

DipTrace

SCHEMATIC AND
PCB DESIGN SOFTWARE

Tutorial

Table of Contents

Part I Introduction	4
Part II Creating a simple Schematic and PCB	4
1 Schematic UI.....	5
2 Establishing schematic size and placing titles.....	6
3 Configuring libraries.....	9
4 Designing schematic.....	10
5 Converting to a PCB.....	27
6 Designing a PCB.....	28
Preparing to route	30
Autorouting	36
Working with layers	39
Working with vias	42
Net Classes	45
Manual Routing	48
Measuring trace length	54
Selecting objects by type / layer	56
Placing Text and Graphics	59
Copper Pour	62
Locking objects	68
Design Verification	70
Design Information	73
Panelizing	75
Printing	78
7 Manufacturing Output.....	80
DXF Output	80
Gerber Output	82
Create N/C Drill file for CNC machine drilling	87
Order PCB	88
Part III Creating Libraries	89
1 Designing a pattern library.....	90
Customizing Pattern Editor	90
Create/Save library	90
Designing Resistor (pattern)	91
Attaching 3D Model	100
Designing BGA-144/13x13	101
Real component Design. SOIC-28 pattern	108
2 Designing a component library.....	111
Customizing Component Editor	111
Designing Resistor (component)	113
Designing Capacitor	116
Designing VCC and GND symbols	123
Designing a multi-part component	126
Using additional fields	134
Designing PIC18F24K20	136
Spice settings	143
Library Verification	144

Placing parts	146
Part IV Using different package features	151
1 Connecting.....	151
Working with Buses and Bus Connectors	151
Working with Net Ports	157
Connecting without wires	158
Connection Manager in Schematic and PCB Layout	162
2 Reference Designators.....	163
3 How to find components in libraries.....	169
4 Electrical Rule Check.....	171
5 Bill of Materials (BOM).....	172
6 Importing/Exporting netlists.....	174
7 Saving/Loading Design Rules.....	177
8 Spice simulation.....	177
9 Checking net connectivity.....	181
10 Placement and Autorouting.....	184
11 Fanout.....	193
12 Hierarchical Schematic.....	196
13 3D Preview and Export.....	206
Part V DipTrace Links	210

1 Introduction

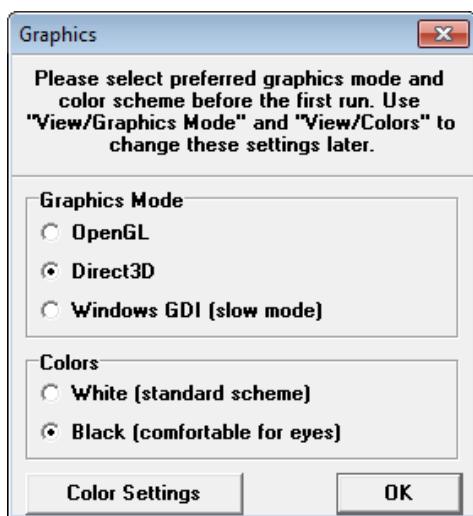
This document allows you to get started with DipTrace by designing simple Schematic and its PCB, pattern and component libraries as well as get basic knowledge about different package features. Tutorial includes step-by-step design guide and many additional insets that allow you to discover program features. Follow step-by-step instructions and create simple board at the end, all the way from schematics to Gerbers. This tutorial was created for DipTrace version 2.4 (June 20, 2014).

2 Creating a simple Schematic and PCB

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

Open DipTrace Schematic capture, go to "Start → All Programs → DipTrace → Schematic" in Windows OS or "Applications → DipTrace Launcher → Schematic" in MacOS.

If it is the first time you launch DipTrace Schematic, you will see the dialog box for graphics mode and color scheme selection.



You can select graphics mode that works better for you:

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

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

OpenGL will be the best choice for high-end engineering/graphics stations with professional OpenGL graphic cards. Anyway, you can try both modes on complex projects and choose the best for you.

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

We will use 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 color scheme or define colors any time with "View \ Colors" main menu item.

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

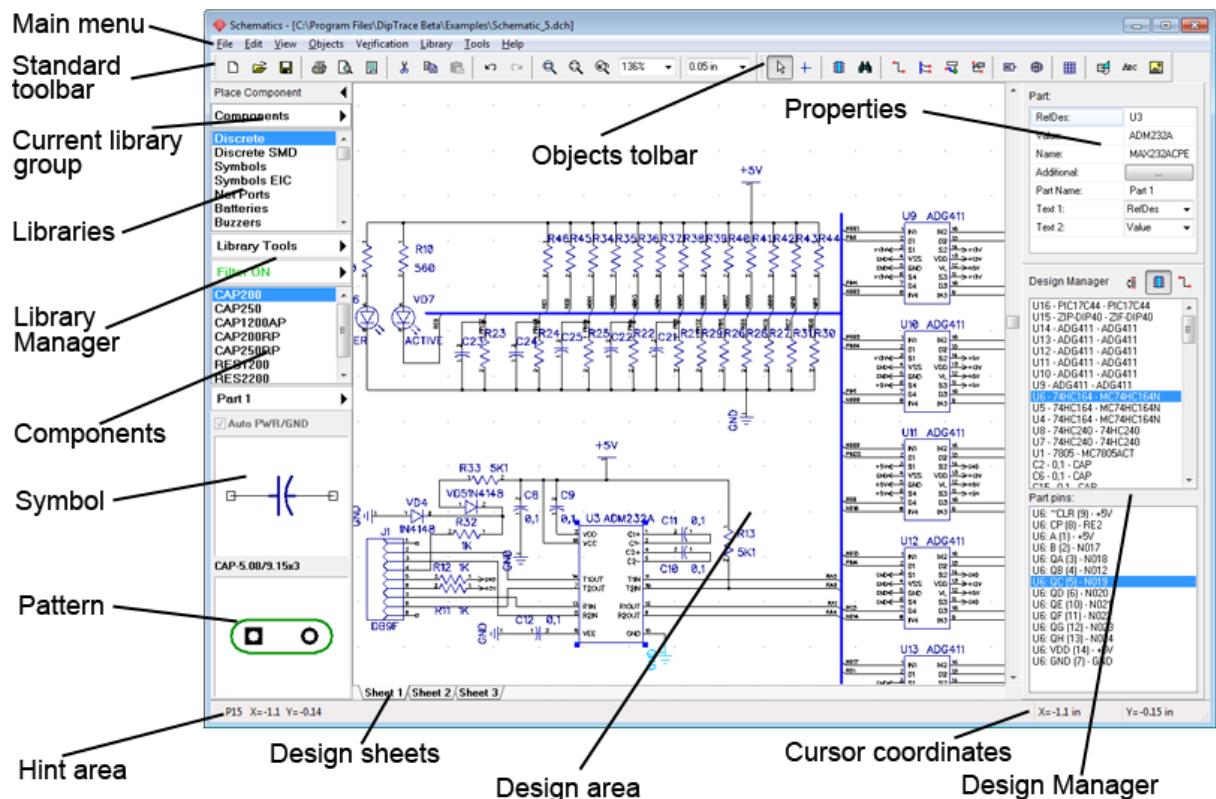
Notice that relative sizes of program panels in screenshots may differ from what you see on the screen due to resolution limitations applied in this PDF tutorial.

Sometimes we will hide 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 Design Manager panel or press "F3" hot key.

Schematic and printed circuit are not real and performed exclusively for DipTrace educational purposes.

2.1 Schematic UI

Schematic main window includes: schematic design area, main menu, toolbars, properties panel, design manager, library manager and status bar.



On the design area you can create and edit schematic objects (parts, wires, buses, shapes, tables etc.). Access to all common functions of the program is done via main menu.

Other interface elements:

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, place parts, find components, create and edit wires, buses, page connectors, hierarchy connectors and blocks, place shapes, text and tables.

Library Manager - all active libraries, user libraries or project components. Select library from the

list, find and place component. Setup libraries panel, search filters, multi-part component placement tools, symbol and pattern previews.

Properties panel - displays properties of current tool or selected objects.

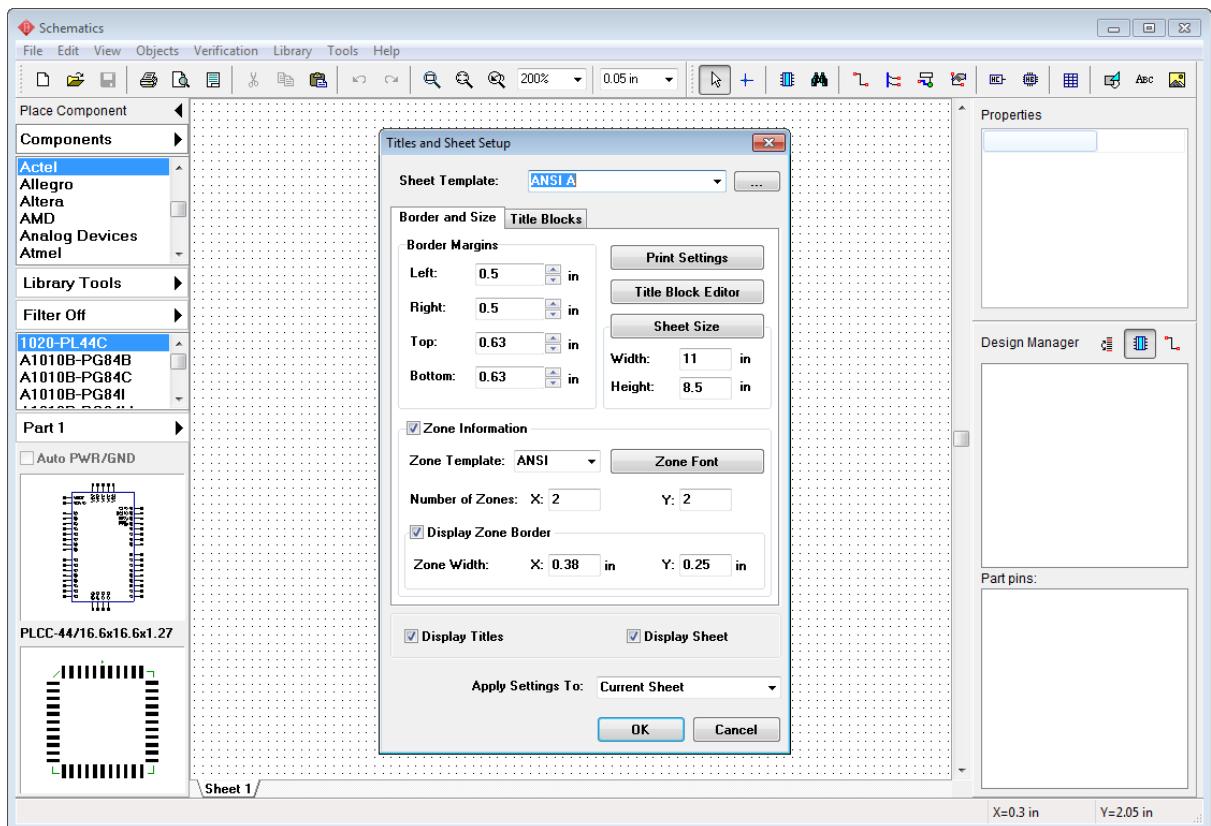
Design Manager - navigates you through the schematic (left click highlights object/s, double click pans circuit to show selected component/net).

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

See DipTrace Help for details ("Help \ DipTrace Help" from main menu).

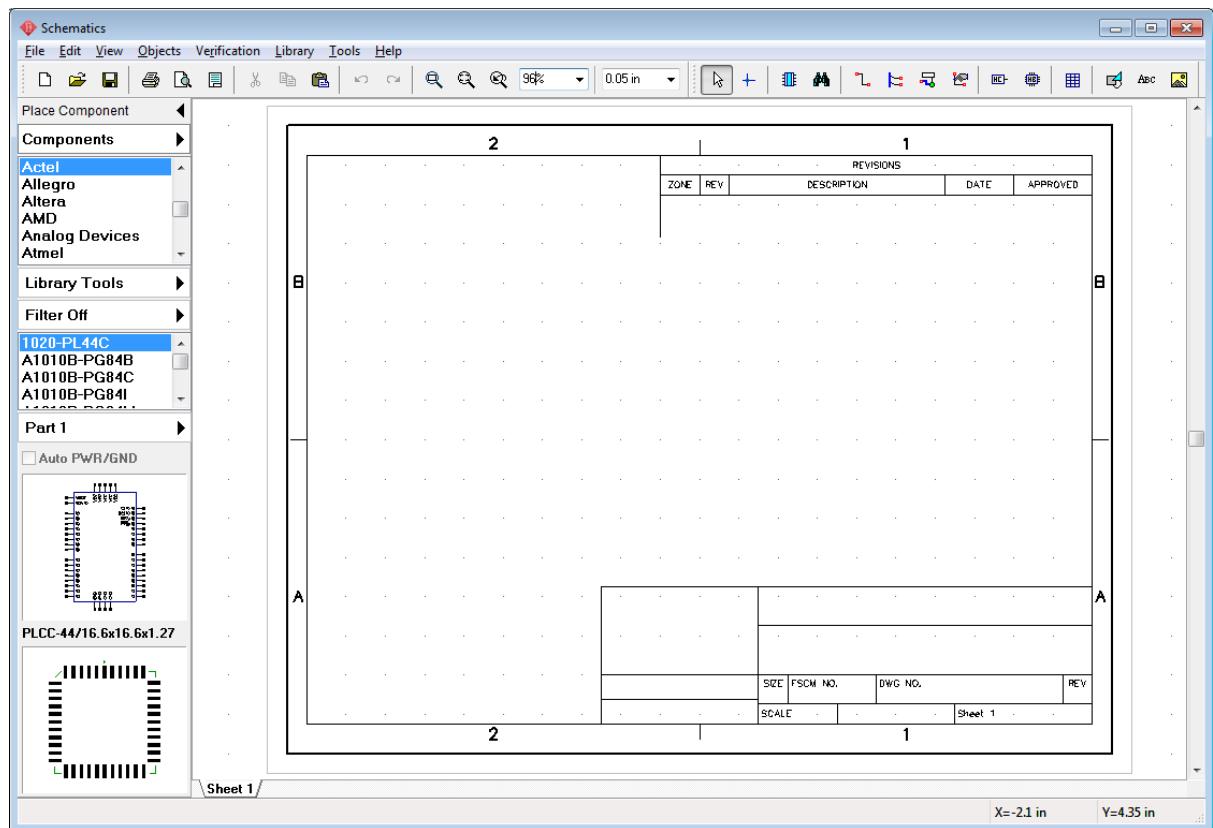
2.2 Establishing schematic size and placing titles

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

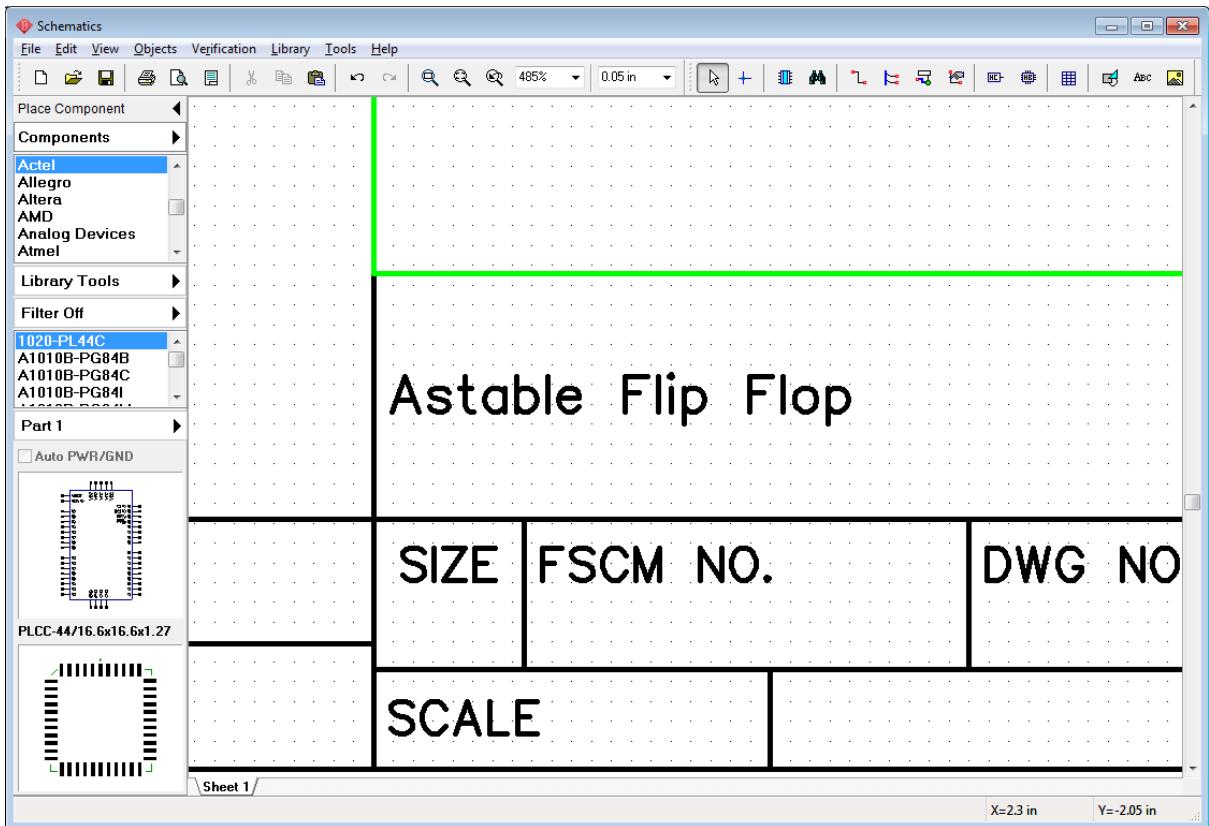


Notice that you can show / hide Titles and Sheet on the design area by selecting "View \ Display Titles" and "View \ Display Sheet" from main menu.

Now press "-" button to zoom out until drawing frame can be seen. "+"/-" hot keys, mouse wheel and scale box on the instruments panel allow to zoom on the schematic. Hover over component or selected area for more precise zooming. Notice that we have hidden design manager panel to the right ("F3" hot key) to allow more space on the design area.



To enter text into the title field simply hover over that field (it should be highlighted in green), left click it to open field properties pop-up dialog box and 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 "Font" button and set font size to "12". Then click "OK" to close the dialog box and apply changes. You can also enter multi-line text into the title block fields.



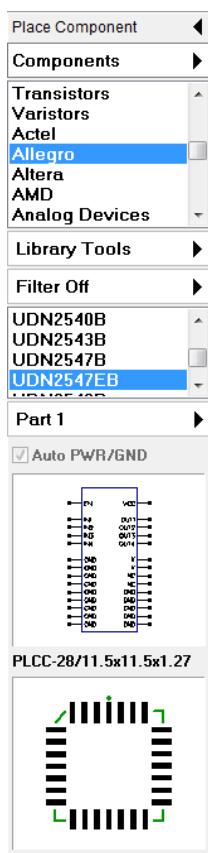
You may zoom on the title block by hovering over it and repeatedly pressing "+" or scrolling mouse wheel.

But let's practice a bit with different zoom options. Click on the "Zoom Window" tool (the third button to the left of the scale box) and draw a rectangle on the design area where you want to zoom.

To return to previous scale and position, use "Undo Scale" tool (button on the immediate left of the scale box).

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

2.3 Configuring libraries



DipTrace incorporates single cross-module library management system. Libraries are organized to user-customized groups with multi-level search filters ensuring that correct component can be found quickly. 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 to the design. The list of libraries is in the upper part of the panel and list of components of selected library is right below, symbol and pattern previews are at the bottom.

Press "Components" (<Current Library Group>) item on the Library Manager 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 group);
 - 3) Project Libraries (auto-generated library with all components from current circuit). It is empty if no schematic file is open.
- "Library Setup" - add / delete / copy / move libraries between groups, add and edit library groups.

Configuring Library Group

Select "User Components" library group. As you can see it is blank. We know all the components and libraries that we will need for the project in advance, hence we can add them to this library group. Press "Library Tools \ Add Library to "User Components" Group". In the pop-up dialog box select "Components" library group in "Add from Group" section.

This group contains all DipTrace standard libraries.

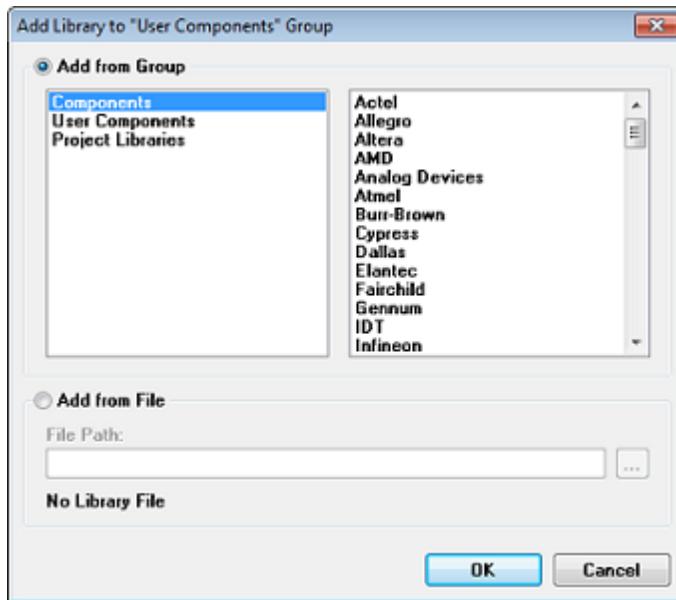
Select "Fairchild", "Discrete" and "Symbols" libraries, use "Ctrl" for multiple selection and press "OK".

That's it - corresponding libraries had been added to "User Components" group and we are ready to start designing schematic.

However, go to "Library \ Library Setup" from main menu or press "Components (<Current library group>) \ Library Setup" on the Library Manager panel to get access to comprehensive library system settings.

Notice that Library Tools panel allows to configure both pattern and component library groups, though only component libraries are visible in Schematic.

More information in DipTrace Help ("Help \ DipTrace Help" from main menu), "Working with Libraries" section.



2.4 Designing schematic

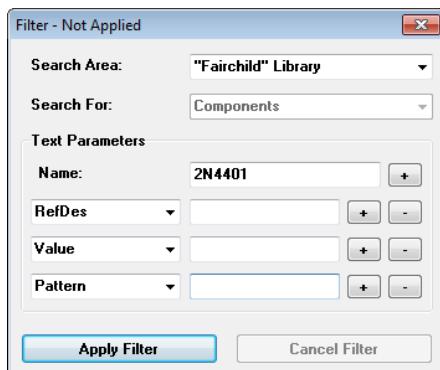
In this section of the tutorial we will show basic principles of working in Schematic module of DipTrace PCB Design Environment.

Please change grid size to 0.1 inch, you can select it from the list of grids (drop-down list with "0.05 in" default text on instruments panel) or press "Ctrl +" hot keys to increase or "Ctrl -" to reduce grid size.

Notice that if there is no appropriate size in the list you can add one by selecting "View \ Customize Grid" from main menu. Measurement units can be changed with "View \ Units" main menu item.

Now let's start to create the circuit. Select "Fairchild" library from "User Components" library group on the Library Manager panel.

Search Component in Libraries

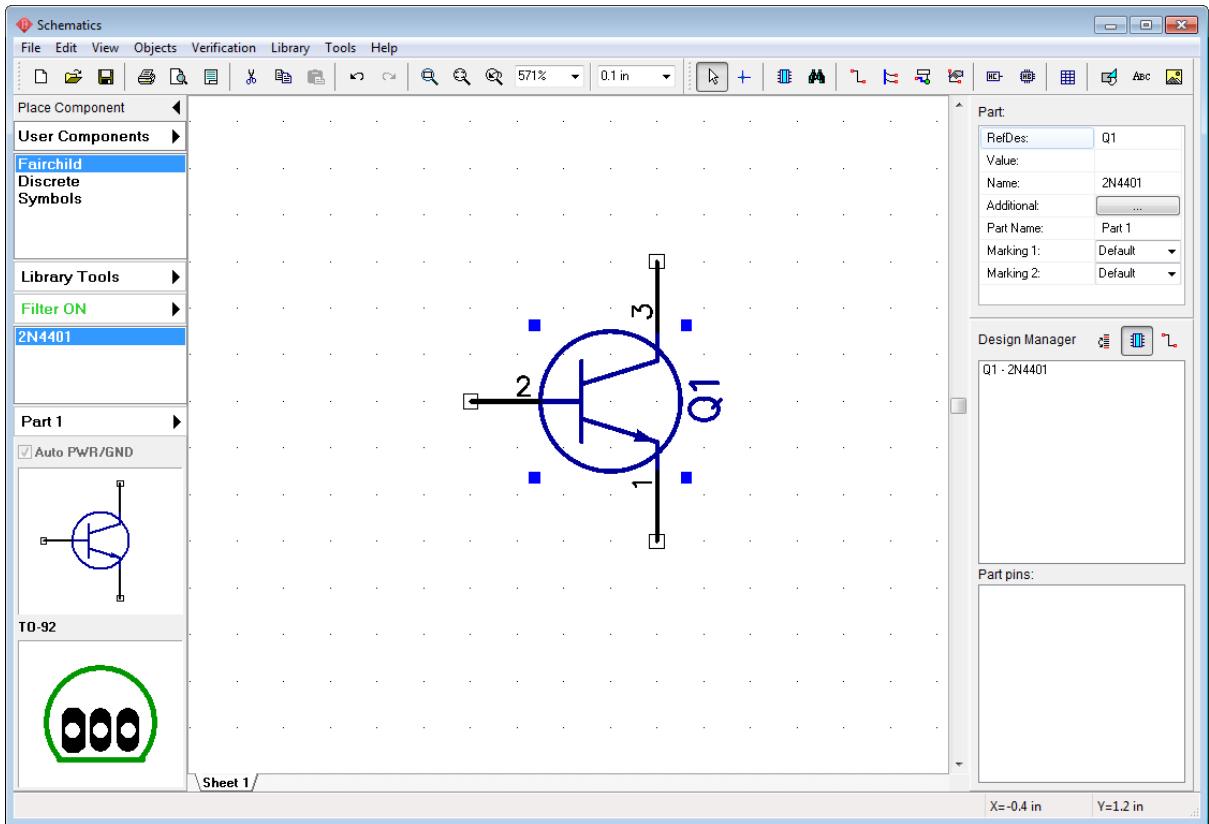


Once library is selected, scroll down components list to find 2N4401transistor or use search filter. Select "Objects \ Find Component" from main menu or just press "Filter OFF" button on the library manager.

In the pop-up dialog box select "Fairchild" library as search area then type "2n4401" into Name field and press "Apply Filter". Close this dialog box when search process is over and you will see only the component you have searched for in the component list and "Filter ON" button glowing green.

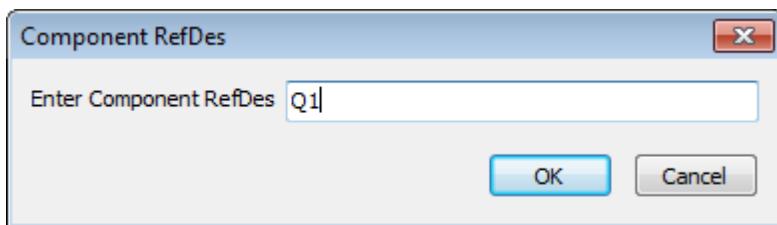
Notice that you can widen search results by entering part of the component name as well as filter components by RefDes, value, pattern, manufacturer, datasheet or additional fields - use "+" and "-" buttons to add or delete search filters.

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

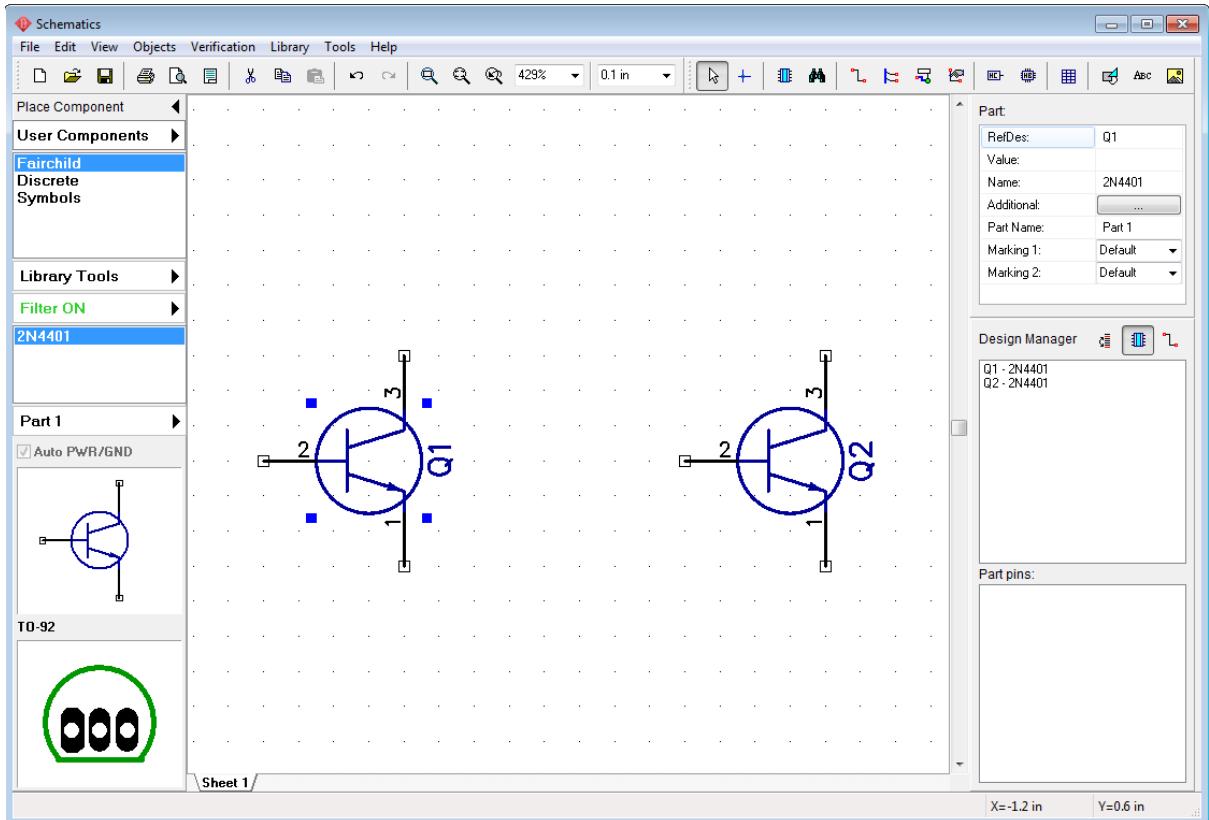


Drag and drop component if you need to move it to another place on the design area. For several components - select them first and drag and drop afterwards. To select several objects, press and hold "Ctrl" button then click on each object that you want to add to selection or move mouse to the upper-left corner of the group, hold down left mouse button and move cursor to lower-right corner. Release mouse button and objects will be selected (if "Ctrl" key is pressed, selection will be inverted).

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

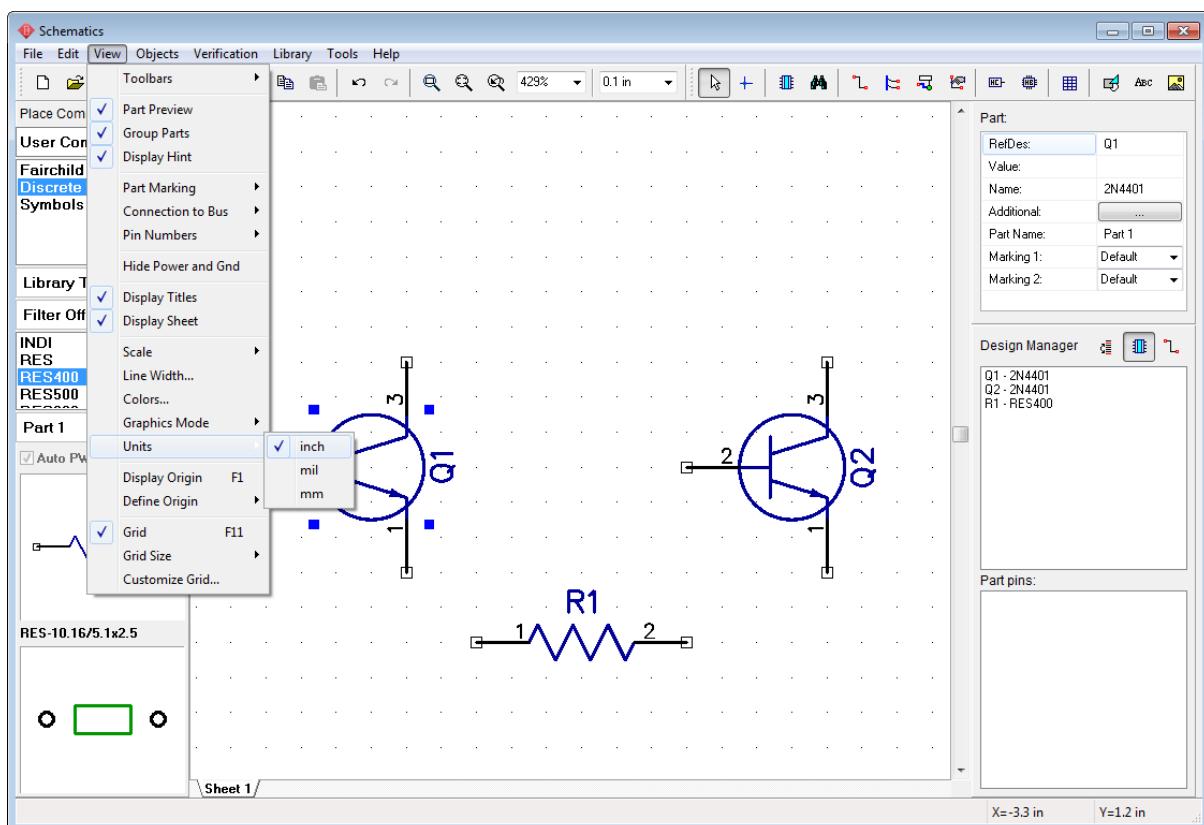


We need two transistors for our schematic, select "2N4401" in the component list again and place it to the design area. If you have changed reference designator, you don't need to rename the second transistor, it will be done automatically. If you want to rotate component before placing it on the design area, press "Space" or "R" button.



When search filter is active you can see only certain (filtered) components of library. Press "Filter ON" button on the Library Manager and then press "Cancel Filter" in the pop-up dialog box to turn filtering OFF. Close search filters dialog box.

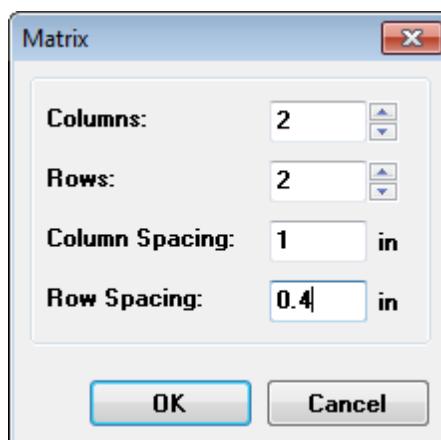
Select "Discrete" library on the library panel, find RES400 resistor and place it on the design area. "400" in resistor's name stands for 400 mils of lead spacing. If you prefer metric units, select "View \ Units \ mm" from main menu, however, we will keep inches as they are the most suitable units for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



Copy Components

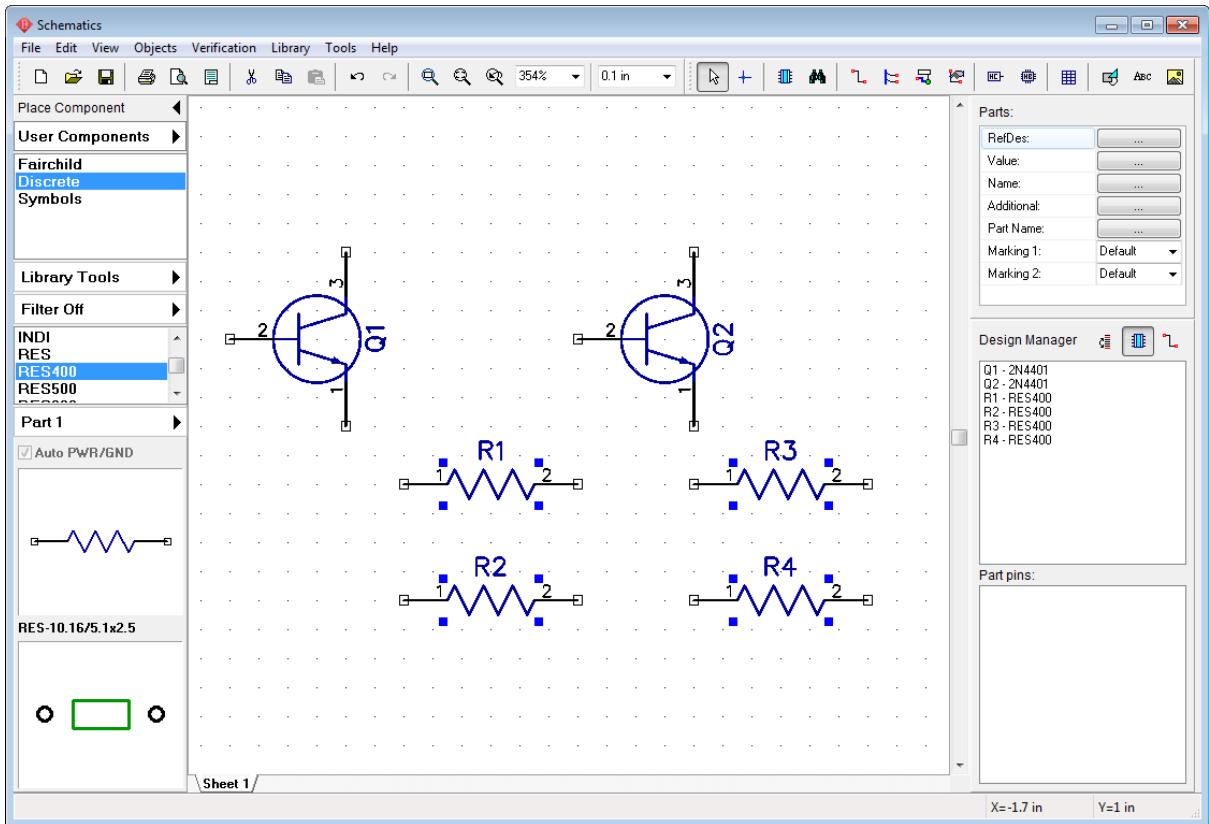
For this project we need 4 resistors. They can be placed manually like Q1 and Q2 transistors, one by one. But this time we select resistor on the design area and copy it 3 times. There are two ways to copy:

1. Once component is selected, go to "Edit \ Copy" from main menu or "right click on the component \ Copy" ("Ctrl+C" hot keys 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 main menu (or press "Ctrl+M").



