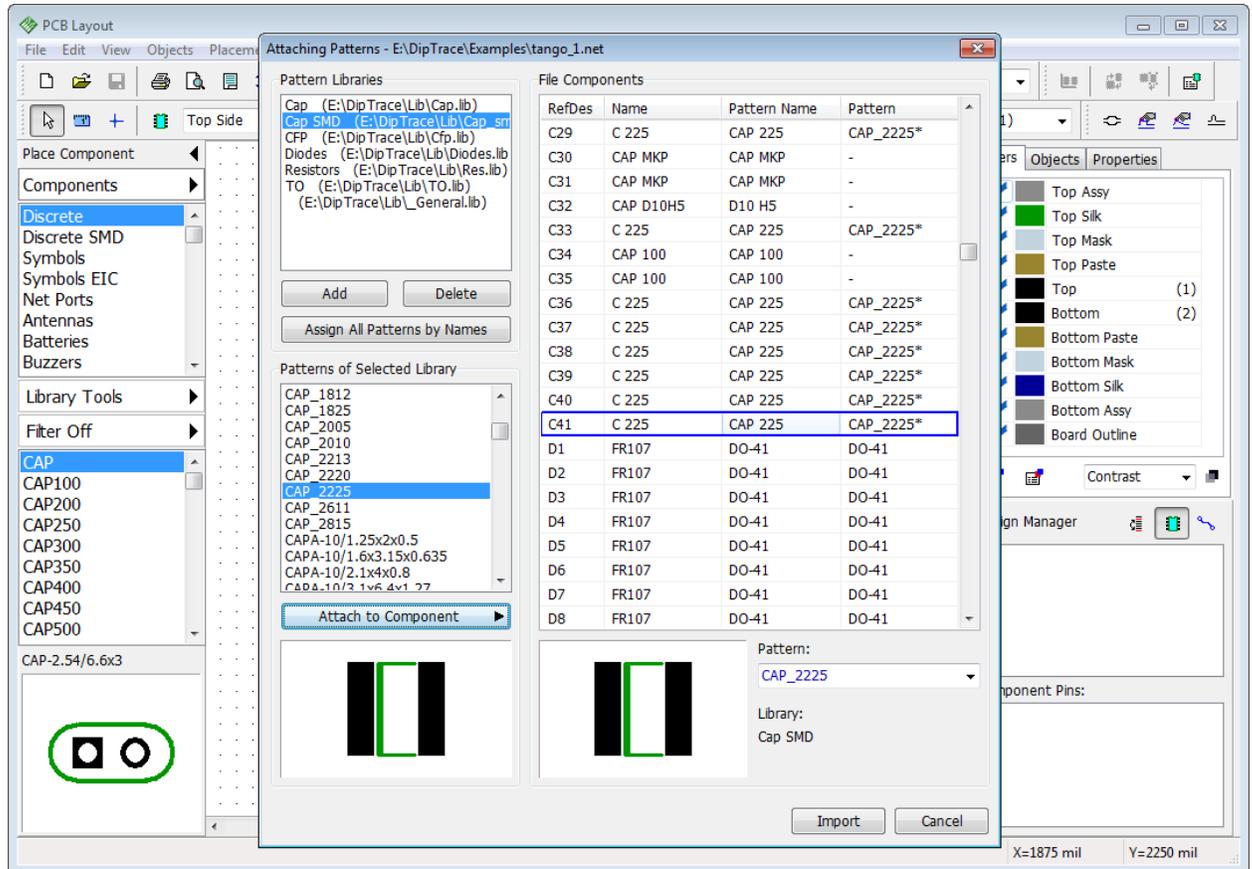
As you already know, all components are represented by patterns (footprints) on the circuit board. The first step while importing the netlist is to **make sure that each component has the pattern (footprint)**. Check all patterns in the File Components list. RefDes column shows component RefDes in the netlist, Name column shows the component's name, Pattern Name column shows pattern's name from the netlist, Pattern column shows attached pattern in the PCB Creator. If it is blank, component does not have a pattern, we need to fix it.

Press Add button to add new pattern libraries to get the patterns from them. For example CAP\_SMD library for C225 components. Add several libraries at a time with "Shift" and "Ctrl" keyboard shortcuts. PCB Creator standard libraries are in the C:\Program Files\PCB Creator\Lib folder and user libraries are in the My Libraries folder by default.

Then press Assign all Patterns by Names – the patterns with corresponding names will be found and attached automatically.

Sometimes PCB Creator is not able to find corresponding patterns, because of partially or completely different pattern names in the PCB Creator libraries and in the netlist. In this case

designer should find and assign all patterns manually. Select component in the File Components list, then select library and pattern from the respective lists. Press Attach to Component button to attach pattern to the component according to their RefDes, name or pattern name. For example, we have added CAP\_2225 pattern to all components with CAP 225 pattern.



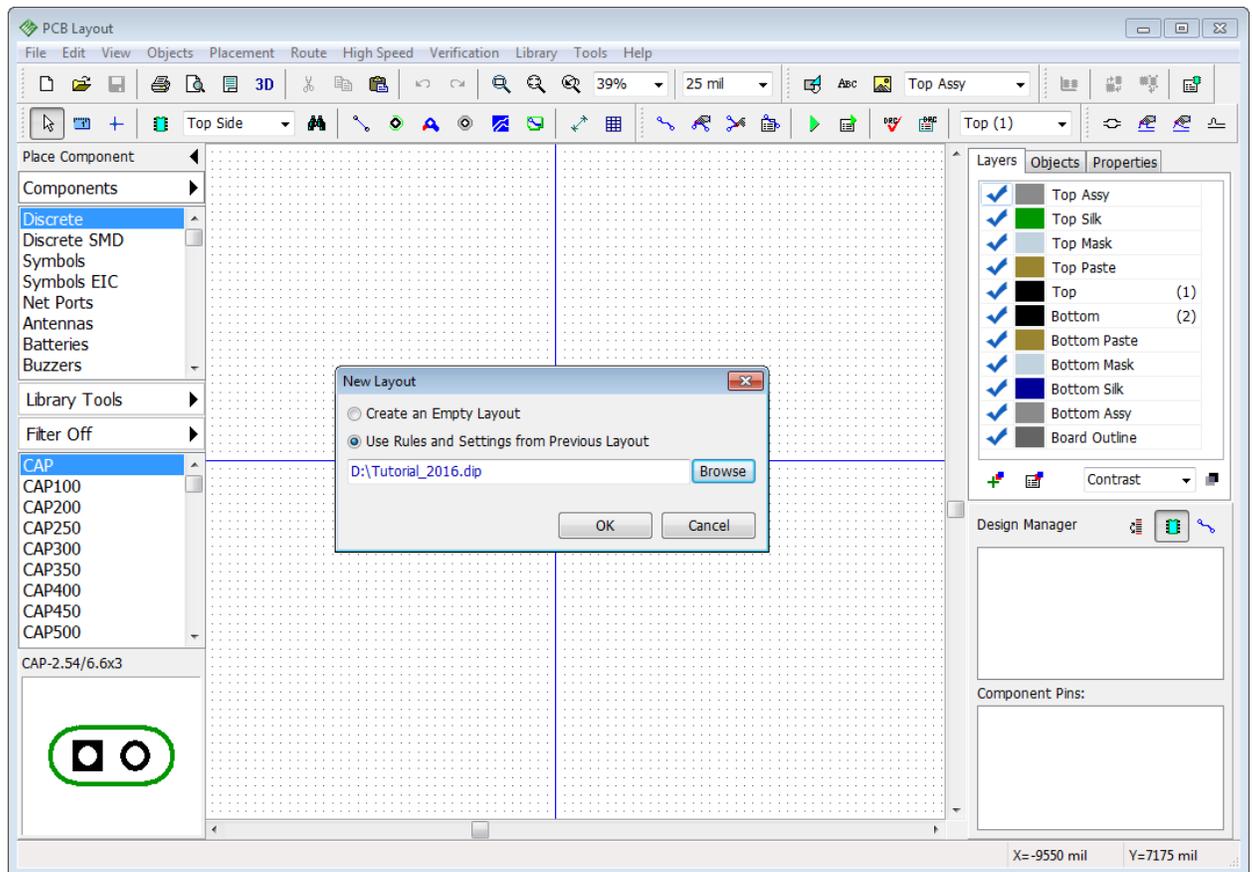
"\*" symbol after the pattern name in the list means that this pattern has been connected manually.

Click Import button when all components have correct patterns.

## 5.9 Saving/Loading Design Rules

In [Converting to PCB](#)<sup>30</sup> topic of this tutorial we mentioned that you can use Schematic rules or load rules from any PCB Layout project, while converting schematic to the PCB. No need to specify all layers, net classes, via styles, and verification design constraints.

Create a new layout, select "File \ New" from the main menu or press "Ctrl+N" default hotkeys. In the pop-up dialog box you can choose to create an empty layout or use settings from the previous project.



Check Use Rules and Settings from Previous Layout, then press Browse, and select \*.bac file of the project which contains layers, via styles, net classes, and design constraints that you may need for this project. Press OK. In our case we have selected the \*.bac file of the project that we have created in the first paragraph of this tutorial. Layer colors and some DRC settings had been changed.

In the PCB Creator PCB Layout you can save settings in the special file, separately from the layout itself. Just go to "Route \ Save Rules", enter the file name, and press the Save button. Rules and settings from this file can be used while creating new projects or loaded later, go to "Route \ Load Rules" and choose \*.bac or \*.rul file.

Please add some via styles, new net class with random parameters, GND and PWR inner layers and save it as a \*.rul file. We will use it later.

## 5.10 Spice simulation

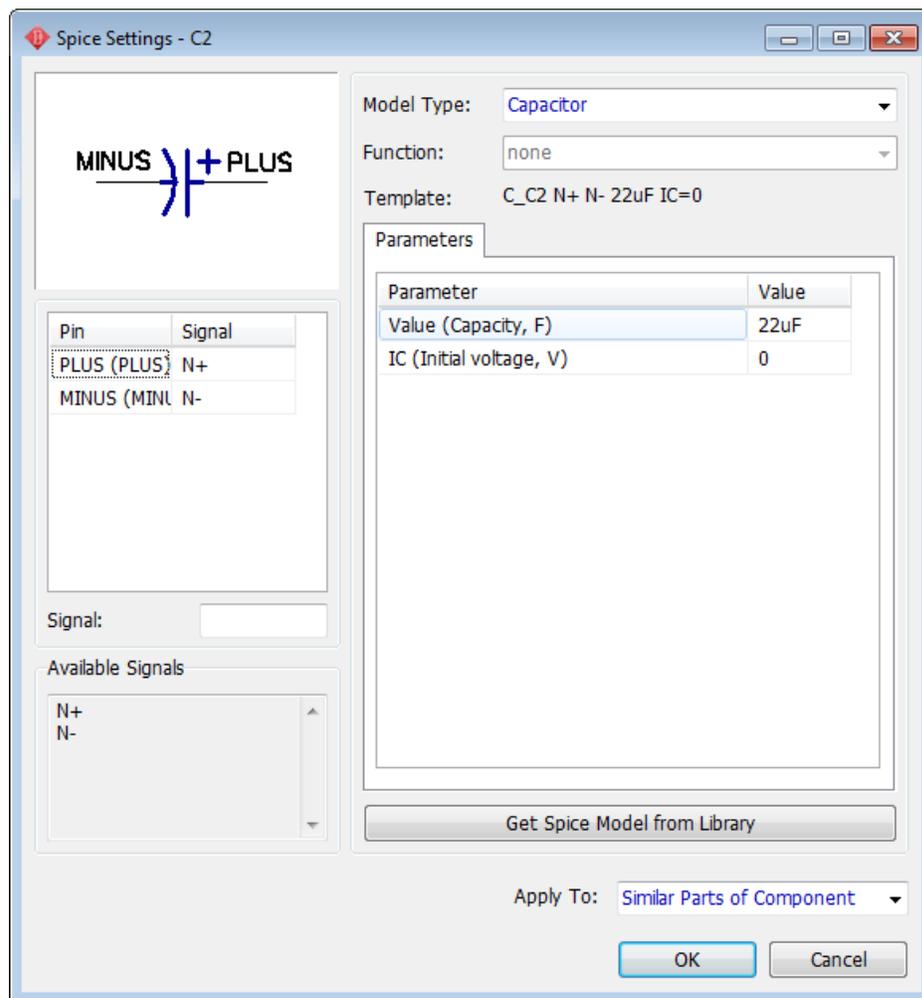
PCB Creator does not have a built-in simulator, but the program allows to define spice settings and export spice netlist to any third-party simulation software. We will try to simulate a stable flip-flop schematic using LT Spice, free and quality simulation software. However, if you have another program that you use for simulation, you're good to go.

### Edit Spice settings

Launch the PCB Creator Schematic and open C:\Users\\Documents\PCB

Creator\Examples\Spice\Astable\_Flip\_Flop\_Spice.dch. We have already defined all Spice settings for this circuit. However, let's review a couple of the components just to learn how to work with the Spice simulation. Right click on the C2 capacitor and select Spice Settings from the submenu. Spice settings for capacitors are pretty simple: you should select the "Model Type : Capacitor", enter values into the parameters table (in our case "22 $\mu$ F "), and specify positive and negative pins (enter values into the pin-to-signal table on the left side of the dialog box, the list of available signals is right below). Notice that you can enter parameters directly into the table cells.

Template field shows how this component is represented in the Spice netlist. You can scroll that field to the right if it is long. Make sure that settings are like on the picture below.

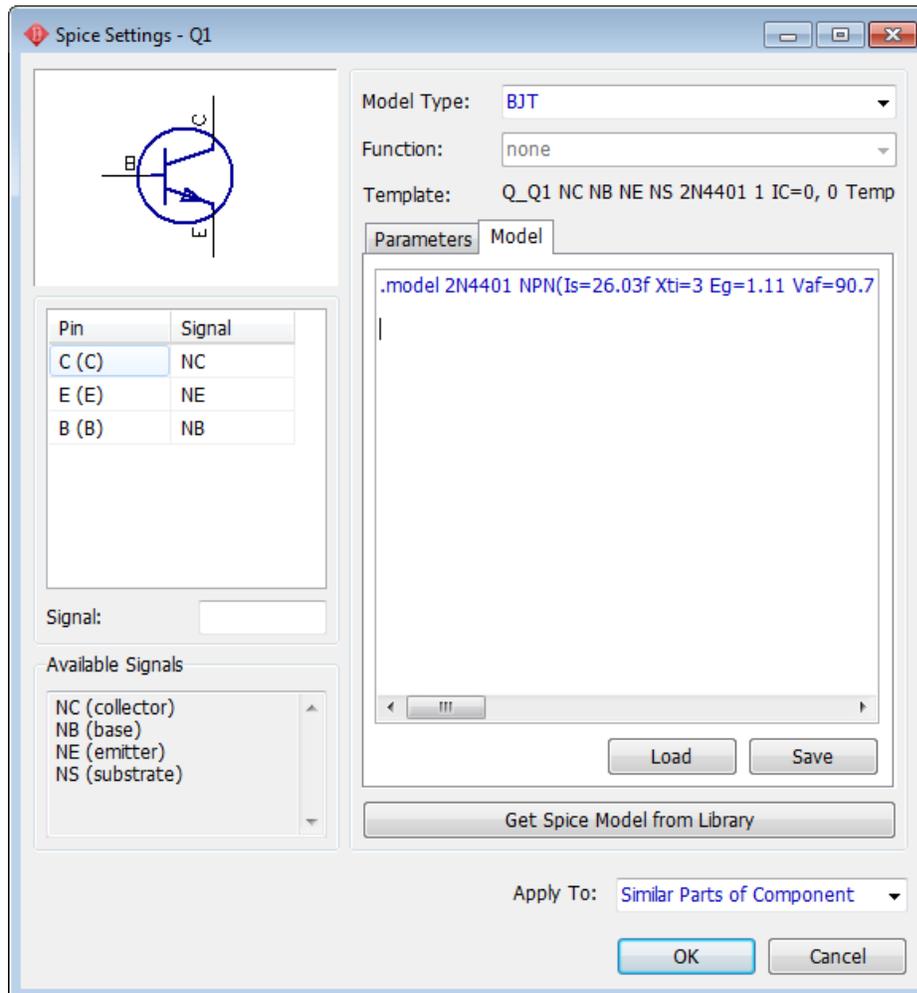


Select another model type (for example, Current Source). For this model type the function can be specified, use the Function drop-down list and select PWL(). In the pop-up dialog box enter the number of points for PWL function and click OK. Now you can enter the values for each point in the Parameters table. Different functions require different parameters (amplitude, phase, etc.). See detailed description in the Spice language documentation.

Now return back to the Capacitor model type, define its value (**discard any changes**) and click OK (or simply close the dialog).

## Spice model

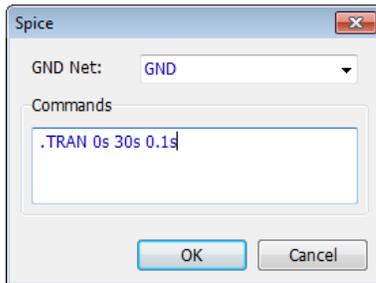
Capacitors do not require additional model description, but transistors do. Right click on the Q1 transistor and select Spice settings. In the pop-up dialog box you can see that there is a Model tab near the Parameters tab. Select it, now you can enter the model text or load Spice model from external file (Load and Save buttons). Some component manufacturers publish Spice models for their components on the web.



Notice that you can get all Spice settings from another PCB Creator library (use Get Spice Model from Library button). Click OK or Cancel to close this dialog box (discard any changes).