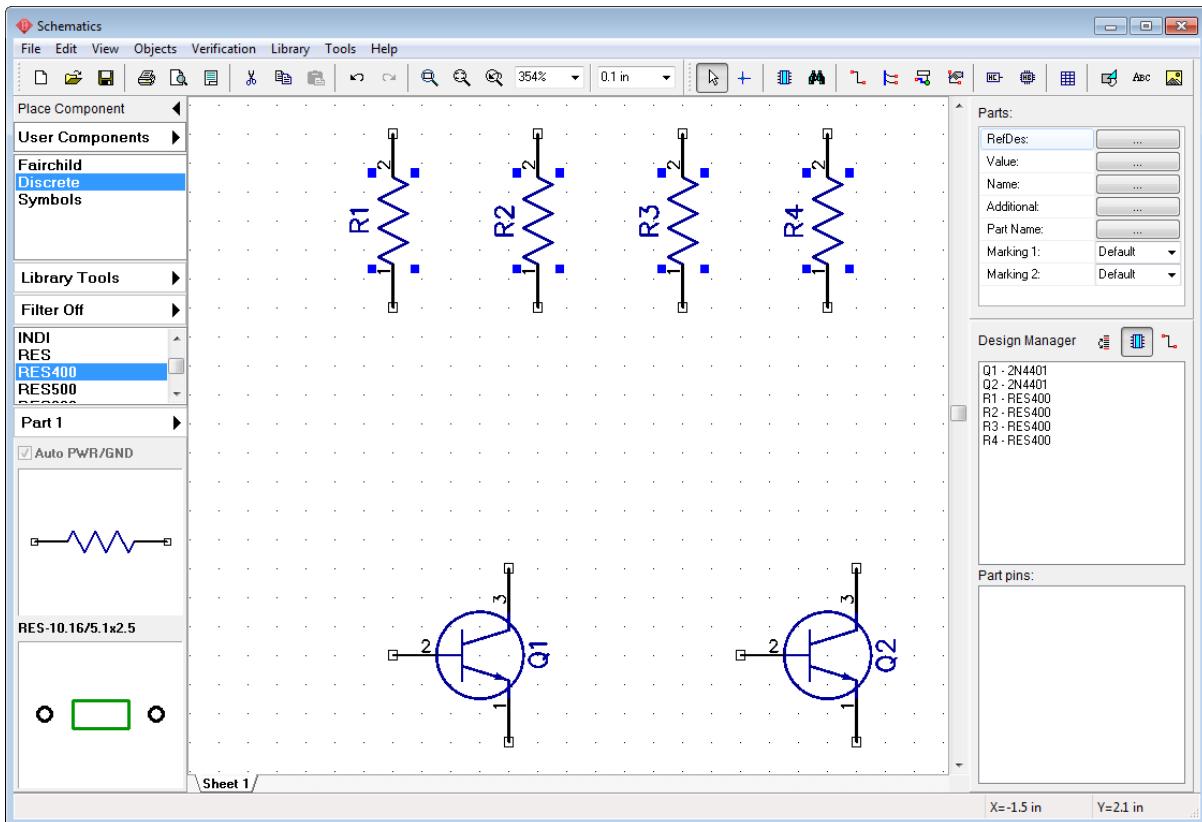
In the "Copy Matrix" dialog box set number of columns and rows ("2" columns and "2" rows to get

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:

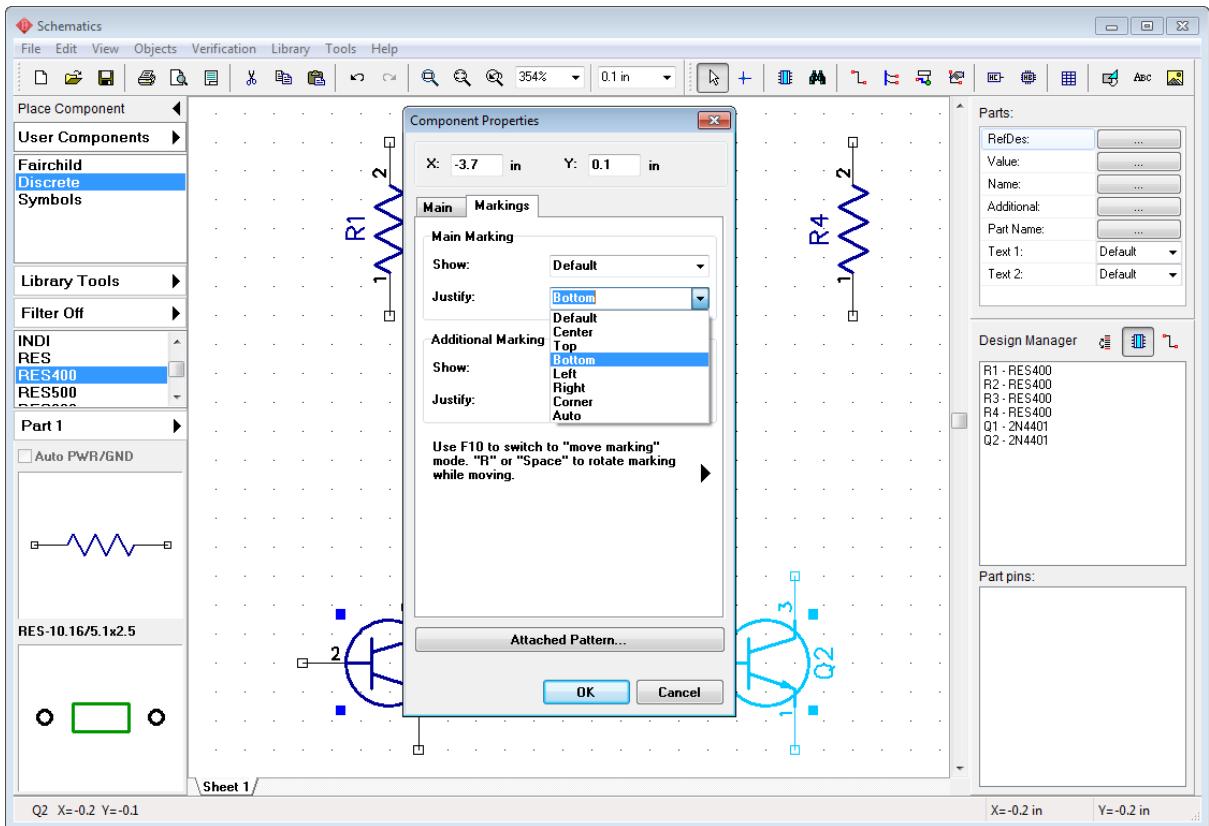


Move resistors to a proper location, like on the picture below (use mouse or arrows on the keyboard) and rotate 90 degrees, use "Space" or "R" buttons to rotate selected components. Another method to rotate objects is using "Edit \ Rotate" main menu item or right click on the object and select "Rotate"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

Notice that you can pan design with the right mouse button or mouse wheel: move mouse arrow to the design area, hold right mouse button or mouse wheel and pan.

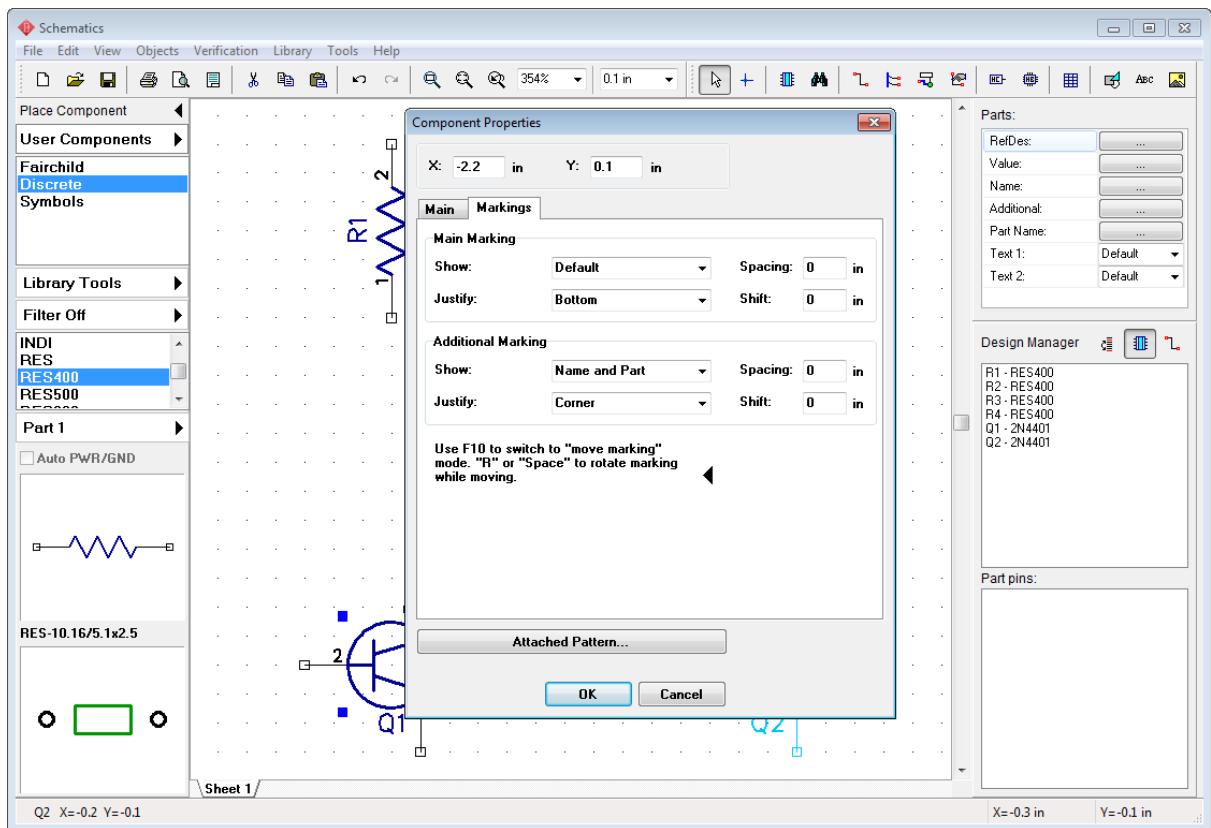


Reference designators of Q1 and Q2 transistors have inappropriate location, they should be under the component symbols. To change RefDes location, select both transistors, right click on one of them and select "Properties" from the submenu. Choose "Markings" tab in the component properties dialog box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



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

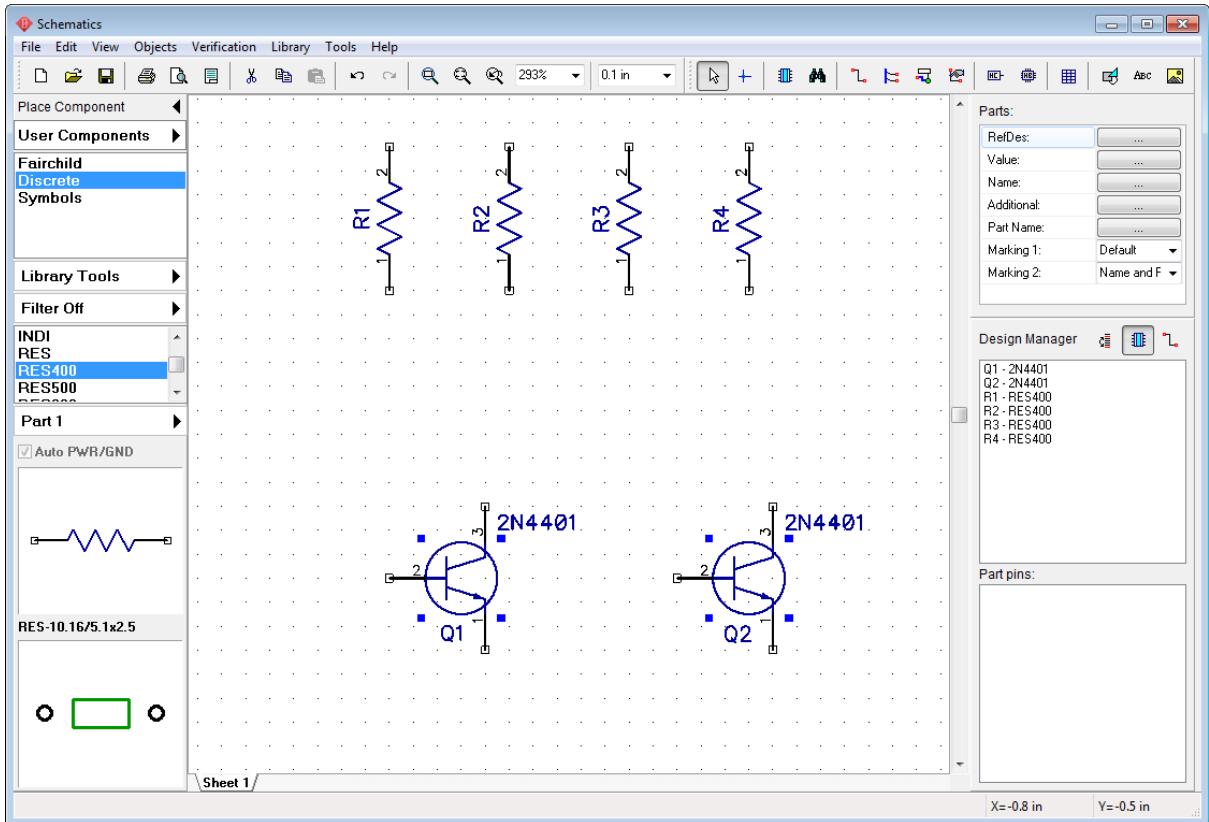
If you want to edit precise markings' positions, press right arrow button under Additional Marking section. The Component Properties dialog box will become wider.



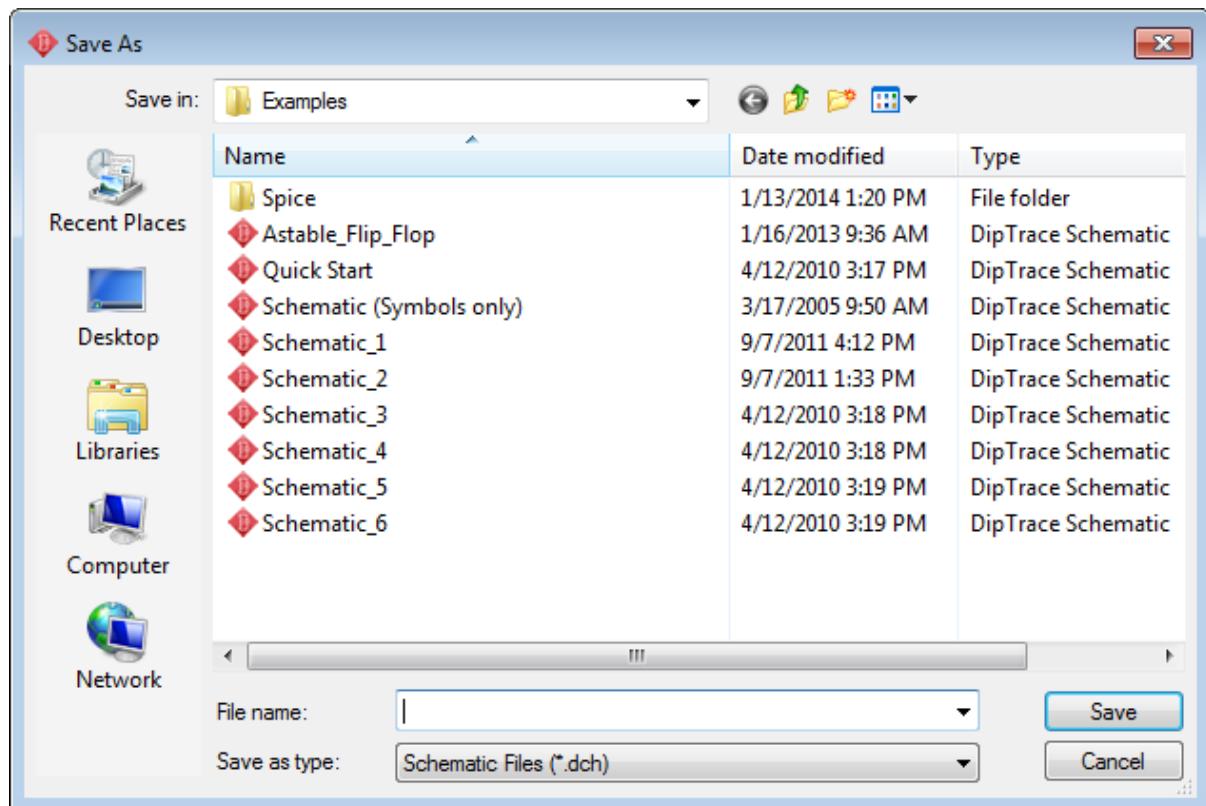
You can show or hide pin numbers for the entire circuit by selecting "View \ Pin Numbers \ Show" if they are not displayed yet. To change pin display settings for selected part, right click it and select "Pin Numbers" from the submenu.

However, if you're 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 main menu or press "F10". It is recommended to turn OFF grid for precise moving ("F11" hot key). You can move and rotate part marking like separate objects with "R" or "Space" key.

"View \ Part Marking" submenu allows to change common settings for part markings. Common settings are applied to all schematic parts, except those with custom properties.

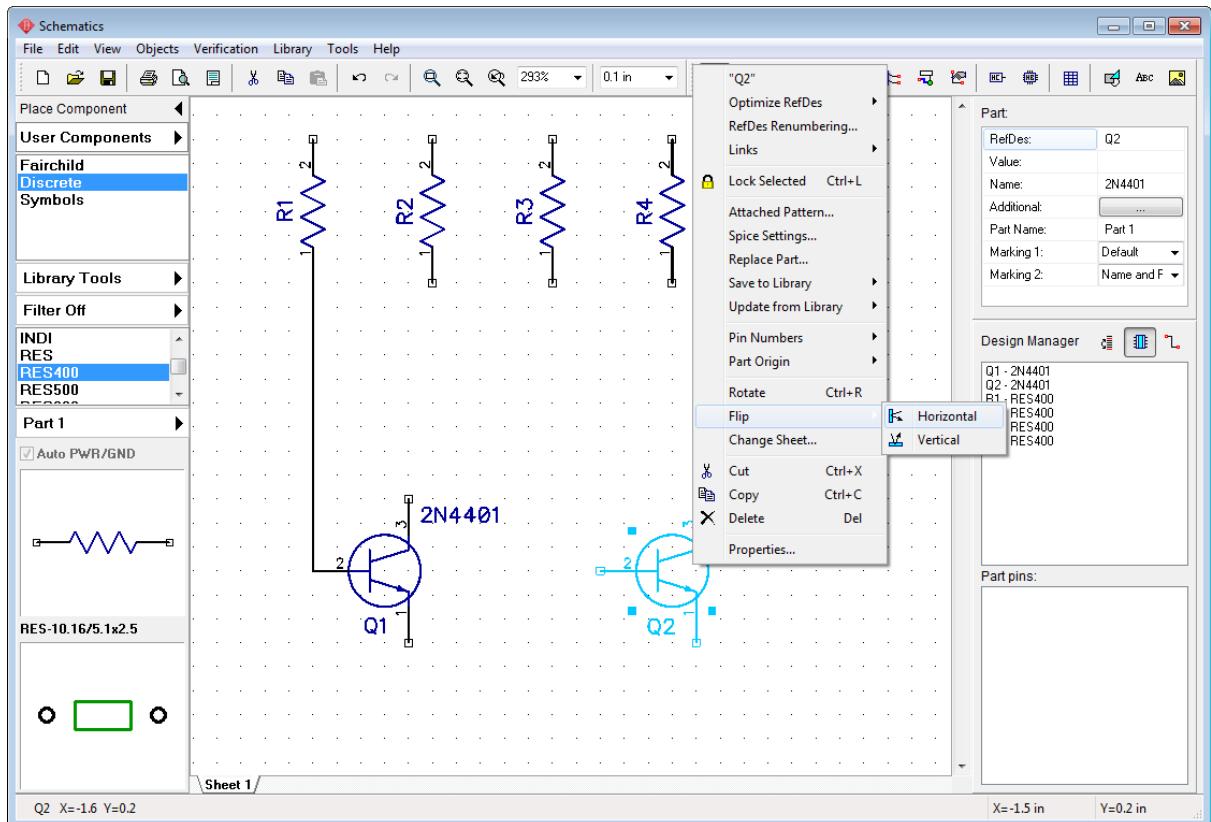


Now please turn ON the grid (if it was turned OFF) with "F11" hot key. Use Undo or Redo options if you are not satisfied with changes you've made, "Edit \ Undo" or "Edit \ Redo" from main menu. DipTrace saves up to 50 steps. Remember to save schematic into the file. Select "File \ Save" from main menu or click "Save" button on standard toolbar. If current schematic has never been saved, "Save As" dialog box will pop up to define file name and location. If file exists clicking "Save" button or pressing "Ctrl+S" is enough. "File \ Save As" can be used for changing filename, for example, for backup e.t.c.

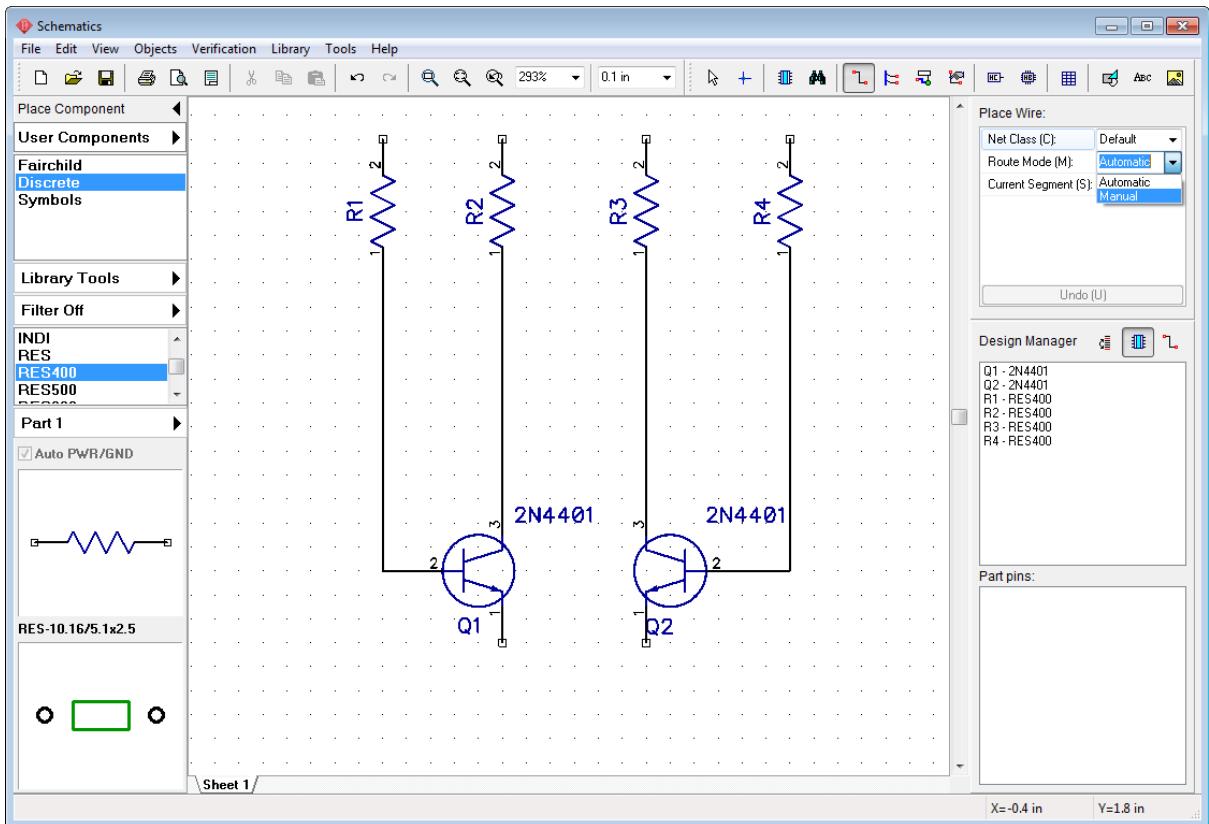


Connect pin 1 of R1 resistor to pin 2 (base) of transistor Q1: hover mouse arrow over the bottom tip of R1 resistor and left click it. Then move mouse arrow down to the base pin of transistor Q1 and left click it to connect wire and create connection between R1 and Q1.

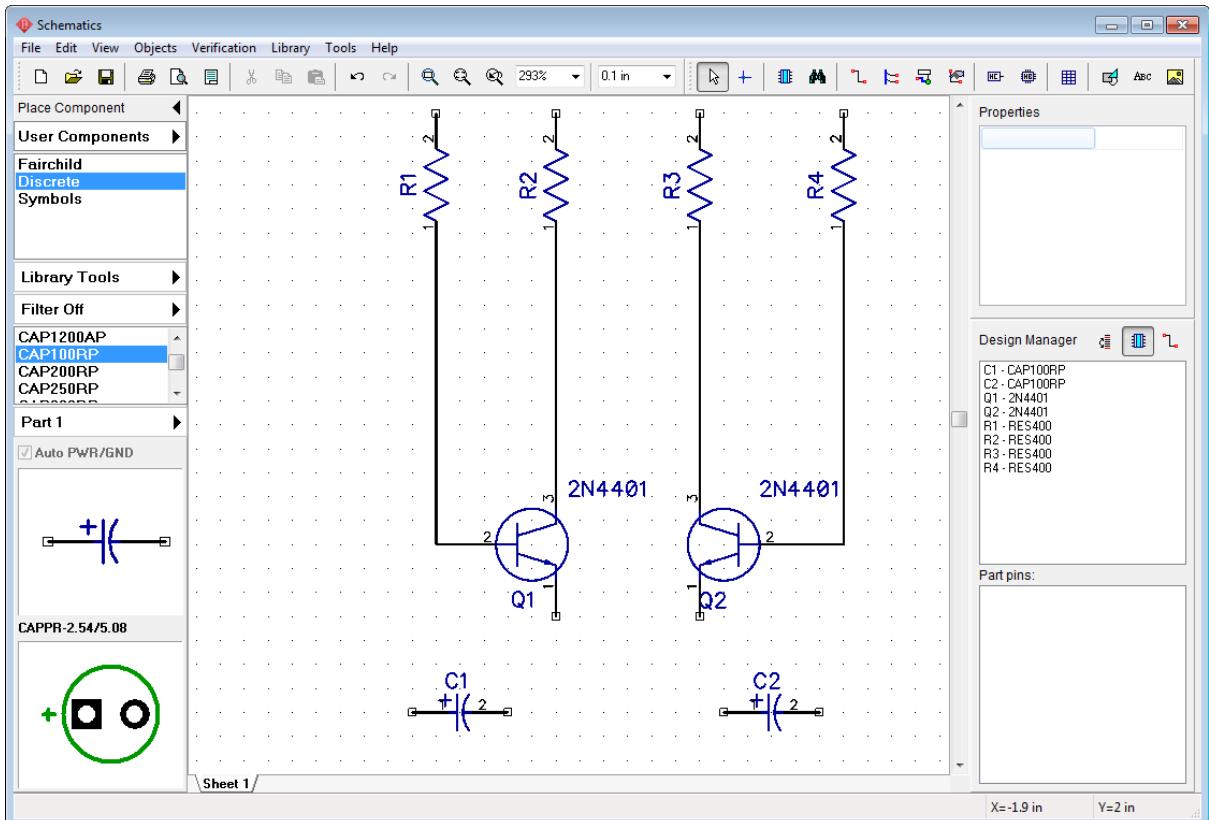
Now we need to mirror Q2 transistor, this will make schematic more easy to understand. Right click it and select "Flip \ Horizontal".



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 "Place Wire" panel on the Design Manager to your right-hand side, set "Manual" in "Route Mode" section or just press "M". You can see "Place wire" panel only when you are in wire placement mode. You may need to move some components for a good-looking schematic.



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

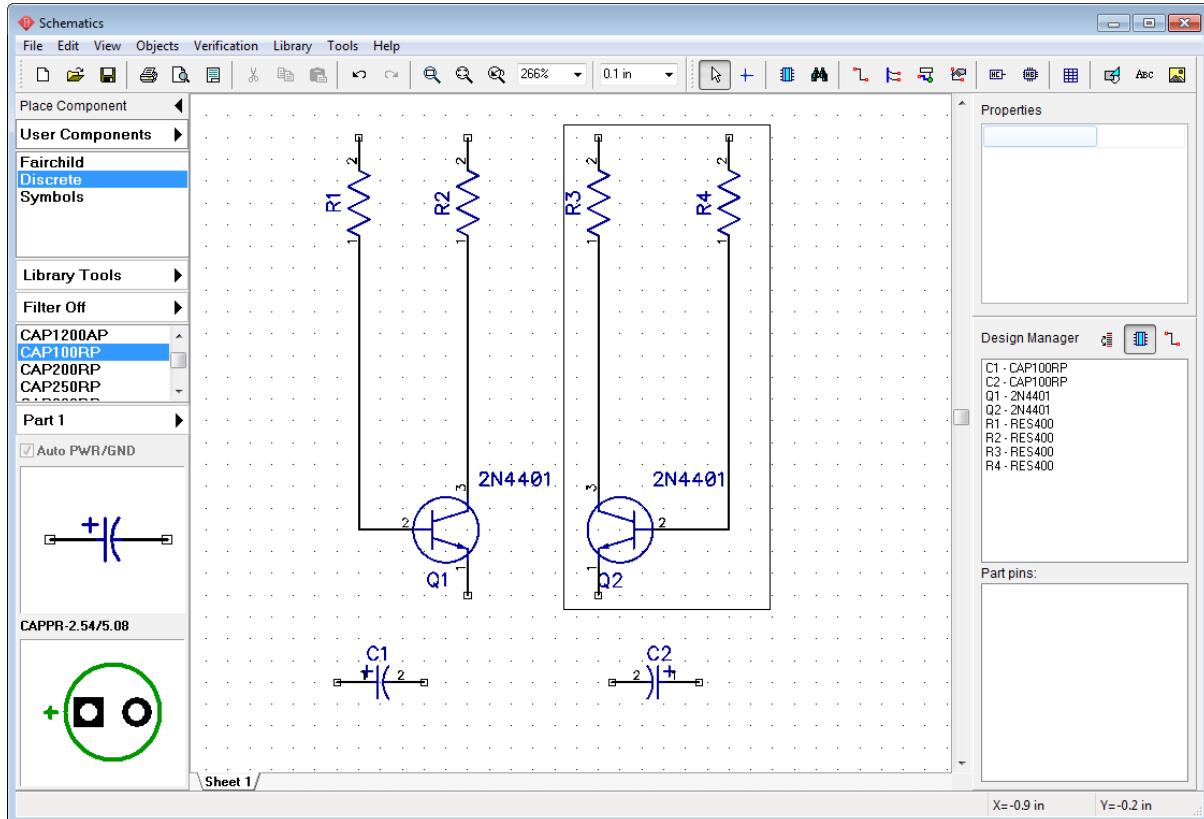


Flip C2 capacitor, "Flip \ Horizontal" feature from the right-click submenu.

C2 capacitor's plus sign should be on the right side.

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

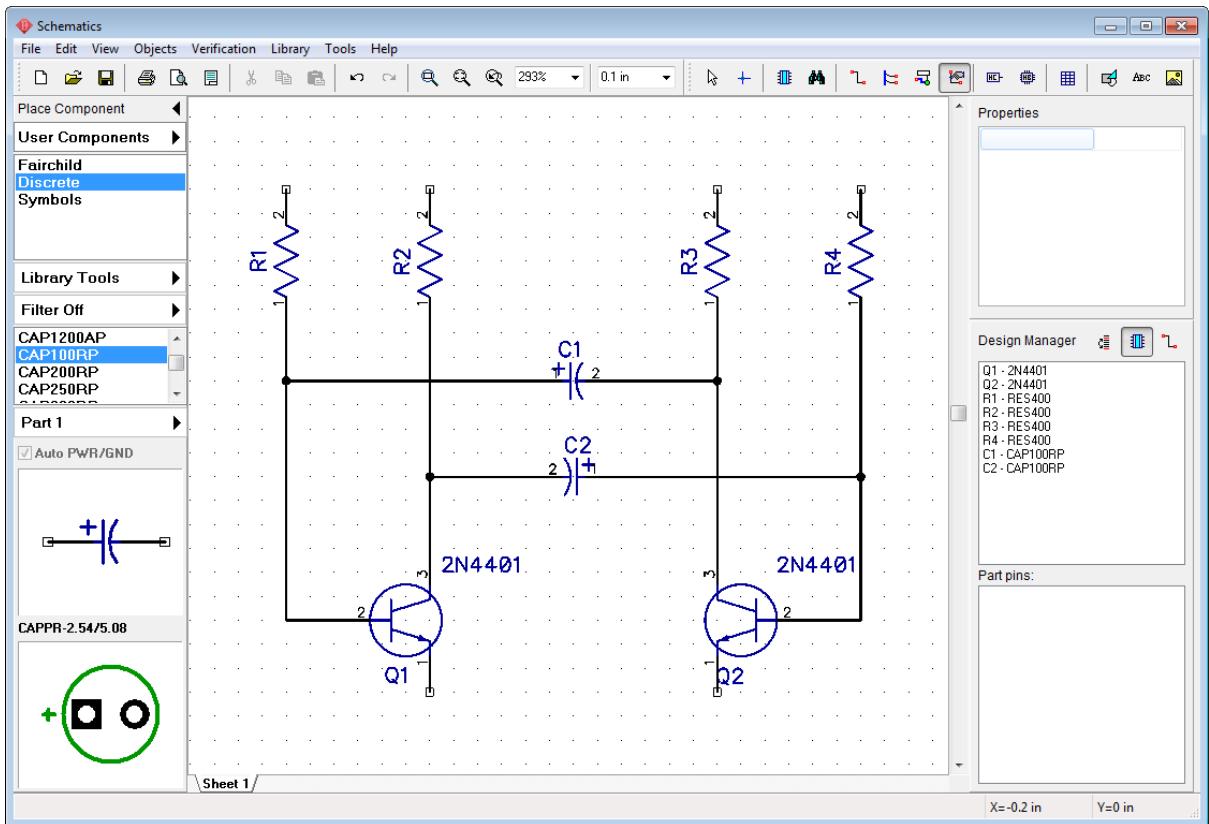
You might need to move some components to give enough space for capacitors and comfortable connections. Move resistors upwards and then select "Q2", "R3", "R4" and related wires and move them to the right a little bit. Place mouse arrow in the upper-left corner of selection, hold left mouse button and move to opposite corner of selected objects. All components and wires inside the rectangle will be selected when you release mouse button.



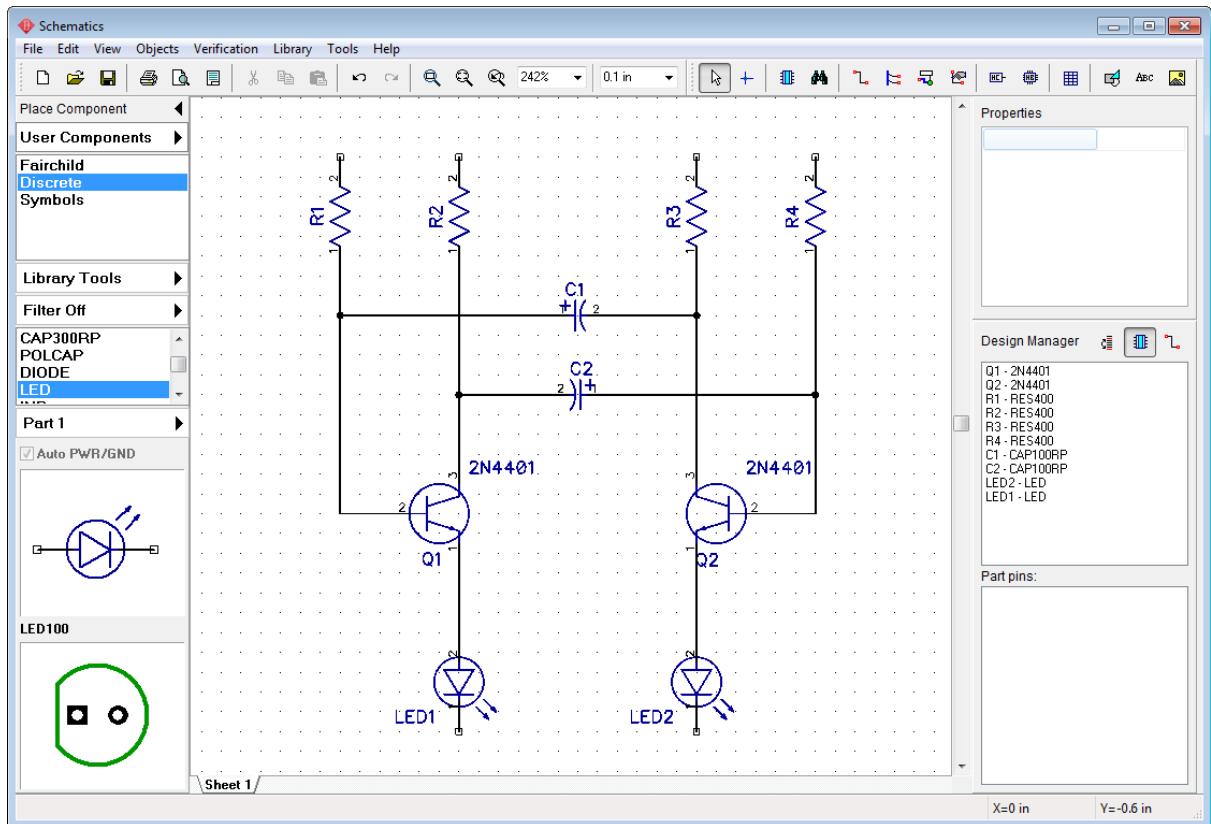
Use right click to deselect all if you are in default mode, or double right-click if you are in another mode (first click to disable active mode and the second click to deselect all).

Connect C1 (+) to pin 2 of 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 circles should appear if wires are connected correctly.

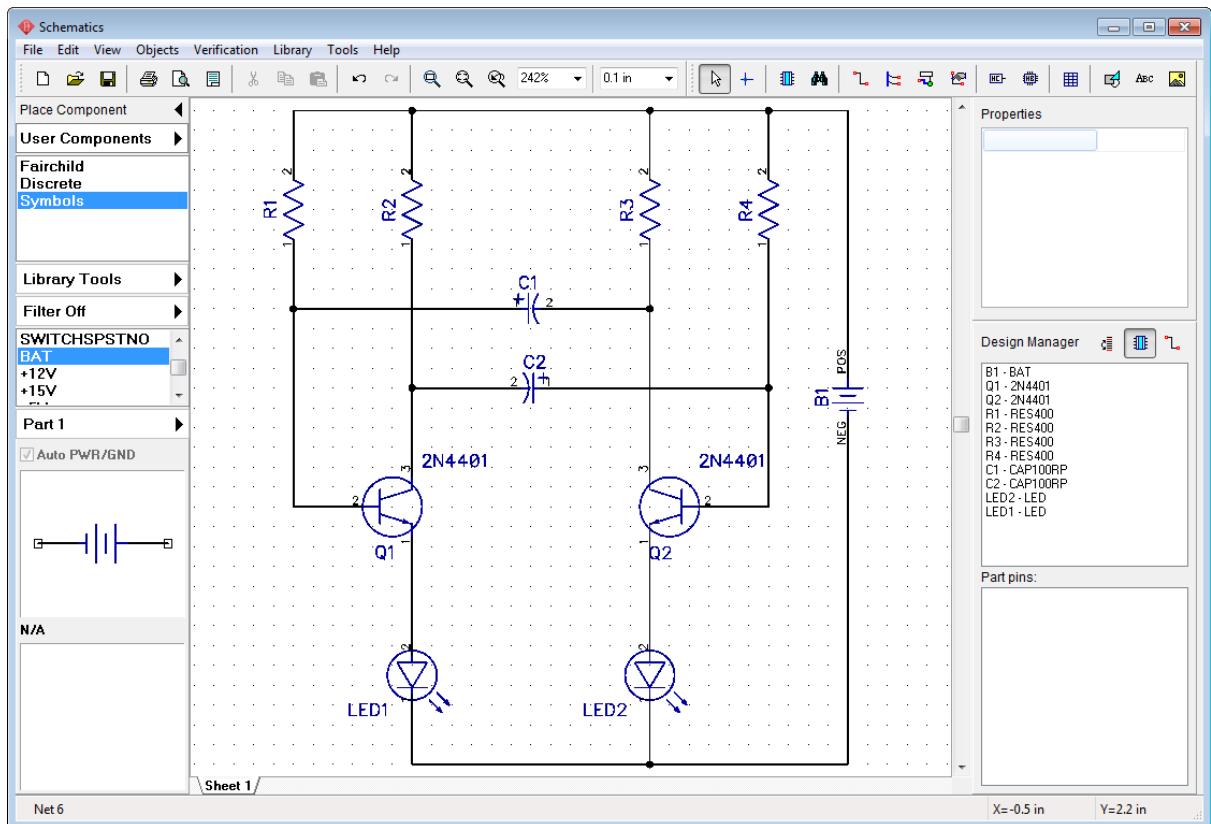
Then connect C2 like on the picture below:



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



Place a battery symbol "BAT" from "Symbols" library, change RefDes and connect wires (see the picture).



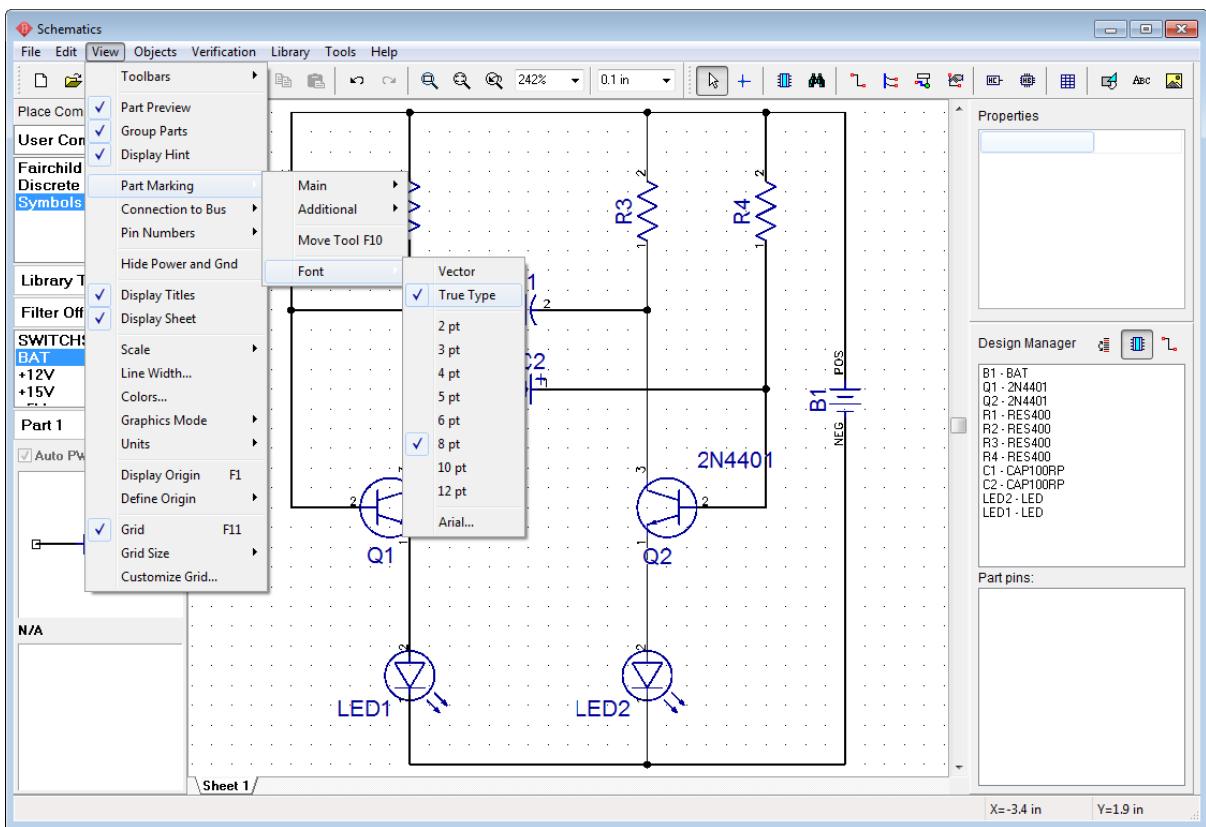
If you want to move existing wire, hover mouse over it (net should be highlighted, mouse arrow shows possible moving directions) then hold left mouse button and move wire to a new position.

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

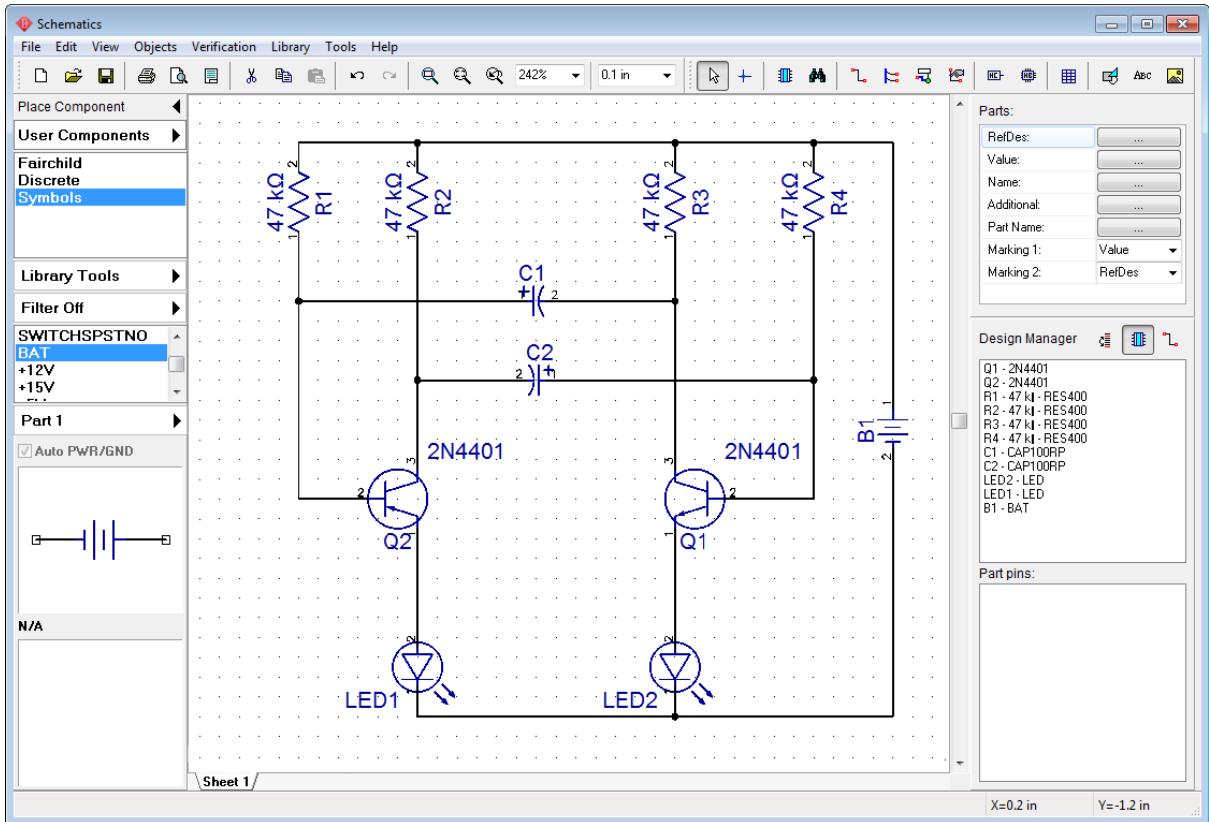
("Place Wire" mode is enabled automatically when you try to place wire by clicking on some component's pin).

If some objects are not highlighted when you move mouse arrow over them, right click on the free area to activate default mode. If you want to delete wire, right click it to open submenu and select "Delete Wire". To delete 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 DipTrace. We need to activate TrueType font for part marking 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.



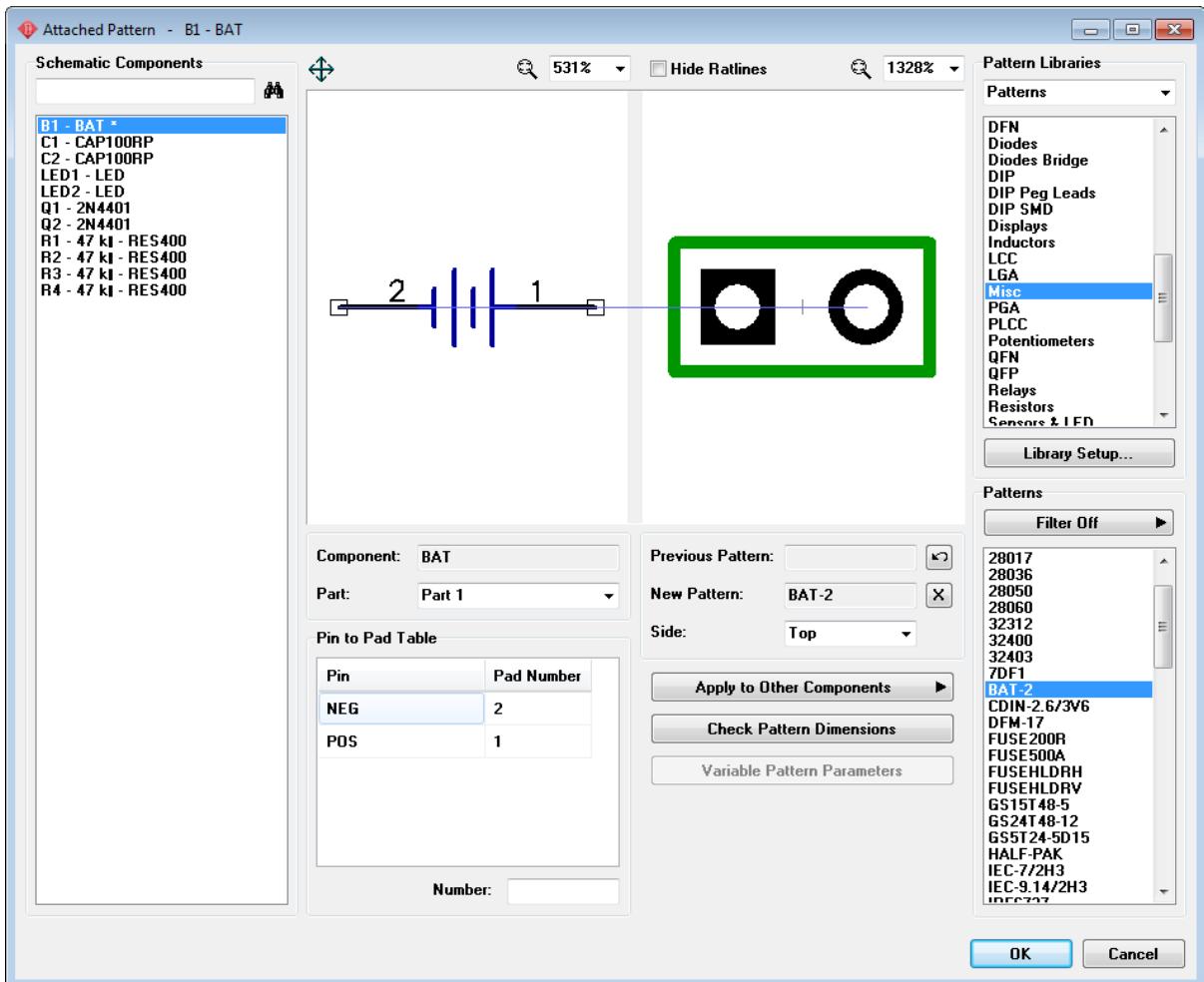
Now select all resistors then right click on one of them and select "Properties" from the submenu. Select "Main" tab and type "47 k " into "Value" field. Click on "Markings" Tab and specify "Main Marking \ Show \ Value", then go to "Additional Marking \ Show \ RefDes" and click "OK".



As you remember, we took battery component from Symbols library. All components in this library don't have patterns, it's just symbols (pattern preview field on the Library Manager is blank). But as you already know, in order to convert schematic directly to PCB you should attach related pattern first, otherwise conversion will proceed with error reports, because program will not know how to show the component symbol on the board.

Move mouse arrow over a 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-BAT" is selected (you will see the battery symbol in the preview field). Select "Patterns" library group in "Pattern Libraries" drop-down list on the right. This library group contains all standard DipTrace pattern libraries. Select "Misc" library from the list and find "BAT-2" pattern from a pattern list at the bottom-right of the dialog box (use search filters if you want). Pin-to-Pad connections will be defined automatically. Just check them: negative pin responds to pad 2 and positive - to pad 1, like on the picture below.

If you need to define pin-to-pad connections: click on the corresponding pad number in the Pin to Pad Table and type in related pad number in the "Number" field or connect pads and pins visually - click on the pin and pad in the preview fields.



Click "OK" when you are done to close Attach Pattern dialog box and "OK" to close Component Properties dialog boxes.

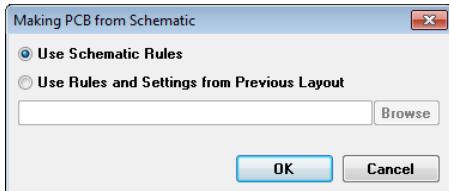
Notice that some symbols may not have attached patterns (for example, VCC, GND or other logical connectors – "Net Ports").

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

Schematic can be printed or saved in BMP or JPG file as well. Select "File \ Preview" from main menu, customize it and press "Print All" to print all schematic sheets, "Print Current Sheet" - to print selected sheet or "Save" to create BMP/JPG file with defined resolution.

2.5 Converting to a PCB

You can open DipTrace schematic files (*.dch) in PCB Layout module, but if you want to save your time it's better to select "File \ Convert to PCB" or press "Ctrl+B" directly in DipTrace Schematic, PCB Layout with your project will be opened automatically. In a pop-up dialog box you can use Schematic rules or load rules from any other PCB layout file.



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

If you plan to use another PCB Layout software you can use netlist export feature of Schematic module. Select "File \ Export \ Netlist" from main menu in Schematic, then select appropriate netlist format. DipTrace supports popular Tango, PADS, P-CAD, OrCAD and many other netlists. This is very useful when checking net structure.

However, we will use DipTrace PCB Layout module to design the board. If you want to hide layers panel and design manager to empty more space, press "F3" or uncheck "View \ Toolbars \ Design Manager" item.

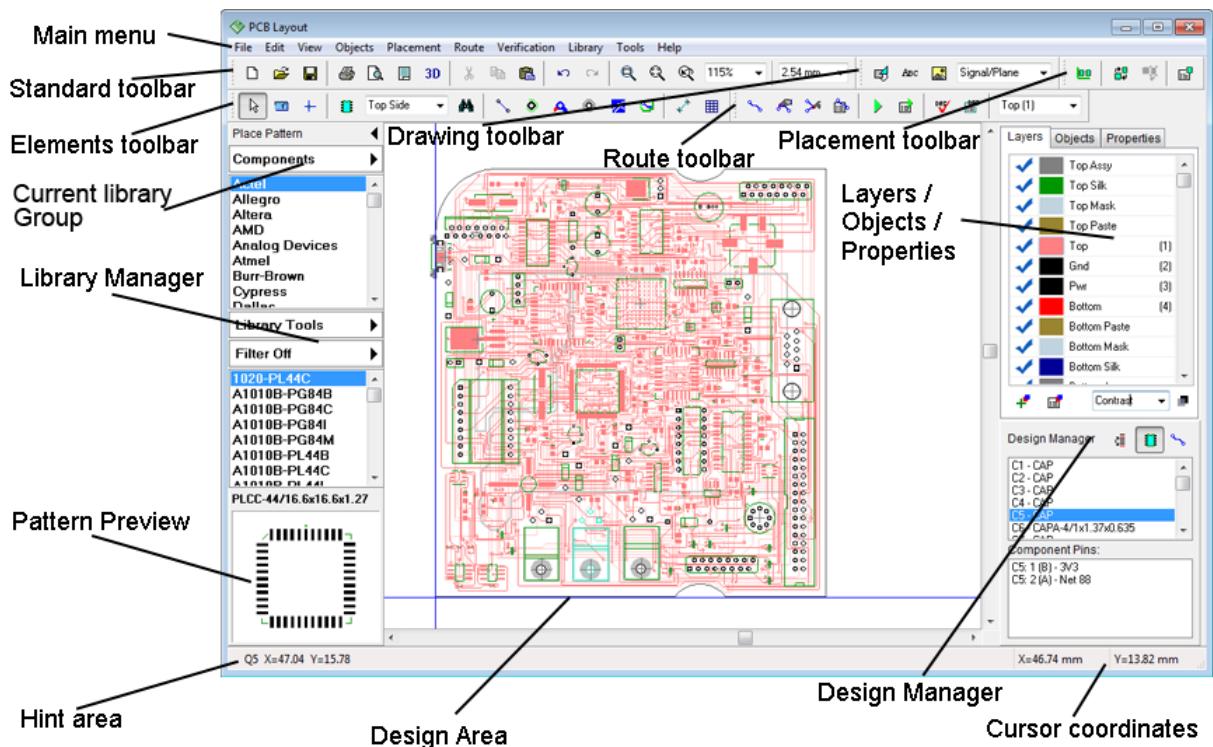
Renew from Schematic

Notice that you can renew PCB from updated Schematic file and keep component placement and current routing. Select "File \ Renew Design from Schematic" then find and open updated schematic file. Renewing from the schematic is possible with three following options:

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

2.6 Designing a PCB

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



You can place and edit different objects (components, ratlines, traces, copper pours, shapes, tables e.t.c) 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.

Library Manager - work with libraries and components: library groups, library tools, search filters, pattern preview. Select library, select pattern and place it to the design area.

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

"Layers" tab allows to work with layers (show / hide, add / delete / edit, change position and color) and change layer display mode;

"Objects" tab allows to show / hide different objects on the design area and block certain objects of being selected;

"Properties" tab shows properties of selected tool / object and allows to edit them.

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 / net.

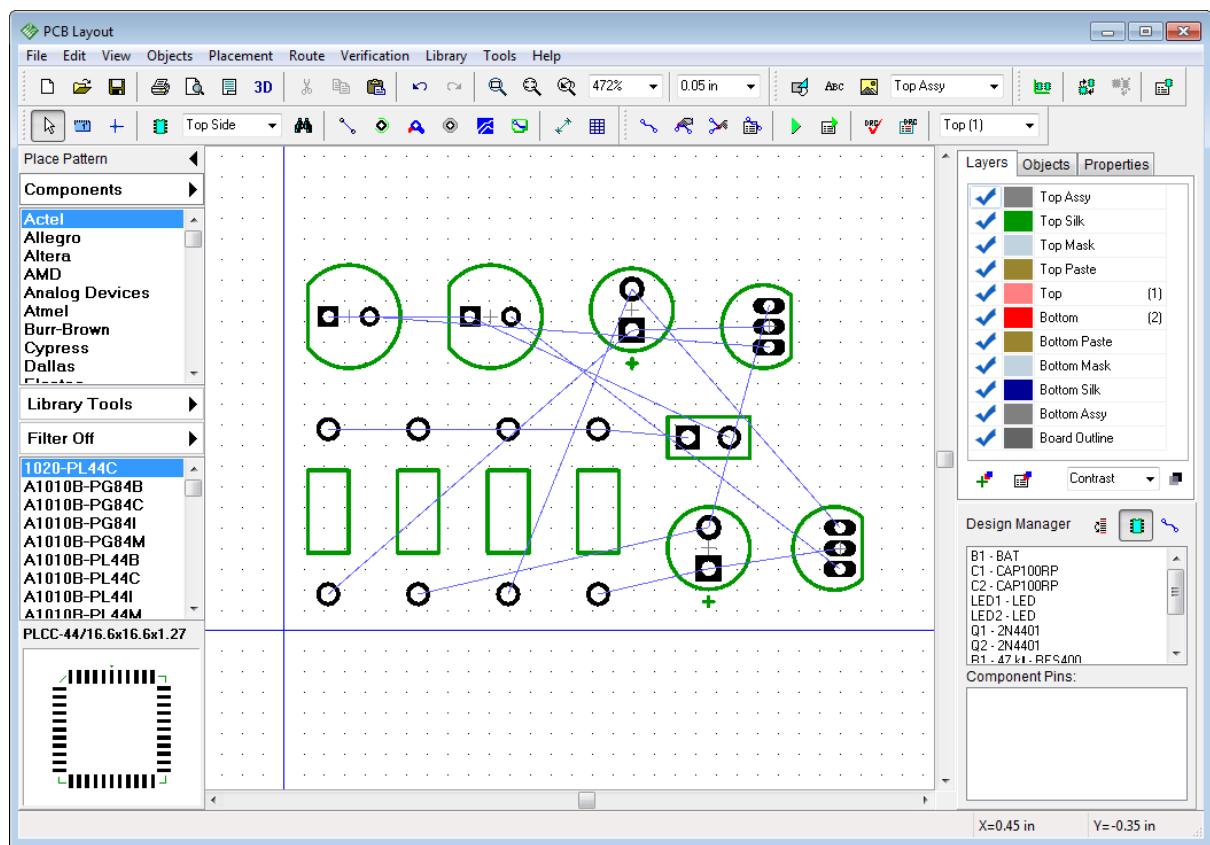
Status bar shows current hint on the left and cursor coordinates on the right.

See DipTrace Help for details ("Help \ DipTrace Help" from main menu).

2.6.1 Preparing to route

Routing itself is one of the final stages of board design. But quality of routing and quality of entire project greatly depend on preparation procedures.

Right after conversion layout looks chaotic. Press "Arrange Components" button on placement toolbar or select "Placement \ Arrange Components" from main menu and components will be placed near the design center (straight blue lines) and arranged according to placement settings. Probably automatic arrangement was not necessary for current design, because it's very simple, but this is tutorial and we try to show you how to use DipTrace even if some features are way to much for this "couple-resistors" project.



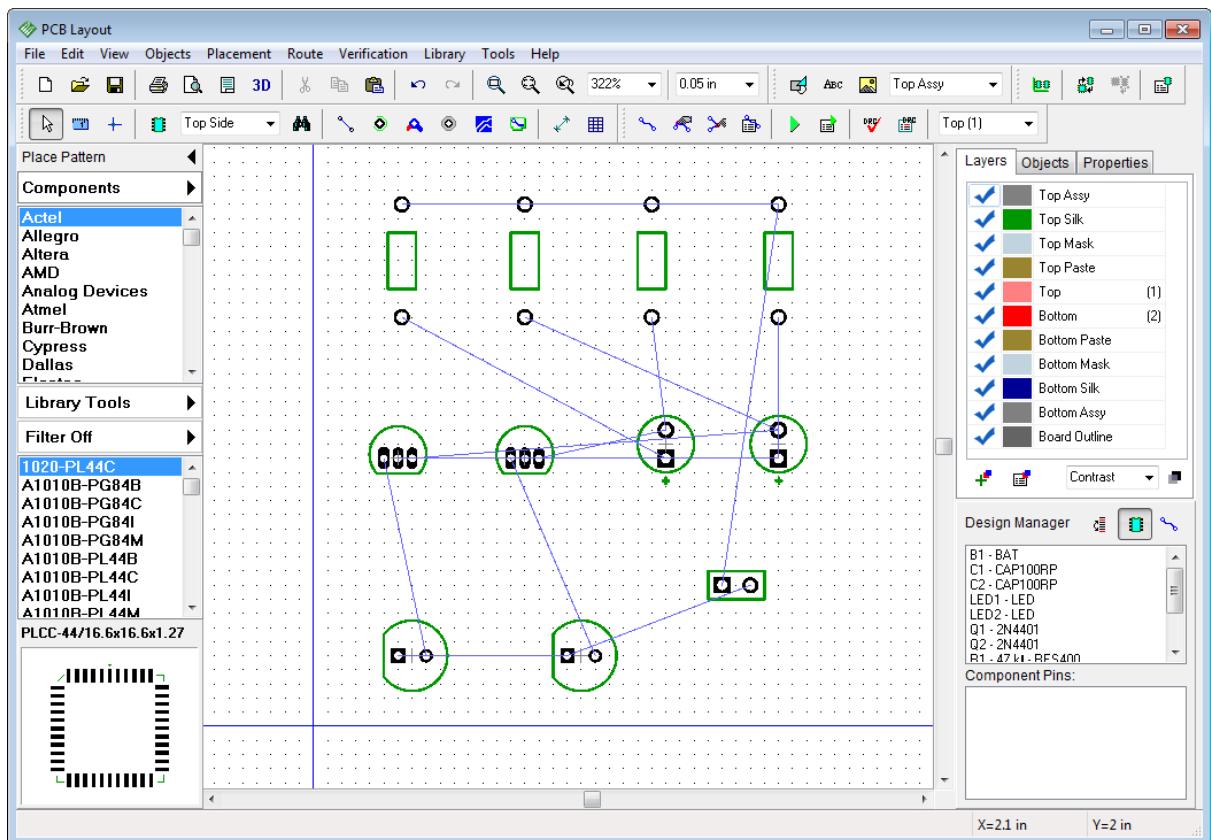
You can use "Auto-placement" or "Placement by list" after converting to PCB. These are very convenient and useful features which allow to get advantages of both automatic and manual placement modes. We will try to place parts automatically in [Part III](#)^[184] of this tutorial with more complex circuits.

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 for high frequency circuits. Notice that we use .05 inch (1.27 mm) grid. Change it if you need with a drop-down list on the instruments toolbar; change measurement units with "View \ Units \ Inch" main menu item.

Create 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" 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.



Make reference designators visible, if they are not visible yet. Select "View \ Component Markings \ Main \ RefDes". This command shows 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 size of the marking text select "View \ Component Markings \ Font \ 5pt" from main menu to make markings more visible. You can choose another size, but don't make it too big.

For PCB Layout Vector font type is strongly recommended, however Unicode and non-English characters are supported only in TrueType. Font settings "View \ Component Markings \ Font" main menu item. Vector and TrueType fonts look a bit different.

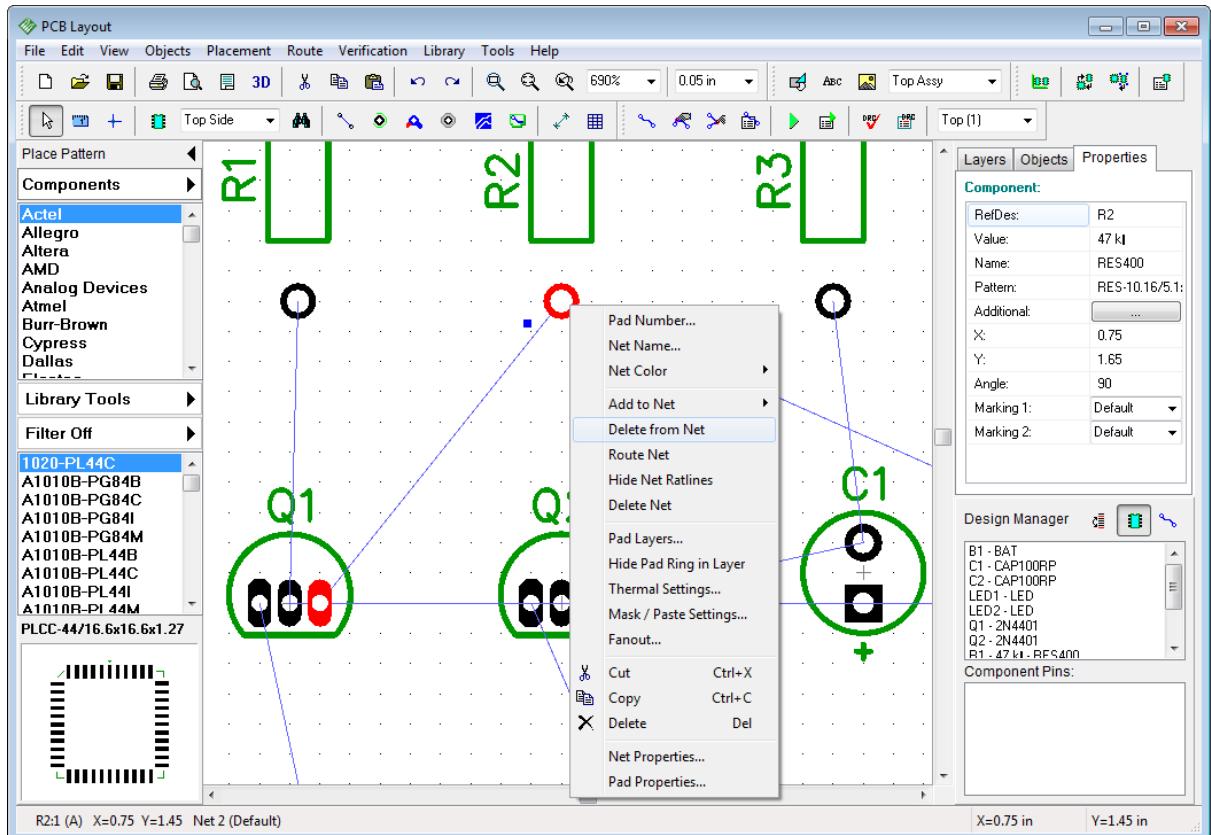
To define custom component marking parameters for selected components right click on one of them and select Properties from the submenu. Then open "Markings" tab in the pop-up dialog box. Remember that you can use move tool - "F10" or "View \ Component Markings \ Move Tool". This option allows to move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

Press "F12" to optimize configuration of connections.

Changing Net Structure

We're going to practice in changing net structure, add and remove connections (blue thin lines between pads, which are called "ratlines"). This step is not necessary for this board, because net structure is good, but we are going to show you how to do it.

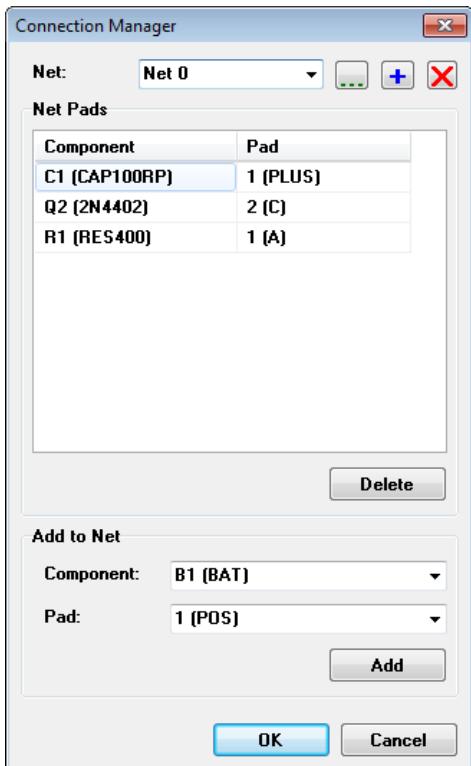
Hover over any pad, right click and select "Delete from Net" and the pad will be deleted from the net. As you can see it's no longer connected with a blue line.



If you would like to add selected pad to the net without creating connection manually (for example, you don't want to search entire design for other pads of that net) hover over this pad, right click it and select "Add to Net \ ...", then select net from the list of all nets of the project. Make sure that you have connected it back to the correct net.

You can create pad-to-pad connection manually. Select "Objects \ Place Ratline" from main menu or press corresponding button on objects toolbar. Then hover over unconnected pad, left click it and move mouse arrow to any other pad and left click on it. A new net, represented as thin blue line (pad-to-pad connection) will be created. By the way, existing connection can be deleted the same way, just select "Delete Connection" from the pop-up submenu which will appear when you click on the second pad.

But 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. It's easy to understand.



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.

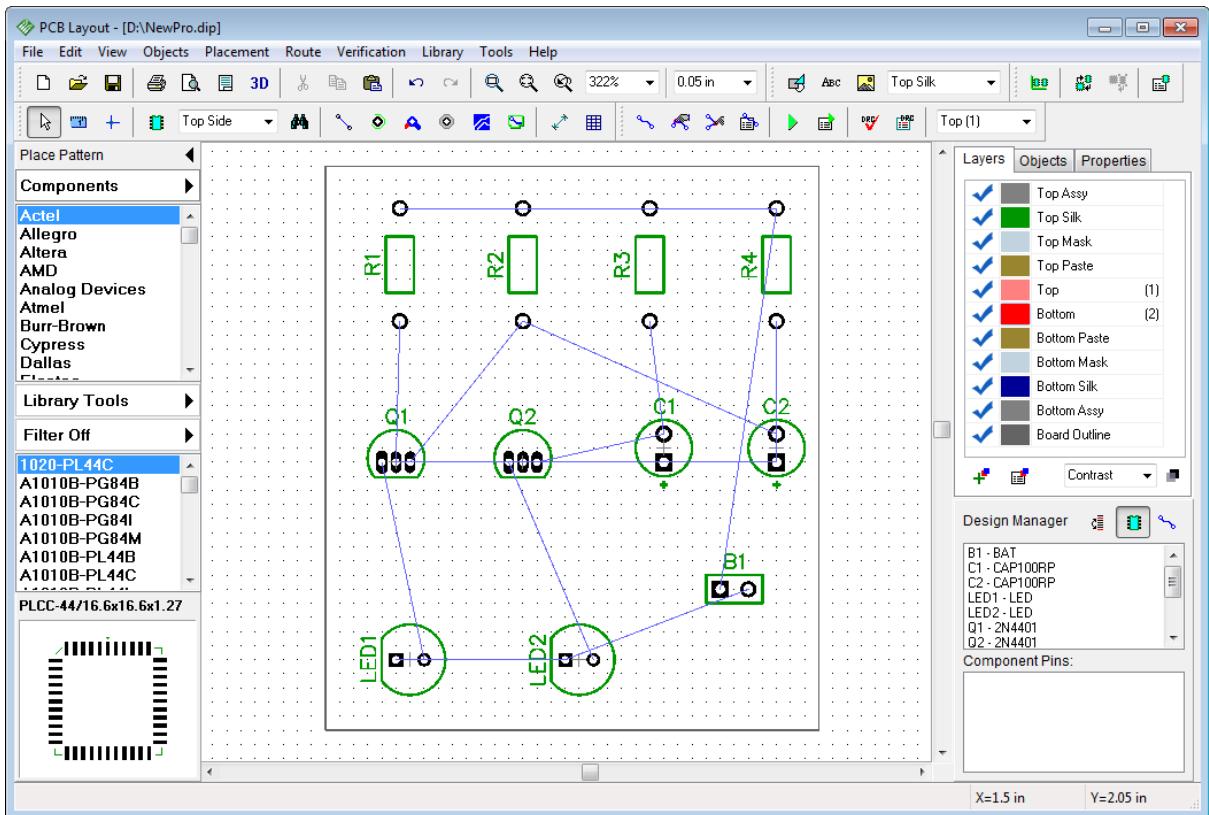
If you have changed net structure, please press "Undo" until previous design is restored. By the way, if you lose design or schematic because of incorrect exit from the program, use 'File \ Recover Board' in PCB Layout and 'File \ Recover Schematic' in Schematic to recover latest project.

To protect 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 appropriate board automatically (rectangle for simple boards). But in real life designer usually has certain board requirements well before starting the project. Board polygon can be created directly in DipTrace or imported from DXF file (complex shapes).

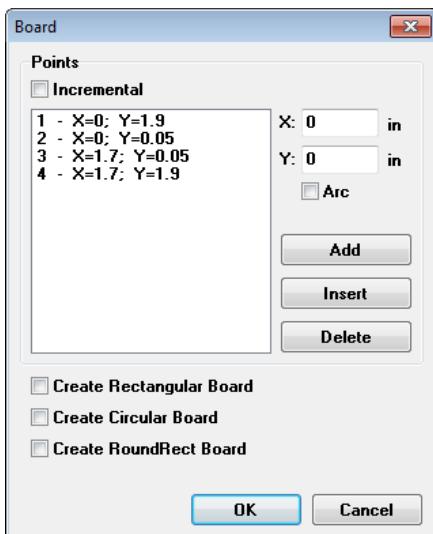
Select "Objects \ Place Board Outline" or press corresponding button on the routing toolbar and place board outline by clicking key points on the design area. Then right click on the final point of the polygon and select "Enter" from the submenu or press "Enter" on the keyboard. For this design we require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



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

You can insert point to ready board outline polygon, move each point or entire polygon. Point coordinates are shown as a hint when cursor is over outline point.

There is a convenient Board Points dialog box in DipTrace, select "Objects \ Board Points" from the main menu to open it.



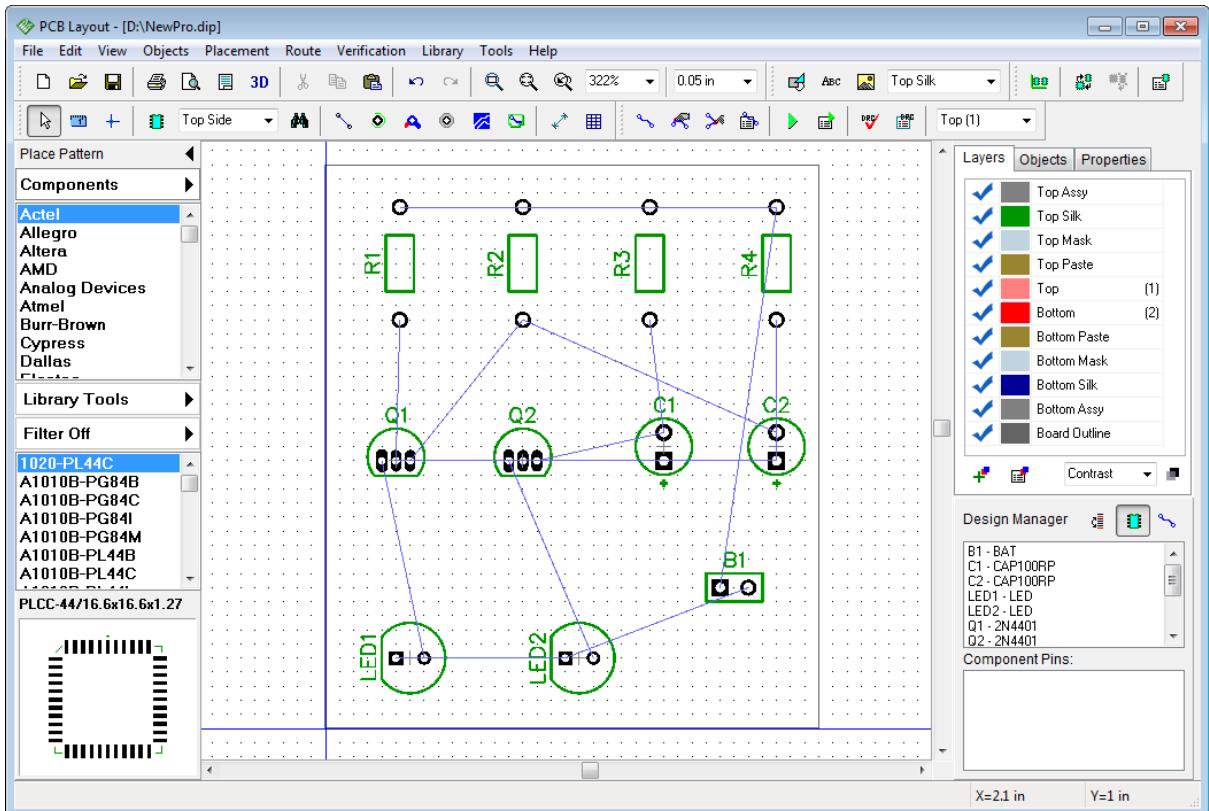
In this dialog box you can Add, Insert and Delete key points. Coordinates can be shown and edited in the absolute or incremental mode. If you check "Arc" box for some point, that point will become the middle point of an arc and neighboring points – arc beginning and ending. For rectangular boards check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

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

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

Board origin should be defined, bottom-left corner of the board outline is the best place for reference point. If you strictly followed these instructions you should see two blue lines crossing in

the correct place. However, in case you don't see origin point or it is not in the bottom-left corner of the board polygon - select "View \ Origin" from main menu or press "F1" hot key to show it. Then select Define Origin tool on the instruments toolbar (button with blue cross) and left click on the design area (DipTrace helps to target on the key points) or go to "View \ Define Origin \ By Mouse Pointer".

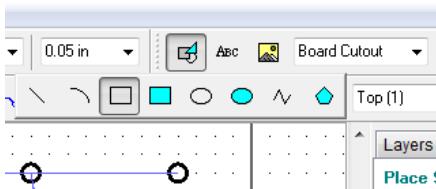


Now all coordinates in PCB Layout will be displayed and edited relatively to the origin. But you can change its position at any moment.

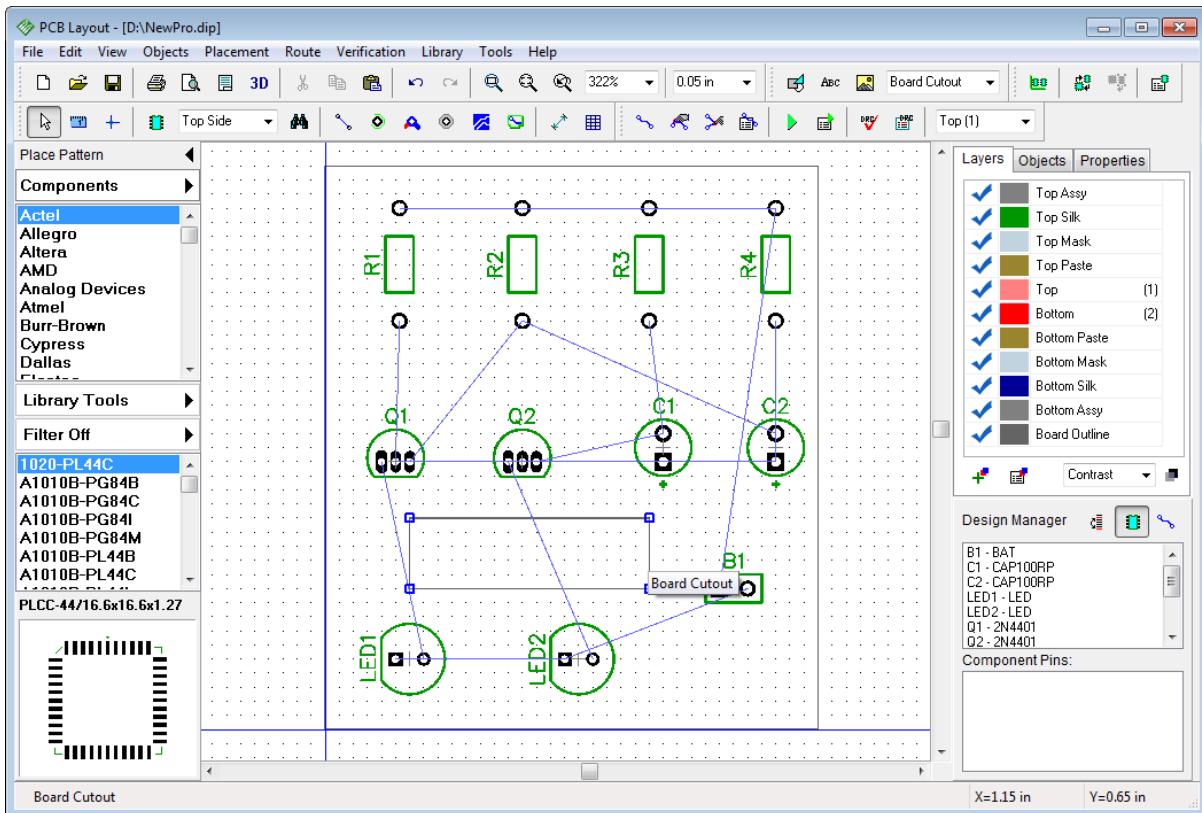
Notice that coordinates of components on the board are calculated by pattern's origin point. It is defined in Pattern Editor. To show or hide origin of selected components, right click on one of them and select "Pattern Origin" from the submenu.

Board Cutout

DipTrace allows to create board cutout polygons, this is very useful for complex projects with holes / cutouts on the board. 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.



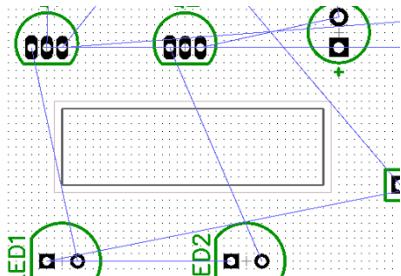
Select Board Cutout layer in the drop-down list on the drawing toolbar then choose rectangle drawing tool and create a rectangle cutout on the board in design area. Pan, zoom and change grid size for precise drawing. Board cutout is ready.



There is another way to create cutouts. Just draw a shape on any layer of the board then right click 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, hence you should be careful not to place cutout instead of outline.

Route Keepout



Please place route keepout around the board cutout. This will allow us to get clearance between copper and cutout. Select bottom layer first then select Route Keepout layer in the dropdown list on the drawing toolbar and draw a shape on the bottom layer of the board. Create a rectangle which is bit bigger than the cutout, like on the corresponding picture. Change grid size for comfortable drawing.