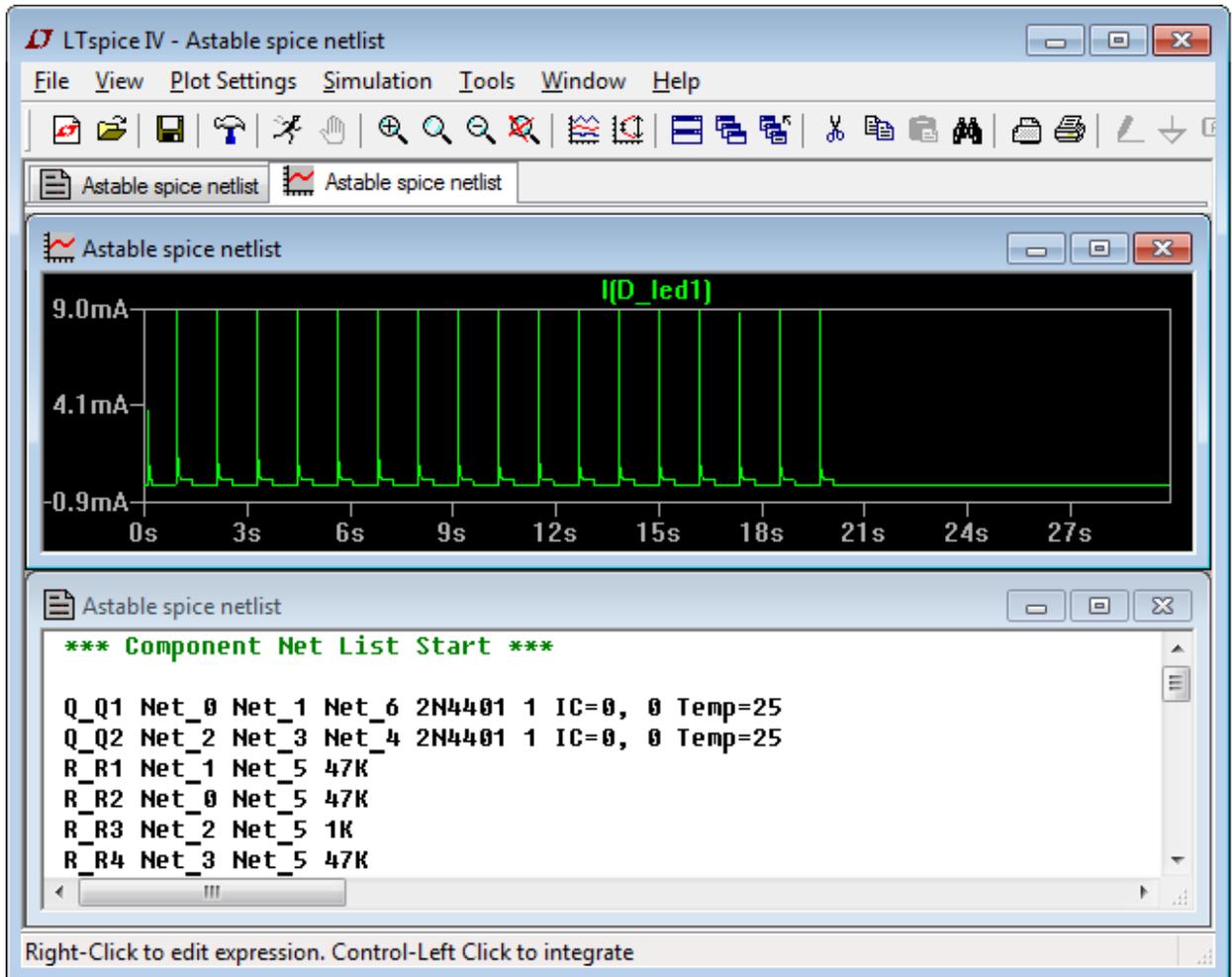
There is no valid spice model for power source (battery) in Astable\_Flip\_Flop\_Spice.dch file, we should define it manually. Right click on the B1 component and select Spice Settings from the submenu. You can see that component has Voltage Source model type, but no valid function. Please select: "Function : PULSE()", specify the following parameters in the table below: "Pulse V2=5", "Pulse PW=20s", "Pulse PER=30s". Keep other parameters and press OK. Now we have the voltage source that produces 5V during the first 20 seconds, then there is a 10-second interval. Now everything is ready for simulation.

### Export Spice netlist



Select "File \ Export \ Spice Netlist" from the main menu. In the small pop-up dialog box select "GND net: GND" (this is a zero point). Specify: "Commands: .TRAN 0s 30s 0.1s" to simulate the circuit from 0s to 30s with 0.1s step. Notice that you can define/change commands directly in the simulation software. Click OK and save the \*.cir file. Launch the Spice simulator that you have. We will use LTspice as an example (download it at the [Linear Technologies](http://www.linear.com) website).

Select "File \ Open" and open the \*.cir netlist you just saved (notice that you should select correct Files of Type). Now you can see the netlist in the text format. Select "Simulate \ Run" and close the error-log window. Select "Plot Settings \ Visible Traces" and choose I(D\_led1) to see electrical current on LED1. This component works during the first 20 seconds then has 10 sec interval. Select other signals to see how they work.



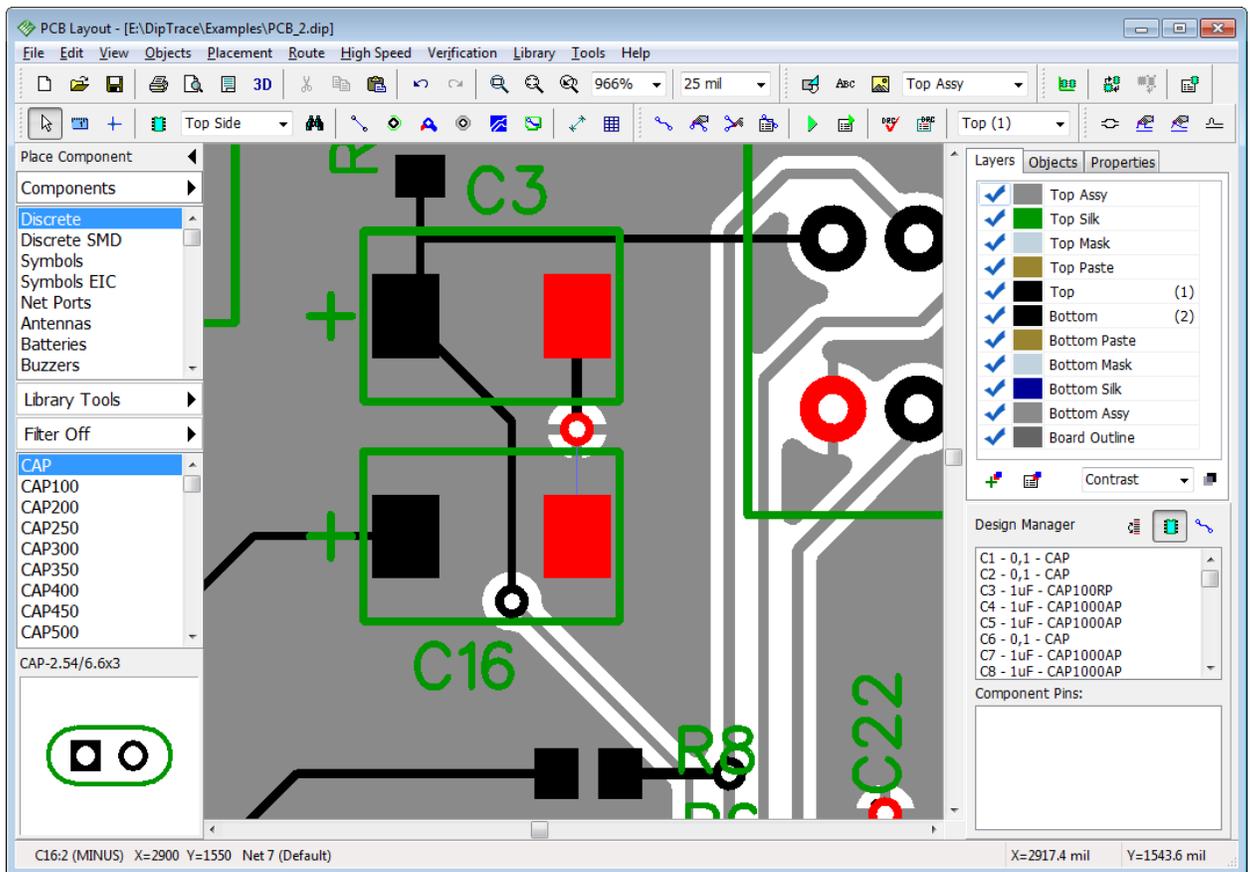
## 5.11 Checking net connectivity

One of the most important features before prototyping is the Net Connectivity verification. It allows the user to check if all nets on the board are connected and reports broken connections and isolated copper areas (not depending on the connection type: traces, thermals, shapes, or copper pours).

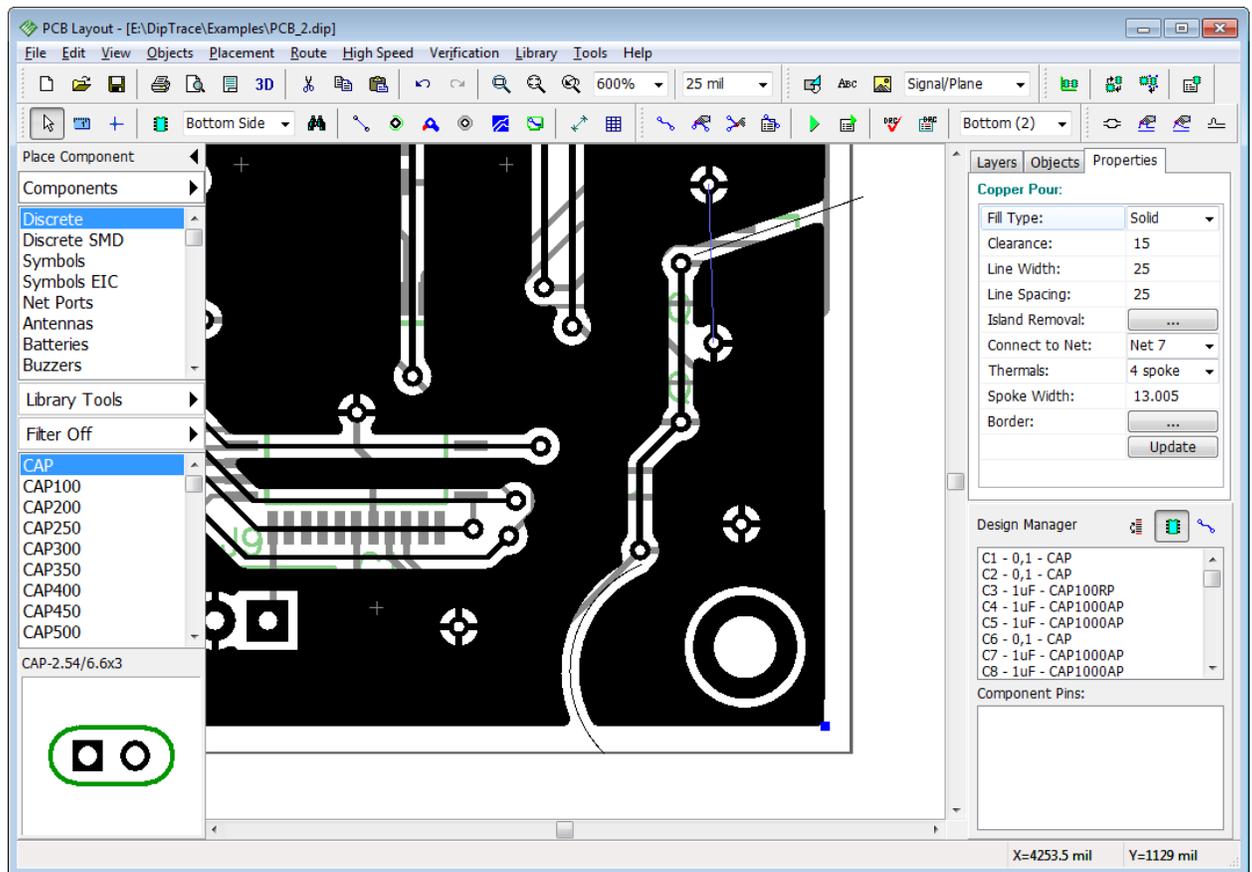
Launch the PCB Layout module and open PCB\_2.dip file from the C:\Users\

You will see a progress bar then "No Errors found" message pops up. Design is correct, therefore we will make some errors intentionally.

Close the error-log dialog box. Select Edit Traces tool on the route toolbar then move mouse to the trace that connects C16:2 to the via and GND copper pour in the Bottom layer, right click on this small trace segment and select Unroute Trace from the submenu. This is going to be the first error.

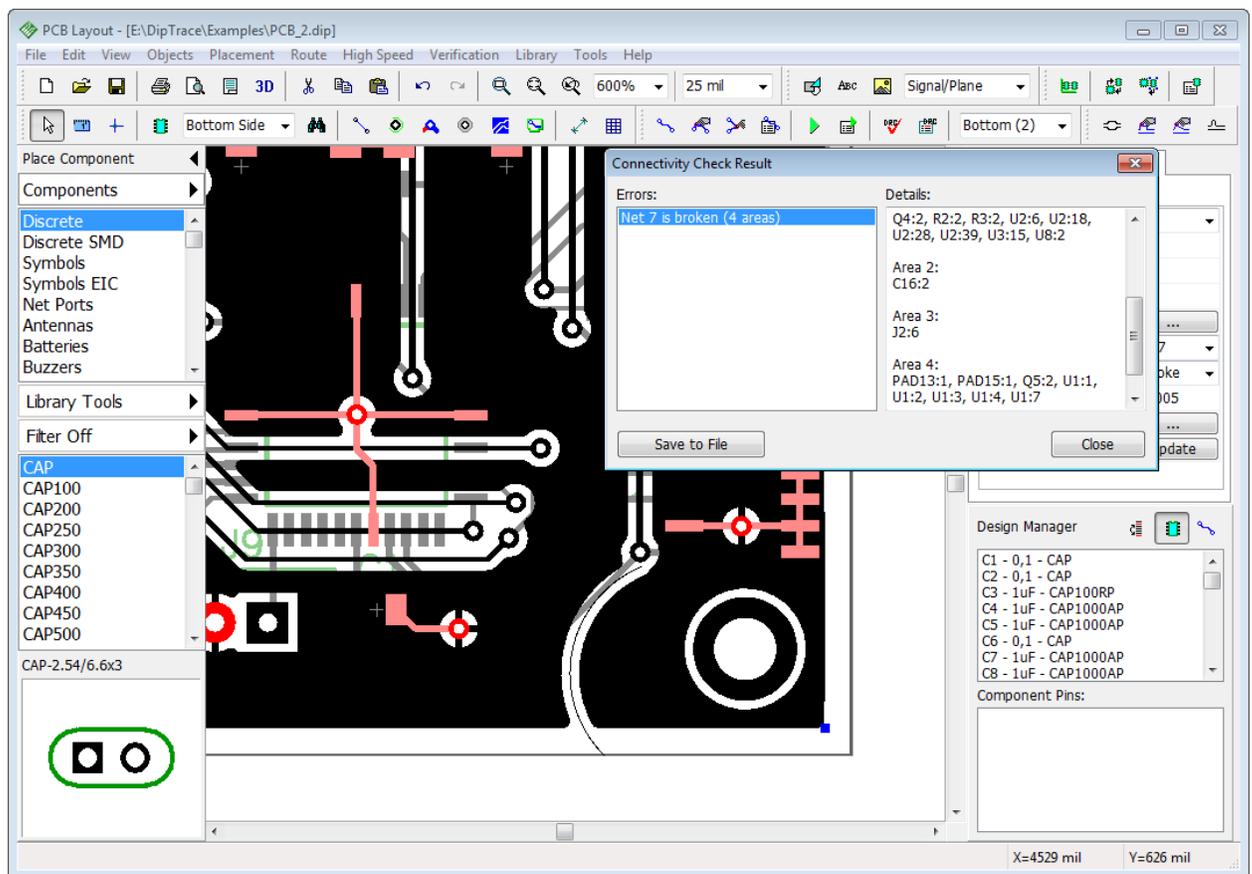


Isolated copper pour area is the second error that we are about to commit. Double click on Bottom layer, pan to bottom-right corner of the design. Place a couple of shapes (arcs or lines) on the signal layer to isolate one of the vias and don't forget to update the copper pour (right click on copper pour's outline, and select Update).



This is a simple situation when a mistake can be visually identified, but most errors of this type (isolated copper pour areas and non-connected pins) go unnoticed on complex projects.

Select "Verification \ Check Net Connectivity", and click OK. You can see connectivity check results. Verification reports that Net 7 is broken to 4 unconnected areas. The first area is the copper pour and all pins connected, the second and the third areas appear as the result of unrouting trace from C16 capacitor (our first error) and the fourth is isolated copper pour at the bottom-right of the board.

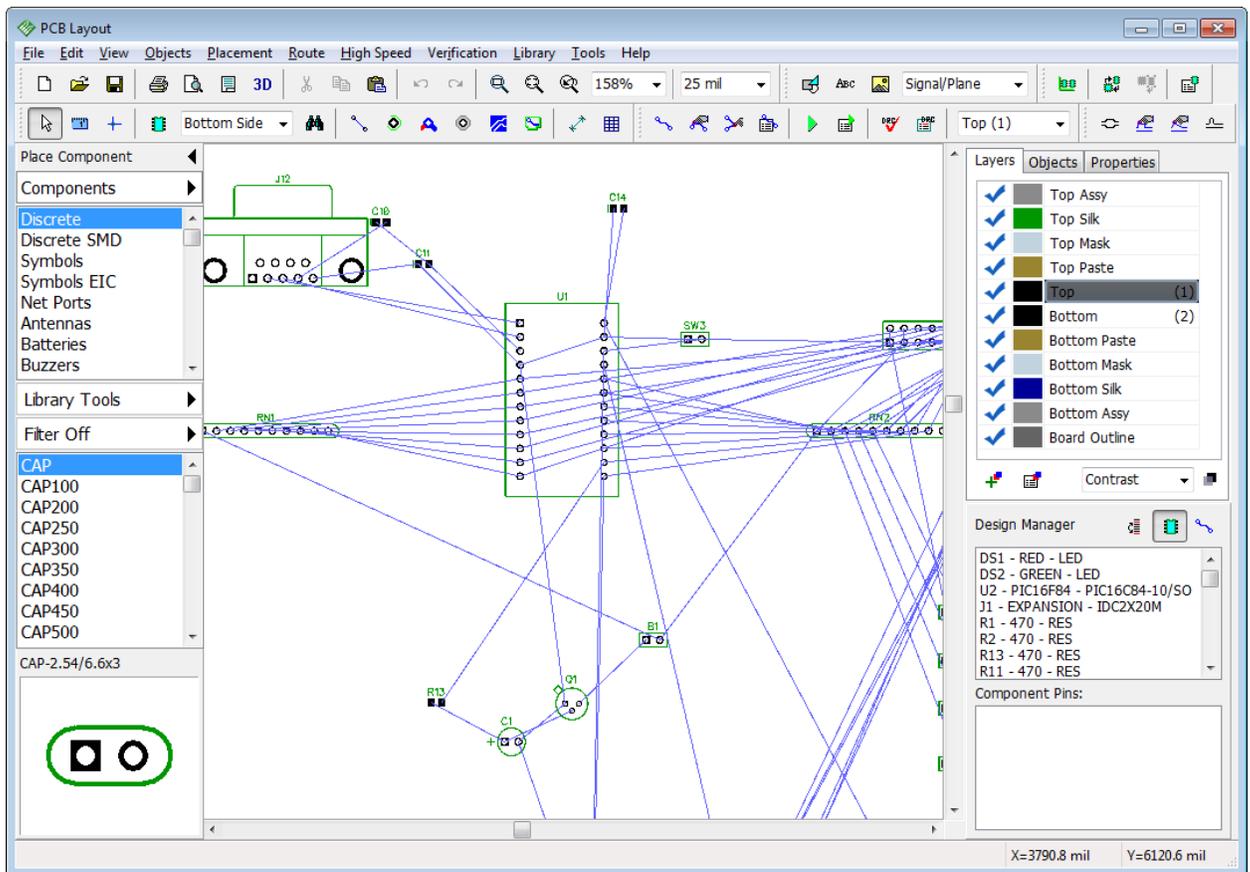


To make further corrections you don't have to close the error-report dialog box. You can save the verification results into the text file.