2.6.2 Autorouting

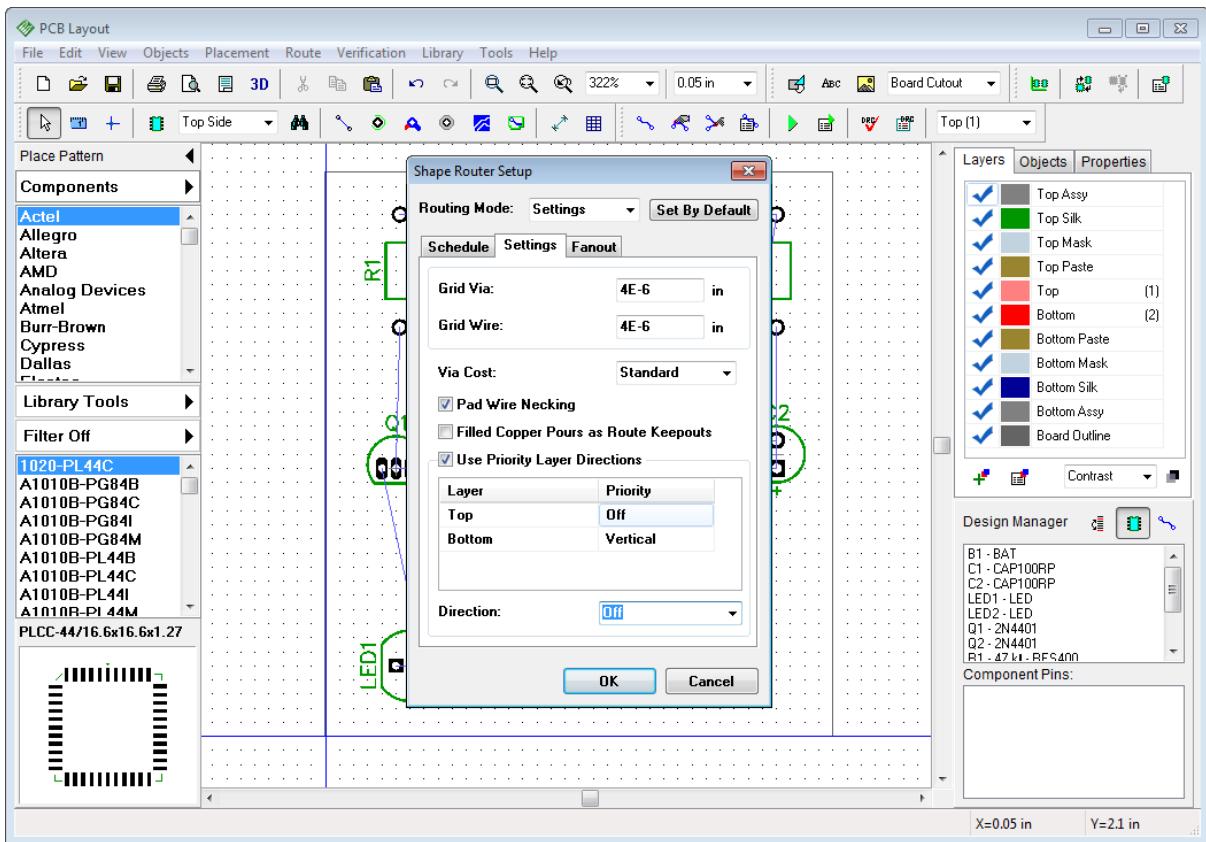
In this tutorial we are working with very simple circuit and discuss some beginner-oriented issues.

Now it is time to route your board. DipTrace has high-quality shape-based autorouter, one of the best on the market, and grid router suitable for simple PCBs and single-layer boards with jumper wire support. Our project can be routed on a single layer (usually it is a bottom side). Single-layer boards give many benefits for prototyping. Traces will be a bit longer but that wouldn't have significant effect for this design.

Select "Route \ Current Autorouter" from main menu and choose Shape Router, it's the best option for complex and simple designs (unless you need jumper cables). Router settings should be defined before starting to route, go to "Route \ Autorouter Setup".

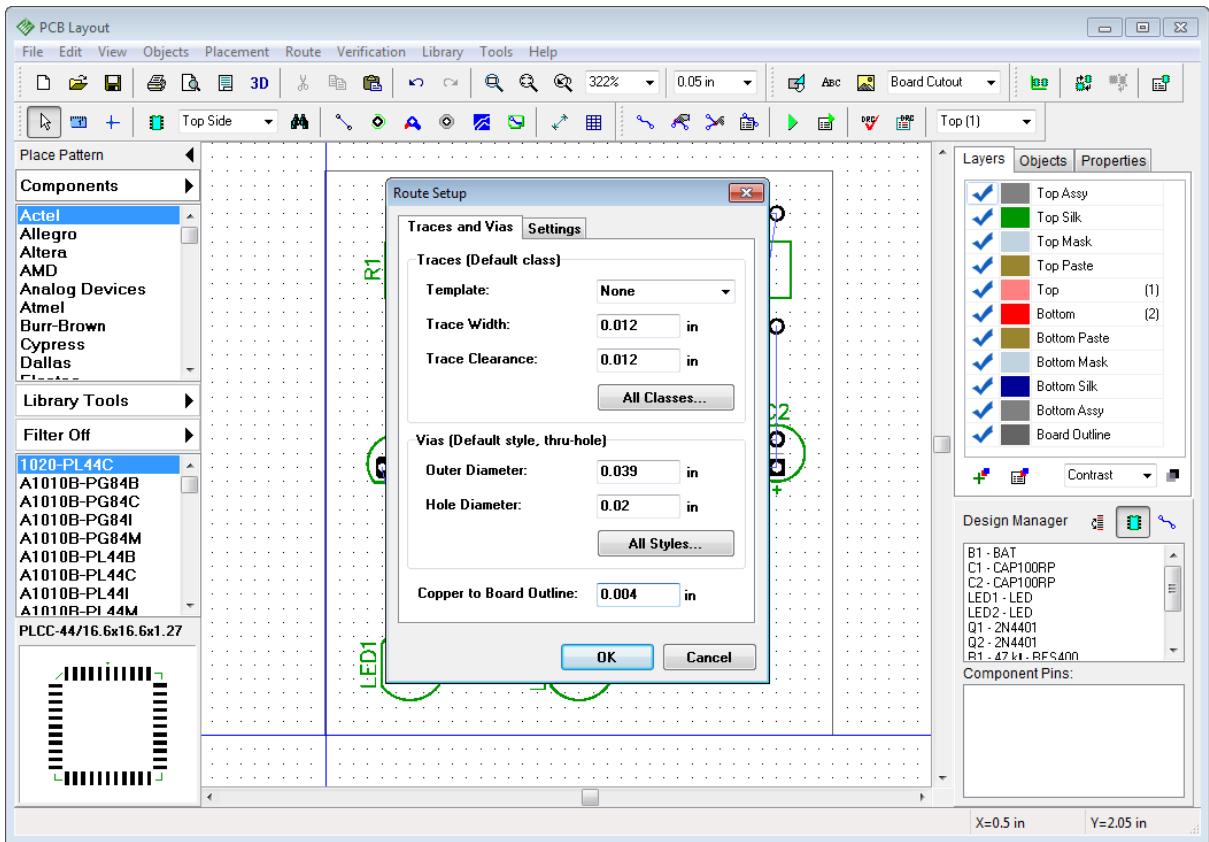
Notice that Autorouter settings depend on selected router (different panels for different autorouters).

In the Shape Router setup dialog box (which is selected now) go to "Settings" tab, check "Use Priority Layer Directions" box, select "Top" in the list of layers and set "Direction: Off" in the drop-down list below. This means that autorouter will not create any traces on selected layer. If you want to route board with jumper wires you need to select Grid Router and check "Allow Jumper Wires" box in Autorouter Setup dialog box. In our case we don't need that.



Press "OK" to apply changes.

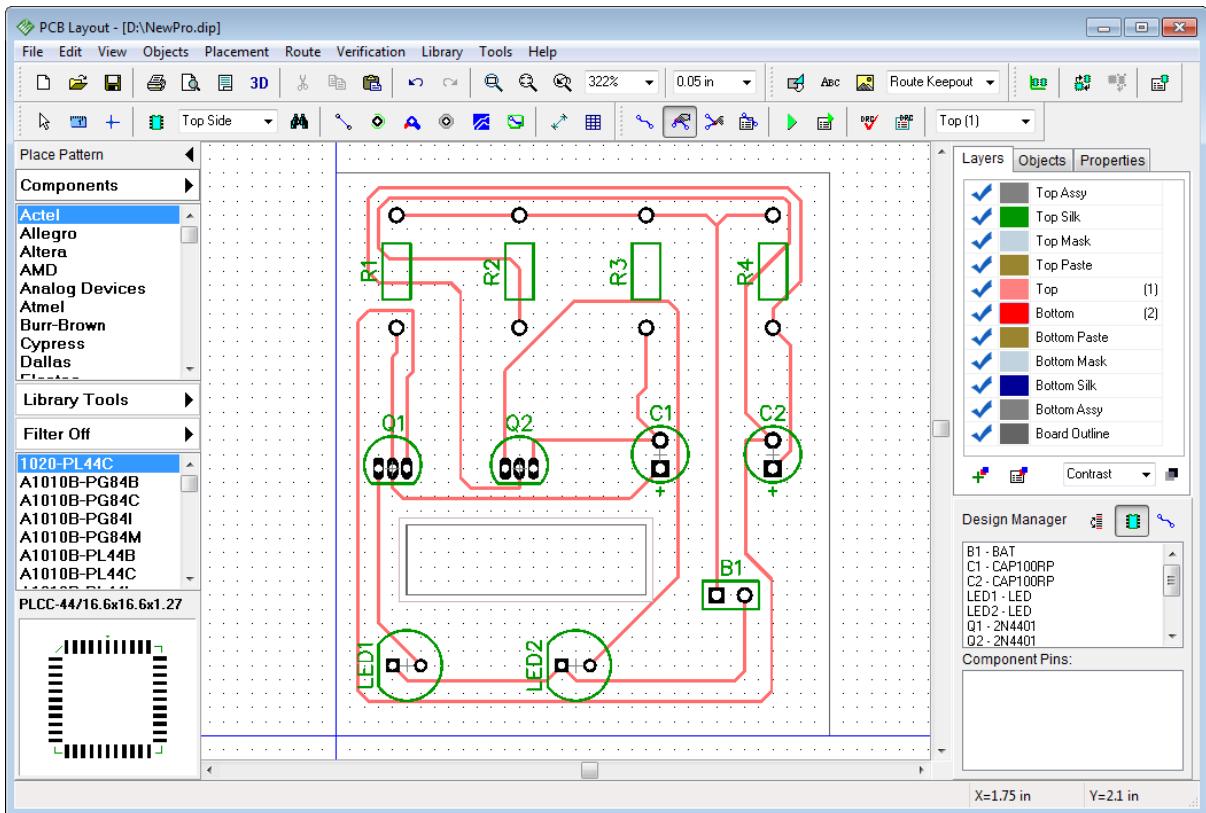
Then select "Route \ Route Setup". In the pop-up dialog box you can change trace width and clearance between traces for **default** Net Class and diameter of vias for **default** Via Style. Route Setup is the quickest way to change that parameters, but more complex projects would require using different [Net Classes](#)^[45] and [Via Styles](#)^[42]. You can press "All Classes..." and "All Styles..." buttons to access Net Classes and Via Styles panels respectively. We will discuss net classes and via styles later in this tutorial.



We strongly recommend to use settings like on the picture above for this tutorial project, it will help to avoid any misunderstandings and errors later. Press "OK" to close this dialog box and apply changes.

Now it's time to route the board. Select "Route \ Run Autorouter" from 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 the one shown, so don't be confused if you are a new to PCB Design and some traces doesn't coincide with the picture.

Notice that color of the traces depends on the layer color.



Automatic DRC

DipTrace has several verification options on different levels of design. For example, Design Rule Check (DRC). It verifies object sizes and clearances between them according to user-defined rules and reports errors in easy-to-understand list. DRC marks rule violations as red and magenta circles directly on the board. Design Rule Check in DipTrace operates in regular (offline) and Real-Time modes. If Real-Time DRC is active, you've probably noticed some red circles while moving components and creating traces. But it is turned OFF by default, therefore we will discuss verification later and concentrate on what we have now.

Regular or Offline DRC (Design Rule Check) runs automatically after autorouting. This project is very simple and you shouldn't get any errors, if there are some - make corrections and rerun DRC by selecting "Verification \ Check Design Rules" from main menu or press corresponding button on the instruments toolbar. To change design rules select "Verification \ Design Rules" from main menu. To hide red circles select "Verification \ Hide Errors". To disable automatic DRC after autorouting, uncheck corresponding box in "Route \ Current Autorouter" main menu item.

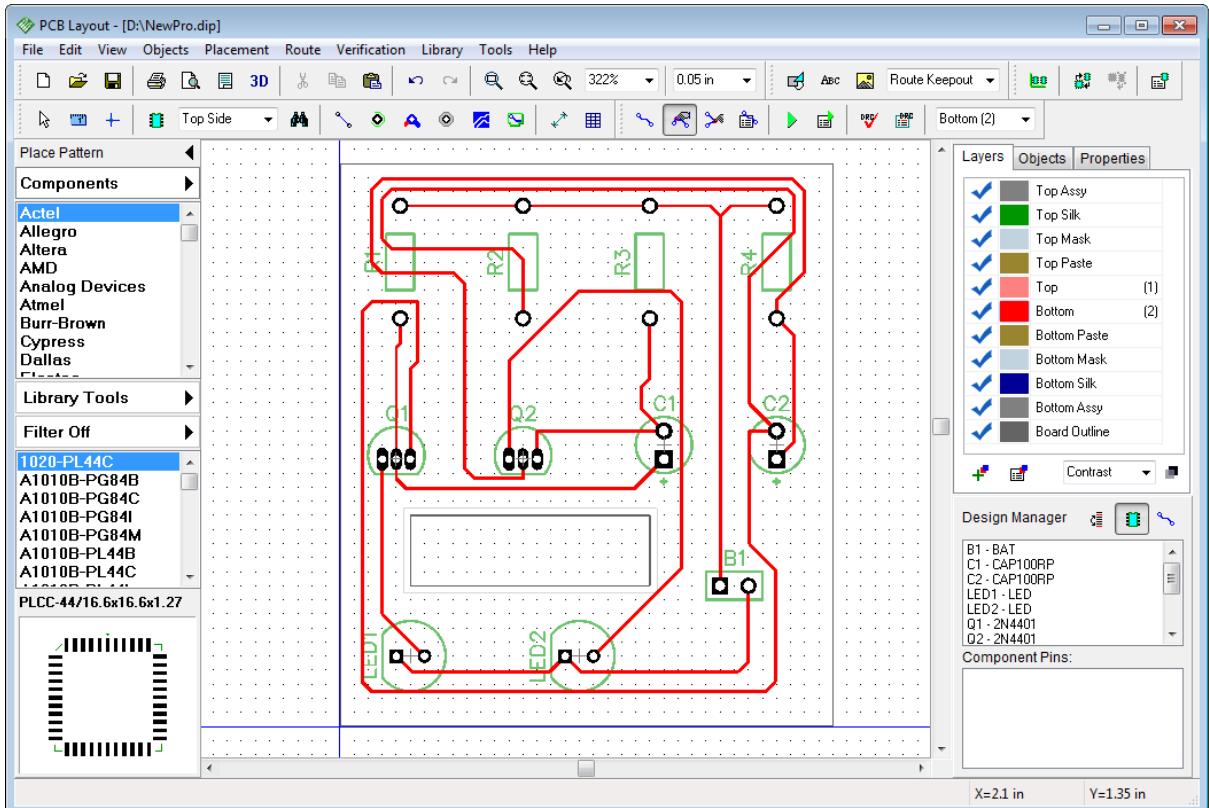
Notice that if you want to finish your project faster, you can skip all topics through ["Printing"](#)⁷⁸, because your PCB is actually ready to output. But if you want to learn some basic useful features of PCB Layout we don't recommend to skip, because your projects probably won't be as easy.

2.6.3 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 Bottom layer and "Contrast" layer display mode is active (Top layer and 50% opacity contrast mode are default settings). Look at Layers tab on the Design Manager (press "F3" if it's hidden). If you want to change active layer, double click it in the list or

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

We double click the Bottom layer in the list to make it active. Make sure you click on the layer name in the list, not on the colored rectangle or blue check mark.



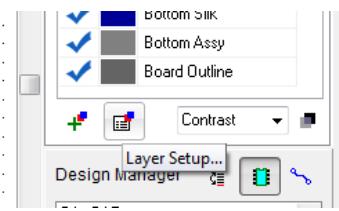
All layers can be divided in two basic types: Signal layers and Non-Signal layers. DipTrace user can easily add, delete and edit both of them.

Our project is a simple 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. DipTrace creates them automatically on both sides of the board (and gives corresponding names to each one - Top Silk, Bottom Paste e.t.c.). Each layer carries special type of information.

Top / Bottom Silk are silkscreen layers, all text and graphical information is automatically added there. Top / Bottom Mask / Paste layers carry information about solder mask and paste. Some non-signal layers are useful or even necessary for certain board manufacturers, some layers provide additional functionality, e.t.c. More information about each layer in [Gerber Output](#)^[82] section of this tutorial.

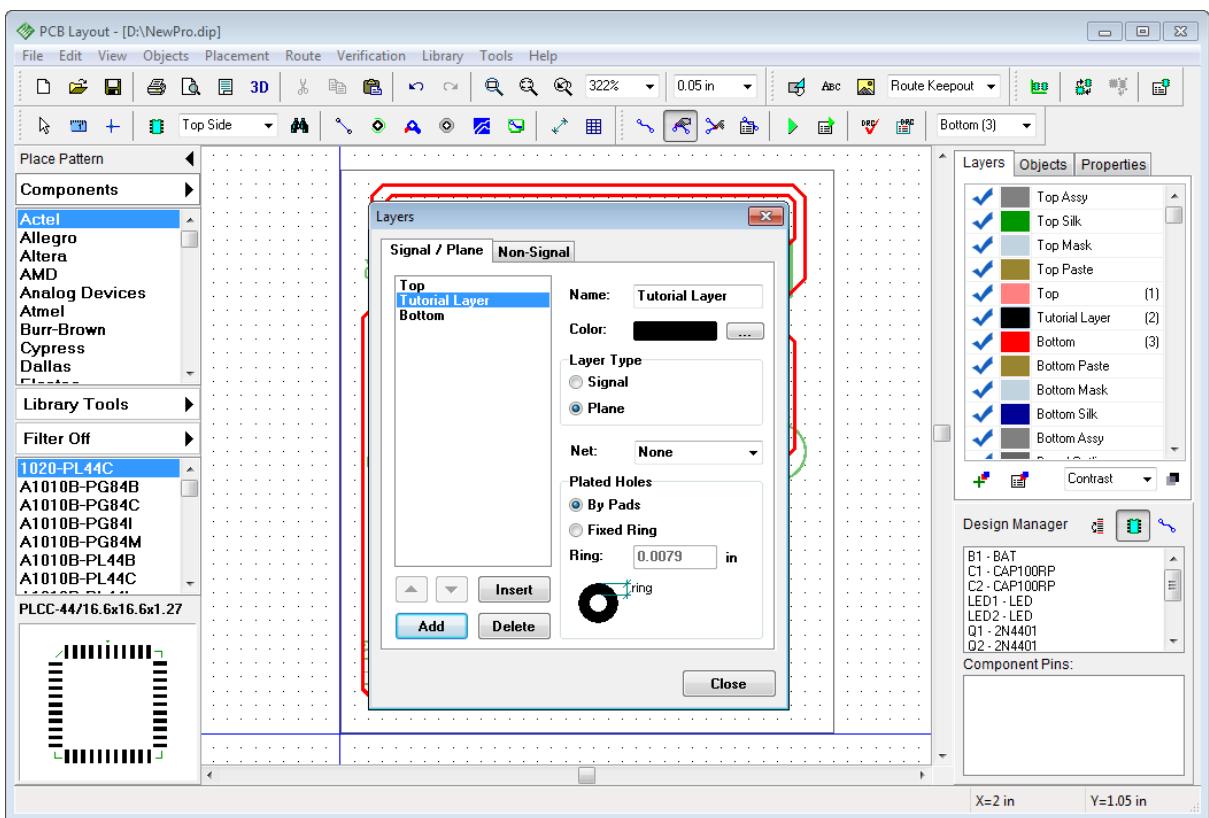
Signal Layers

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, not on planes.



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

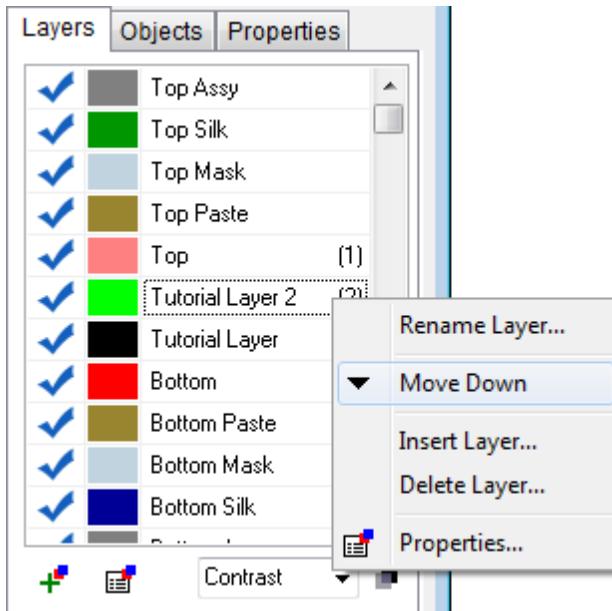
We will add a new plane layer just to show you how it works. Press "Add" button in Layers Setup dialog box, enter its name, select type and color. If you choose Plane type you can connect layer to one of project nets, usually it is Ground or Power but it can be unconnected. You can also specify details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



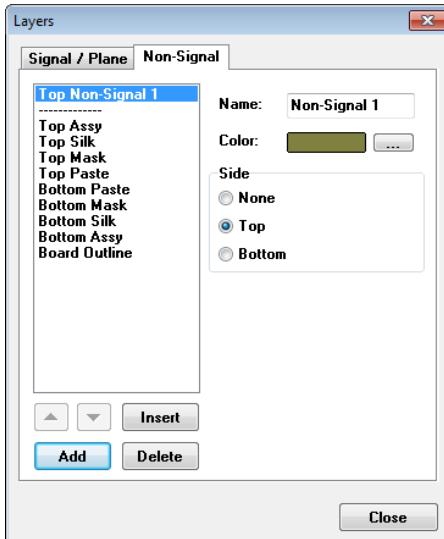
Press "Close" button.

Our new Tutorial Layer will appear 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 layer stackup. Click on the color rectangle on the panel to quickly change color of 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 design with DipTrace. If you need to create non-signal layer select "Non Signal" tab in Layers Setup dialog box ("Route \ Layer Setup" from main menu). Press "Add" button, enter 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 will not create any non-signal layers. Close this dialog box now.

There are some quick-access buttons on the Layers tab of Design Manager: first from the left - "Add Layer",

second - "Layer Properties", third is a drop-down menu of layer display mode and contrast level setup is the fourth. Remember to use 1,2,3,4 e.t.c. buttons to get access to selected layer quickly.

Pick "View \ Mirror" from the main menu to see the bottom side of the PCB.

Delete non-signal layer if you have created one, because it's not needed for this project. But do not delete plane layers, we'll use them for vias practice.

2.6.4 Working with vias

DipTrace supports Through and Blind/Buried vias (by physical properties). Vias are also divided in two logical types that don't depend on their physical parameters:

Trace Vias (regular vias), which are technically parts of traces and appear automatically when you

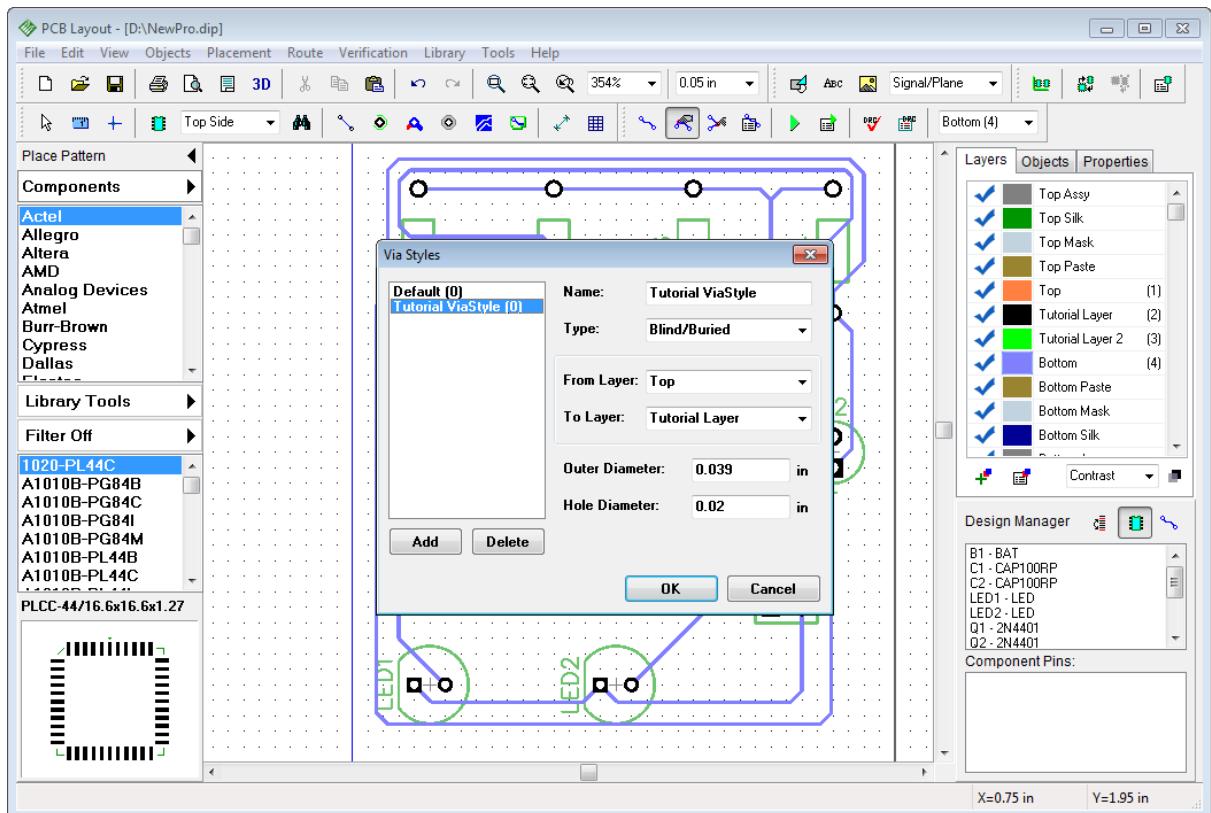
move trace segment to another layer;

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

All vias in DipTrace, disregarding their logical type are organized to Via Styles.

We don't need a lot of different via styles for current project, but we want to show you basic principles of working with vias. Go to "Route \ Via styles" and 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 layers involved (top and bottom layer of the via). In our case we make blind vias from Top layer to Tutorial Layer. Press "OK".

Notice that we've changed layer colors in previous topic.



Blind vias are impossible on printed circuit boards with only two layers, that is why we did not delete tutorial layers from the previous section of this book.

Trace Via

Now unroute one of project nets (we will manually route it, this will help us to show how to work with vias). We have chosen the net connecting resistors' pads with battery.

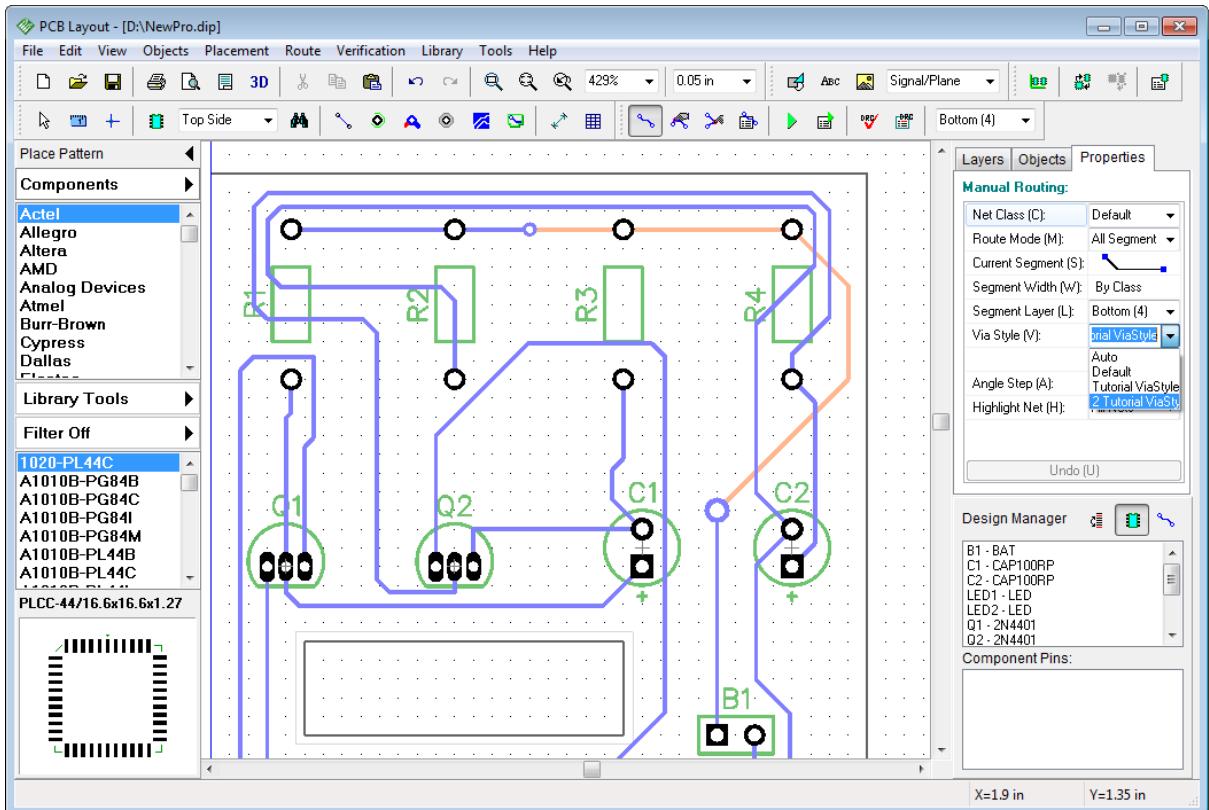
Return to Bottom layer then right click on the net that you want to unroute and select "Unroute Net" from the submenu. Go to "Route \ Manual routing \ Add Trace" from main menu or press "~" hot key. Then left click on one pad and trace starts to appear on the board. Create **only part** of the trace to some point between two pads, left click again to set part of the route and right click.

Choose "Segment Layer \ Top" from the pop-up submenu (if you're routing on the bottom layer and vice versa). Trace via will appear automatically and we can continue routing on the opposite side of

the board to another pad and then left click it. Do not route entire net.

Now let's add another via style with through-hole vias of bigger diameter than default (to clearly see difference on the board).

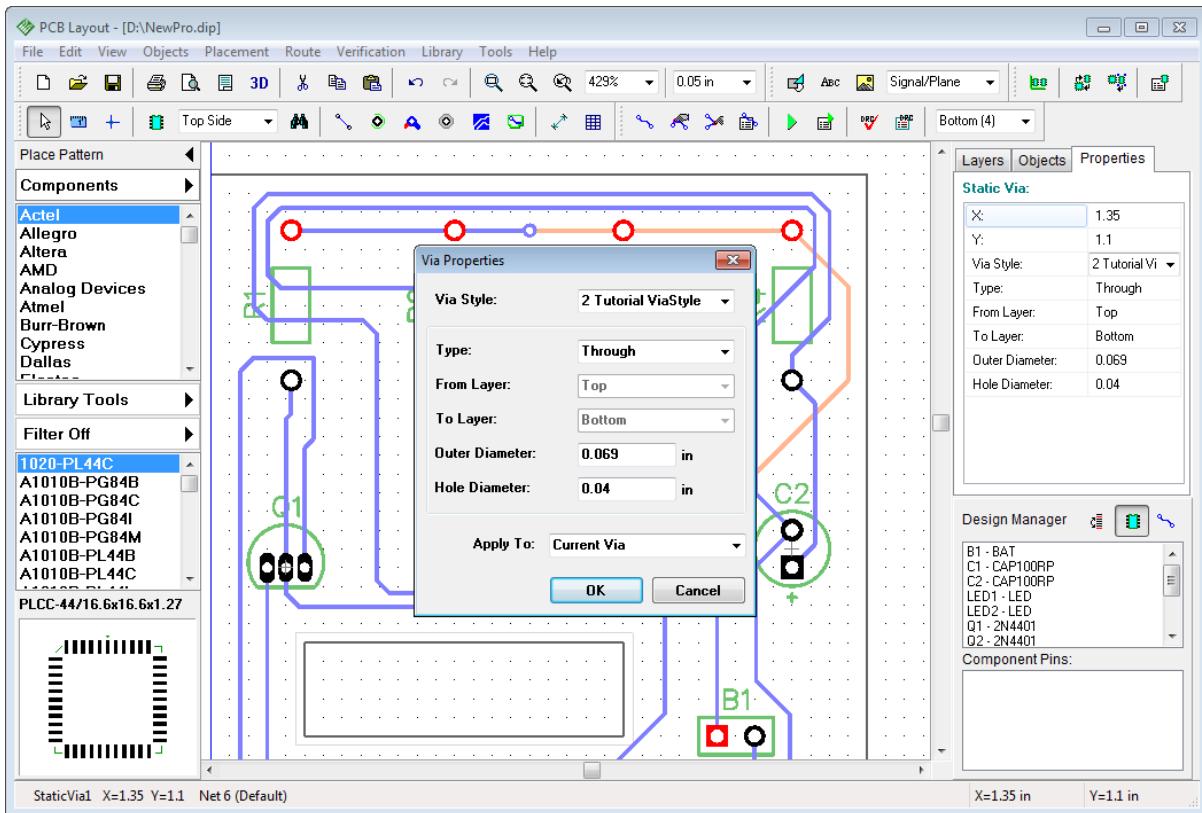
We continue manual routing of the net further and create one more trace via (picture below). While in routing mode, Manual routing panel is on the right side of the screen. In the drop-down "Via Style" menu you can choose which of the available via styles will be used for routing this net. "Auto" means that DipTrace will use via style that takes less space on the board. But in our case we select style with a big via (2 Tutorial Via Style). Then create a trace via (go to opposite board side) and finish trace.



Static Via

Go to "Objects \ Place Static Via" from main menu to create a static via or you can make it directly from a trace via, just right click on the trace via and select "Convert Via to Static". Then specify which vias to convert: Current via, Selected segments e.t.c. Static vias are basically like pads. If you change parameters of via style, all vias of that style, even those on the design area, will be changed automatically.

We can change style, type, diameter of the via and apply new settings to current or selected vias or nets on 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, DipTrace 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 (didn't convert trace via to static), you can not convert it to trace. Delete this via/s.

2.6.5 Net Classes

All nets of the project in DipTrace can be organized to Net Classes. This feature allows to apply certain parameters to any net/s with nearly one click. Net Classes can be used while routing board manually or automatically (Autorouter). Parameters of net classes should be specified **before** routing.

We are going to practice working with net classes using same project, hence 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 Default net class is available and all nets belong to this class. Press "Add" button and new net class will appear in the list of all net classes, right under Default. Left click it and type in the name.

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

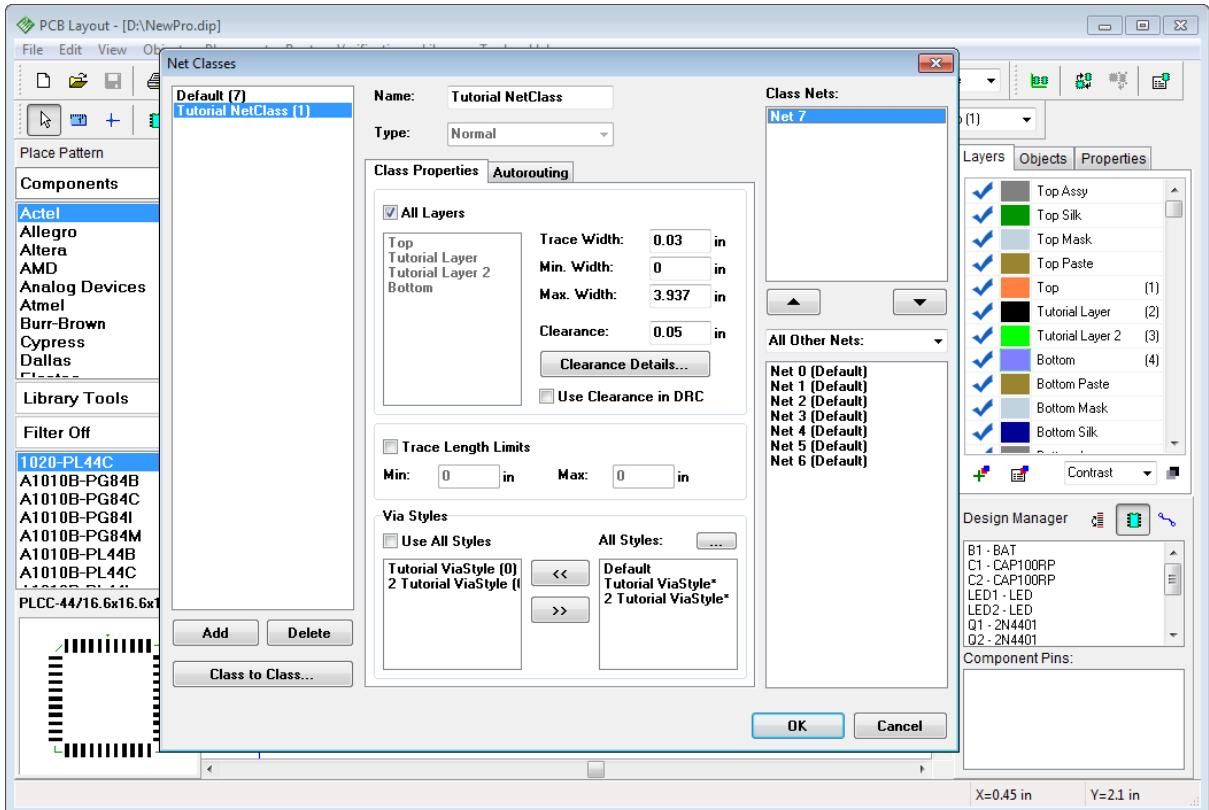
Notice that "" symbol in the input field means that it is a default parameter.*

If you uncheck "All Layers", the list under this checkbox will become active, allowing you to set different parameters of traces on each layer of the board. We don't need this now.

Uncheck "Use All Styles" in 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. "... " button allows to preview parameters of each via style. We have allowed "Tutorial NetClass" to use

only vias of tutorial styles that we've created earlier (see the picture below).

Net Classes exist, but do not have any sense if no nets belong to them. So we're going to add some. In the lower-right of the Net Classes dialog box you can see the list of nets of the project and name of net class in the brackets. In our case it is Default net class. Select one or several nets with "Ctrl" and press arrow up to add them to the net class (Class Nets list right above).

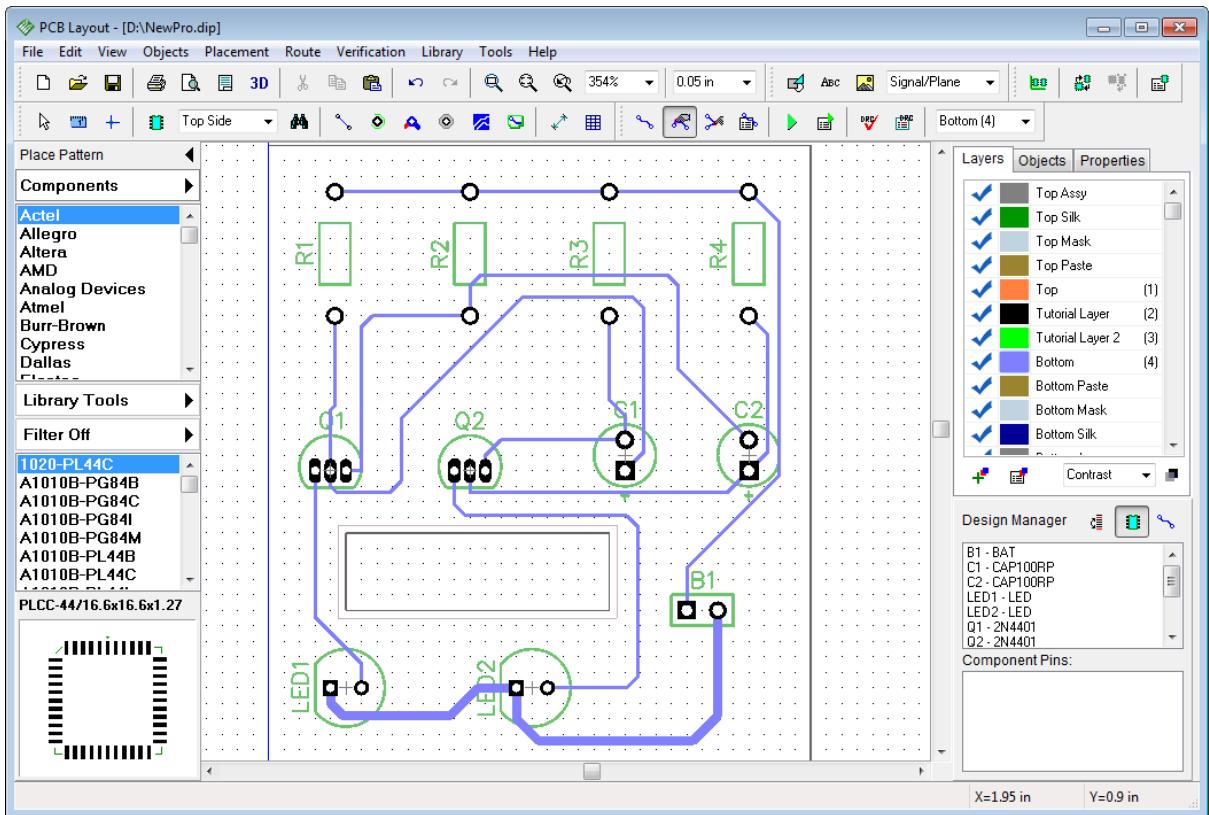


As you can see Tutorial NetClass contains one net (Net 7) with .03 inch trace width.

"Clearance Details" button allows you to set clearances between different objects. "Class to Class" - specifies clearance between 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 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" hot keys and you'll get something like on the picture below. As you can see traces on the PCB have different width, because they belong to different net classes with different parameters.

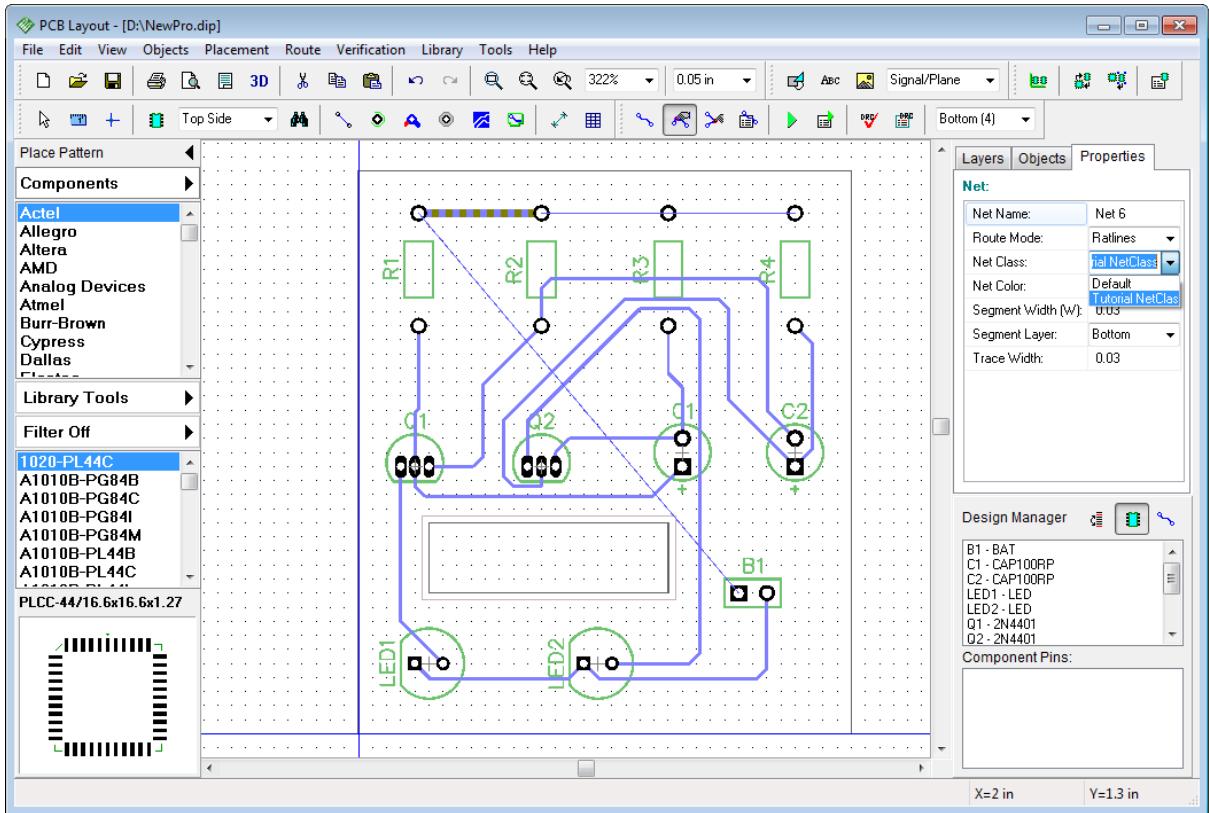


If everything is clear, unroute the board again ("Route \ Unroute All"), open Net Classes dialog box and reassign all nets (Net7) from Tutorial NetClass to Default class (use down arrow button below Class Nets list). Press "OK" then launch Autorouter again and you'll get board with traces of the same width. Tutorial Net Class still exists but it doesn't influence the board, because no nets belong to it.

Manual routing with Net Classes

Select Bottom layer and left click on one of the nets (for example Net 6), you'll see Net Properties panel on the Design Manager to your right-hand side. In Net Class drop-down list change net class to Tutorial Net Class. Then right click on the same net and select "Unroute Net" from the submenu. Now go to "Route \ Manual Routing \ Add trace" or press "~" hot key to activate manual routing mode. Left click on one of the pads and create trace to another pad and left click it to create a trace segment. You'll notice that trace is much wider because it is in another net class than the rest.

Notice that DipTrace allows to change 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 diversity on the board. Please **Undo ("Ctrl+Z")** several times to get board right after autorouting with all nets of the same width, no Tutorial Layers and don't forget to delete all net classes, except, of course, Default.

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

2.6.6 Manual Routing

Simple projects like ours can be routed automatically but for complex boards manual routing becomes inevitable. Actually, entire board can be routed manually but because of low speed of manual routing usually combination of two methods is the best choice for complex projects if you want to get a well-working prototype in reasonable terms. Critical nets are routed manually and the rest - with autorouter.

Our simple board is good without manual routing but the goal is to show you how to work in DipTrace. Moreover, sometimes you may need to correct traces after autorouter.

Editing modes

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

Editing traces is a bit different. Go to "Route \ Manual Routing \ Edit Traces" from main menu to enable "Edit Traces" mode 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 to move traces with 45 or 90 degrees angles. This is very convenient for

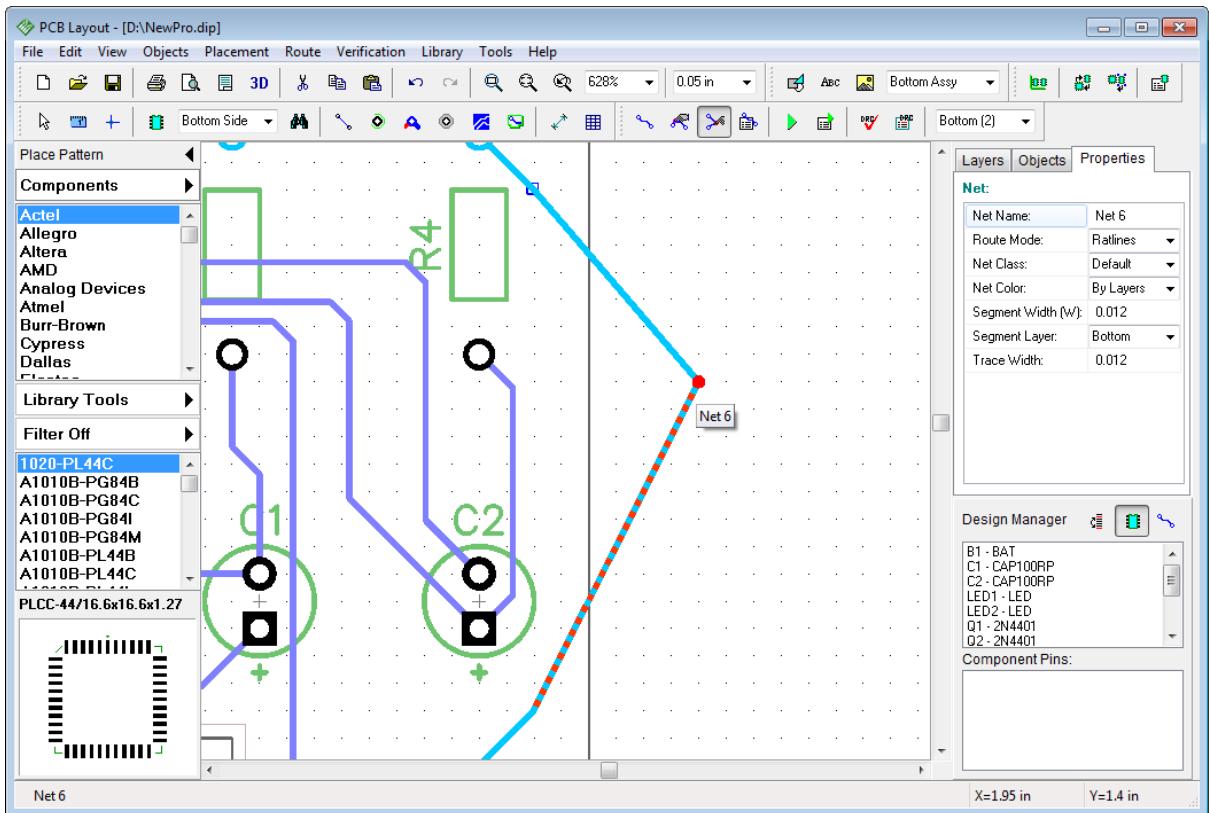
almost any designs, but sometimes you might need traces editing tool with more capabilities. Go to "Route \ Manual Routing \ Free Edit Trace" or press corresponding button on the Route toolbar. Now you can edit traces without any restrictions.

Don't forget to change grid size (on standard toolbar or "Ctrl+" and "Ctrl-" hot keys). To configure list of available grids, select "View \ Customize Grid" from main menu. "F11" - to hide grid.

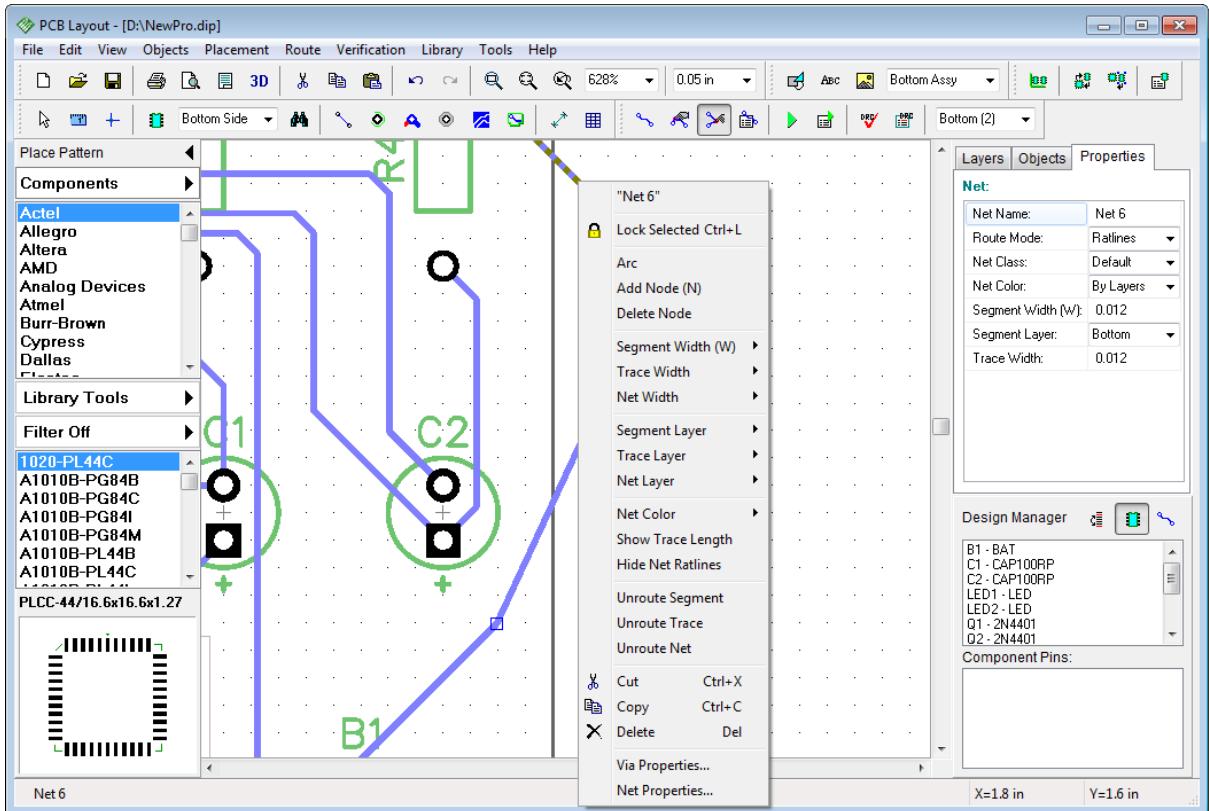
Remember, if you don't know which tool you are working with, right click couple times on a free area of the board and DipTrace will return to default mode.

Nodes

Any routed net is divided into traces and segments. Trace is the route between two pads of the net and segment is the route between two nodes. Node is a point on the route which divides trace to segments (red dot and 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' hot key to add new node in selected place, then select and move this node to some point outside 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 net name, select color, change width and layer of the net, trace or segment e.t.c.

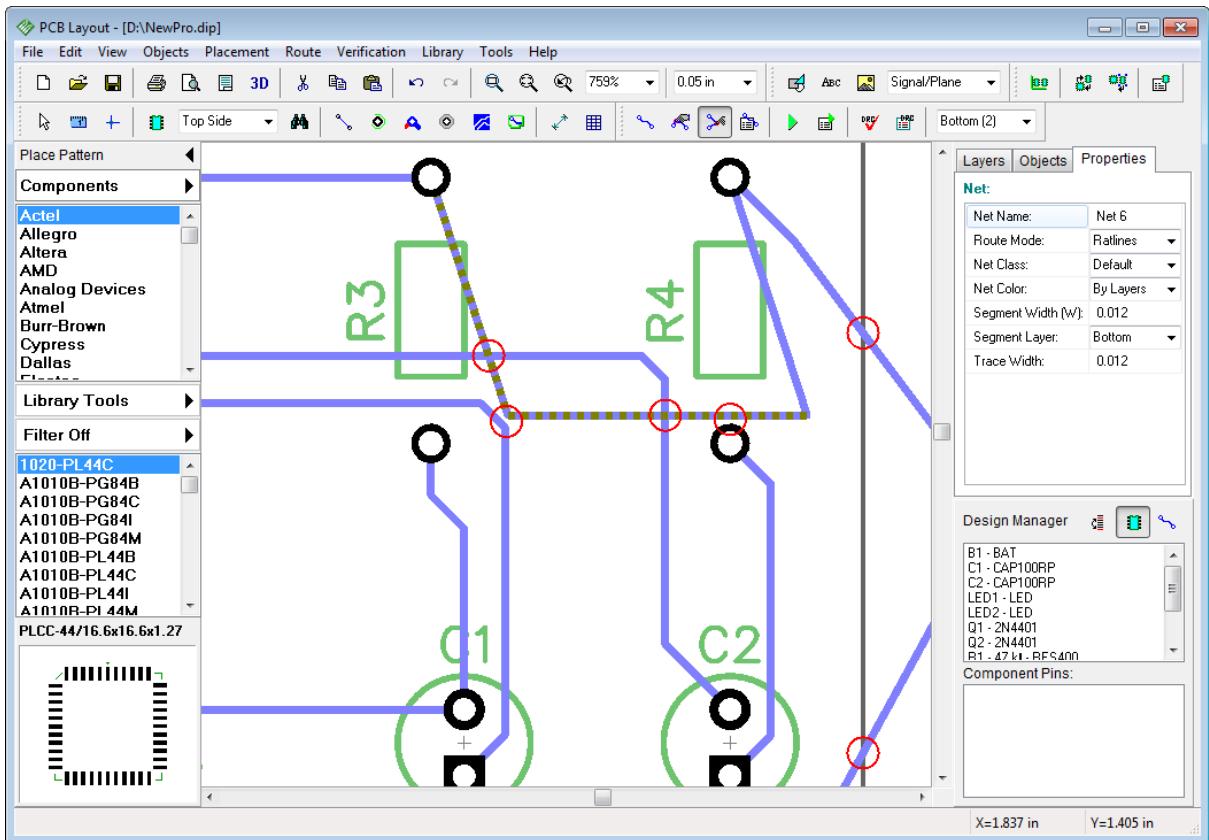


Manual routing offers great opportunities thus providing more chances to commit an error. Fortunately DipTrace has Real-time Design Rules Check, which verifies board in real time and enables user to see errors **before** actually making them. We will have a detailed discussion about DRC [later](#)⁷⁰, but we can't avoid some basics..

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 Design Rules dialog box. You can see that two red circles have appeared where 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 which report about clearance errors will appear **before** you place trace to a new position, the same happens when you move objects or edit components if corresponding items in Real-time DRC tab have been checked.

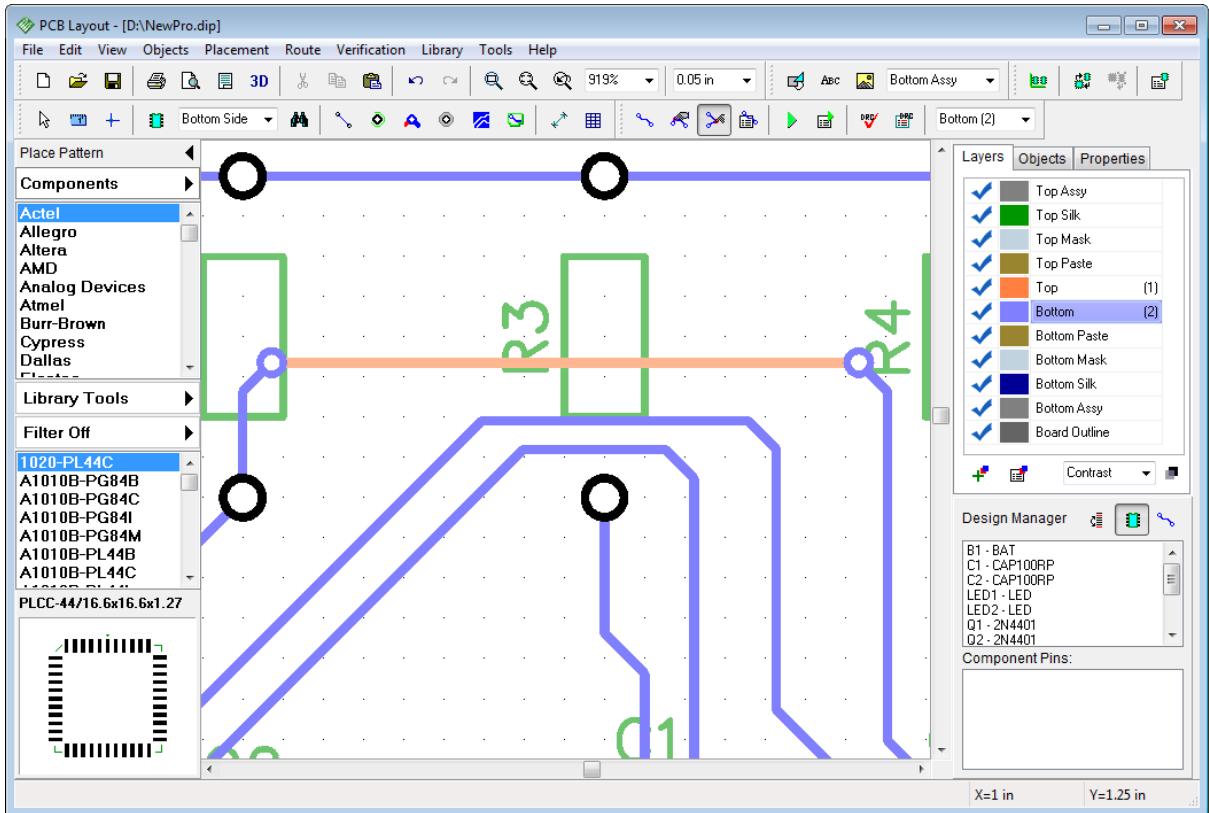


Notice that we have hidden grid on the picture above ("F11" hot key).

Now return trace (between R3 and R4) to initial position.

Change Layer

DipTrace allows to move existing net (trace or 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 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.



Change current layer to Top, right click on that segment and move it back to bottom layer. Then return to bottom layer again.

Manual Routing Panel

It's time to manually route some net. Right click on one of the nets and select "Unroute Net" from the submenu. We have selected Net 6, but you can choose another. "Unroute Net" command from net submenu is applied to all selected nets. Select "Route \ Manual Routing \ Add Trace" from the main menu or press corresponding button on the Route toolbar.

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

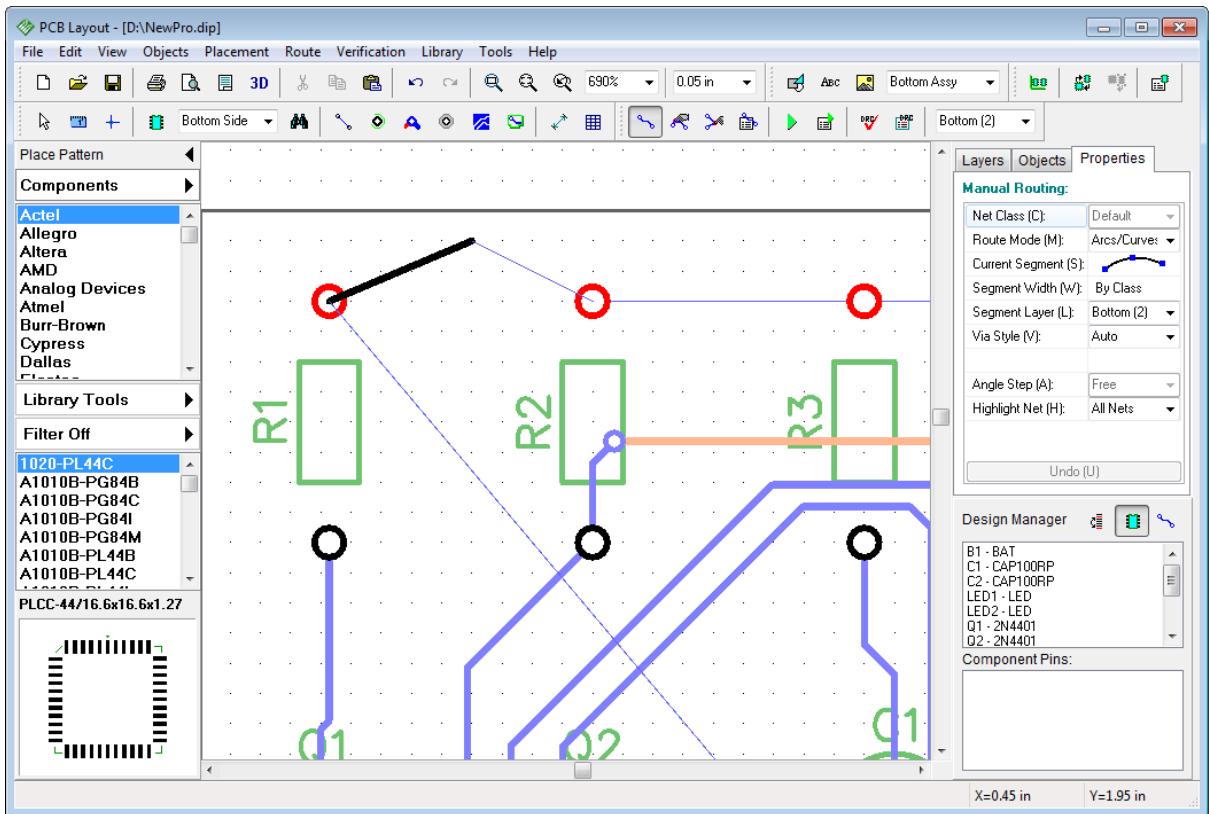
Notice that you cannot change net class of existing net in Manual Routing Panel. This change will be ignored and applied only to the new net. Don't forget that net exists not depending on whether it is routed or unrouted.

Fortunately, we have only Default net class, this helps newbies to avoid confusion and concentrate on the subject.

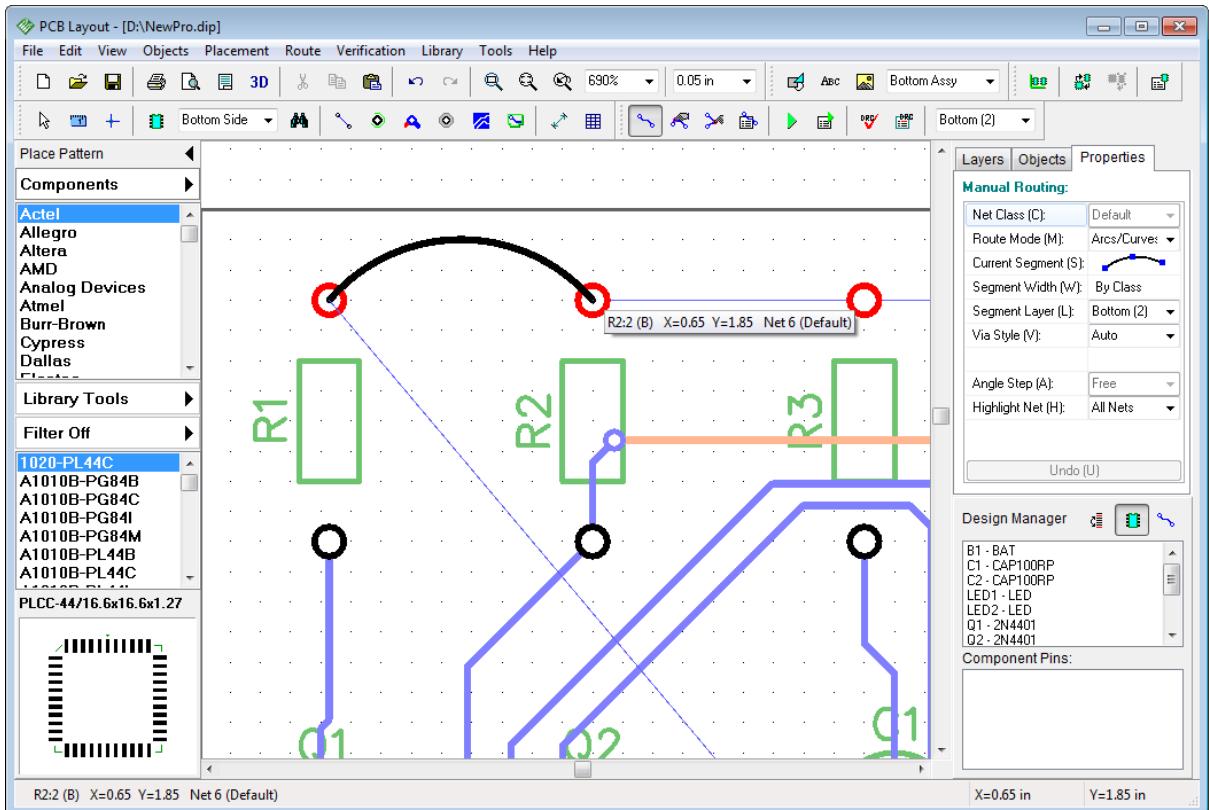
In "Route Mode" drop-down list we can specify the group of trace segments that we are going to need, therefore we will select current segment not from the list of all segments but from the list of segments of one mode. Custom route mode is available.

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 the arc), then left click on some point between two pads higher than the blue connection line (this is the second point).

Like on the picture below.



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



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

Notice that there are hot keys that will make manual routing really easy and quick.

"M" - switch between 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 wire or back (if you are in Bottom layer, jumper will be placed to Top and vice versa),

"A" - angle step,

"H" - highlight net,

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

You can undo by pressing "U" button while routing.

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

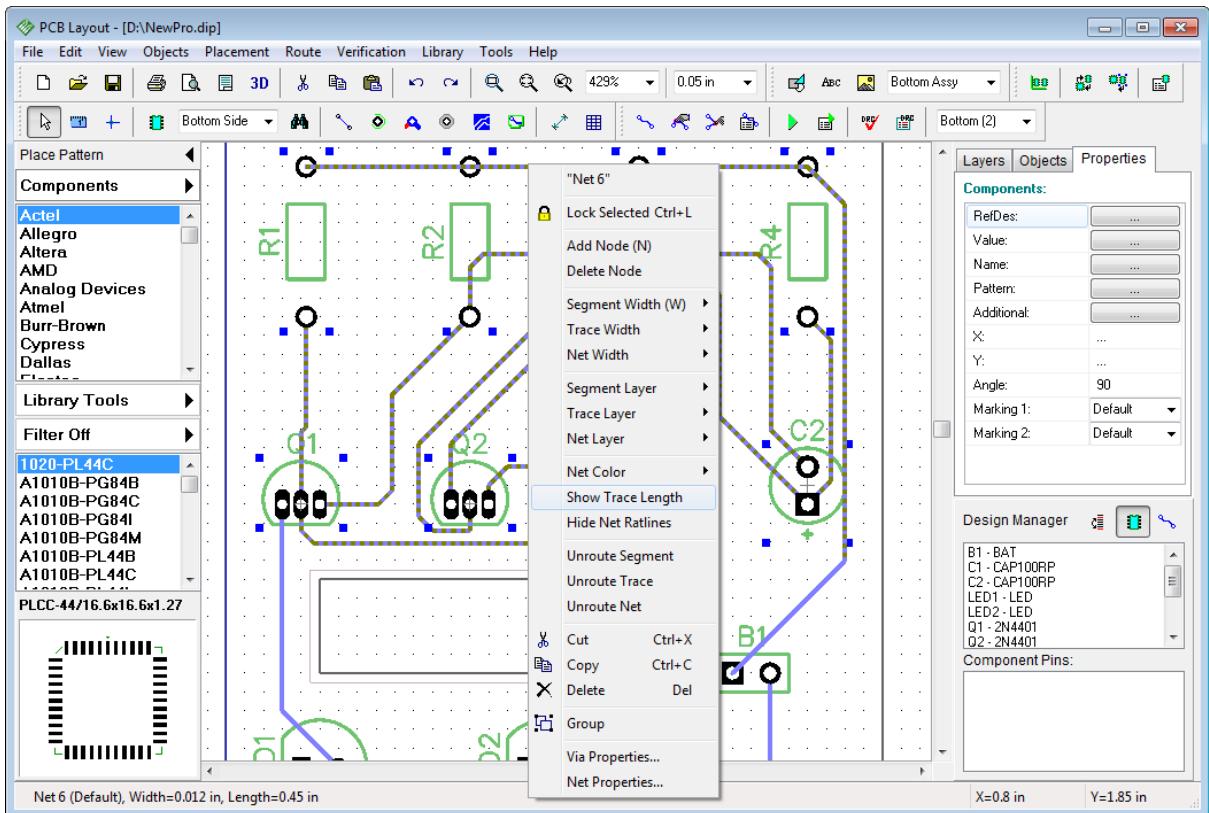
2.6.7 Measuring trace length

DipTrace allows for easy and convenient trace measuring option. Current project is simple and low-speed, thus we don't need to use this tool, but if you design high-speed circuits, video devices, e.t.c. trace length becomes very important.

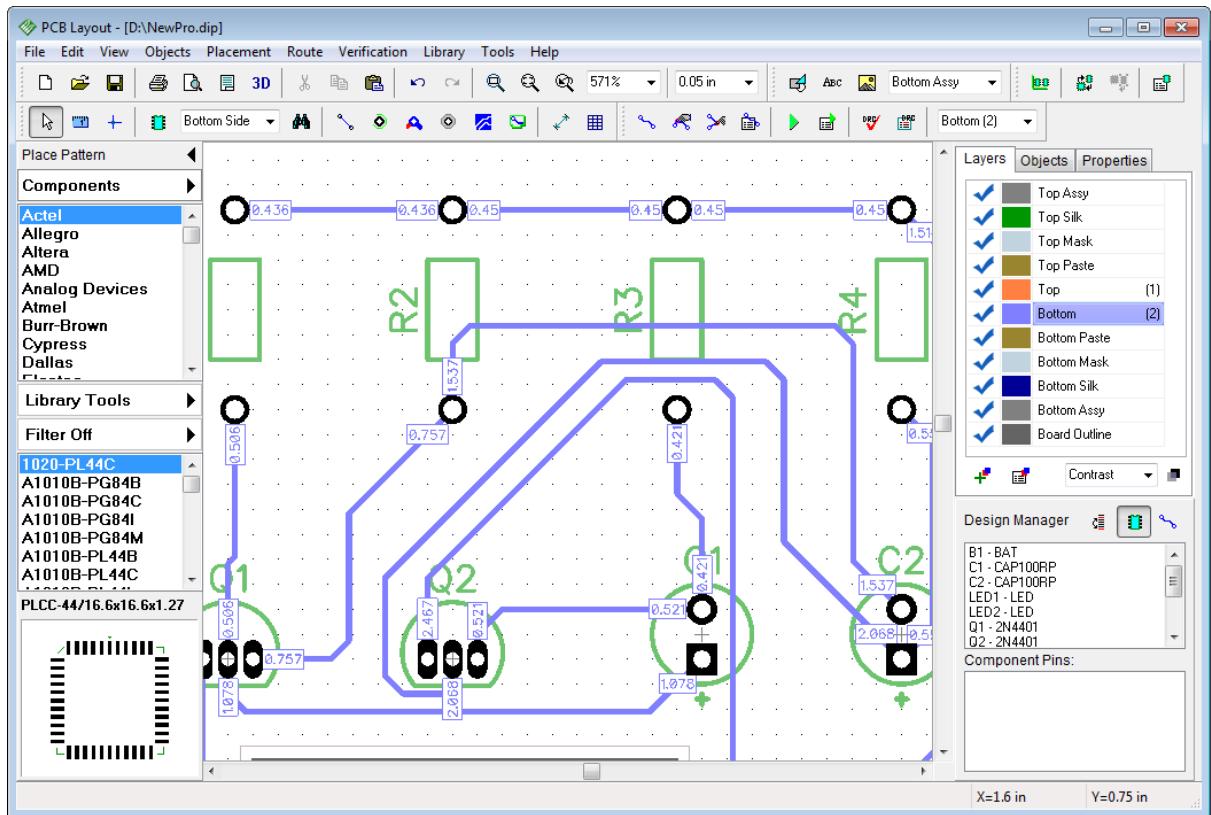
Notice that hint of each trace already includes its length.

This is helpful, but not enough for comfortable routing of complex designs.

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



You will see small boxes with trace length values near all pads of selected nets, they are also highlighted when hovering over the trace. 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. Traces color depends on layer color.

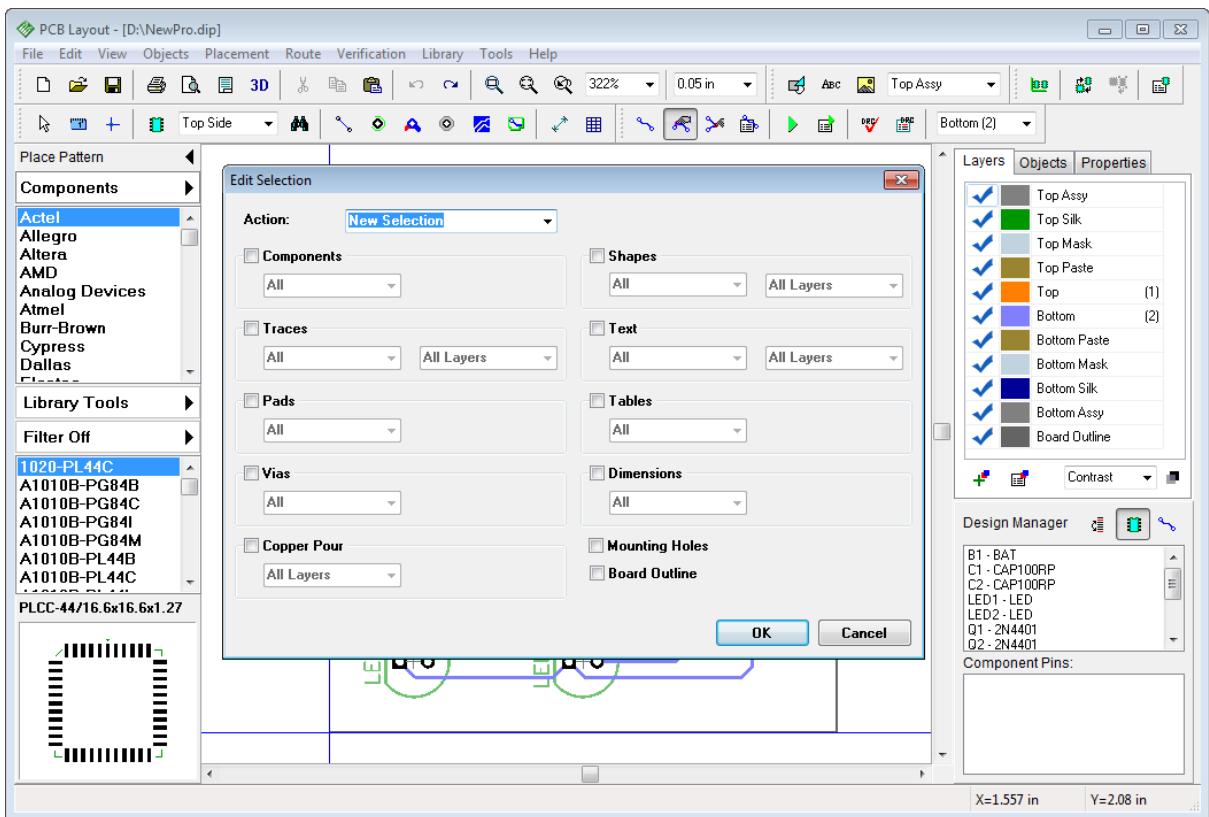


Now please hide trace length, using net submenu (select the same item) or **Undo**.

2.6.8 Selecting objects by type / layer

Sometimes it is necessary to select all objects on one layer or exclusively components, nets, vias e.t.c. 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 main menu.

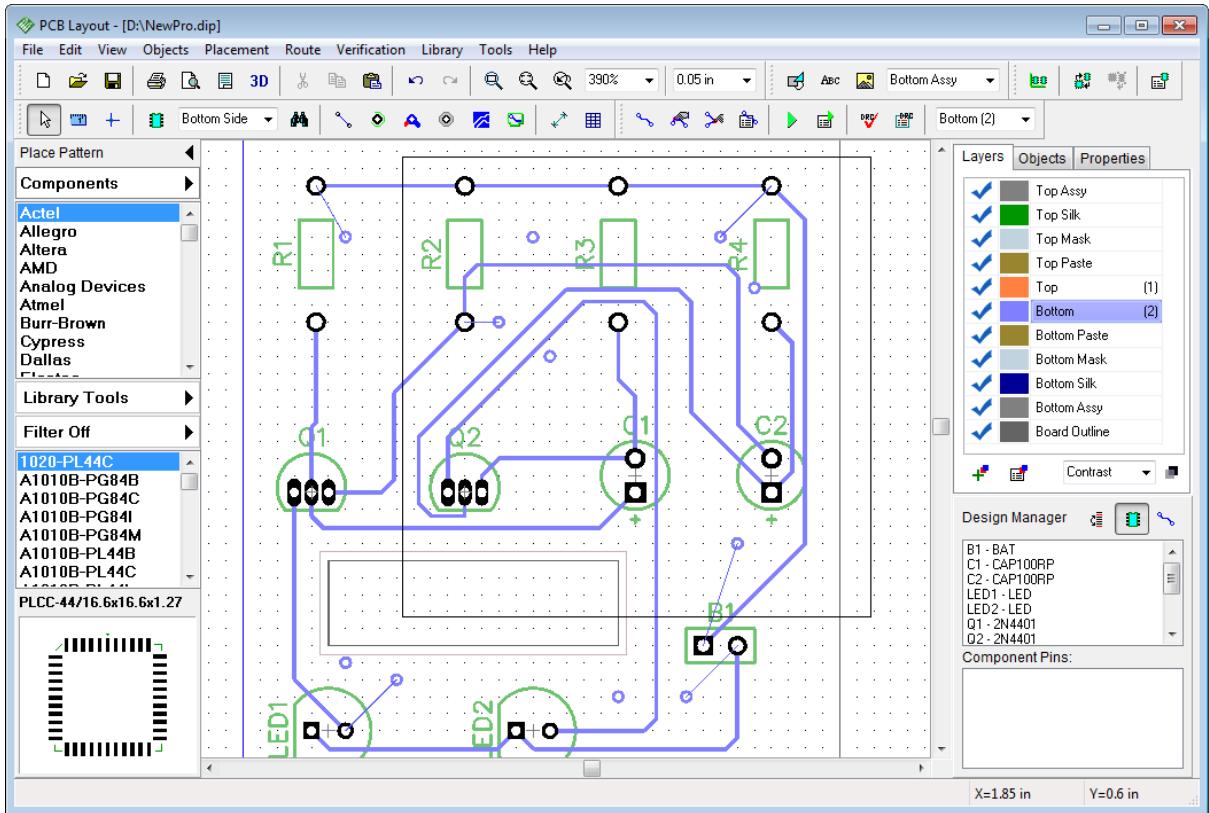


We will select, for example, all components of the layout. Check components item and click "OK" - components will be selected, but this is very simple example.

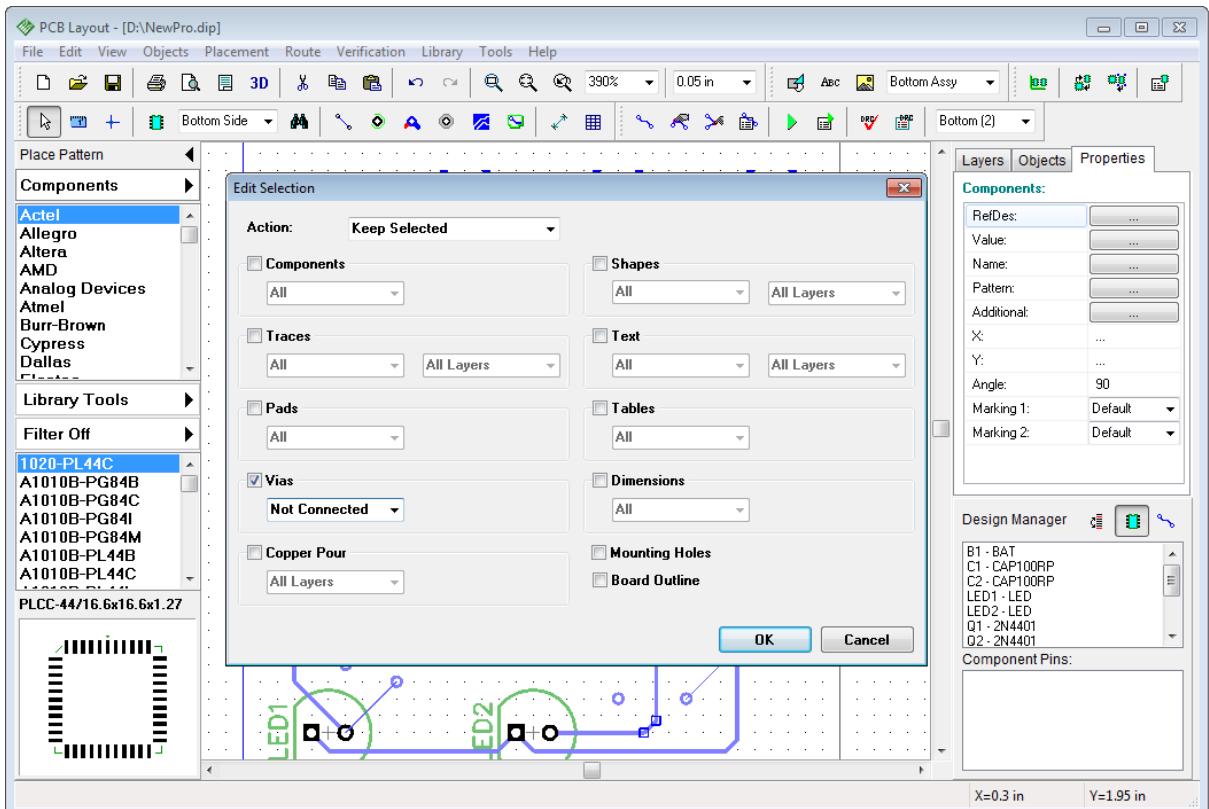
Our task is to select only unconnected vias in predefined area of the board.