## 5.12 Placement and Autorouting

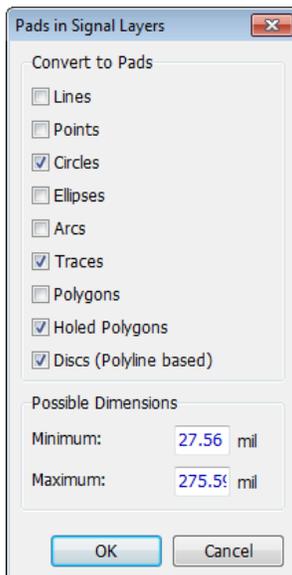
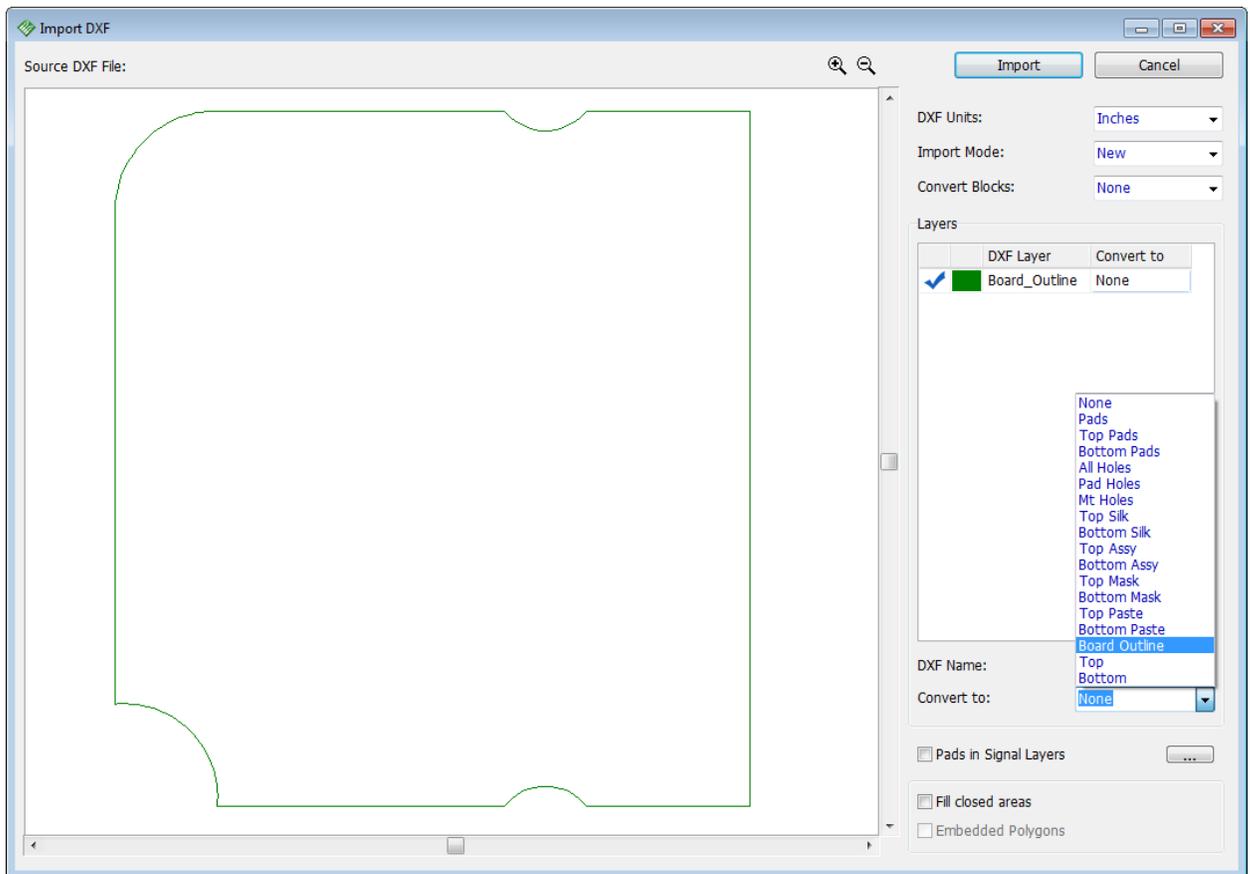
PCB Creator has advanced placement features and integrated auto-placer. This makes placement and layout optimizations much easier.

Launch the PCB Layout module, select "File \ Open", then select C:\Users\



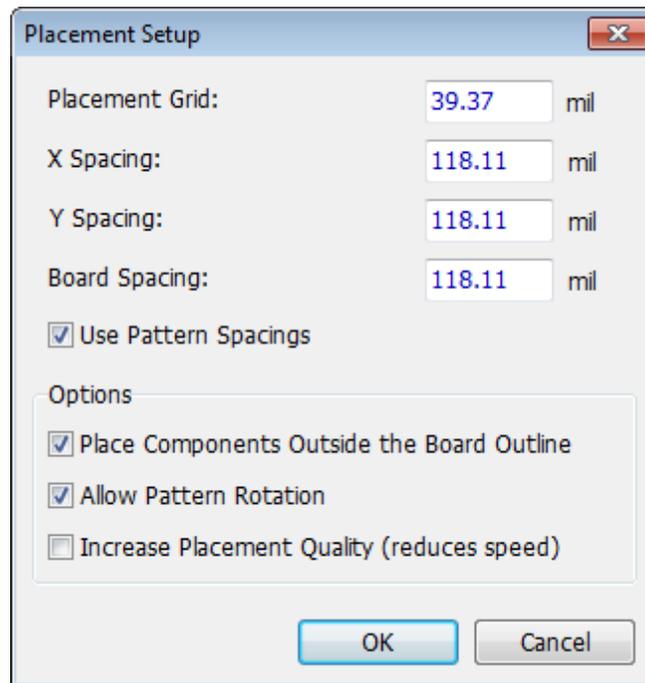
### DXF import

We will import the board outline from the DXF file. Select "File \ Import \ DXF" from the main menu and open C:\Users\

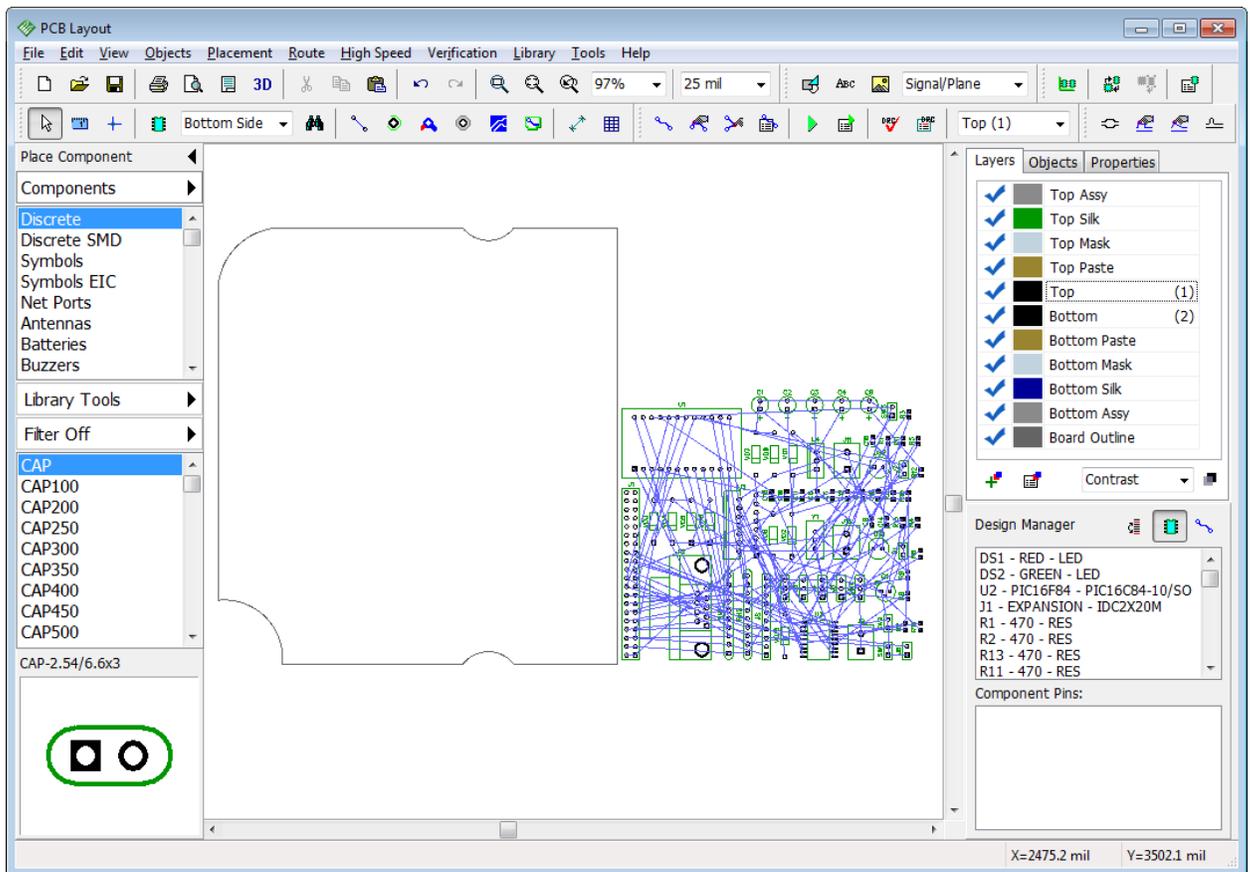


When importing component's drawings or entire layout from DXF, you can Fill closed areas and cut holes using Embedded polygons check boxes (usually DXF designs are made from the outlines without fills). This feature works for copper and mask/paste layers only. If you're importing pads in signal layer, you can check corresponding box and press "... " button to specify what shapes will be automatically converted into pads and their possible dimensions.

Select "Import mode: Add" to add board outline to existing layout, make sure inches are selected and press Import button in the upper-left. Board outline will appear on the design area, but components are still messed. Select "Placement \ Placement Setup" from the main menu:



Check Place Components Outside the Board Outline box to arrange components near the board outline. Keep other settings like on the picture above (notice that values are in mils, you can change units in "View \ Units" from the main menu). Click OK to apply changes and press the Arrange Components button on the placement toolbar or select "Placement \ Arrange Components" from the main menu.



All components are now located in one place near the board outline.

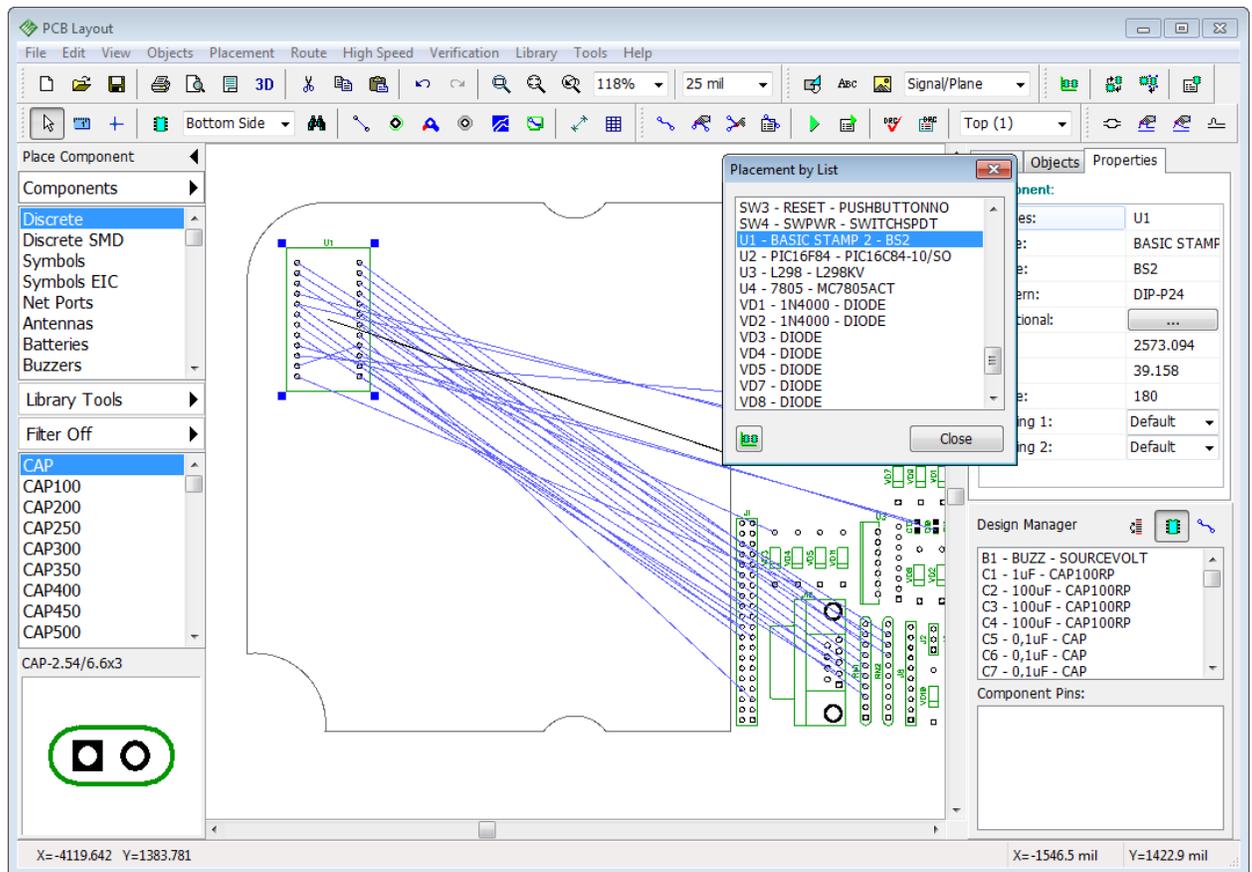
*Notice that Arrange components feature isn't the same as Auto-placement.*

Automatic placement creates a layout with minimum possible total length of connections between the pads of components. Arrange Components simply brings all components to one place and makes it easier to work with them.

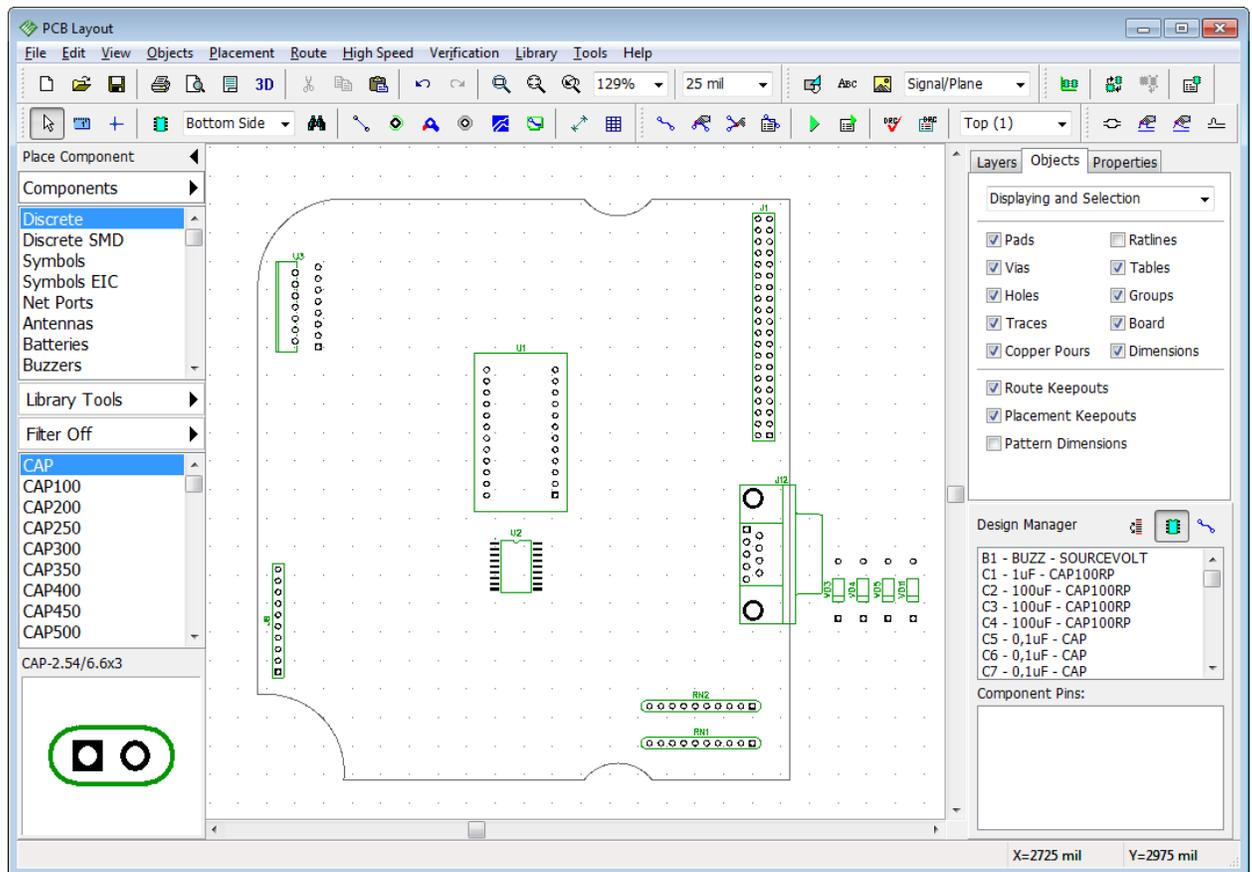
However, in real life manual placement is widely used, because not all components can be placed anywhere on the board. PCB Creator allows the user to combine automatic and manual placement opportunities.

### Placement by list

Select "Placement \ Placement by List" from the main menu, then select component from the list (left click), move mouse to the board outline and click inside the board outline to place selected component.

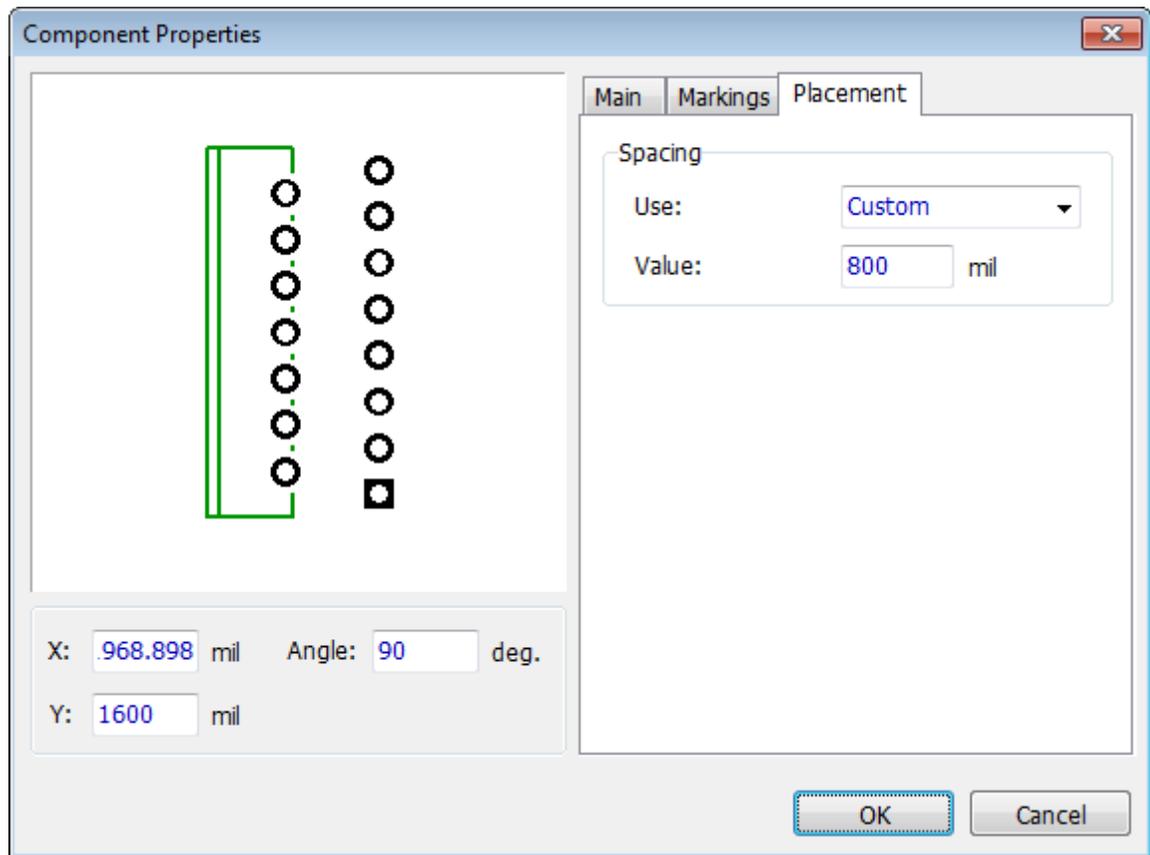


Component disappears from the list after placement (list shows only components outside the board outline). Place U1, U2, U3, J1, J8, J12, RN1, and RN2 components manually, like on the picture below (you can optimize connection lines with F12 hotkey or hide the ratlines, uncheck Ratlines item on the Objects tab in the design manager). Close the Placement by List dialog box when done.



### Custom component clearance

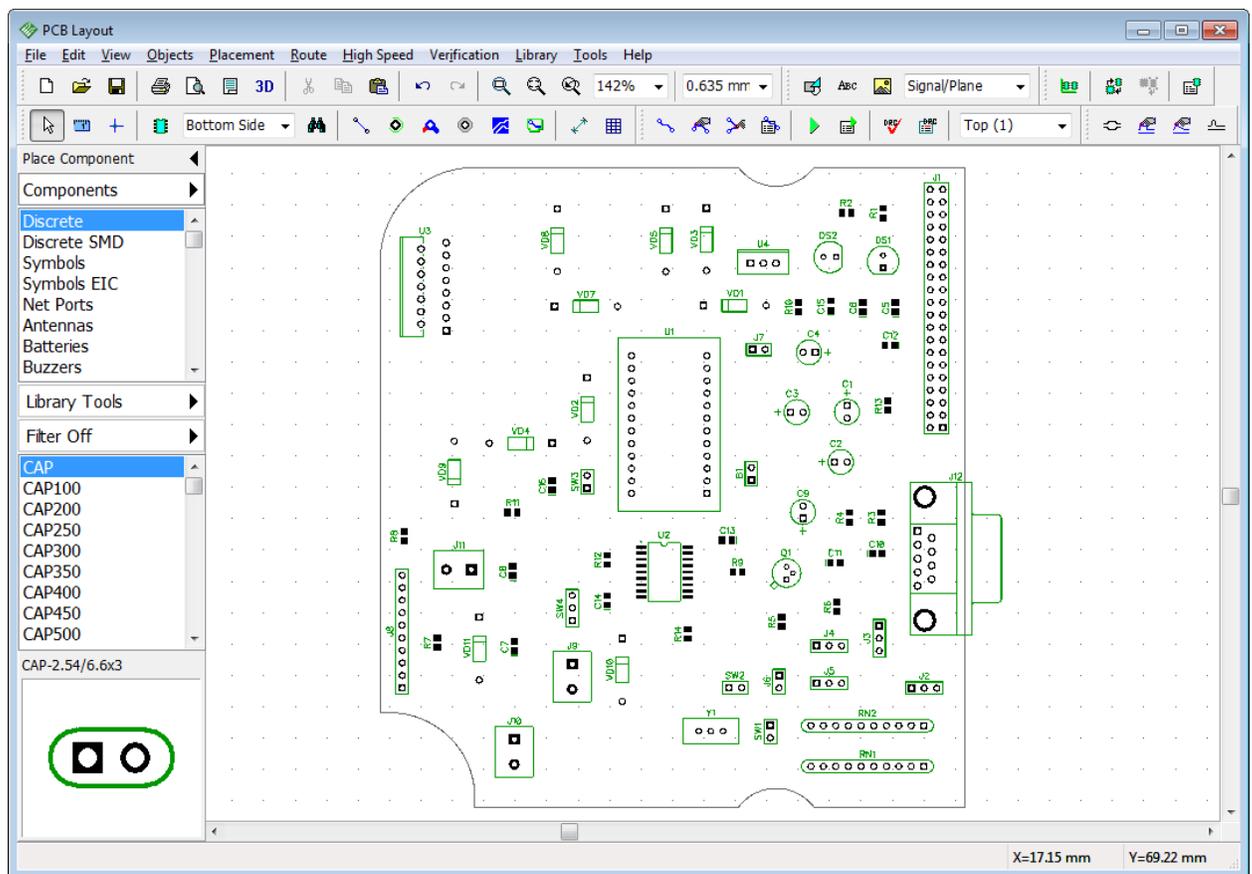
Placed components should have fixed positions, except the U3. Select and lock them ("Ctrl+L"). Now right click on the U3 component and select Properties from the submenu. Go to the Placement tab and define: "Use: Custom" and "Value: 800 mil" (approx. 20 mm). This means that all other components should be located minimum 20 mm (800 mil) away from the U3 component. Click OK and lock this component.



### Auto-Placement

We do not have special requirements for other components, therefore we can place them automatically with 5 mm spacings. Change measurement units ("View \ Units \ mm" from the main menu) Select "Placement \ Placement Setup", change X Spacing and Y Spacing to 5 mm, and set 3 mm board spacing. Make sure that Allow Pattern Rotation option is checked (sometimes it is better to turn it OFF, for example, for single-sided boards with jumper wires). Uncheck Place Components Outside the Board Outline and make sure Use Pattern Spacings item is checked, this will allow the program to use 20 mm custom clearance of the U3 component.

Press OK to apply changes and then press Run Auto-placement button on the placement toolbar or select "Placement \ Run Auto-placement" from the main menu. PCB Creator looks for the best location for each component. You'll get something like on the picture. Notice that design manager is hidden ("Ctrl+2" hotkeys).

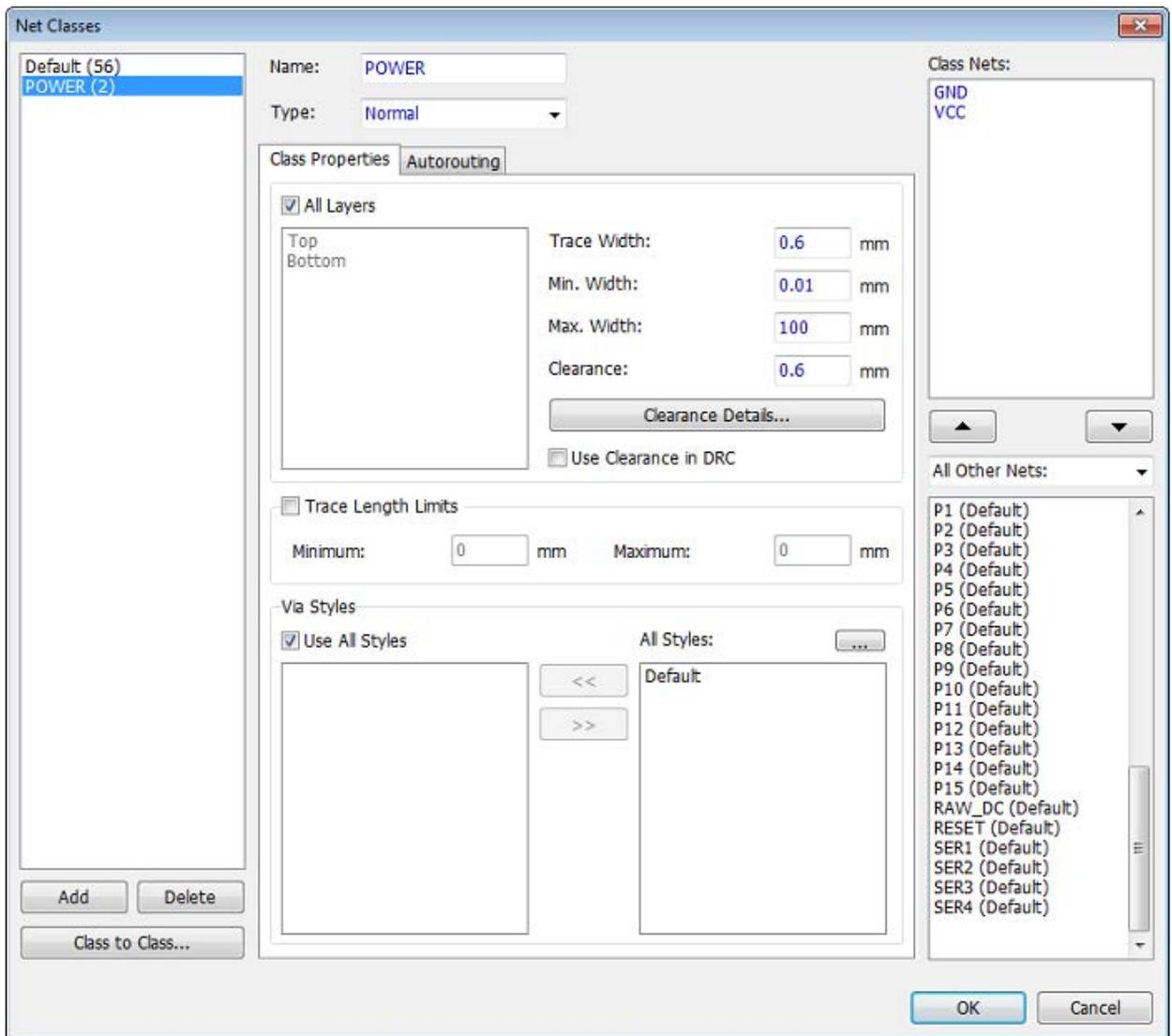


Some connections are not optimal, because we have placed large components manually. If you auto-place the entire board, you can get better results, but usually this is not the option in real life.

It's clearly visible that there is no component on the board closer than 20 mm to the U3, because of the custom clearance.

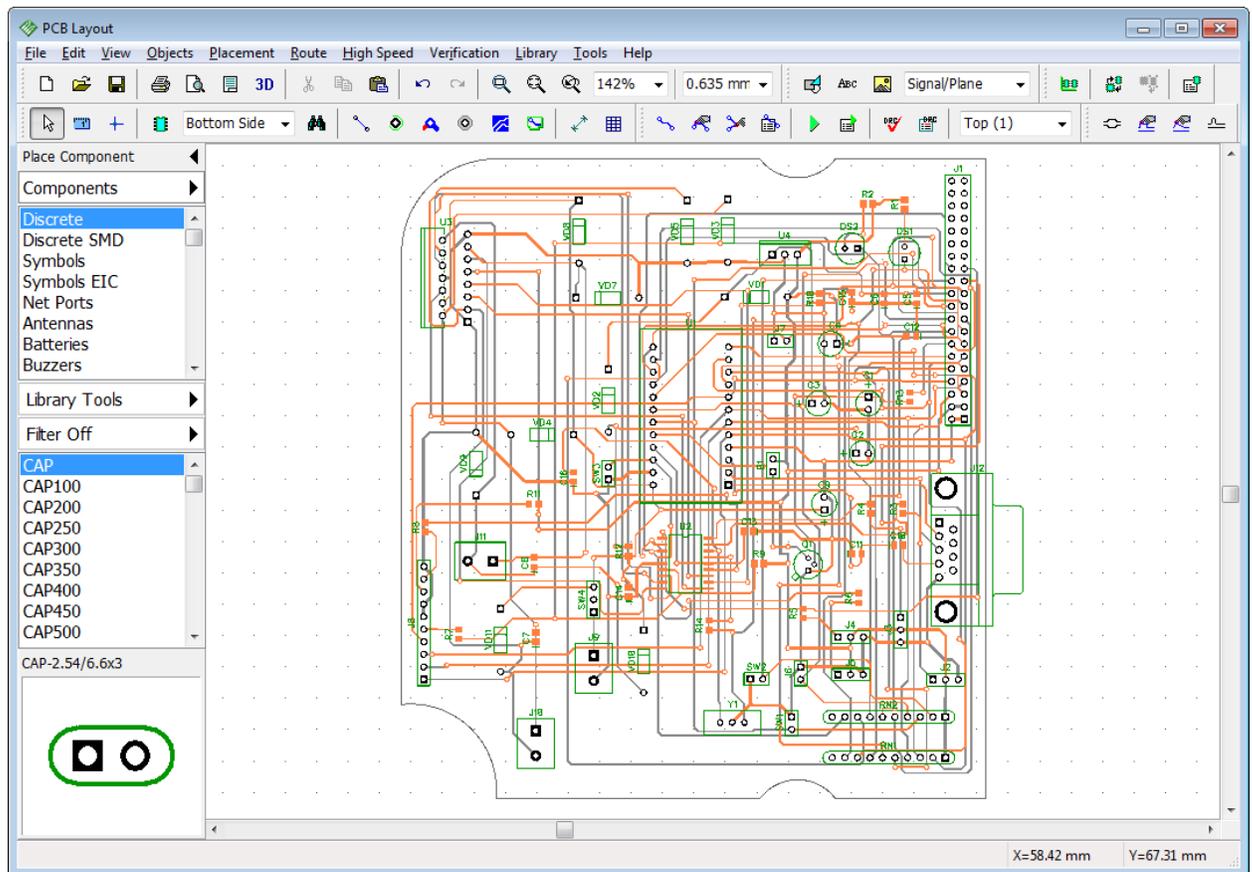
### Autorouting with Net Classes

Check via properties in the "Route \ Via Styles". One via style is enough for this project (we use 1.2 mm via with 0.6 mm hole). Now we need to create a separate net class for Power and Ground nets, because traces of these nets should be a bit wider. Select "Route \ Net Classes" from the main menu. All nets belong to the Default net class. Press Add button to create a new net class then select it from the list and enter its name ("POWER"). Then specify: "Trace Width: 0.6 mm", "Clearance: 0.6 mm". Press Clearance Details button and set "Trace to Pad: 0.5 mm" in the pop-up dialog box. Press OK. Select VCC and GND nets from the list of all nets of the project in the lower-right corner of the dialog box (use "Ctrl" button for multiple selection) and add them to POWER net class.



Now select Default net class and specify the following parameters: "Trace Width: 0.4 mm", "Clearance: 0.4 mm", "Trace to Pad: 0.3 mm". Use all via styles for both net classes (as you know, we have only one via style). Press OK to close the Net Classes dialog box. Make sure that Shape Router is active ("Router \ Current Autorouter") then go to Autorouter setup and uncheck Use Priority Layer Directions in the Settings tab of the Autorouter setup dialog box.

Now press "Ctrl+F9" or the green arrow on the route toolbar to launch the autorouter. In a few second you will get the results. Change layer colors if you want.



Notice that all autorouter settings are described in the Help file for the PCB Layout ("Help \ PCB Layout Help" from the main menu). If you still have some unrouted nets, undo, change trace width/clearance, placement or other settings then launch the autorouter again. However, if you follow the aforementioned instructions you should not get any problems.

## 5.13 Fanout

Fanout allows the user to connect pads of selected component (BGA, SOIC, QUAD) or SMD pads of selected net to inner plane layers automatically with vias of selected style.

Open the PCB Layout module or if it is already open, select "File \ New" from the main menu or press New button on the standard toolbar. Load rules from the \*.rul file we created at the end of ["Saving/Loading Design Rules"](#)<sup>[201]</sup> topic of this tutorial. It should have default, through-hole and blind/buried via styles, custom net class, and two inner plane layers.