First of all deselect components with a right click on empty area. Then place several static vias and connect **only some of them** to nets randomly, while leaving couple of vias unconnected. Use "Objects \ Place Static Via" from main menu (or press corresponding button on the objects toolbar) to place via and "Objects \ Place Ratline" - to create connections visually. Left click on the via and then left click on the pad to add via to pads net.

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



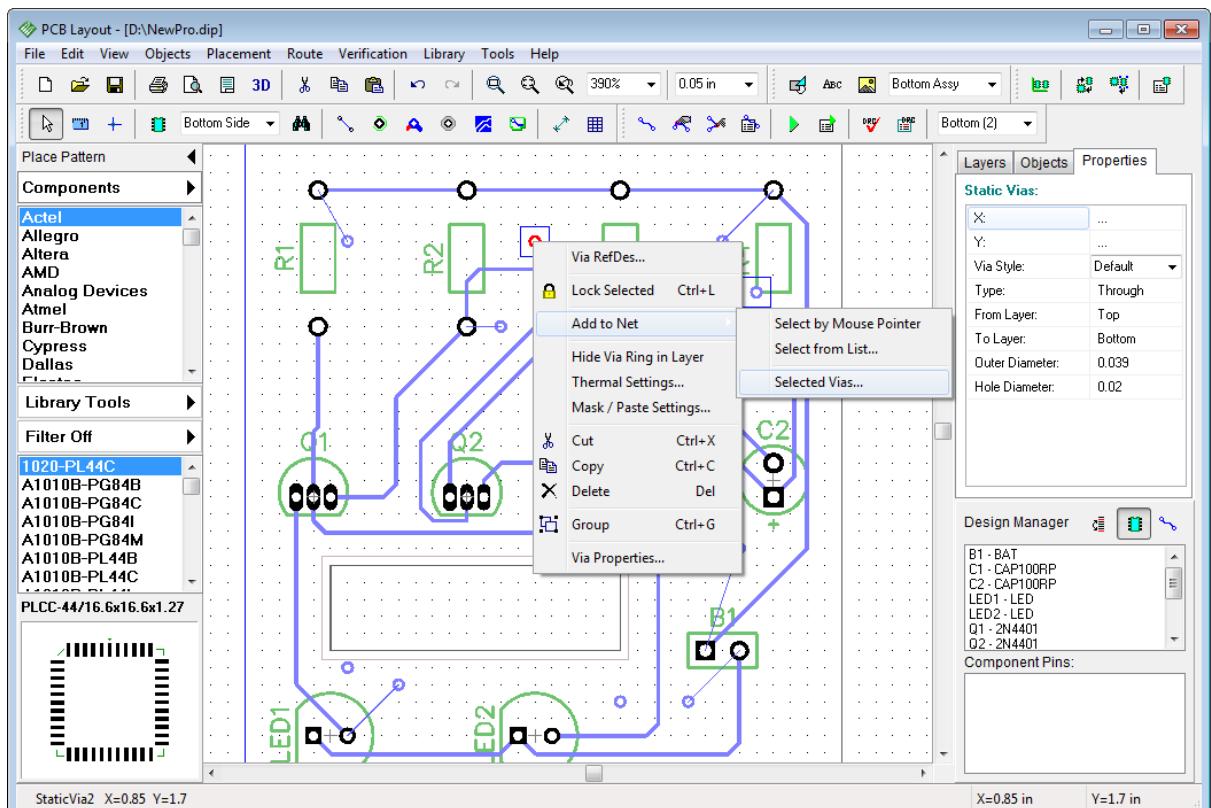
All objects in the box are selected. We need to extract only non-connected vias from the selection. Open "Edit \ Edit Selection", choose "Mode: Keep Selected", check only "Vias" box (other boxes should be unchecked) and then select "Not Connected" from the Vias drop-down list.



Click "OK" and only unconnected vias will be selected now.

Next step - for example, connect these vias to the 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 net in the pop-up dialog box.



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

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

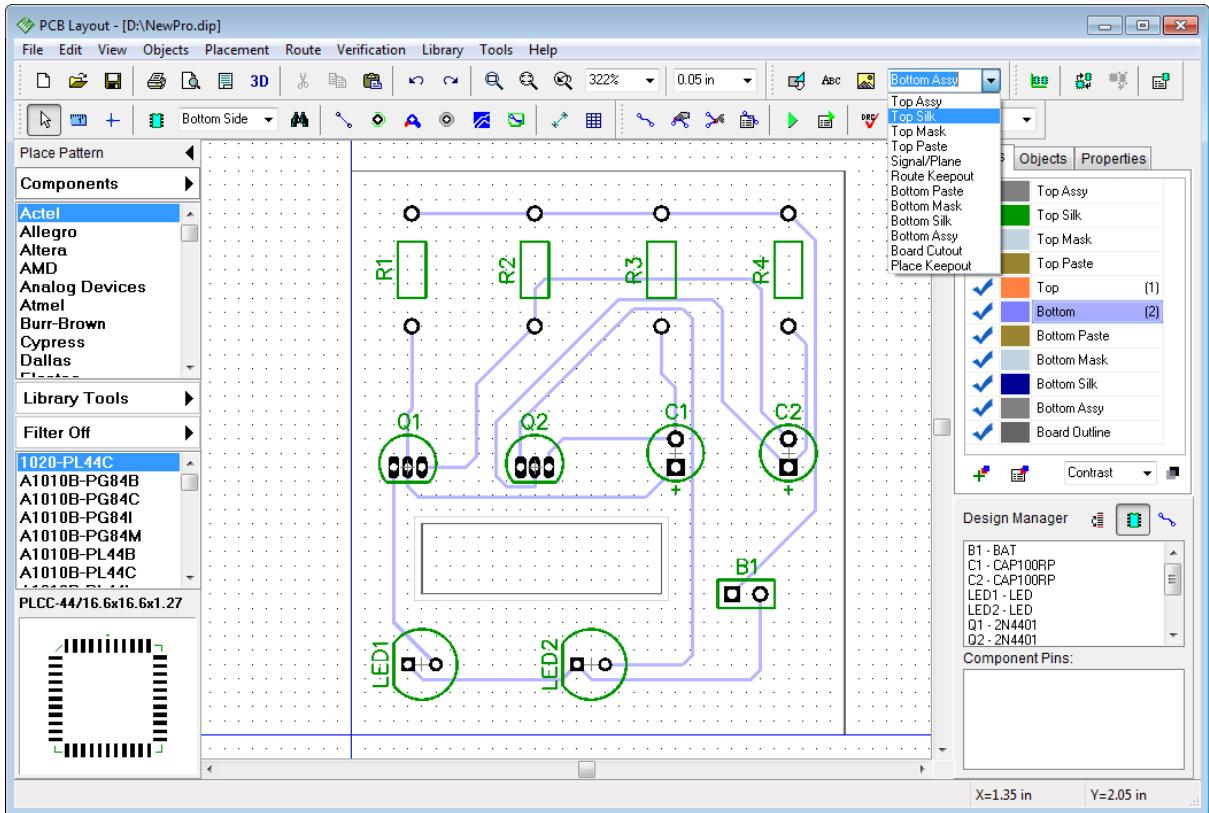
2.6.9 Placing Text and Graphics

With DipTrace you can add text, shape and logo in BMP or JPEG format directly on the board and export it to Gerber. We will practice with adding text.

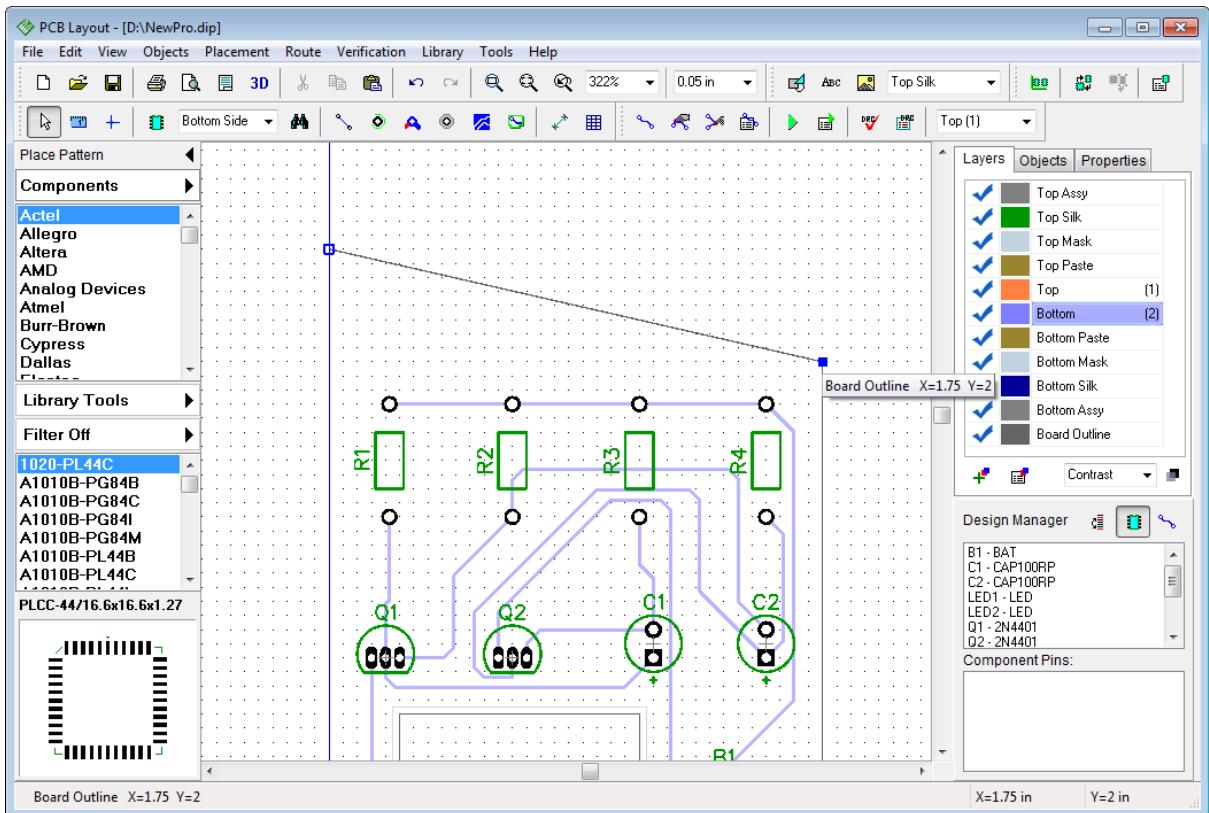
Select layer where you're going to place graphics, usually it's silkscreen layer (Top Silk in our case), double click it in the Layers tab of the Design Manager panel.

PCB Layout program allows to change layers with two drop-down lists on the instruments toolbar and in 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 to place 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 drop-down list on the route toolbar.



Let's make board polygon a little bit bigger to place additional text. Drag and drop upper-left and upper-right vertices of the board outline a little bit upwards. DipTrace makes visual editing very easy with appropriate grid size.



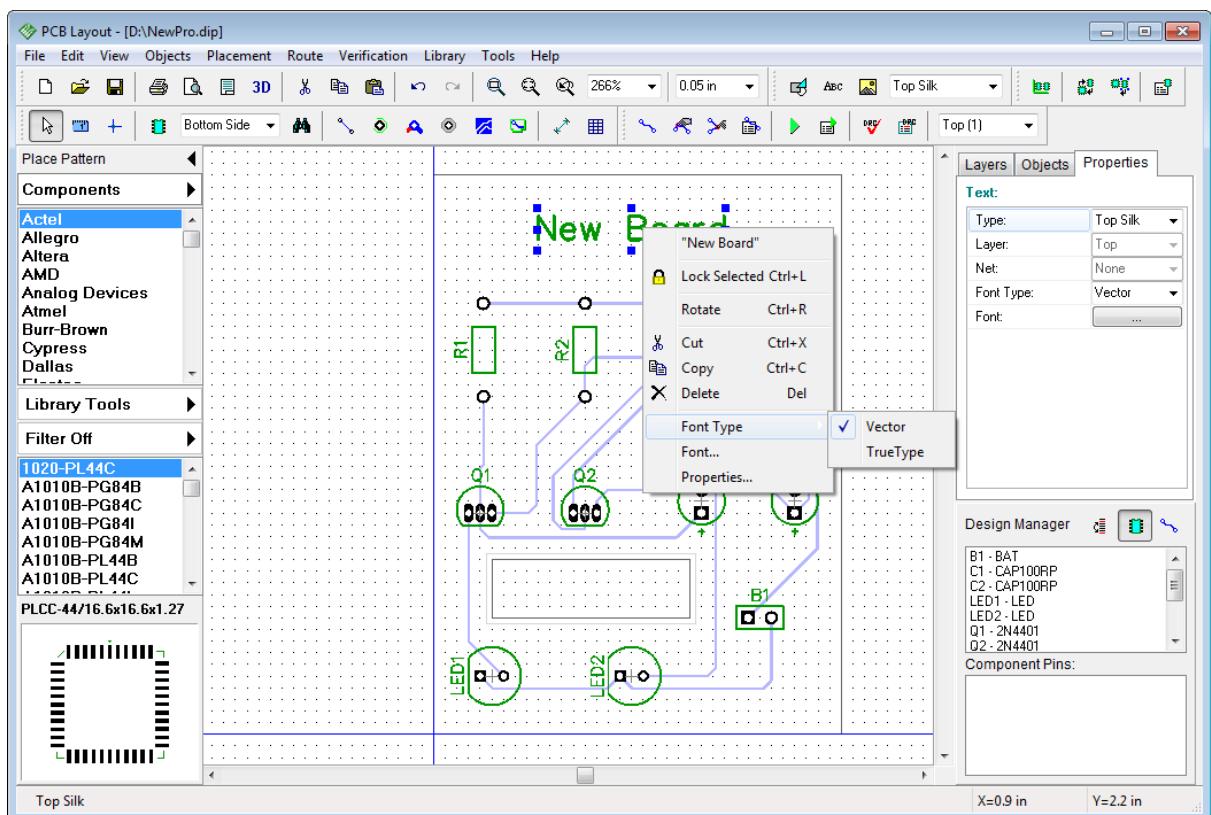
Board outline can be moved - left click on the outline (not the vertex) and drag and drop it (don't do this now).

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

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

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 text layer can be changed in Text properties panel on the Design Manager or in right click submenu. Use vector font, because it is directly exported to Gerber. For Unicode and Non-English characters TrueType should be selected, however it will be exported to Gerber as small lines (created by special recognition algorithm).

Some PCB manufacturers do not accept TrueType text objects in copper layers.



Text object is on the silk layer, hence it has the same color as layer. If you are going to change color of the text, we recommend to move it to "Top Assy" layer and then change Top Assy layer's color.

You can change layer of the graphics and text object at 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, e.t.c.).

In our case we just leave 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 shape properties dialog box.

2.6.10 Copper Pour

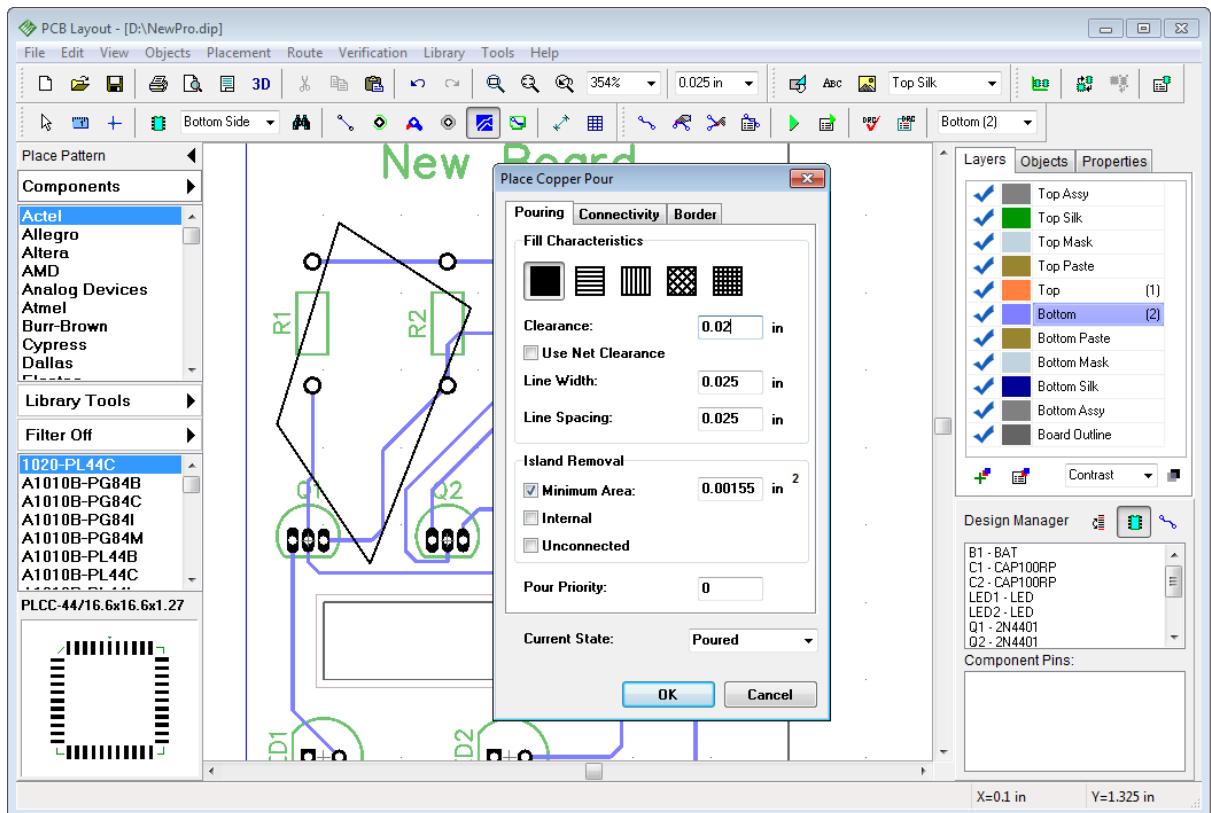
Copper Pour is used as a low-impedance conductor for Power and Ground nets. Pours are usually located on inner layers of the board, but can be placed on top and bottom as well.

Obviously, we do not need copper pour for current design, but we add it anyway, just to show you how to do that.

Place Copper Pour

Change grid size to comfortable .025 inch. Select Bottom layer then go to "Objects \ Place Copper Pour" from main menu or press "Copper Pour" tool button on the elements toolbar. Now you can draw a copper pour polygon borderline by defining its key points. You can draw a precise polygon or, - in case if you need to cover entire board with copper pour, - create random shape (for example, like on the picture below) and use "Depending on Board" feature which will pour entire board automatically (regardless of what shape you have drawn).

Right click on the last polygon point and select Enter from the submenu to finish drawing. Place Copper Pour dialog box pops up.



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

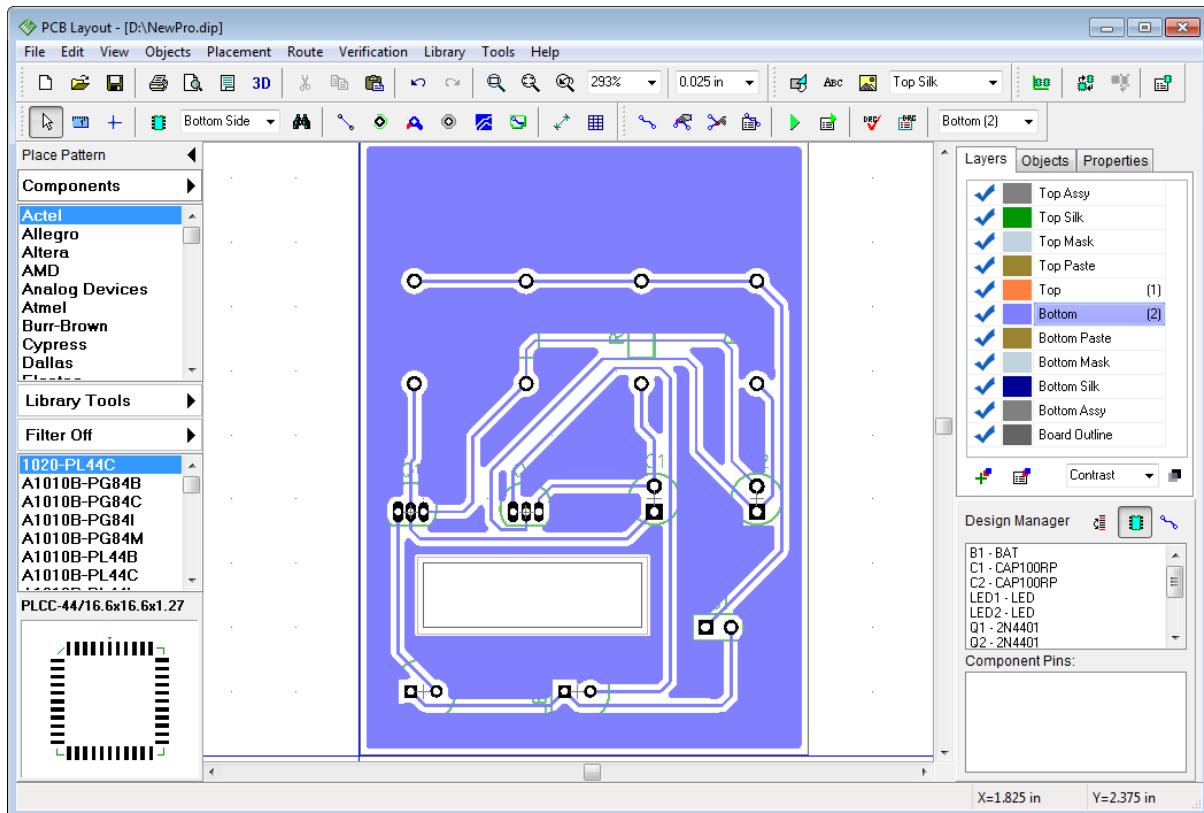
"Pouring" tab allows to specify different non-solid fills for the copper pour, clearance line width, spacing, island removal options, pour priority and current state (poured or unpoured). DipTrace has Shape-based copper pour system.

"Connectivity" - here you can connect copper pour to the net, select thermals and change their

settings, Separate thermals for SMD pads are supported. Select "Hide Net Ratlines" regime - it can automatically show ratlines only for unconnected traces or other if specified.

"**Border**" tab allows to define border points. "Depending on Board" check box can be used to save your time and build copper outline automatically; "Snap to Board" option means that copper pour will change its shape depending on the board outline.

Check "Depending on Board" item and keep all other settings like on the picture above. Click "OK" to place Copper Pour.



Board outline clearance specified in copper pour settings is not applied to board cutouts. Always use route keepout to allow certain clearance between copper pour and cutout, like we did.

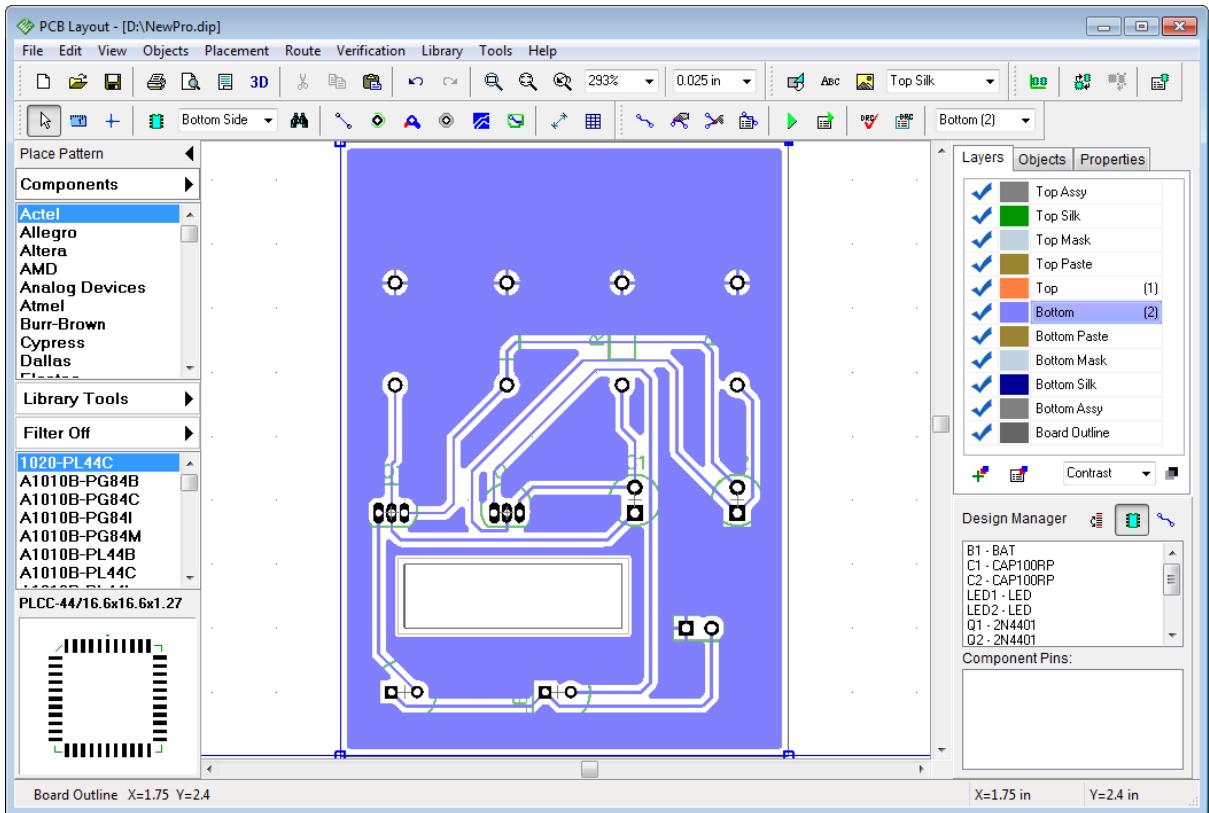
Copper Pour object can be in two states: Poured and Unpoured. The second state is often used for editing objects, because only copper pour border is visible. To change copper pour state right click on the copper outline, select "State" and choose the item you need.

We have copper pour but it is unconnected. 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 fast.

Connect net to copper pour

Unroute one of the nets (Net 6 in our case), right click on the trace and select "Unroute Net" from the submenu. Remember net name ("Net 6"). Right click on copper pour border and select Properties from the submenu. Go to "Connectivity" tab and select "Connect to Net: Net 6", select appropriate thermals (for example, "4 spoke") and press "OK" to update copper pour.

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



You can see that connection lines (ratlines) are hidden now and net (Net 6) is connected to copper pour with thermals of selected type (4 Spoke thermals in our case).

Now we will place second copper pour. Select another net that we will connect with a copper pour (Net 2 in our case) and unroute it. Then right click on the edge of existing copper pour and open Pour Properties dialog box. Select "Current State: Unpoured", do not close this panel.

Pour Priority

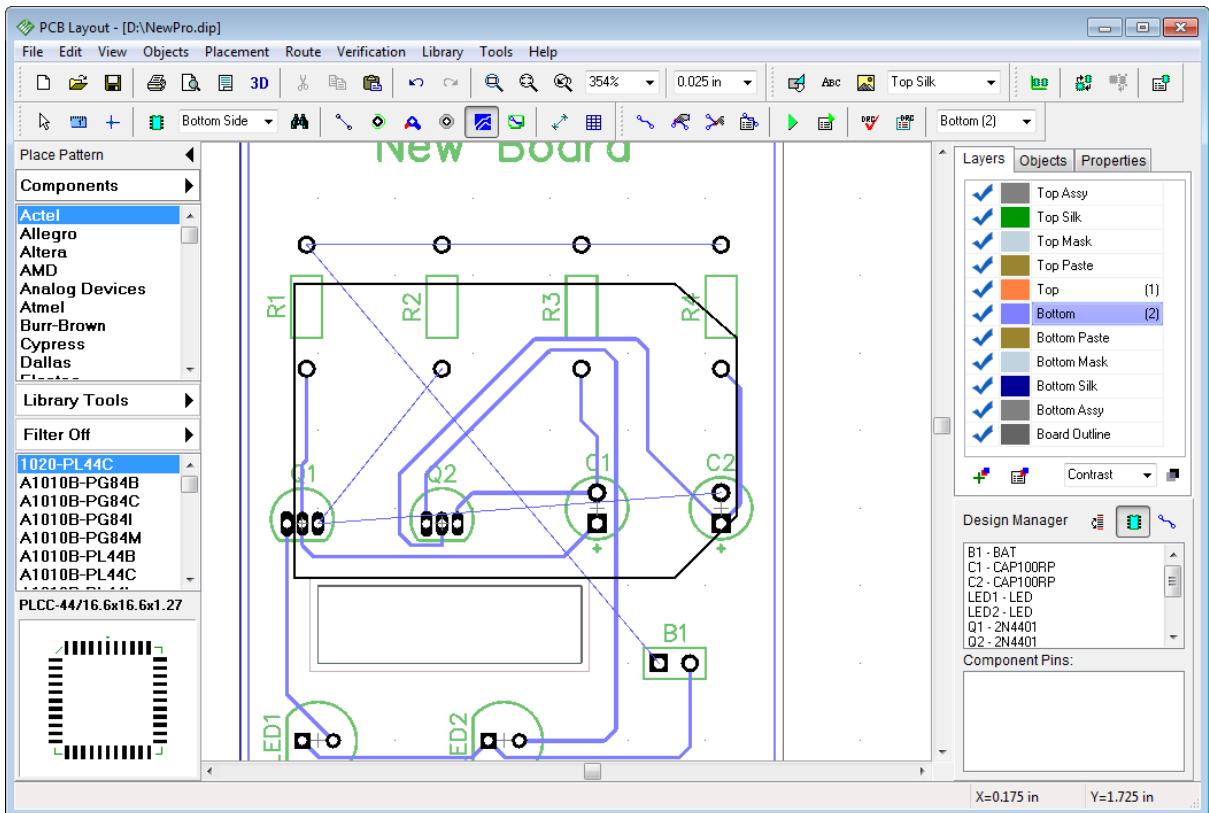
Now it's time to change pour priority for existing polygon. Specify: "Pour priority: 1" in "Pouring" tab.

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

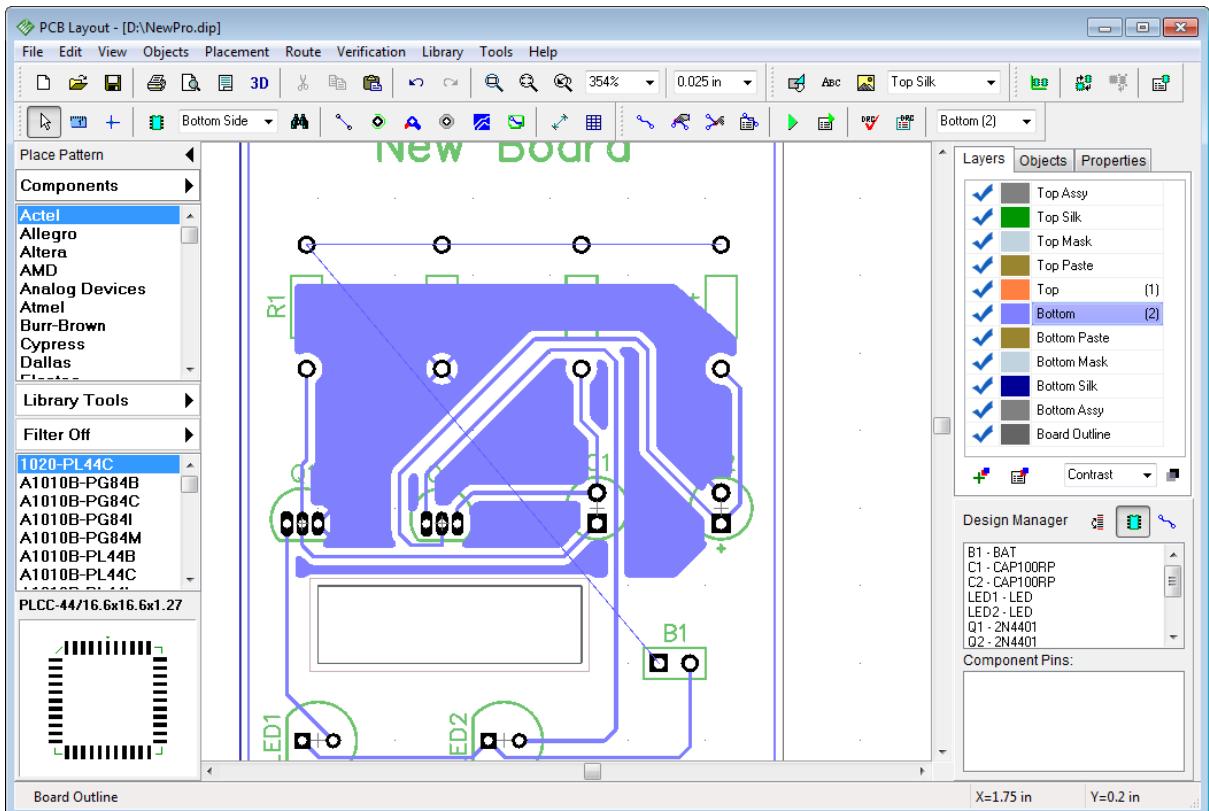
Notice that two different-net copper pours with the same priority level will intersect. Real-time Design Rule Check will show numerous errors in that case.

Press "OK" to apply new settings. Notice that in unpoured state ratlines are displayed automatically.

Draw polygon of second copper pour. Select copper pour placement tool ("Objects \ Place Copper Pour") and draw a second polygon to connect pads of Net 2, like on the picture below (pads of C2, R2 and Q1 components):

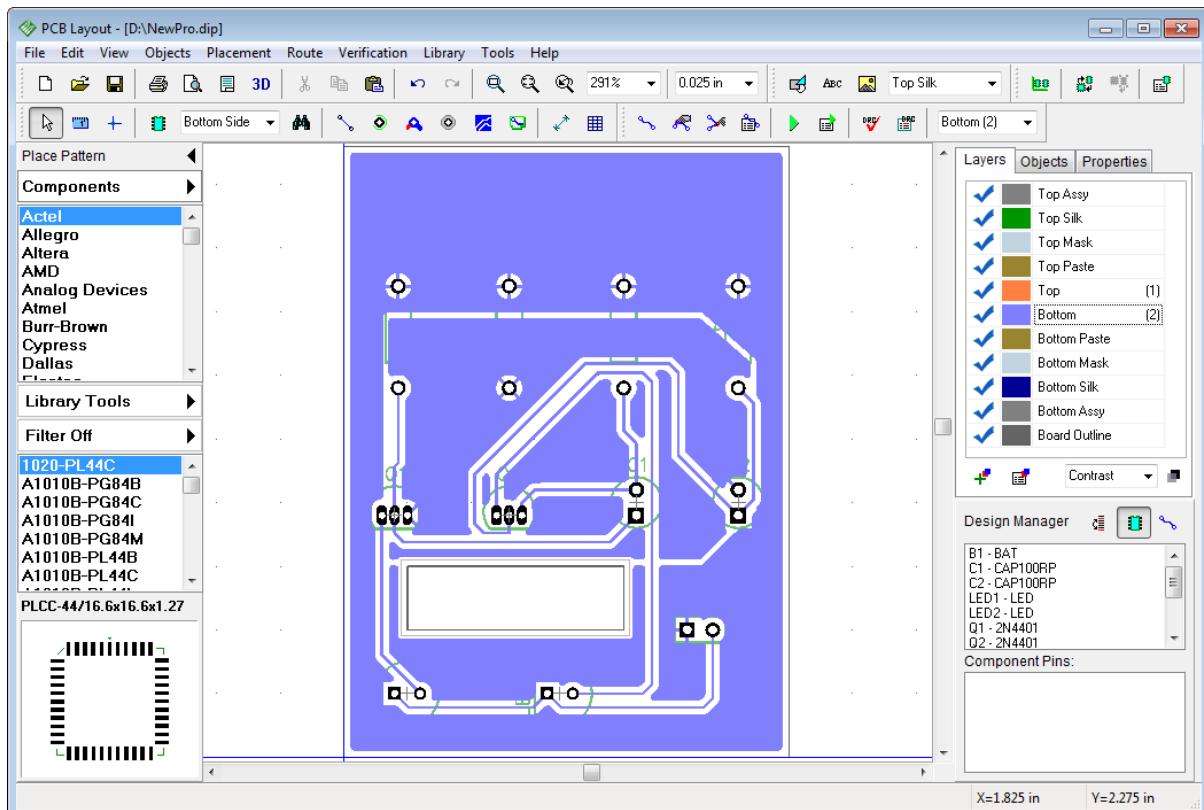


In the pop-up dialog box connect second copper pour to Net 2 and specify thermals type applied (for example "4 Spoke 45"). Press "OK" to close dialog box and create copper polygon.



Now select Net 6 copper pour, which is unpoured now. Right click on its border and select "State \

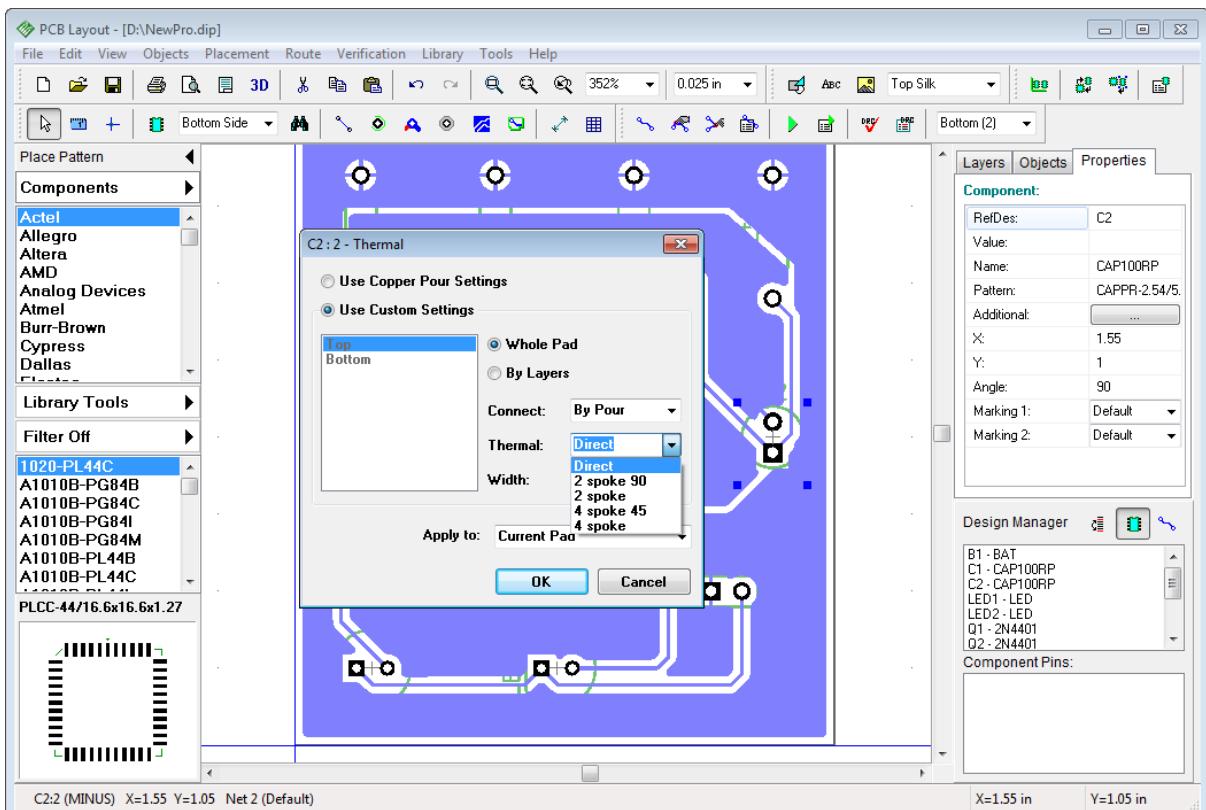
Poured" from the submenu and you will see that two copper pours connecting two different nets are independent and Net 6 copper has changed its shape according to the Net 2 pour polygon which has higher priority level.



Thermals

Sometimes it is necessary to make separate thermal settings for single or several pads. To set pad thermal settings right click it (when pad is highlighted) and select "Thermal Settings" from the submenu. Then check "Use Custom Settings" and select new thermal connection.