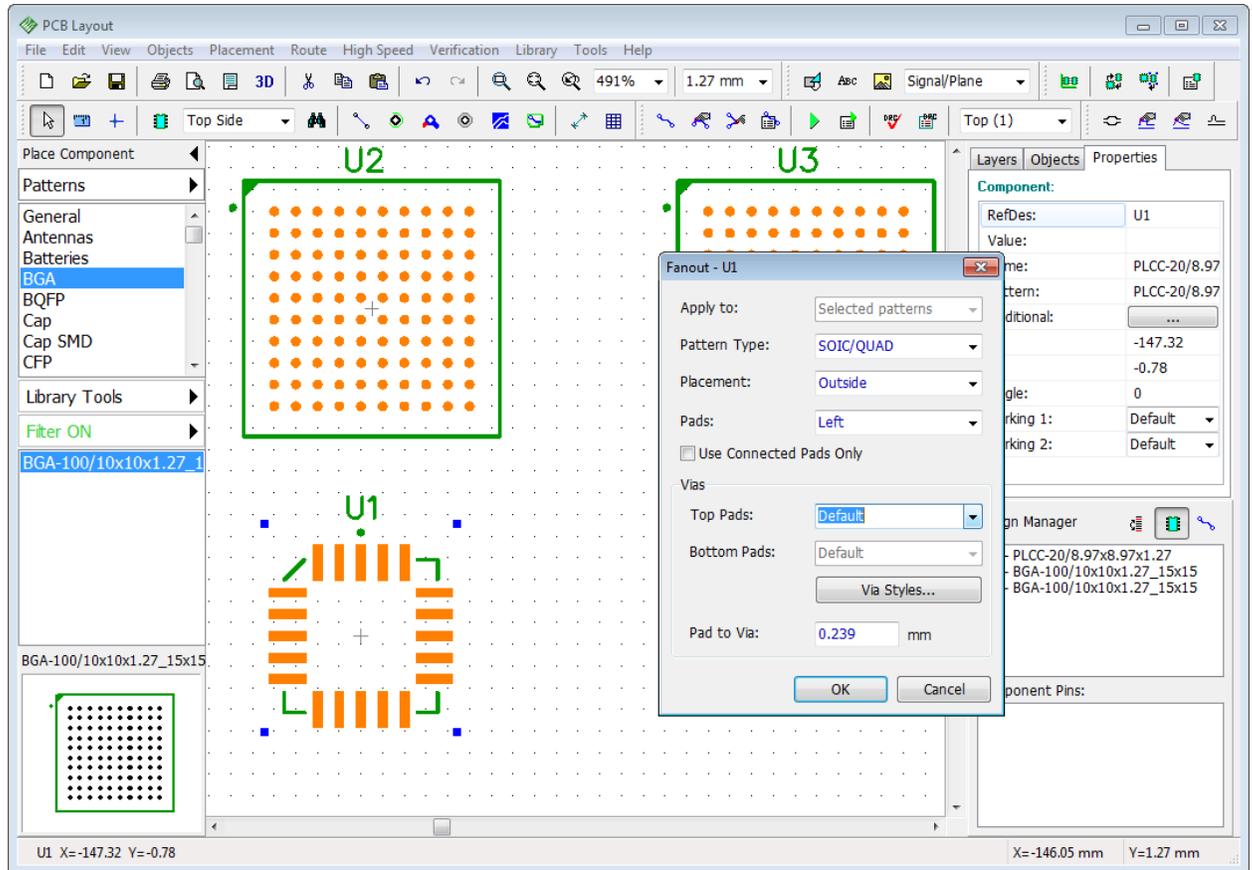
### Fanout a component

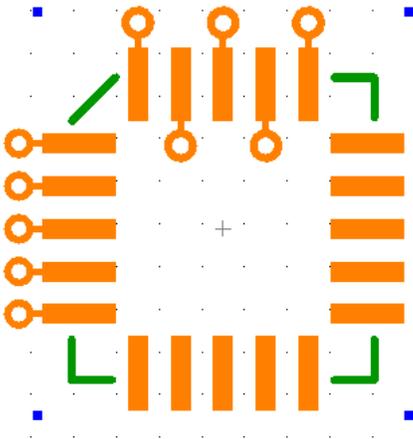
Now select Patterns library group, it contains all patterns available in standard PCB Creator libraries. Notice that these are just patterns without schematic symbols. Place one PLCC-20/8.97x8.97x1.27 from the General library and two BGA-100/10x10x1.27\_15x15 from the BGA library. We will use these patterns for demonstration, but you can select another patterns / components for practicing with fanout.

Right click on the PLCC pattern and select Fanout from the submenu. In the pop-up dialog box

specify: "Pattern Type: SOIC/QUAD", "Placement: Outside", "Pads: Left" (this means that vias will be created only for the left row of the pads) and make sure that Use Connected Pads Only box is unchecked (vias will be created for all pads, not only for connected ones).

Select different Via Styles for the pads on the top and bottom side of the board (inactive if there are no pads on that side). Preview the parameters of existing via styles by pressing Via Styles button. In our case we have three via styles: one with through-hole vias, another with Blind/Buried vias and the Default via style with relatively big vias. Select the one which fits the size of current component (Default via style in our case).





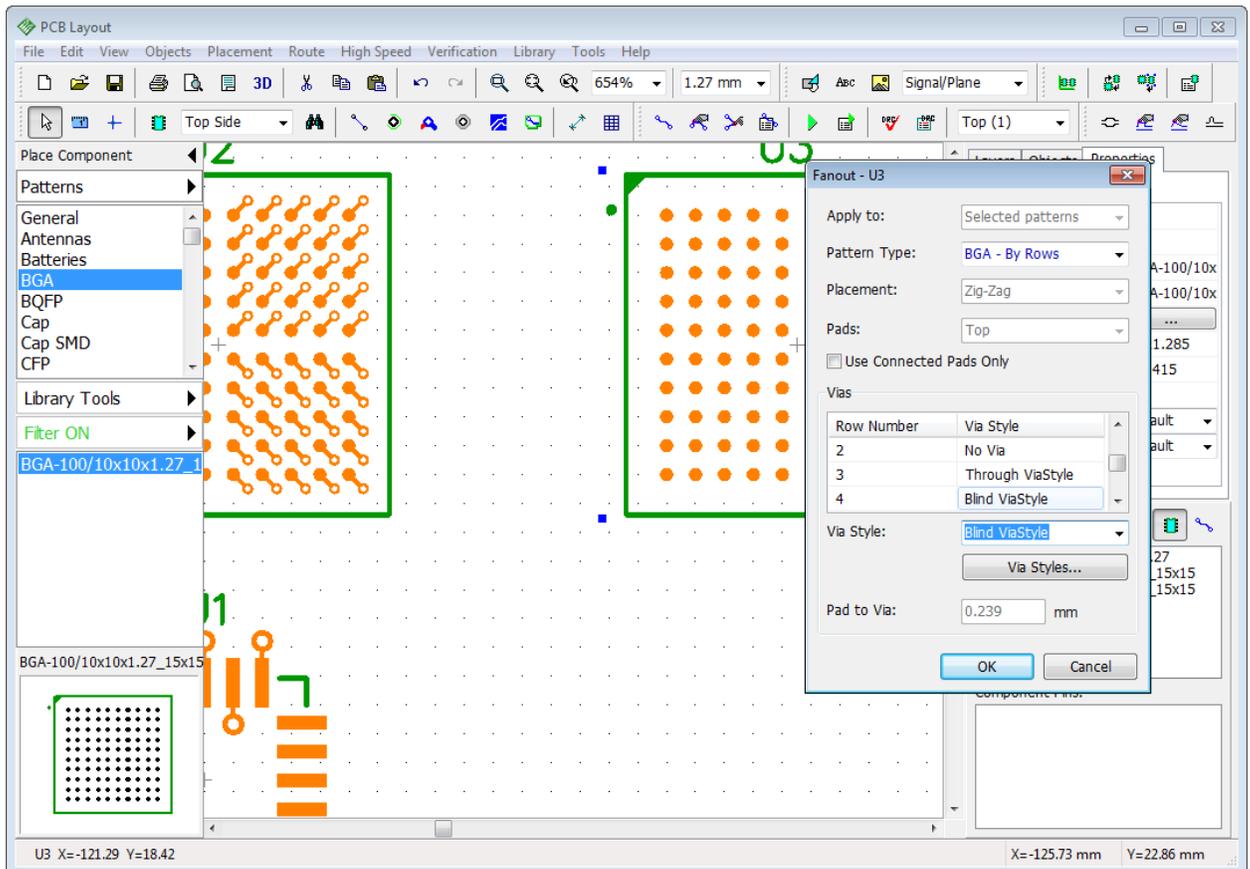
Press OK and vias will appear outside the left pad line of the pattern. Right click on the same pattern and select Fanout again. Now we will place zig-zag vias for the top pads. Select: "Placement: Zig-zag" and "Pads: Top", keep other settings and click OK.

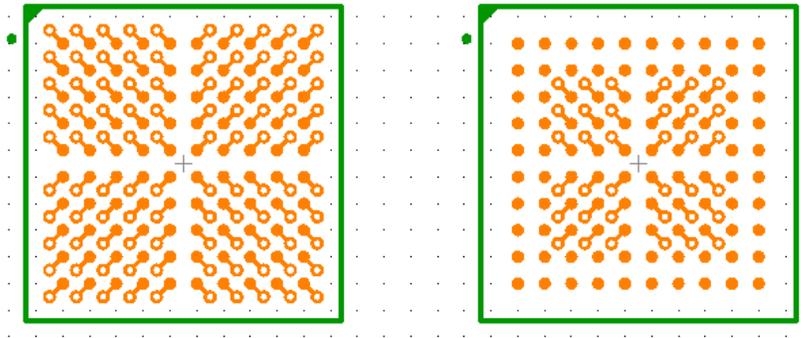
Now we will make through-hole vias for one of the BGA patterns and blind/buried vias for another. Make sure that you have corresponding via styles first. BGAs will need smaller vias (we used 0.5 mm vias with 0.25 mm holes for this example).

Right click on the first BGA pattern and choose Fanout, select "Pattern Type: BGA – All pads" and select

custom via style with through-hole vias. Press OK.

Now select the second BGA package. Right click it and choose Fanout. Select: "Pattern Type: BGA - By rows". This allows the user to apply different via styles to different pad rows of the same pattern or even exclude some rows from the fanout. Left click on the row number and select Via Style from the drop-down list.





We will not create vias for rows #1 and #2 and use different via styles for different rows. Press OK.

We can see, that for the first BGA pattern all pads are connected to vias, for the second one – two rows are

without vias (they are usually connected in the top layer of the board).

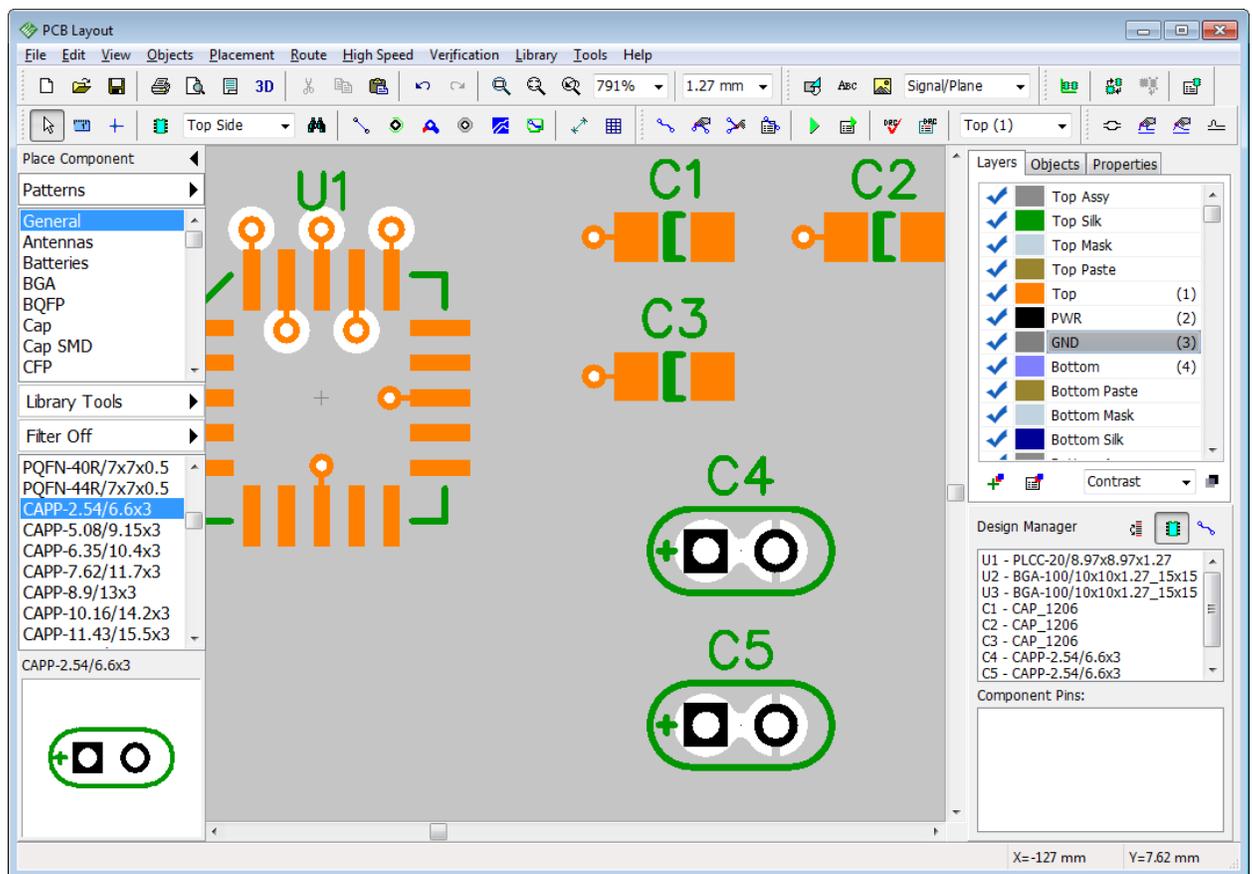
### Fanout a net

We will connect several SMD pads to the GND plane layer using the fanout feature. Place several SMD and a couple of through-hole patterns on the design area. Create a net that connects some pins of these components (we suppose this is our GND net that we should connect to the GND plane layer). Select "Objects \ Place Ratline" from the main menu or press



button on the objects toolbar to create ratlines (connections). Rename net to "GND" if you want. Check the Ratlines item on the design manager Objects tab if you don't see the ratlines on the design area.

Make sure that there is a via style with blind/buried vias from Top to GND layer. Then right click on one of connected pads (not on the pattern) and select Fanout from the submenu. In the pop-up dialog box select appropriate via style and click OK. Now all SMD pads of selected net have vias connecting them to the GND inner plane layer, where we've placed a [copper pour](#)<sup>[68]</sup> connected to the GND net.



## 5.14 Hierarchical Schematic

We will design a very simple two-level hierarchical schematic just to show you how this feature works in Schematic and PCB Layout modules of the PCB Creator.

### Hierarchy blocks

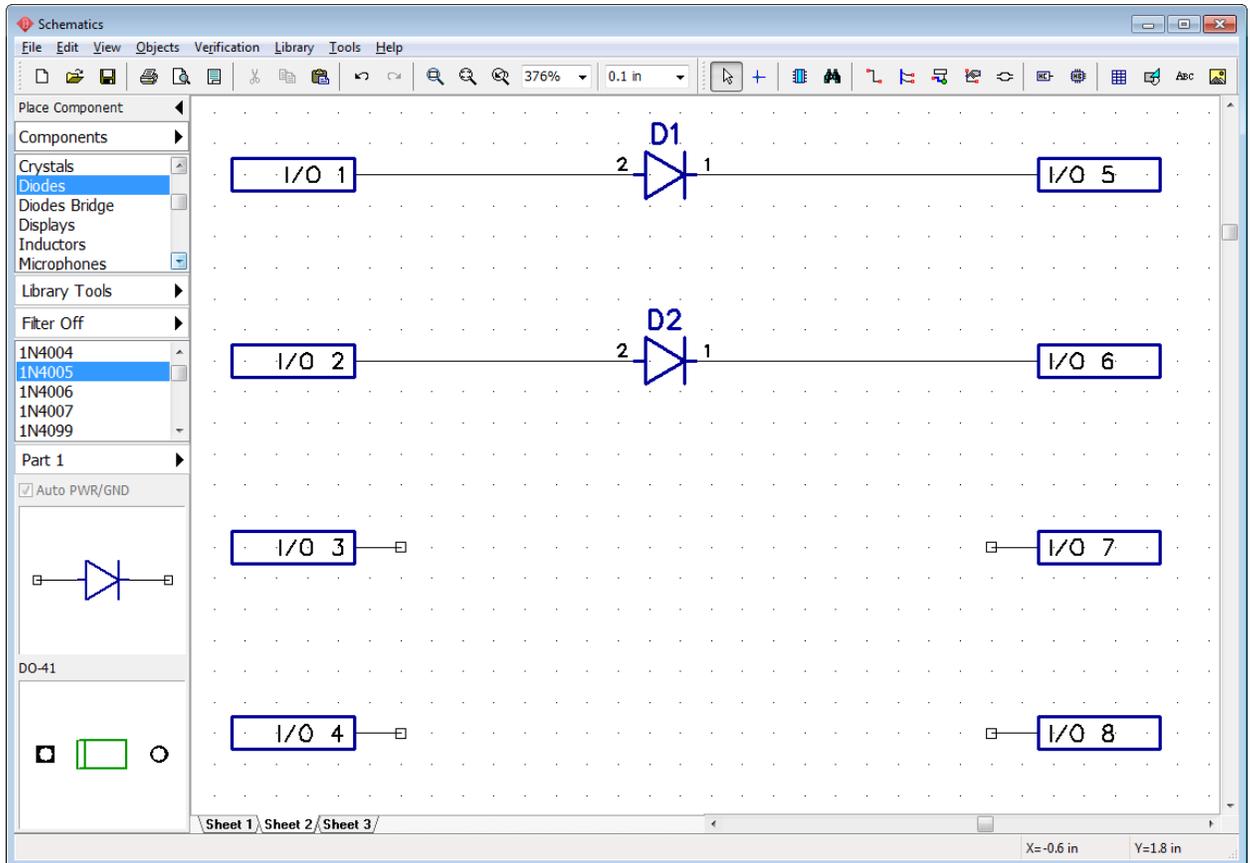
Open Schematic module. Hierarchy blocks are associated with sheets, so first of all we have to add two sheets to the blank schematic, select "Edit \ Add Sheet" from the main menu twice. Then convert additional sheets to hierarchical blocks. Select the second sheet in the bottom-left corner of the design area and go to "Edit \ Sheet Type \ Hierarchy Block" from the main menu. Do the same for the third sheet.

Select the main (first) sheet and place several components there (for example, three UGN3275K components from the Allegro library). This will be our main circuit. It doesn't have any hierarchy blocks yet.

Select the second sheet. Choose "Object \ Hierarchy \ Place Connector" from the main menu or

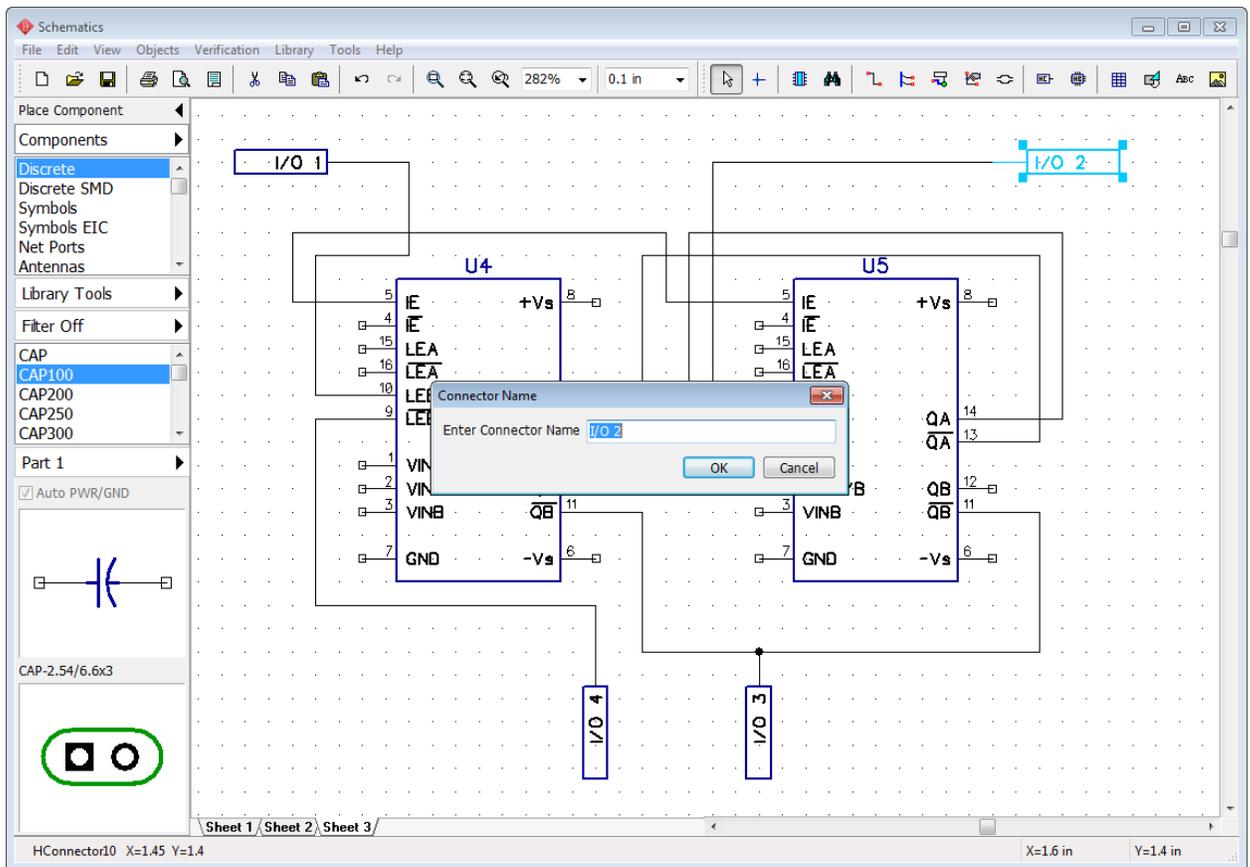
press  button on the objects toolbar. Place several hierarchy connectors to the second sheet (notice that you can not place hierarchy connectors to non-hierarchical sheets). These connectors are the inputs and outputs of the hierarchy block, position and rotation of connectors is the location of hierarchy block pins on the main sheet.

Place 8 connectors, 4 on the left side and 4 on the right. Add two diodes from the Diodes library, connect them to the connectors and leave a free space for the upcoming hierarchy block of the second level. Use "R" hotkey to rotate hierarchy connectors.



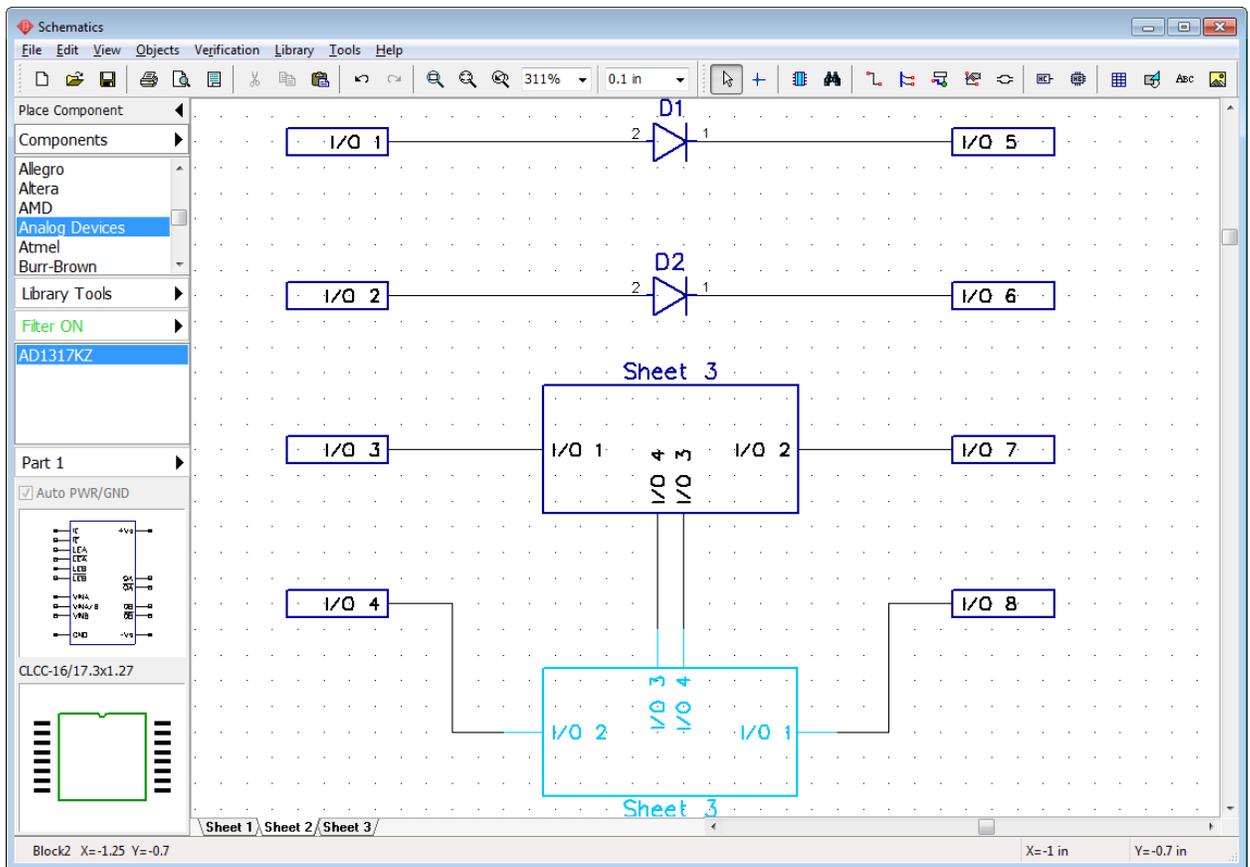
Select Sheet 3 and create the second hierarchy block here. Place several hierarchy connectors (for example two on the sides and two at the bottom), add a couple of components (for example, two AD1317KZ components from Analog Devices library) and connect them.

*Notice that hierarchy connectors can be renamed in the right-click submenu (select the first item from the drop down). Connector name corresponds to pin name of the hierarchy block.*



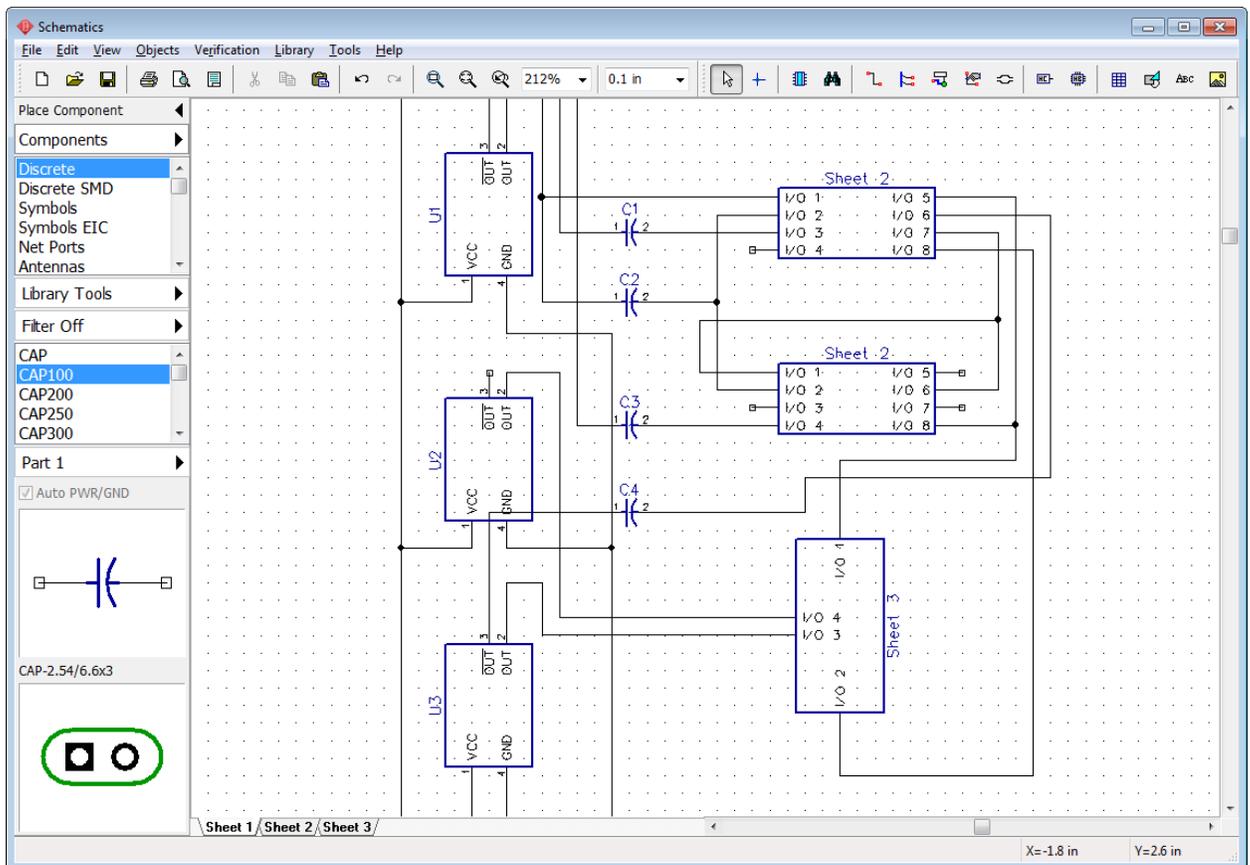
PCB Creator supports multi-level hierarchy, i.e. hierarchy blocks can be inserted into the main circuit and into each other as many times as needed.

Select Sheet 2 then go to "Objects \ Hierarchy \ Place Block" or press  button on the objects toolbar. In the pop-up dialog box with the list of available hierarchy blocks select Sheet 3 and place two blocks (Sheet 3) into the second sheet. Use "R" hotkey to rotate blocks. Notice that you can place Sheet 2 inside itself or make a closed loop of hierarchy blocks, but it is an error. To avoid this situation, use "Verification \ Check Hierarchy" from the main menu. PCB Layout also checks hierarchy for closed loops and displays the warning message when you open schematic with hierarchical errors. We will not make loops right now, just place two Sheet 3 blocks into the Sheet 2 and connect them to connectors like on the picture below.



Hierarchy blocks can be renamed just like regular sheets. Just right click on the corresponding Sheet in the bottom-left of the design area and select Rename from the submenu.

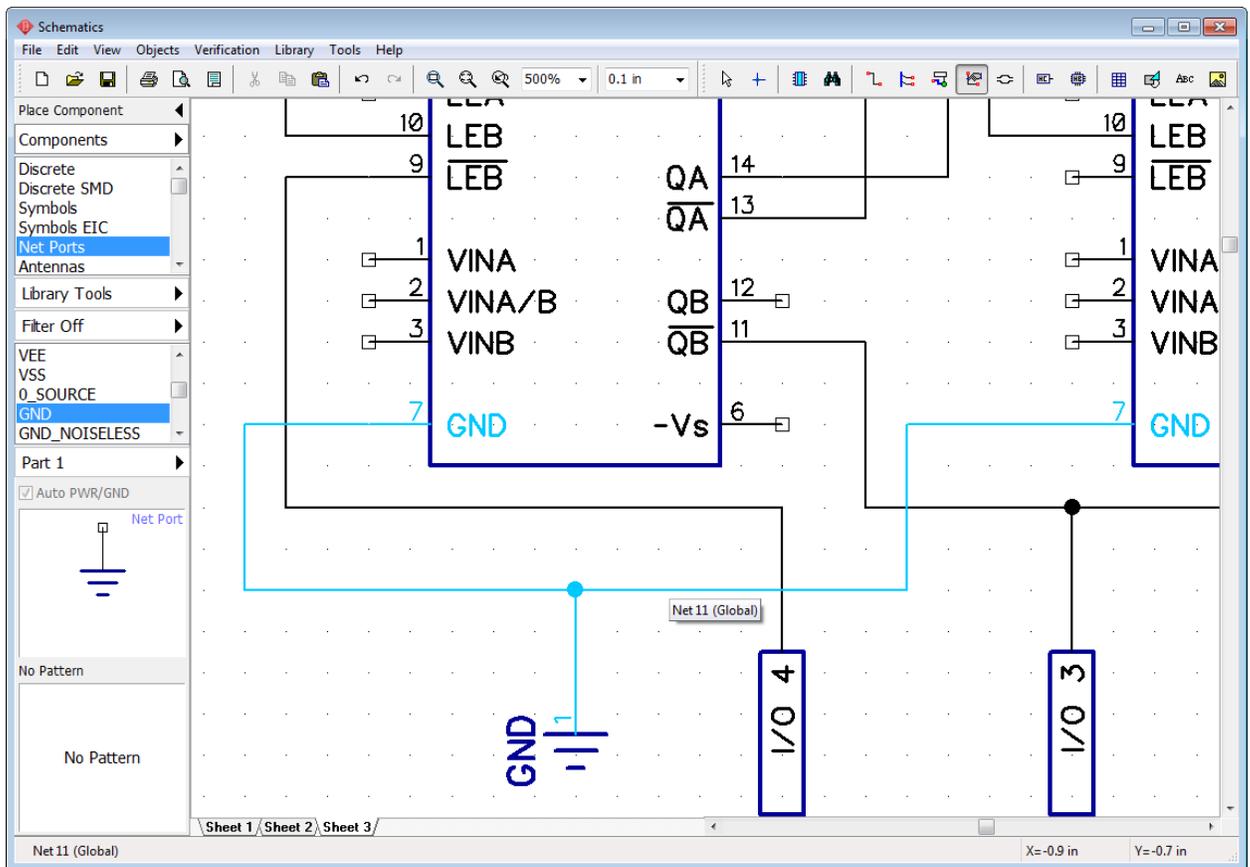
Select main sheet and place hierarchy blocks to the main circuit (for example, add two Sheet 2 blocks and one Sheet 3, like on the picture below). Connect hierarchy blocks with other components of the schematic. Notice that hierarchy blocks are similar to regular components, they have pins and can be easily rotated or moved around. These circuit is not a real-life project, it's just a demo example for this tutorial.



### Global nets

As you already [know](#)<sup>[165]</sup>, pins on different levels of hierarchy **can not** be connected with a single net, unless it is a special-type net called global. Global nets exist on different levels of hierarchy and **do not** depend on hierarchical structure of the schematic.

Return to Sheet 3 and place a ground (GND) net port from the Net Ports library. Then connect it to GND pins of the U4 and U5 components. Notice that net has automatically become global.



Select Sheet 1 (the main circuit) and place GND net port in there, then connect it (create a wire from the net port to some free GND pin/s). You'll notice that this net has also become Net 11 (Global). Now we have a single global net on two hierarchical levels. We can continue this net into Sheet 1 etc. Rename it to "GND".

*Notice that the same net ports anywhere on the circuit are automatically connected into a single net (Global, – if in hierarchy).*

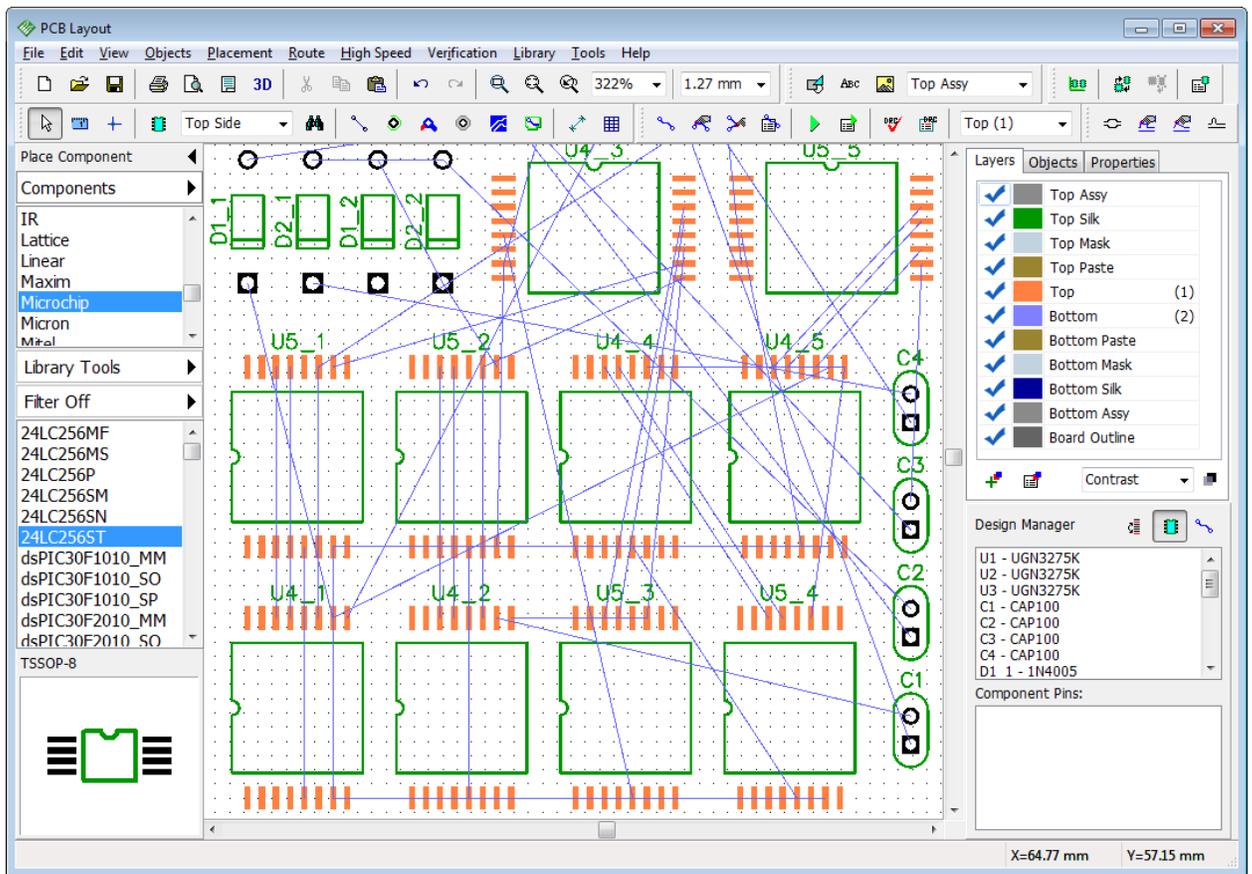
You already know how to [connect nets by name](#)<sup>[165]</sup>, creating global nets does not differ a lot. Right click on a random net and select Properties from the submenu. In the pop-up dialog box check Global Net for Hierarchy and Connect Nets by Name boxes. Type in the name of the global net that already exists and press OK.