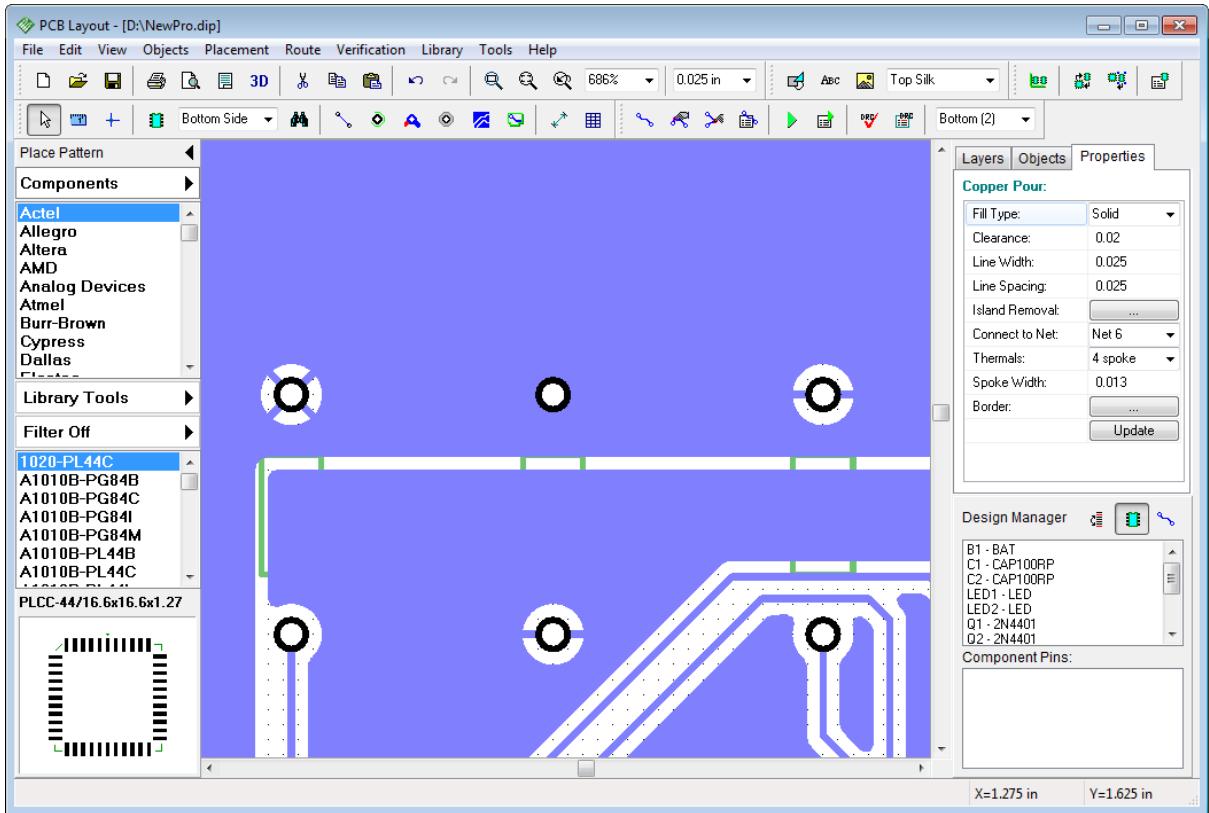
Some pads can become unconnected after placing copper pour, because of selected thermal type and layout structure (net connectivity check will report this), so selecting separate thermal settings for pads is very useful feature.



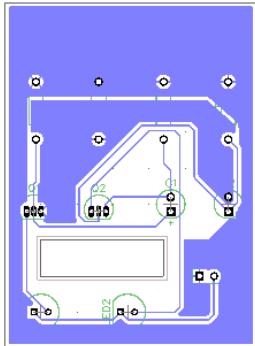
After changing thermal settings click "OK" to close dialog box, then right click on the copper pour border and choose "Update" from the submenu.

Notice that each copper pour should be updated separately.

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 thermal, another one is connected directly and the third pad - 2 spoke connection.



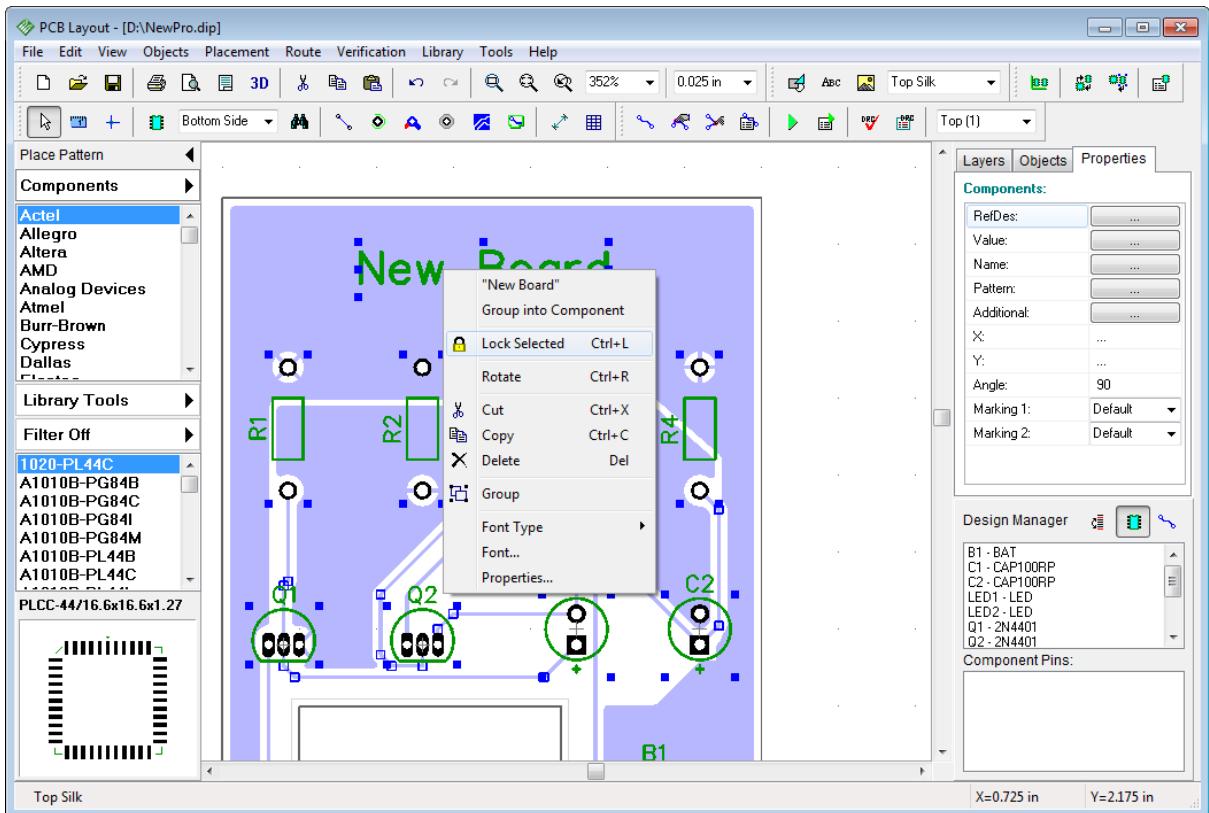
When copper pours are used as Ground and Power planes, SMD vias are connected to them by fanouts. Fanout can be made manually with "[Fanout](#)" feature or automatically by "[Shape Router](#)".

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

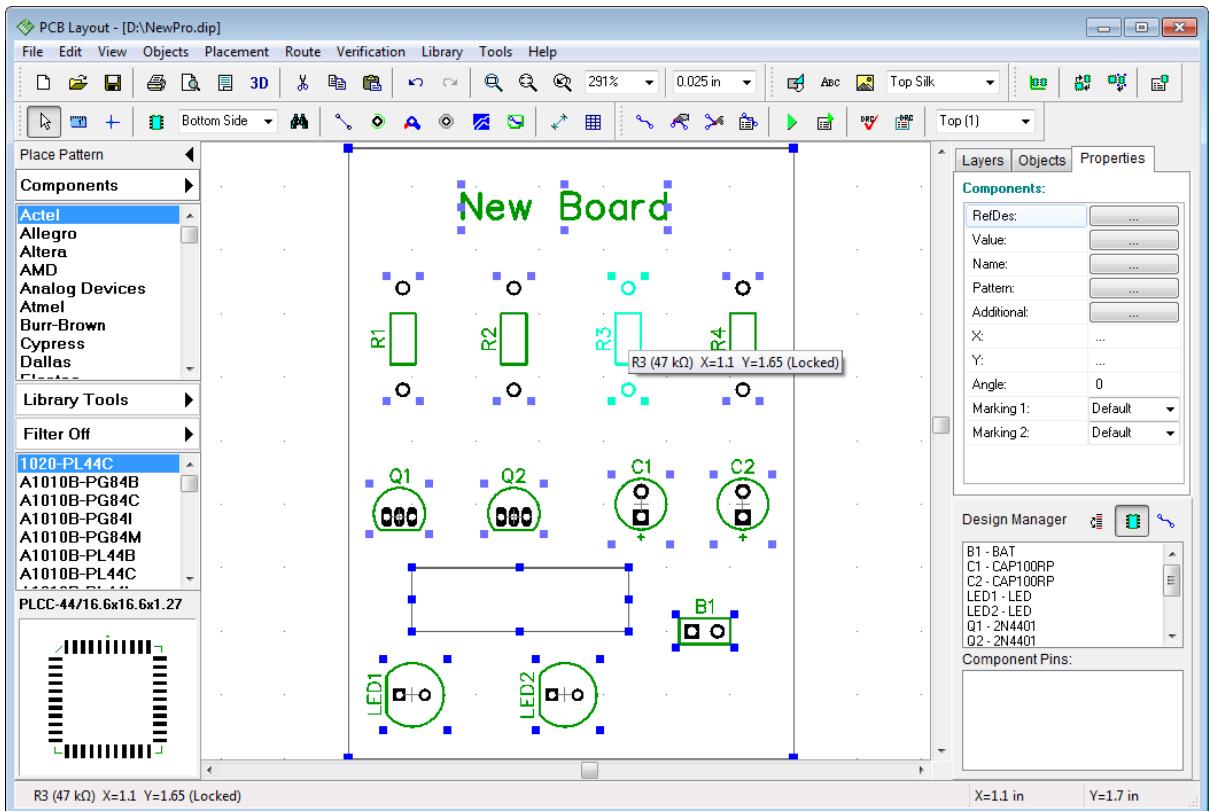
2.6.11 Locking objects

Sometimes when you edit schematic or PCB you need to lock objects to prevent further editing of their positions and properties.

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



Notice that locked objects have low contrast of selection rectangles (in our case color is similar to copper pour, so we've made only current layer visible (with drop-down list on the Layers panel). "Locked" text appears in the hint of locked objects.



You are unable to move, resize or edit locked object. Now please unlock all objects, select all with "Ctrl+A" and unlock ("Edit \ Unlock Selected" from main menu or "Ctrl+Alt+L").

Components can be locked after placing them on top or bottom side. Select "Edit \ Lock Components \ Top" to lock top components. Using this mode you can route board and do not worry that some components can be moved accidentally. Select it again from main menu one more time to unlock components on the board side and return to contrast layer display mode.

2.6.12 Design Verification

DipTrace has several verification procedures united in "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 to check clearances between objects and allowable object sizes according to the set of design rules. DRC works in regular (offline) and real-time modes. Real-time DRC checks all user actions on the go. For example, when you move some component or create new trace too close to another object Real-time DRC shows red circles which mean that clearance between these objects (trace, pad, copper pour) is smaller than specified parameter. 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 main menu or pressing "F9" hot key. 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 rules and various options. There are 4 tabs in the pop-up dialog box: "Clearances", "Sizes", "Real-time DRC" and "Options".

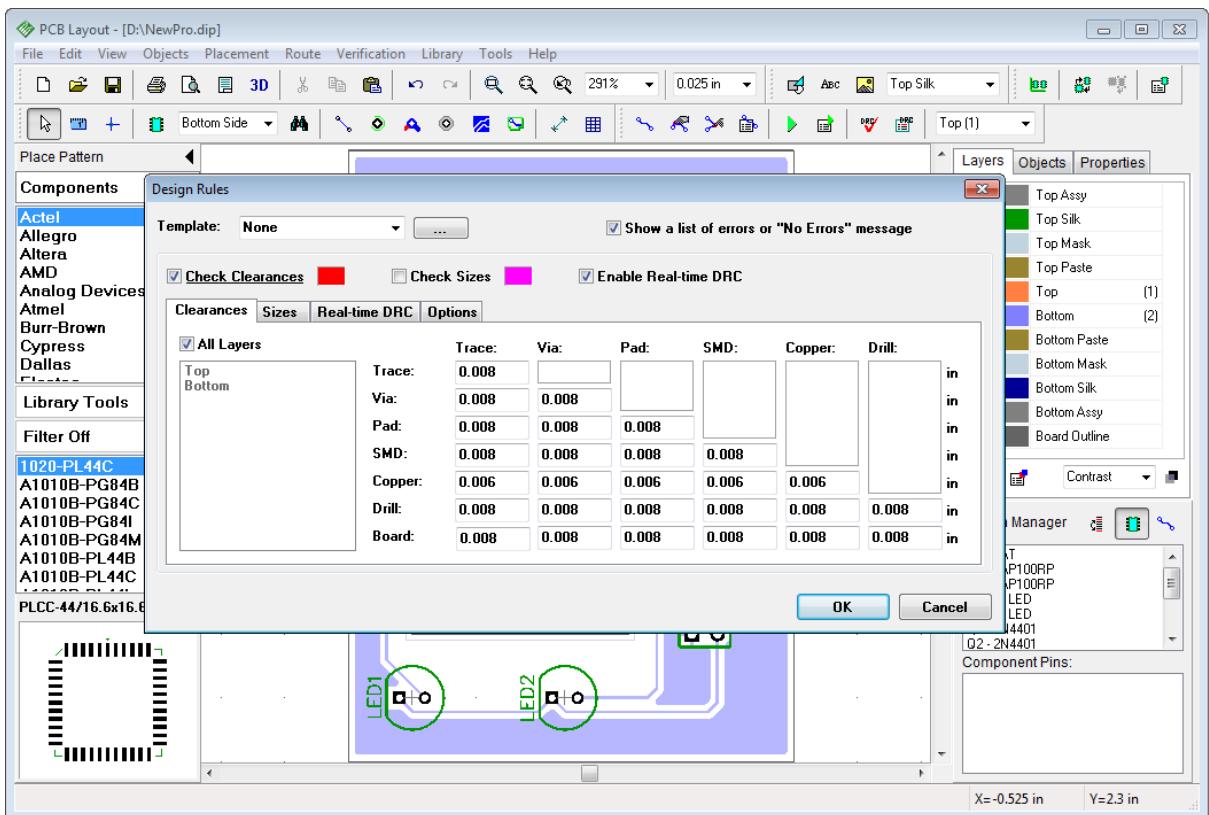
"Clearances". Specify object-to-object clearances. Uncheck "All Layers" item, select layer from the list below and define different object-to-object clearances on different layers.

Notice that clearance settings are NOT applied to nets with custom net class clearance ("Use Clearance in DRC" option in [Net Classes](#) dialog box) or Class-to-Class settings.

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

"Real-time DRC". It can be easily turned ON/OFF for actions like 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 corresponding tab and leave only "Enable Real-Time DRC" active, you will see errors right after completing 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.

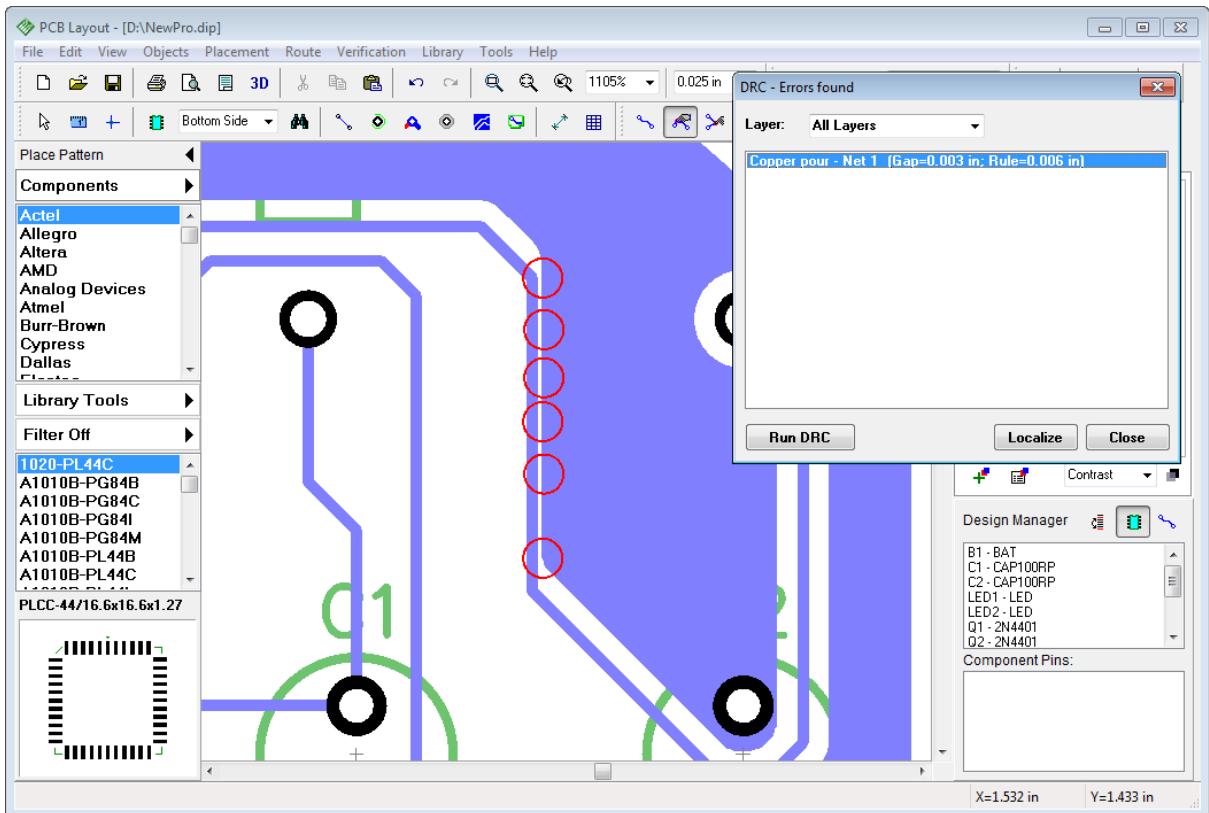
"Options". Setup verification options.



For this tutorial example, please turn OFF size verification and Real-time DRC by unchecking corresponding items. Make sure "Check clearances" and "Show a list of errors..." are checked. Keep settings like on the picture above.

Open "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 lets try to see how DRC works in regular mode. Even if Real-time check was ON during design process, we recommend to verify project with regular DRC at least once before exporting to Gerbers just to make sure everything's fine. Press "OK" to apply changes and close dialog box.

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



Errors can be displayed by layer. Left click on the error in the list and press "Localize" button - DipTrace will target the error's site and place it at the center of the screen. Red circles mean clearance errors, magenta circles - size errors.

Move trace back to its 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 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 should be performed. It checks entire design and displays 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 "No Errors Found" message. More information about [Net Connectivity Check](#)^[181].

Comparing to Schematic

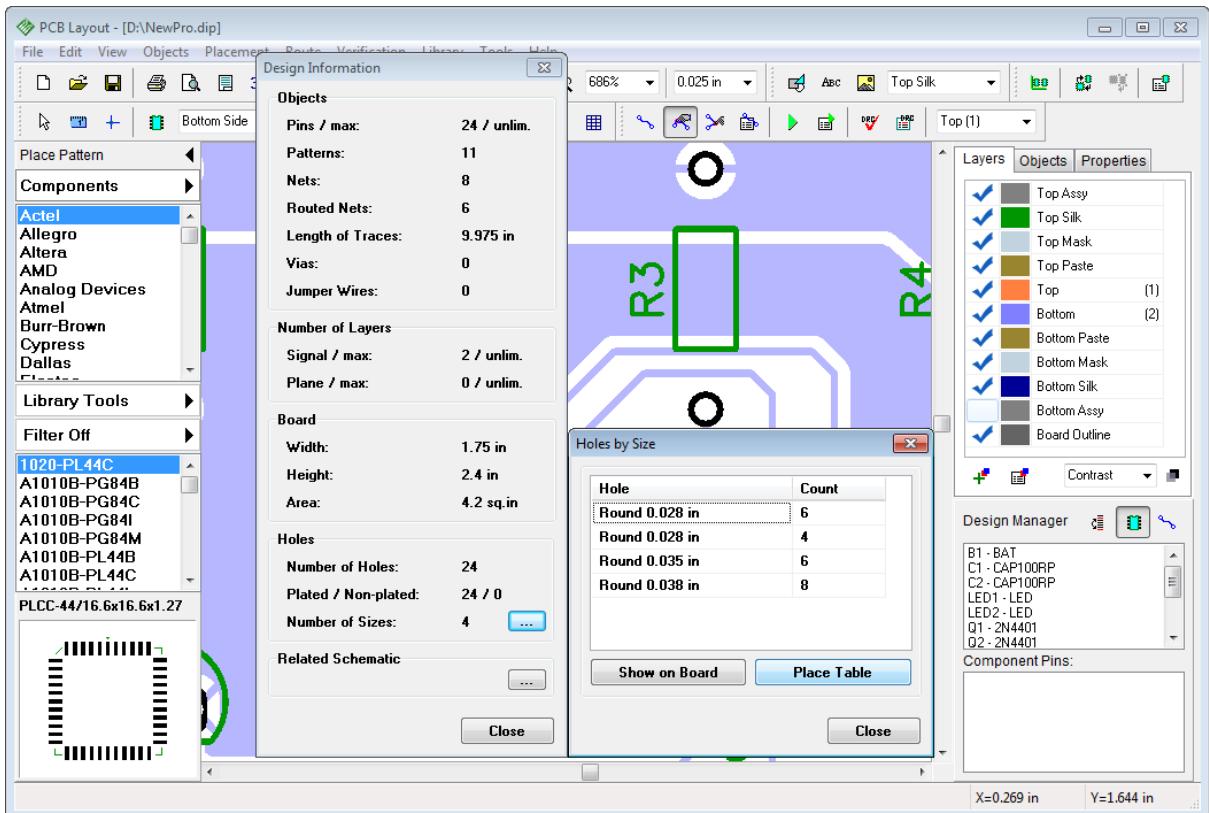
Compare procedure allows you to check if PCB corresponds to the source Schematic file. Verification shows net structure errors and unknown components. Select "Verification \ Compare to Schematic" from main menu then choose source Schematic file and press "OK". If your net structure was not changed and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

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

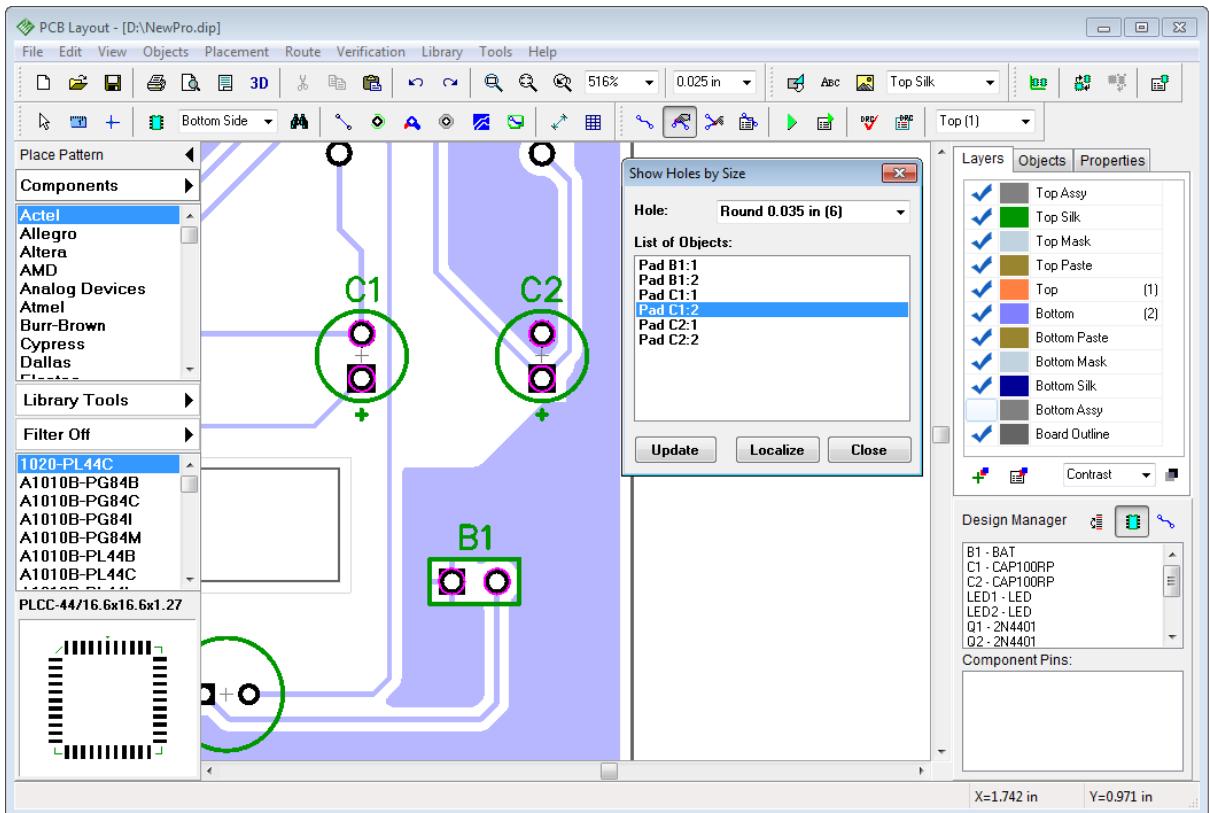
2.6.13 Design Information

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

Drill / hole sizes can be shown on the design area - this may be useful when optimizing drill table and removing some hole sizes.



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



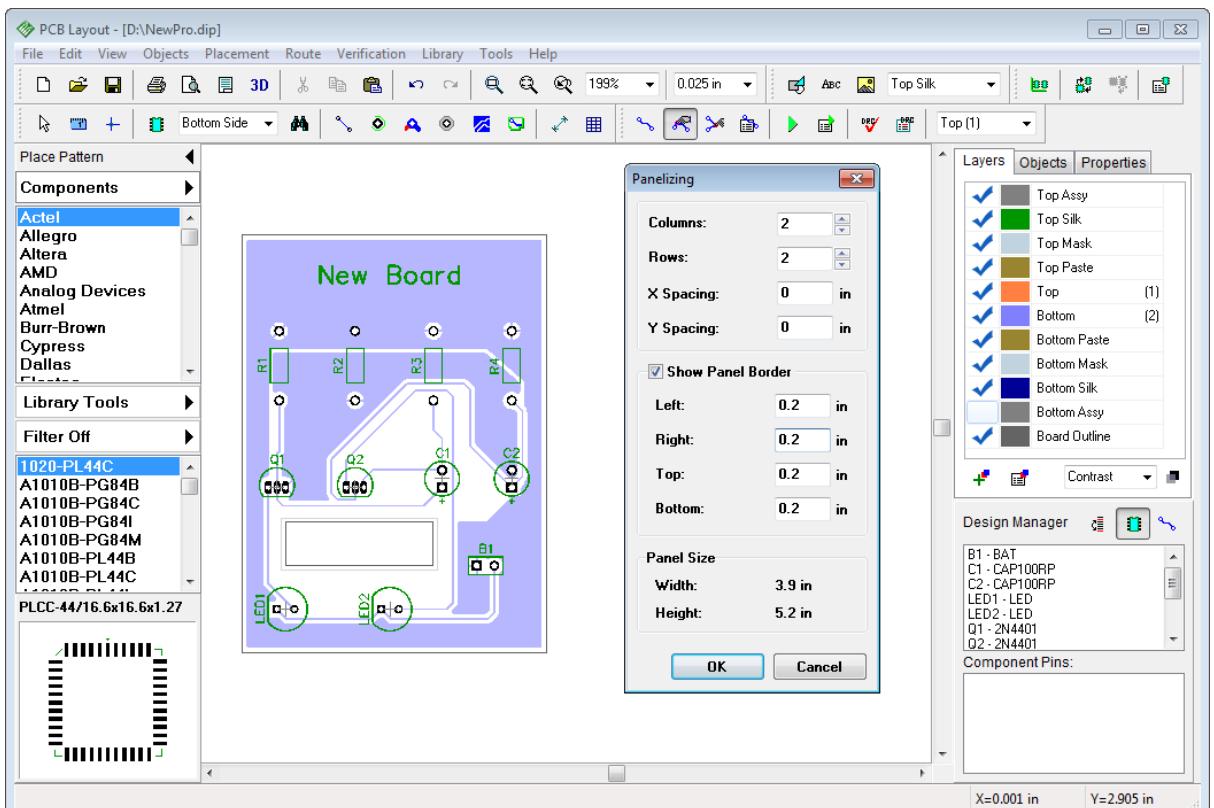
Close the dialog box and save design.

2.6.14 Panelizing

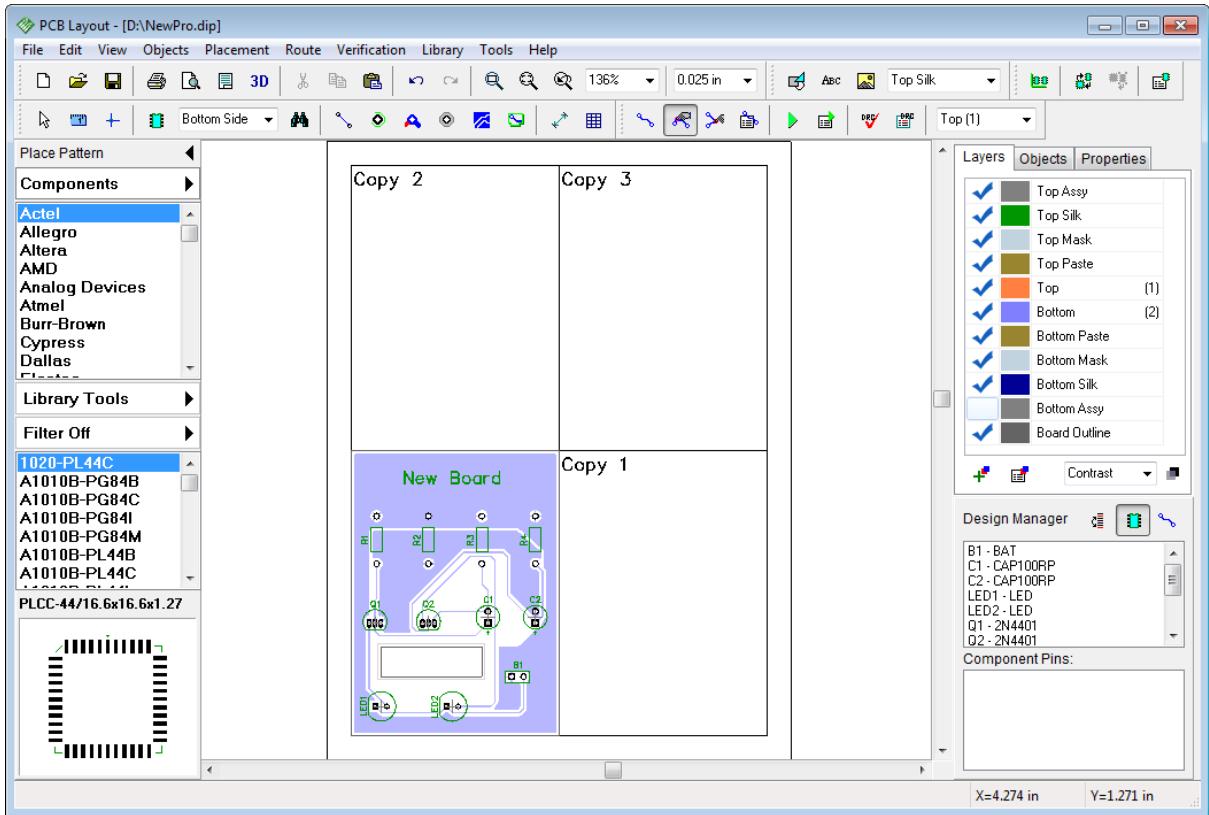
With DipTrace you can panelize similar or different boards on a single layout.

Panelize project

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

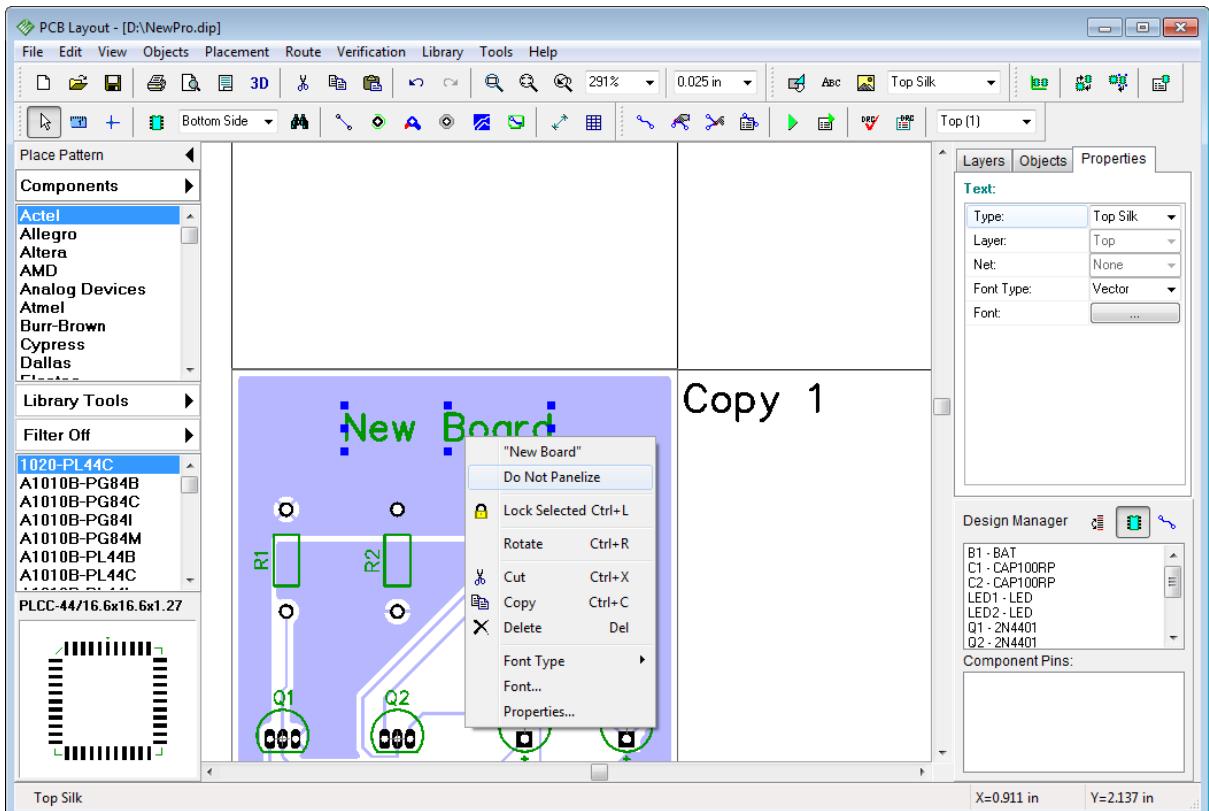


We will make 4 copies of the PCB, 2 columns and 2 rows. Spacings between boards will be zero. Showing panel border may help to determine approximate panel size. Rail edges (board to panel border distance) for all sides will be 0.2 in. Some manufacturers need panel border in the board outline layer, so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



In the design area we can see only boxes with "Copy #" text, but in print preview dialog box ("File \ Preview" from main menu), while printing or exporting Gerber, DFX or N/C Drill file, complete copies of the board will be inserted and you will see 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". This item is available only if panelizing is ON.

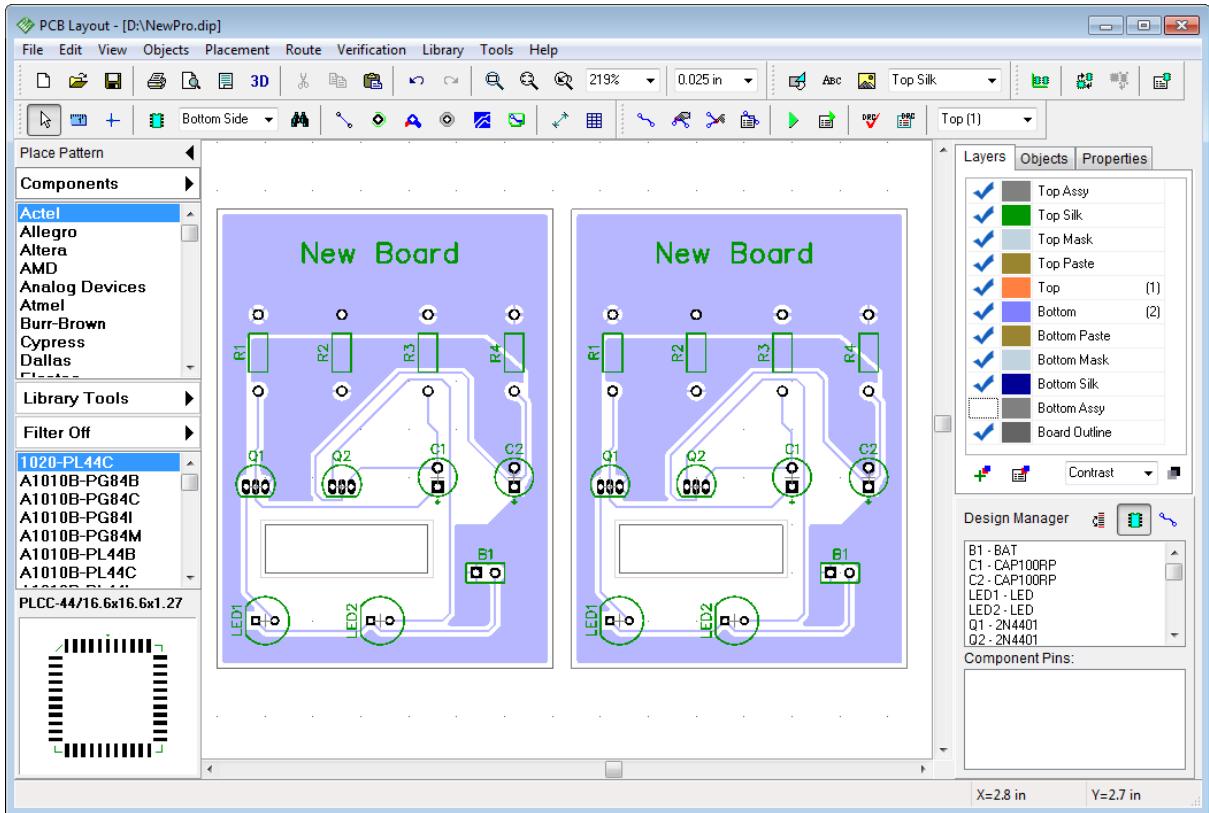


Notice that panelizing works only if PCB has board outline.

Open Panelizing dialog box and change Number of columns and rows to "1" (this will remove copies and let us practice with panelizing different boards on a single layout).

Panelize different projects

Check 'Edit \ Keep RefDes while Pasting' item from 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 upper-left corner of the second board) and select "Paste".



We got second copy of our PCB (it can be 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.

If "Keep RefDes while Pasting" item is checked, pin limitations of your DipTrace edition (Free, Lite, Standard, Extended editions) do not apply for copying, so you can easily panelize several 250 pin layouts even with free DipTrace edition.

2.6.15 Printing

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

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

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

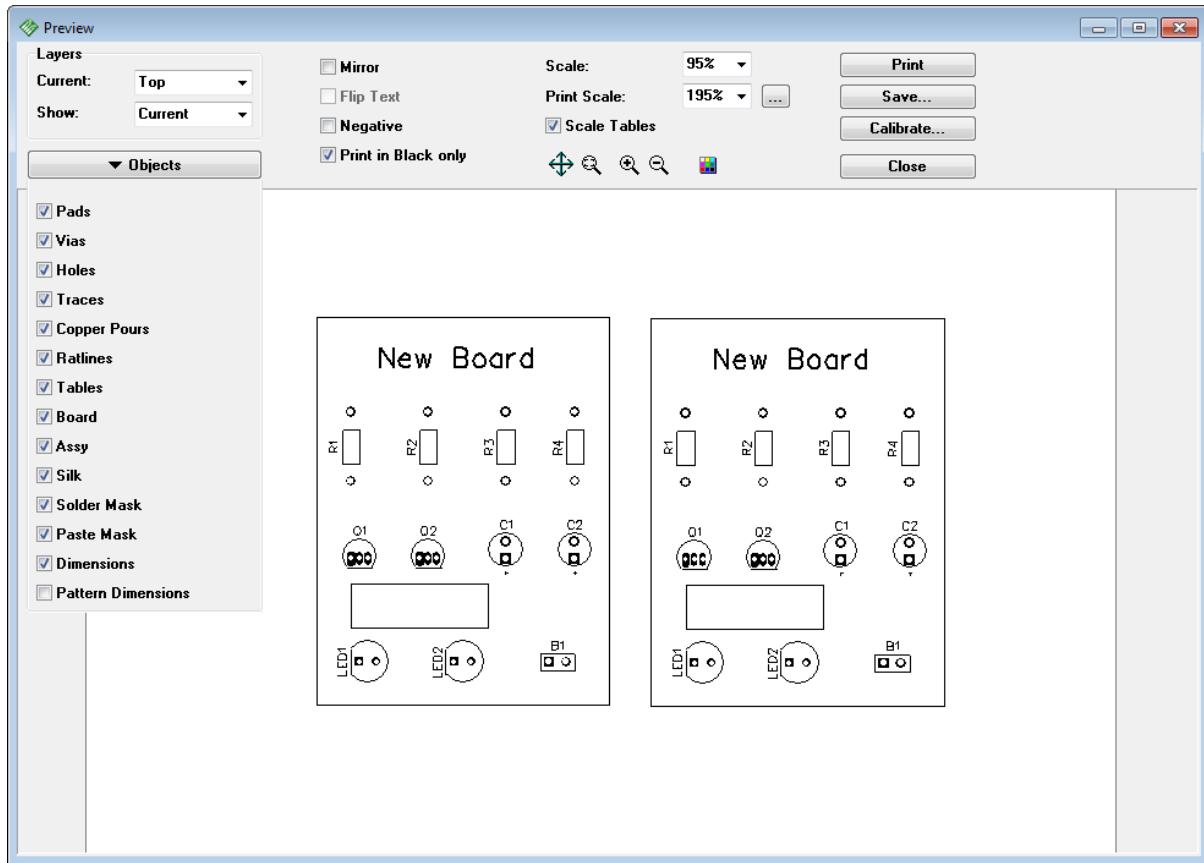
Use "Print Scale" drop-down list or "Zoom In" / "Zoom Out" buttons to change scale of the layout on the sheet and "Move Board" tool (button with four arrows) - to move design on the sheet.