### Hierarchy in the PCB Layout

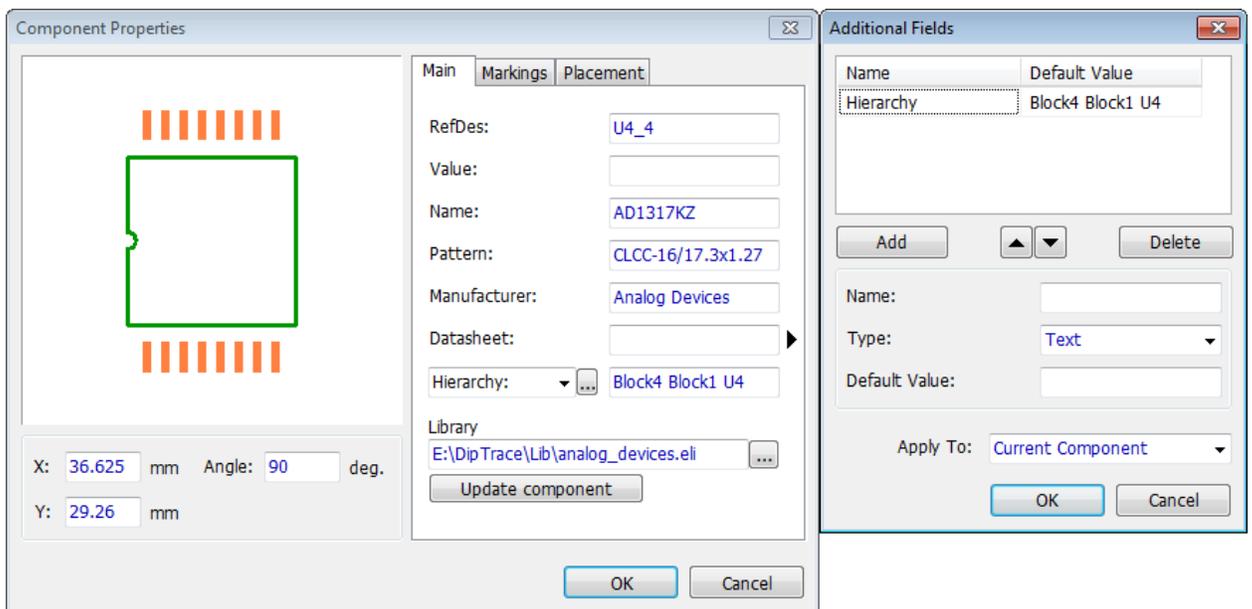
Convert this hierarchical schematic to the PCB. Press "Ctrl+B" hotkeys and select Use Schematic Rules. In the PCB Layout module components that were in the hierarchy blocks are superimposing each other, [arrange](#)<sup>[209]</sup> them (  button on the placement toolbar).

*Notice that all components have the same reference designators as in the Schematic + hierarchy block index.*

Use "View \ Component Markings \ Main \ RefDes" to display designators if they are hidden.



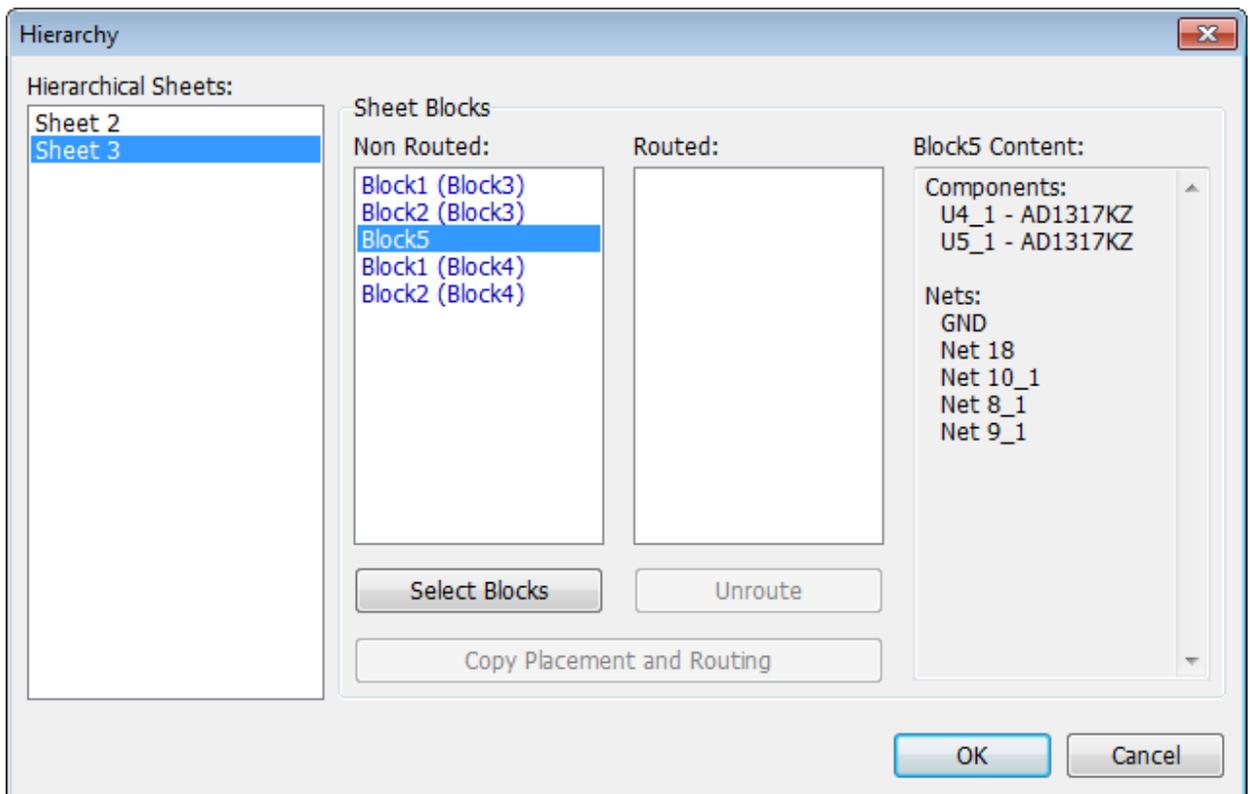
Right click on one of the components that were in hierarchy block and select Properties from the submenu. Notice that component involved in hierarchy has additional field with each hierarchy block RefDes and component RefDes (path). This additional field is used while updating PCB by RefDes ("File \ Renew Design from Schematic" from the main menu).



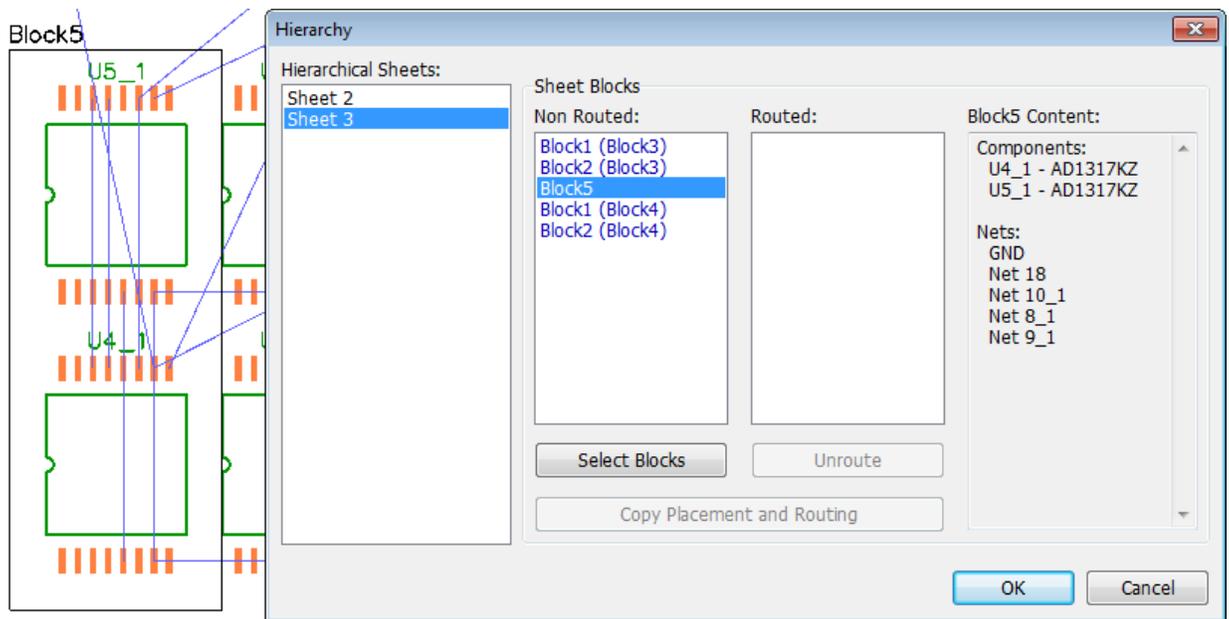
PCB Creator works with hierarchy circuits in the PCB Layout module, components can be automatically **arranged by hierarchy blocks, routing from one block can be applied to another similar block** in a blink of an eye. On the printed circuit board all components, regardless of their hierarchy level are on the same surface layer.

Select "Route \ Hierarchy" from the main menu. There are two hierarchical sheets available (exactly like hierarchy sheets in the Schematic). Select Sheet 2 and you will see two actual hierarchy blocks (because Sheet 2 was inserted two times into the main circuit in the Schematic). Select Sheet 3 and you will see five blocks inside it (because Sheet 3 was inserted two times into each Sheet 2 hierarchy block and inserted once directly into the main circuit). Notice that the name of the block of higher hierarchical level is listed in the brackets. When you select the block from the list, you can see components and nets that belong to this block to the right.

None of the blocks is routed at the time.



First of all we need to arrange components by blocks on the board. Select block in the list and its components will be highlighted on the design area. Press Select Block button to arrange components by selected blocks.

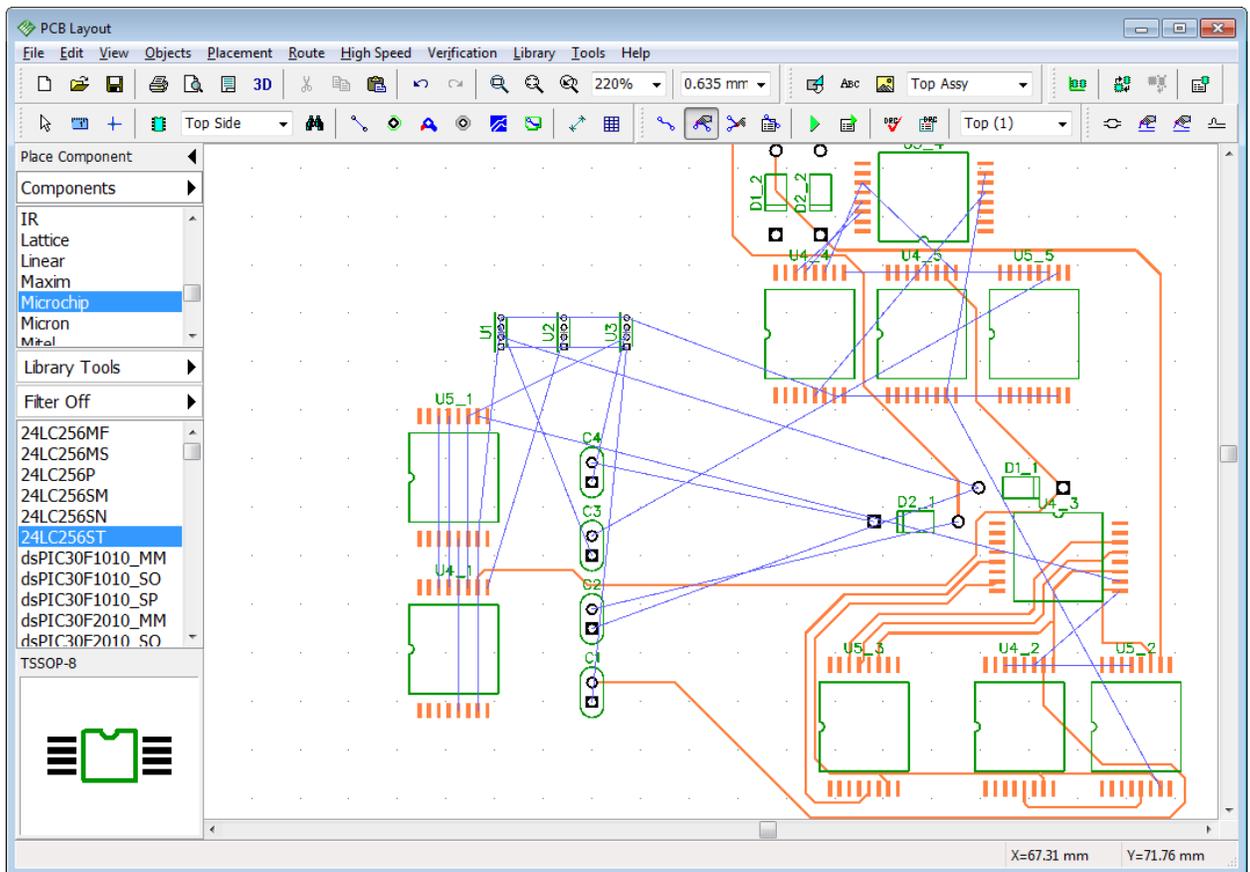


We will arrange components of two hierarchy blocks on the board. Select Block3 and Block4 from the Sheet 2 (use "Ctrl" hotkey to select two blocks at a time). Press Select Blocks button and then press OK to close this dialog box and apply arrangement. Now two blocks of the components are clearly visible on the design area. We will work with Block 3, which is right below the Block 4.

Change the layout of the components and then route traces in automatic mode. But first, **make sure GND net (or any global net) is unrouted.**

*Notice that global nets should be routed only when layout is arranged and routed by blocks.*

We exclude GND net from the autorouting. Right click on any pad that belongs to the GND net, select Net Properties from the submenu and in the pop-up dialog box specify "Auto-route Mode: Don't Route". Press OK. Now select other components, right click on the component pad and select Route Net from the submenu or right click on the component and select Route Traces (rectangular board outline appears). Edit traces manually if needed. Notice that we didn't route the entire board. We have mostly routed the traces inside the Block 3, and some connections to Block 4 and to the components from the main schematic sheet.

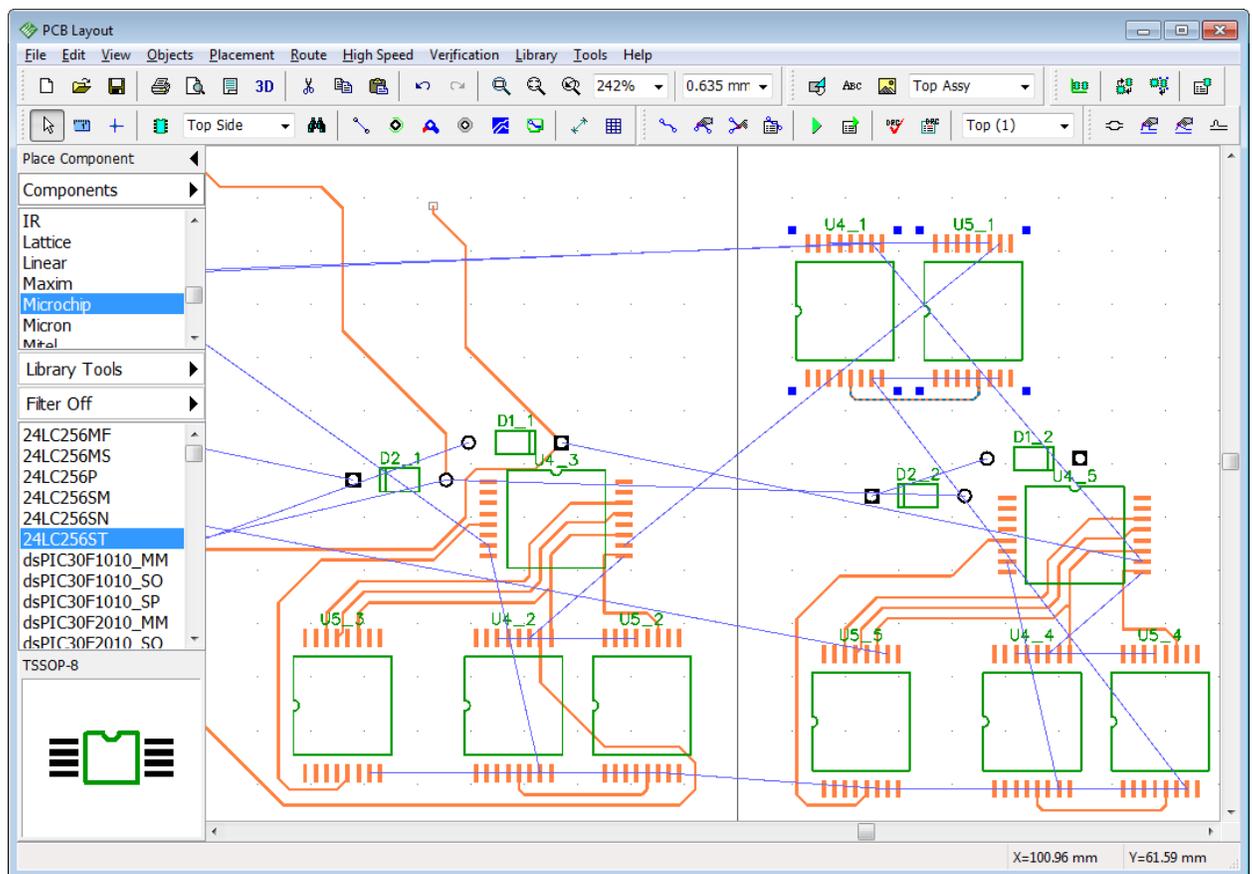


Go to "Route \ Hierarchy" from the main menu, select Sheet 2 again. This time Block 3 is routed. Select Block 3 from the Routed list and Block 4 from the Non Routed list, then press Copy Placement and Routing button to apply Block 3 placement and routing to the Block 4.

If you have routed a lot of traces from Block 3 to Block 4, the Block 4 can appear in the Routed list on the hierarchy dialog box. Just select it, press Unroute, then apply routing and placement.

Let's also apply placement and routing to Block 5 without closing the hierarchy dialog box. Select Block 5 on the Sheet 3 and one of the routed blocks, then press Copy Placement and Routing button again. Now press OK to apply routing.

Routed blocks are located next to the board outline, use box selection to drag them to a correct position and finish the project. Notice that PCB Creator didn't copy traces heading outside selected hierarchical block.

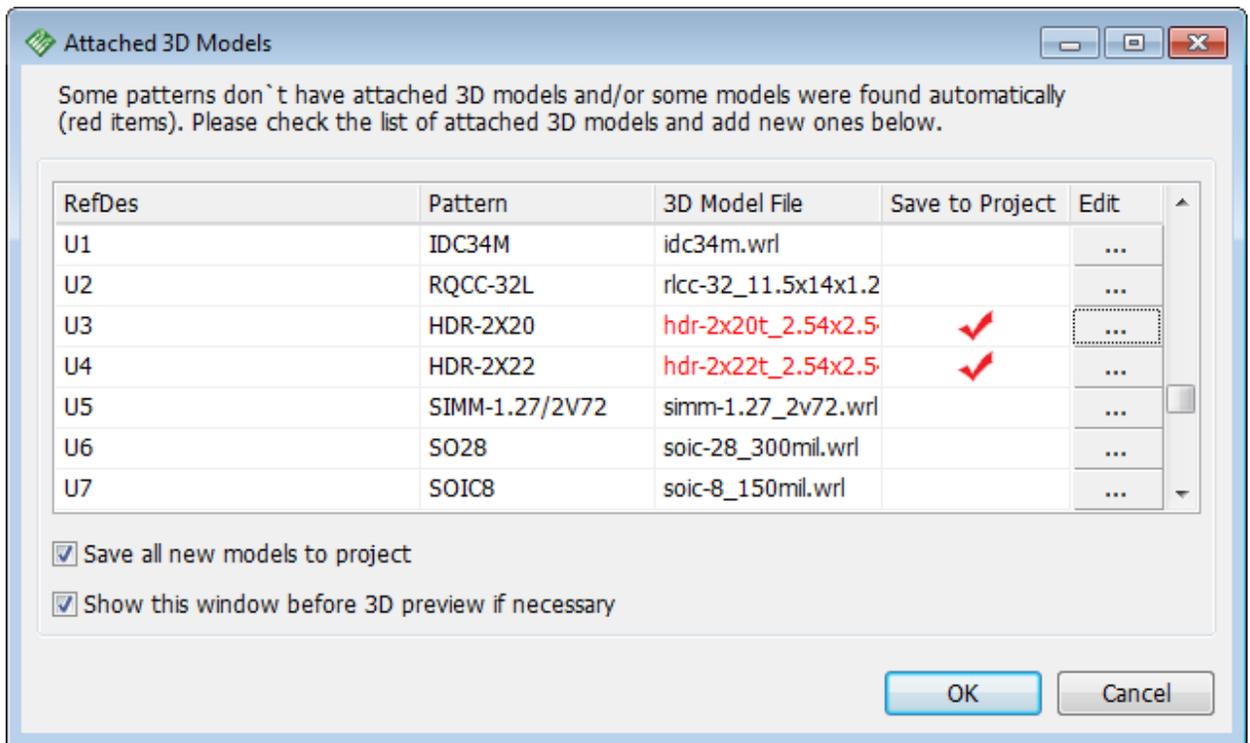


## 5.15 3D Preview and Export

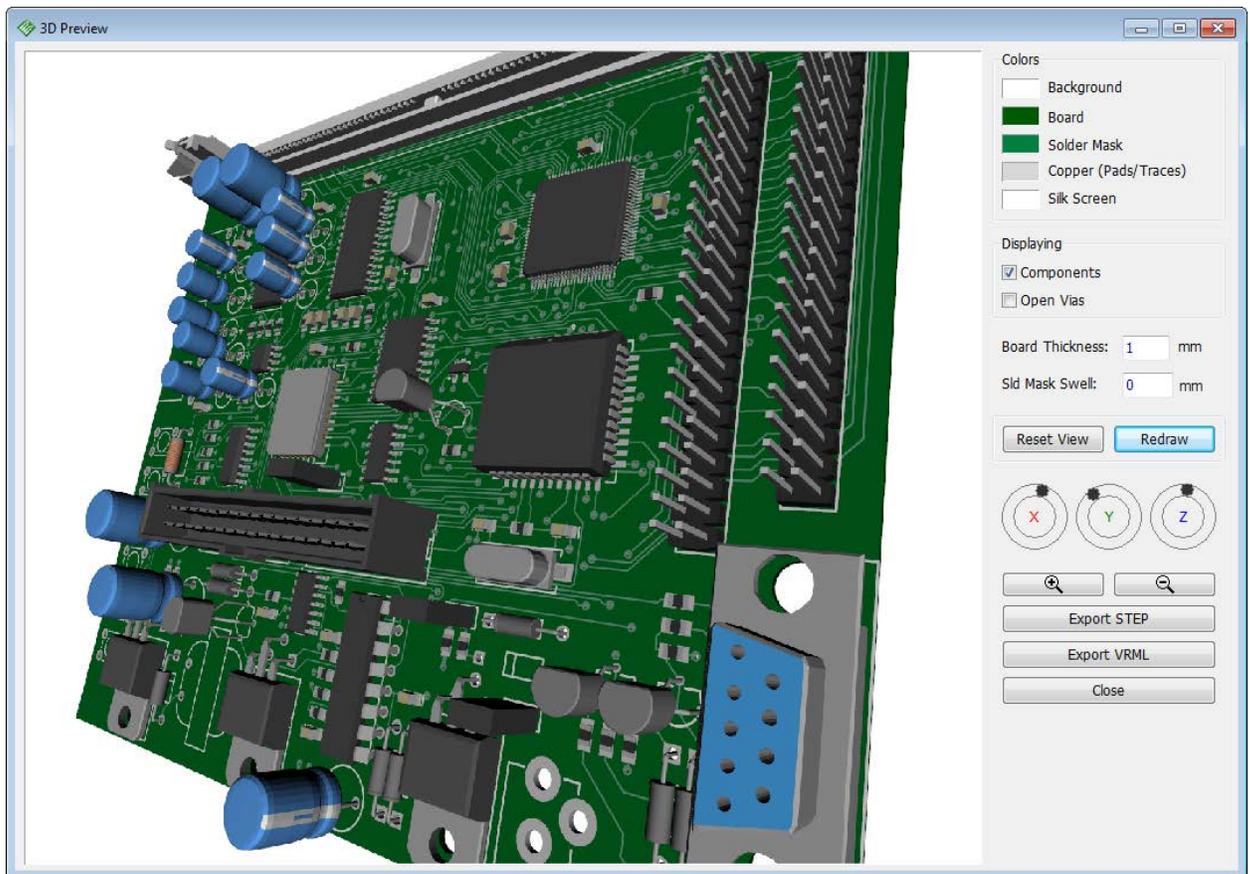
PCB Creator features built-in real-time 3D visualization with STEP and VRML export. This tool allows the user to visually check the circuit board with all components installed and export PCB to mechanical CAD programs for developing the device housing etc.

**Download and install 3D model libraries** from the Bay Area Circuits [website](#). Components without 3D models will be appear only as footprints on the circuit board in the 3D mode.

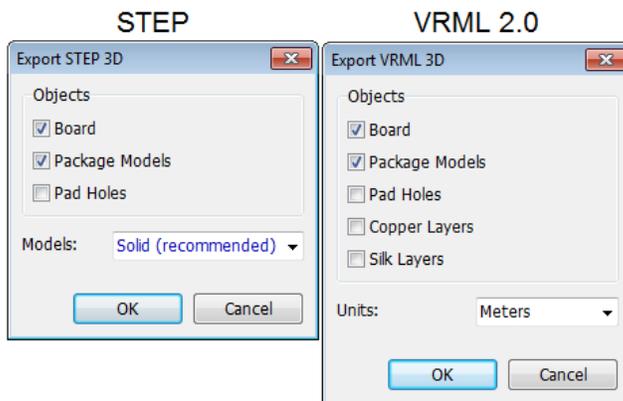
Go to "File \ Open" (or press "Ctrl+O") and select C:\Users\



Press OK to see a 3D model of the board. You can rotate PCB in three axes, move it with the mouse, zoom in and out with the mouse wheel. Colors of the background, board, solder mask, as well as component and via display options can be changed (press Redraw button). Press corresponding buttons to export board model to STEP and VRML formats which are standard for mechanical CAD systems.



### 3D export



PCB Creator 3D Preview module allows the user to export 3D model of the board to STEP (\*.step) and VRML 2.0 formats (\*.wrl) supported by most mechanical CAD software.

When in the PCB Creator 3D Preview & Export module, press Export STEP or Export VRML button. In the pop-up dialog box specify the export objects, model export mode, or export units.

Press OK to specify the filename and the

folder. We recommend to export project as a solid body in STEP format.

*Notice that exporting holes dramatically slows the process.*

Press Export VRML button, select which objects will be exported (Board, Package Models, Pad Holes, Copper Layers, Silk Layers) units and file location. Export complex boards with a lot of holes can be slow.

Step and VRML files can be opened in various Mechanical CAD software.

### Map 3D model

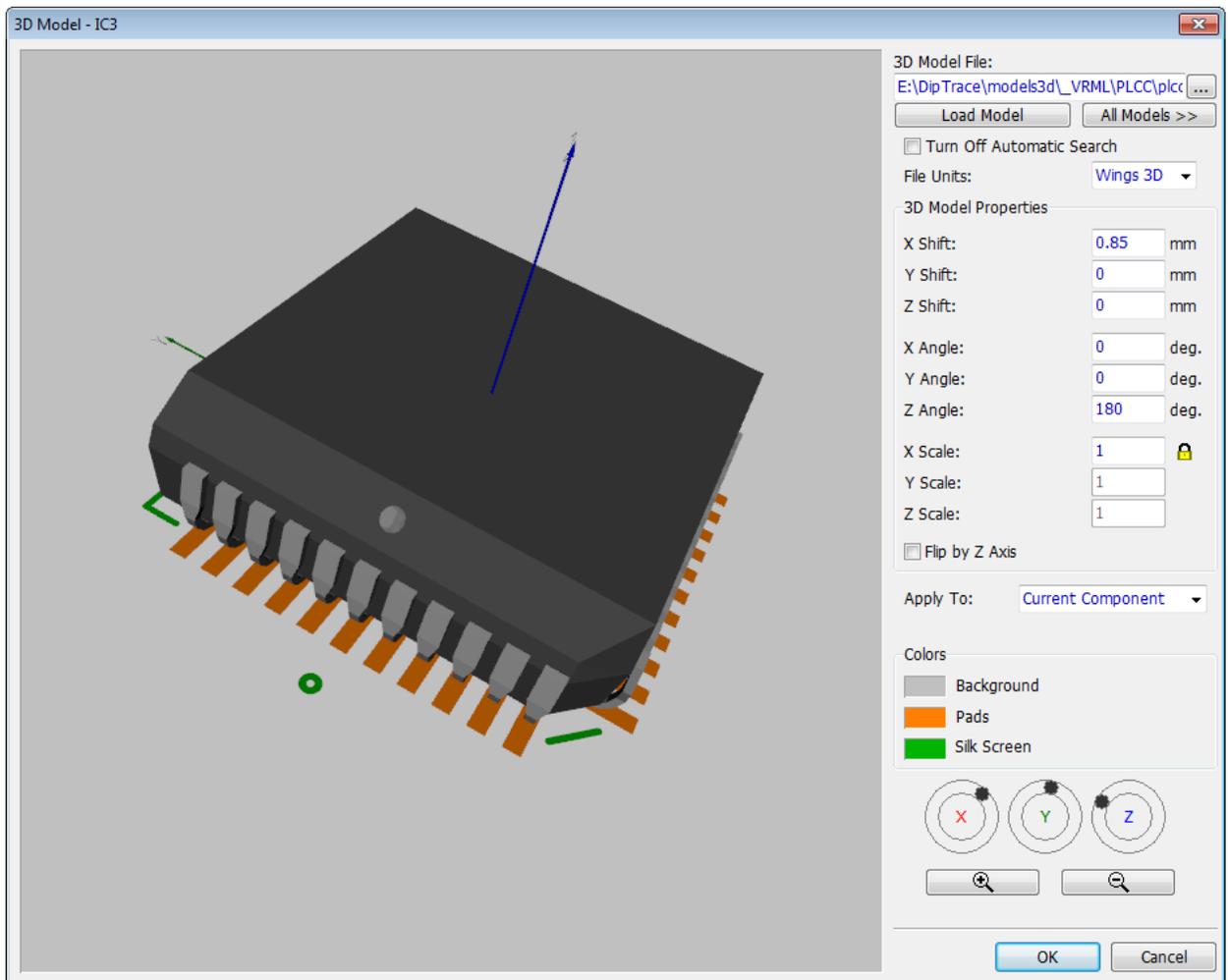
Press "..." button in the Attached 3D Models dialog box or right click on any component on the design area and select 3D Model from the submenu. Rotate model in three axes, zoom in and out, move model by holding the right mouse button and change the colors of the preview in the pop-up dialog box.

If you need to change a 3D model, press All Models button and select a 3D model from the list of all available models sorted by pattern libraries. If model file is in the separate folder on your computer, press "..." button to find it or type in the file path into the corresponding field and press Load Model button.

PCB Creator supports \*.3ds, \*.wrl, \*.step, \*.iges 3D model files, which can be designed in 3D CAD software packages or downloaded from the manufacturer's websites.

If you see that attached model does not align with the footprint, enter new values in the corresponding fields of the 3D Model Properties section: rotation angle by different axes, shift and scale.

Please refer to [Attaching a 3D Model](#)<sup>[102]</sup> topic to see how we mapped a STEP model in the real-life example.



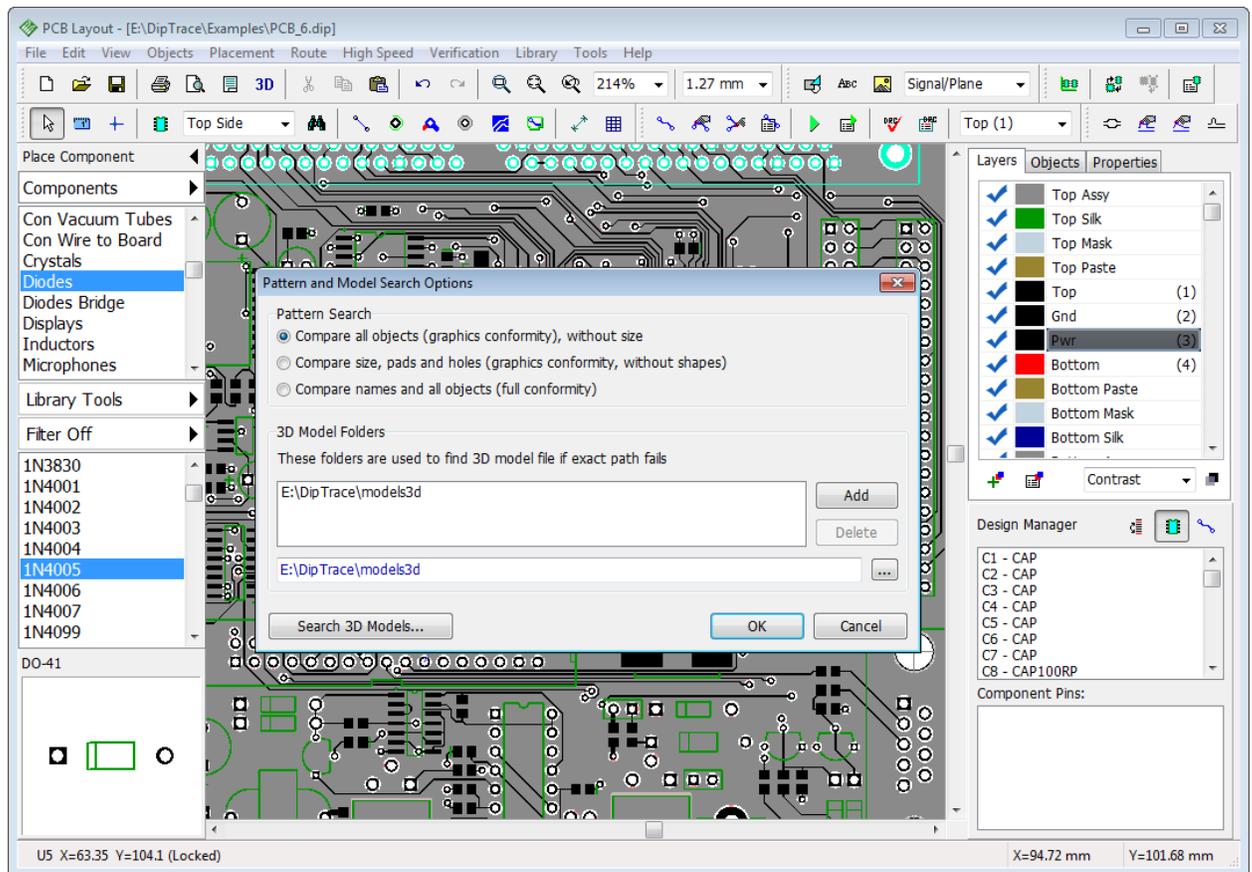
Press OK to close the 3D model dialog box.

### Search 3D models

Go to "Tools \ 3D Preview \ Patterns and Models Search". In the pop-up dialog box you can change the search accuracy and active search folders.

In the Pattern Search section you can select the level of conformity that will be used by PCB Creator while searching for the models (more strict requirements means less possible models found). Press Search 3D Models... to check the results.

However, for all components we recommend to attach 3D Models in the Pattern Editor while drawing the footprint.



Thank you for using PCB Creator.