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

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

Notice that only layers with default color depend on color scheme. If you changed layer color it will not change when new color scheme is applied.

For printing all in black without changing layer colors, check "Print in Black Only" box.



For hobbyists attention: please be aware that a laser printer introduces some degree of dimensional distortion due to heat expansion of paper. It depends on your laser printer and quality of paper. 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 it is not the issue, since ink-jet technology does not heat up paper. Laser printers do not always distort image visibly, but you have to be ready that it can happen. You can use "Calibration" feature of print preview dialog box to minimize heat distortion.

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 printer.

Close "Print Preview" dialog box and use **Undo** several times to remove second PCB and return board without panelized copies. Save Project.

2.7 Manufacturing Output

2.7.1 DXF Output

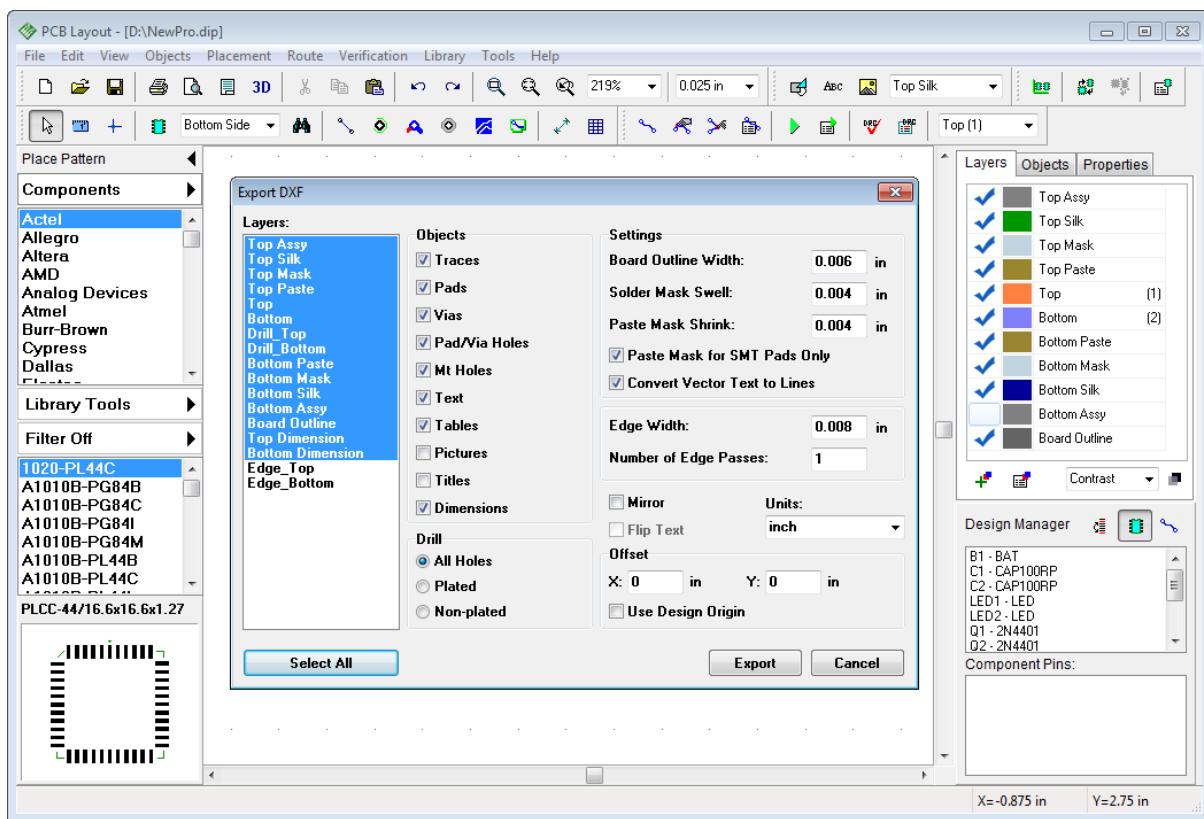
DXF output allows to export design to many CAD / CAM programs which support DXF import (AutoCAD and others).

DXF Export

To export board to DXF format, select "File \ Export \ DXF" from main menu. Select layer from the list of all project layers and check / uncheck to show / hide different objects for selected layer (text, pictures, vias, e.t.c.). Setup offset (distance between zero and bottom-left corner of the board: design origin or custom value), mask and paste layers if needed. Each layer can be saved into separate DXF file, but in order to export entire board into single multi-layer DXF press "Select All" (layers) button.

Notice that "Edge_Top" and "Edge_Bottom" are not selected. Technically these are not layers of your design. They are exported only if you are going to manufacture board with milling method.

Press "Export" button, specify file name, location and save board in DXF file.



All layers of the board will be exported into a single DXF file. You can open it with AutoCAD or another program capable to read AutoCAD DXF.

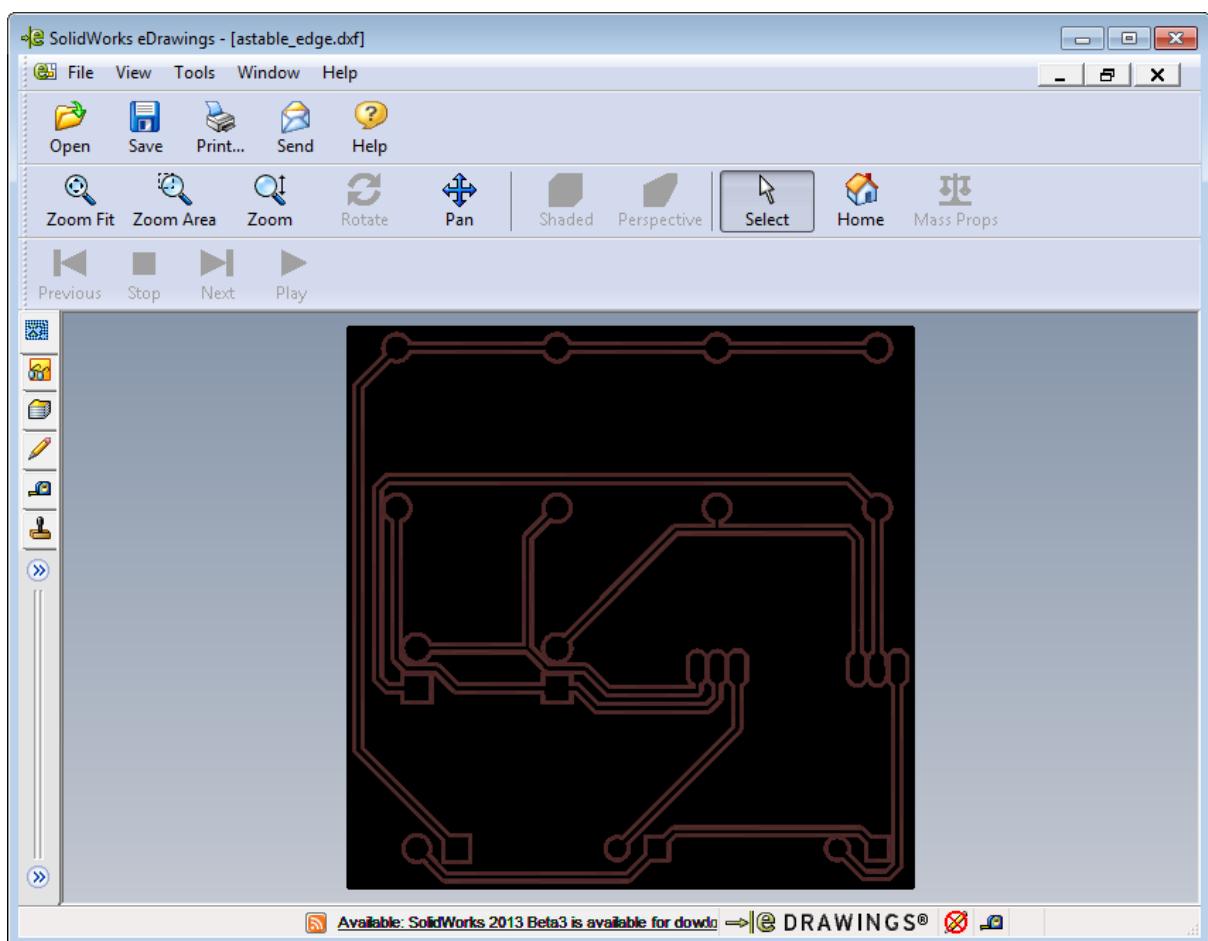
Export for Milling (DXF and G-Code)

Milling method is convenient and cheap for non-complex boards.

Notice that copper pours (unlike thermals) are not considered, when exporting edges for milling.

Select "File \ Export \ DXF" to open DXF Export dialog box. Then select "Edge_Bottom", because all traces of that PCB are in Bottom layer (if you have traces in top layer, Edge_top should be exported). Check "Mirror" box to mirror the design (for bottom layer). This will allow us to see the actual bottom side of the board. Then define "Edge Width" parameter. Center line of milling will be calculated as a half of Edge Width parameter value from design objects (traces and pads). Milling depth depends on edge width and instrument angle. You can leave default values if you're not familiar with specific milling settings.

Press "Export" button and save DXF file. Now open it with AutoCAD or another program to view the result.



The edge exported from DipTrace is a set of polylines with defined width. Before export DipTrace checks design and if object-to-object clearance is less than edge width DipTrace shows warning message and enables designer to correct errors.

Notice that CAD programs usually show polylines with sharp angles, therefore picture can be different. But when you mill the board or simulate milling with CAM program, everything will be good because of the instrument radius.

CNC Drilling machines work with G-Code files. Convert your edge from DXF format to G-Code, using [ACE converter](#) (it is free).

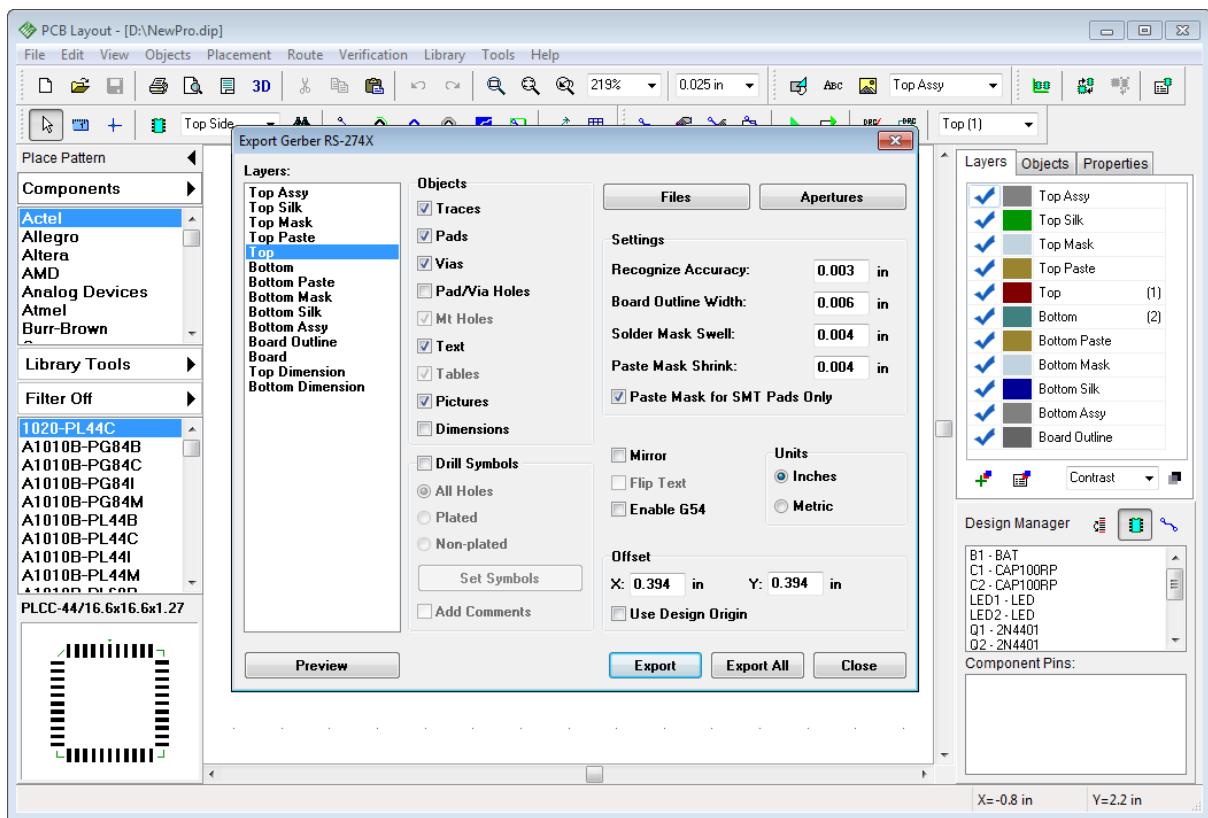
2.7.2 Gerber Output

DipTrace allows to export project in Gerber format, accepted by almost all PCB manufacturers around the world.

Select "File \ Export \ Gerber" from main menu. In Export Gerber dialog box you need to select layers, which you are going to export (use "Ctrl" and "Shift" for multiple selection).

Notice that NOT all layers are necessary for successful board manufacturing. It depends on your project and additional features that you order.

Specify which objects will be exported to Gerber for each layer (select layer from the list and use corresponding check boxes in Objects section). Press "Preview" button to preview selected layer. Unlike DXF, all layers should be exported to Gerber separately, one layer per file. Press "Export All" and DipTrace will generate files for each layer automatically.



Gerber Layers

1. Top Assy – this is assembly layer, it includes all shapes/texts placed in Top Assy as well as objects defined in "View \ Assembly Layers". Assembly layer in this project includes board outline because it is selected in "View \ Assembly Layers" main menu item, but you can select another object or hide board outline in this layer.

2. Top Silk – includes pattern shapes, texts and all other shapes and texts placed in Top Silk layer. Do not change settings and click "Preview". Notice that if you use TrueType fonts parts of the text can be invisible (depends on the font and size), you should make "Recognize Accuracy" value a bit smaller, but do not apply minimums.

3. Top Mask – this is solder mask layer. It is generated automatically, based on pads, custom pad

settings and common "Solder Mask Swell" parameter, defined in Export Gerber dialog box. This layer includes shapes placed in solder mask layer as well. We should uncheck "Vias" box (exclude vias from export to this layer), because vias are usually covered with solder mask. To change custom solder mask settings for pads, right click on the pad and select "Mask \ Paste Settings" from the submenu.

4. Top Paste – this layer is used for SMD pads only, so we can check "Paste Mask for SMT Pads only".

5. Signal layers (Top, Bottom, etc.) - these are copper layers. Please check "Vias" box for all of them and preview each one to make sure that layers are displayed correctly.

Notice that "Pad/Via Holes" item in Objects section should be checked only if you plan to drill holes manually (not using PCB house). If "Pad/Via Holes" box is checked, two Gerber layers will be created inside one Gerber file: Positive Drawing and Hole Clearing. The second layer is used to remove artefacts over the drill holes. Manufacturers prefer Gerber files without pad/via holes.

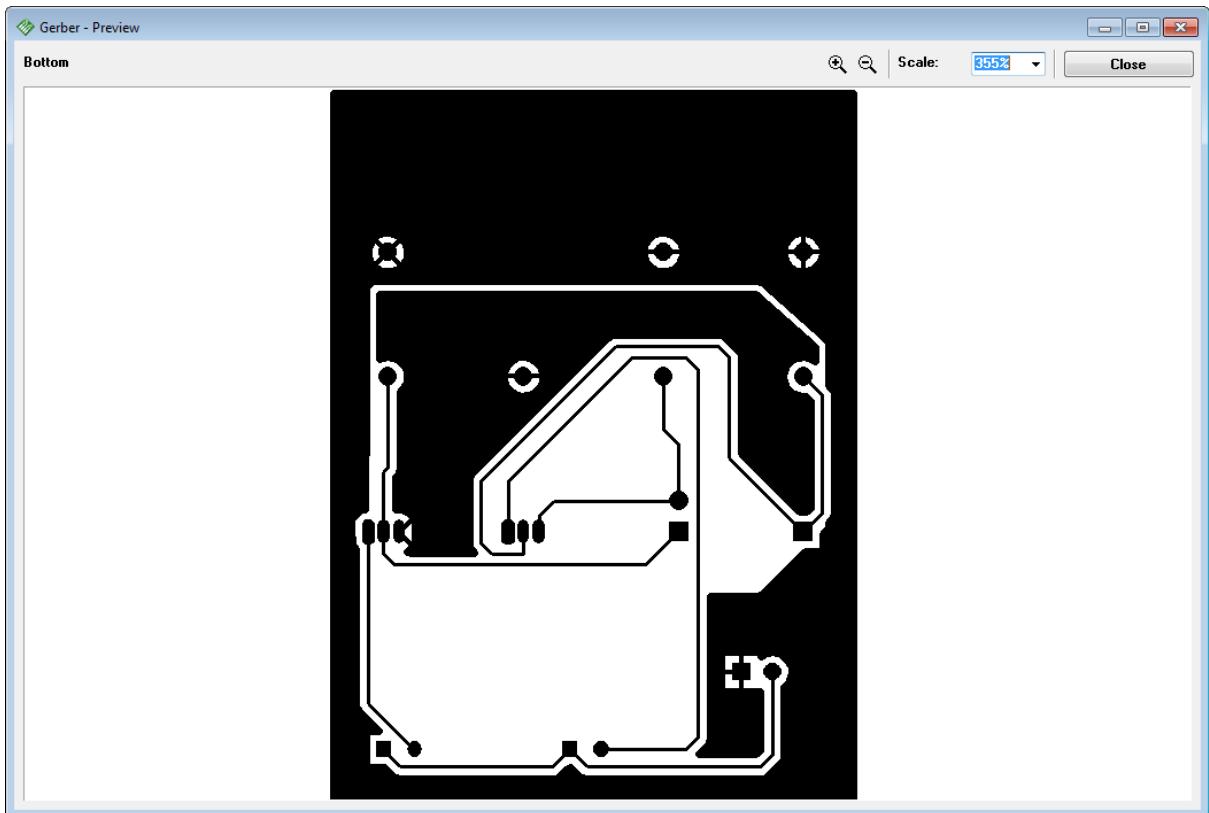
6. Bottom Paste, Mask, Silk and Assy layers are just like their analogs from the Top side. By default all text objects in Bottom layers are flipped - "View \ Flip Text Automatically" option in main menu, however, if that option is OFF, you can flip text for selected layers manually ("Flip Text" box).

7. Board Outline layer includes board outline with defined width.

8. Board layer includes board as a filled polygon.

9. Top/Bottom Dimensions - layers created specially for dimensions. These layers are blank in our case, because current project does not have placed dimensions. Top / Bottom Dimensions can help some manufacturers to avoid mistakes in sizes.

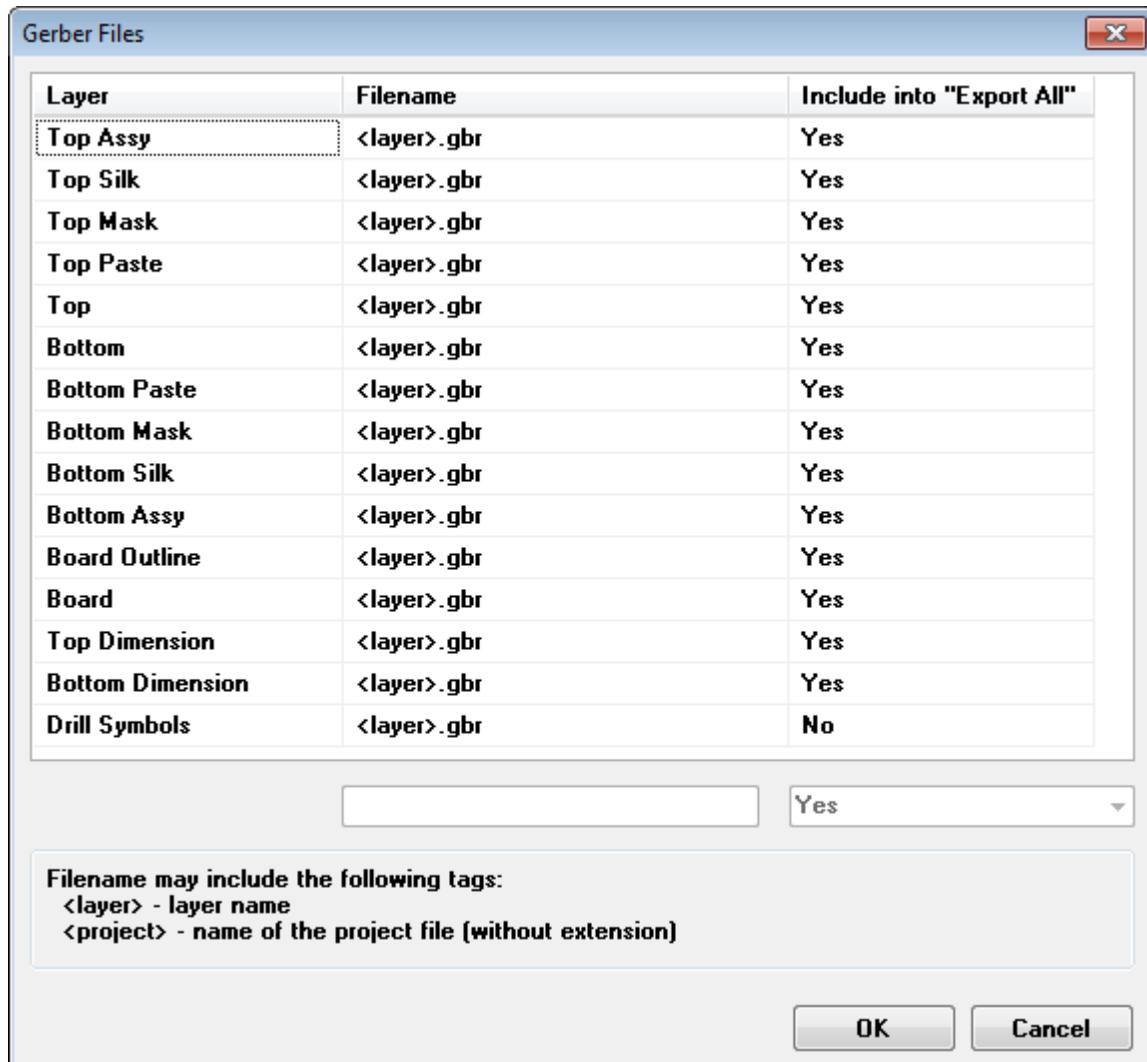
Select "Bottom" layer in the list and click "Preview".



You can zoom in and out. Press "Close" button to close preview panel.

The Offset parameter in DXF, Gerber, N/C drill and "Pick and Place" export dialog boxes is the distance between zero and bottom-left corner of the board. Custom value or design origin can be used (check corresponding item or enter values in "Offset" section of the export dialog box).

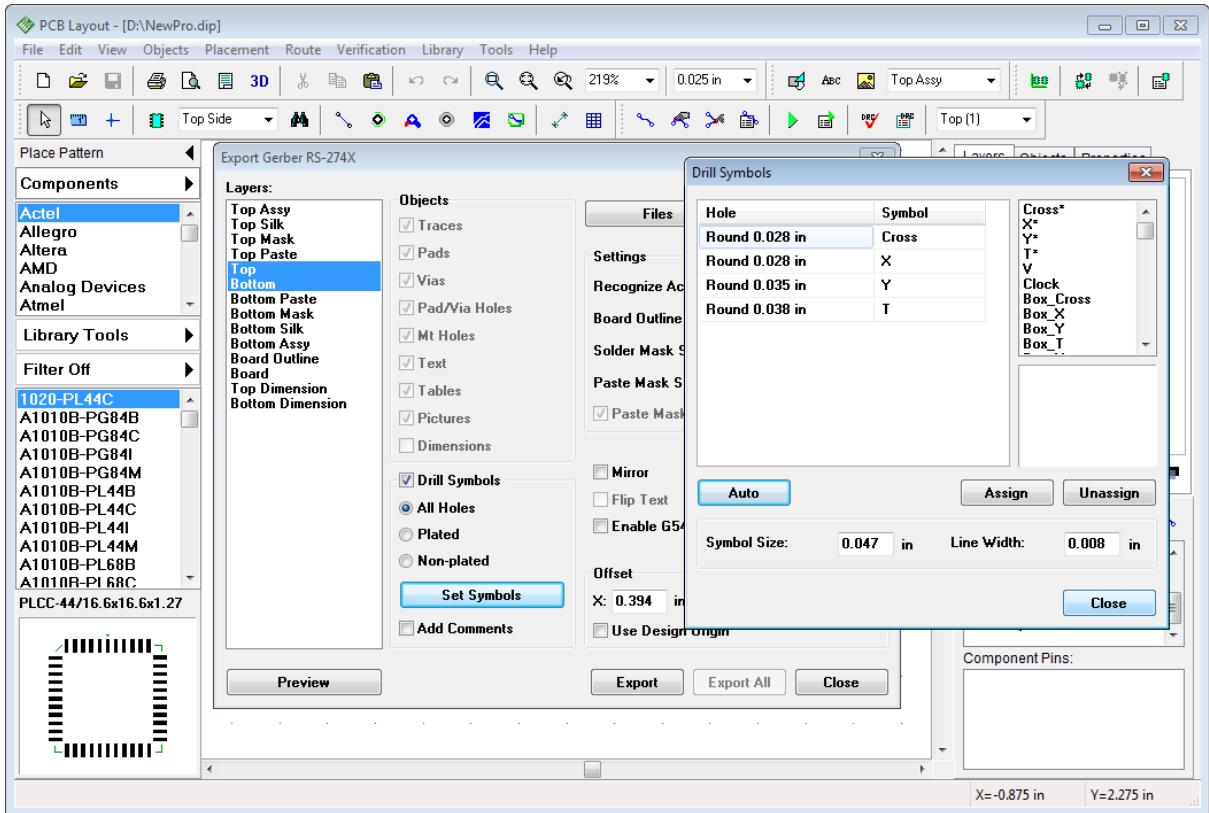
Press "Files" button in the upper-right corner of Export Gerber dialog box. Define filename and extension for each Gerber layer and include or exclude certain layer/s from exporting, when you press "Export All" button. Select layer in the list and type in new name and extension. Layer name and project name tags are supported. We will not change anything, keep <layer> tag - all files will be named as layers. Press "OK".



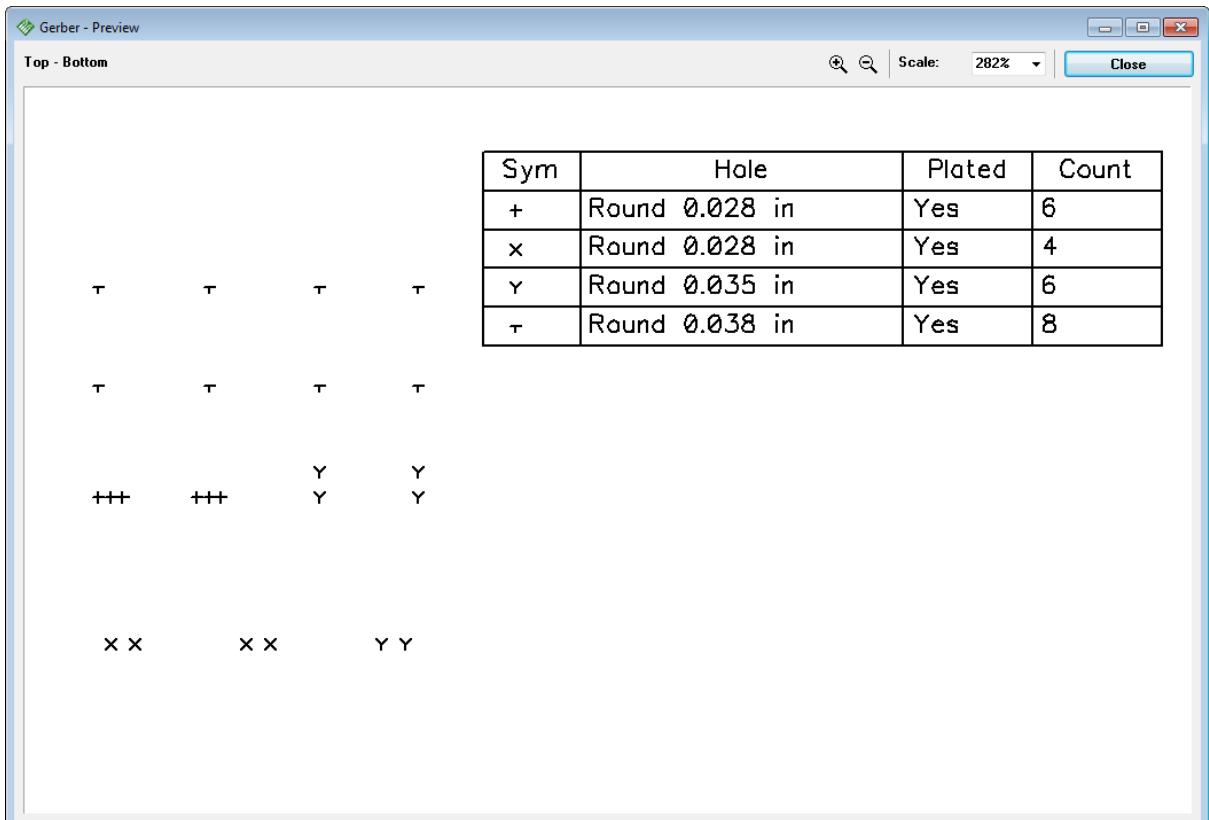
Now it's time to export project. Press "Export All" button in Gerber Export dialog box and save all Gerber files one-by-one (one layer per file). These files together with [N/C Drill Files](#) can be sent to manufacturer for board production.

Drill Symbols (Legend)

Some PCB houses require Drill Symbols. DipTrace allows you to export them as a separate Gerber file. Check "Drill symbols" box in Export Gerber dialog box then press "Set Symbols" button. In the pop-up dialog box you need to assign each hole with the symbol from the list in the right side manually or press "Auto" button to get all symbols automatically. Close this dialog box.



Check "Add Comments" checkbox and press "Preview" button. You will see drill symbols and a table with parameters of the holes.



Close preview window and press "Export" button to save Drill Symbols file. Drill symbols will be exported in separate file just like any other Gerber layer. If apertures are not defined, DipTrace will ask you to set them automatically.

Uncheck "Drill Symbols" box when done. Otherwise you will get blank file / preview while exporting silk, assy, signal layers, etc.

DipTrace allows to export any texts, fonts and Unicode symbols (even Chinese hieroglyphs) as well as raster black and white images (company logo, etc.) to Gerber, but you should define "Recognize Accuracy" for such objects. "3 mil" is set by default, up to 0.5 mil accuracy can be used.

We recommend to check Gerber files with third-party viewer, before sending them to manufacturer. The best option is to use the same software (or free viewer, based on the software) as your board manufacturer, because some programs may read Gerber files a bit differently from official RS-274X specification.

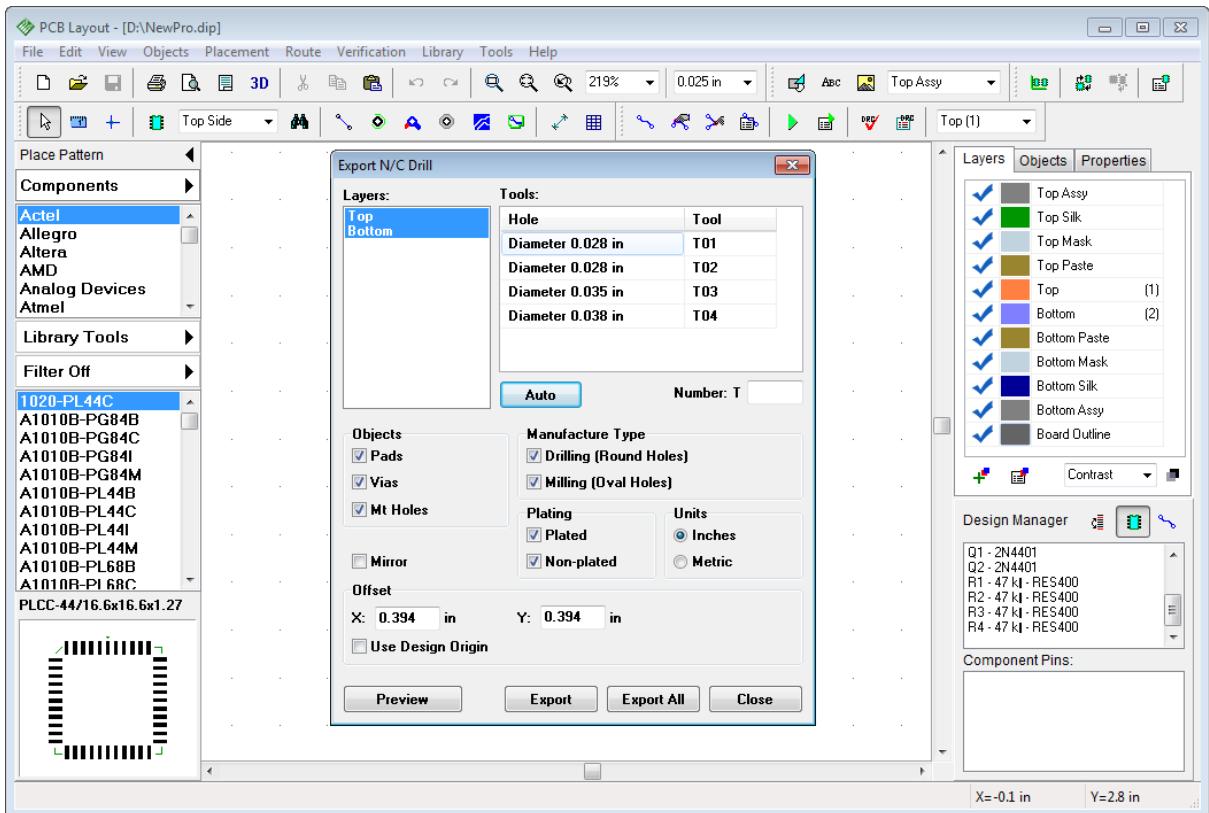
We tried to take into account specifics of different manufacturing software in DipTrace Gerber export feature, but verifying files is a very good practice.

If you don't know what software your manufacturer uses, we recommend [Pentalogix Viewmate](#), it has strict RS-274X conformity.

2.7.3 Create N/C Drill file for CNC machine drilling

If you order board manufacturing at PCB house you need to provide **Gerber and N/C Drill files** to them. Select "File \ Export \ N/C Drill" from main menu. Then press "Auto" button to define tools and "Export All" button to save all necessary files automatically. Use "Preview" to check holes location visually.

If you want to export some holes separately - select layers pair of the via style and press "Export" button. Notice that for through holes all layers should be selected, for Blind/Buried vias - only top and bottom layers involved in the via style.



2.7.4 Order PCB

For those who doesn't want to look for a PCB house to manufacture project, DipTrace allows for a simple ordering tool with our partner PCB manufacturer. No need to export Gerber or N/C Drill files, enter few details and manufactured board will be delivered to your place. Go to "File \ Order PCB" from main menu in DipTrace PCB Layout, review board parameters, specify quantity, manufacturing time, shipping address, name, phone, e-mail and some additional options. Price will be automatically calculated, depending on entered details. Press "Place Order" button. Order page will be automatically opened in web browser to review total cost, including shipping. Online payment - PayPal.

Order PCB from BayArea Circuits

Board Parameters

- Board Size: 1.7 x 2.25 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
- Electrical Test
- Solder Mask Both Sides
- Silk Screen Both Sides if wanted

Additional Options

No Solder Mask (2-layers only)

Comments

Quantity: 5 **Price:** Per Unit: \$10.98 Total: \$54.91

Turn Time: 5 days

Order Details

First and Last Name *	Email Address *
John Doe	John_Doe@*****.com
Company	Confirm Email Address *
*****	John_Doe@*****.com
Phone *	
+1*****	
Shipping Address	
Country *	City *
United States	Sacramento
Address *	State *
	California
	Zip Code *

Close **Place Order...**

Congratulations! You have finished designing a simple project with DipTrace.

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

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

3 Creating Libraries

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

Important:

A regular component in DipTrace consists of schematic symbol, pattern drawing connected to schematic symbol and 3D model, connected to pattern drawing. All three integral components are saved in three different library formats with different file extensions (*.eli - components; *.lib - patterns; *.wrl, *.3ds, *.iges, *.step - 3D models). Component Editor is used to manage components - draw schematic symbol and connect it to pattern drawing. However, pattern drawing is not editable in Component Editor, Pattern Editor should be used instead. Patterns, symbols and models can exist as separate stand-alone units but correct component always have all of them properly connected.

In most cases you can find appropriate pattern in DipTrace standard libraries and attach it to new component, but for educational purposes we will create new pattern in this tutorial section.

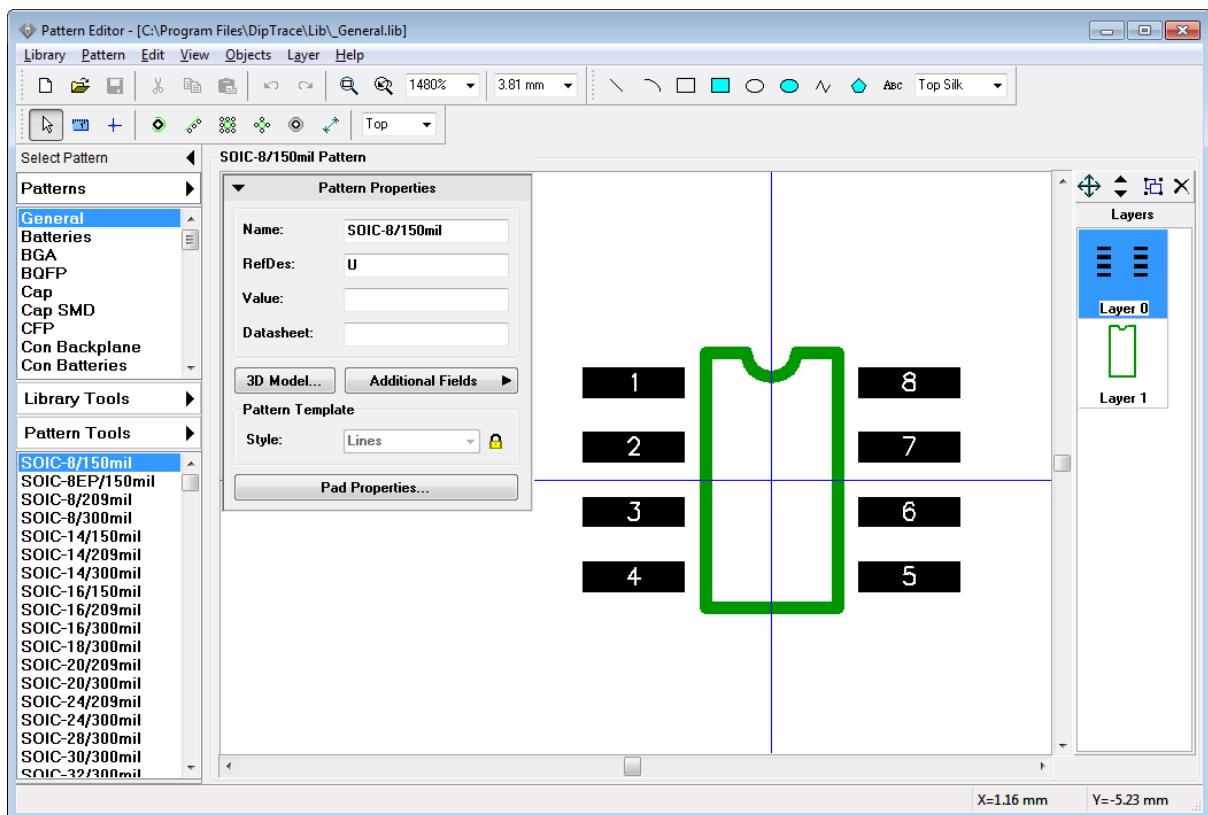
3.1 Designing a pattern library

Open DipTrace Patter Editor, i.e. go to "Start → All Programs → DipTrace → Pattern Editor".

3.1.1 Customizing Pattern Editor

First of all we need to show center point and X,Y-axes, select "View \ Display Origin" from main menu or press "F1" hot key (if it is not displayed yet). Notice that you can change origin at any time while designing pattern ("View \ Define Origin \ ..." from main menu or quick-access button on the instruments toolbar). Origin is a zero point of the pattern when you place, rotate or change pattern position by coordinates in **PCB Layout**.

Pattern Properties panel in the upper-left of the design area allows to design pattern 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 panel click arrow button in its upper-left corner. To hide / show it - "View \ Pattern Properties" from main menu.

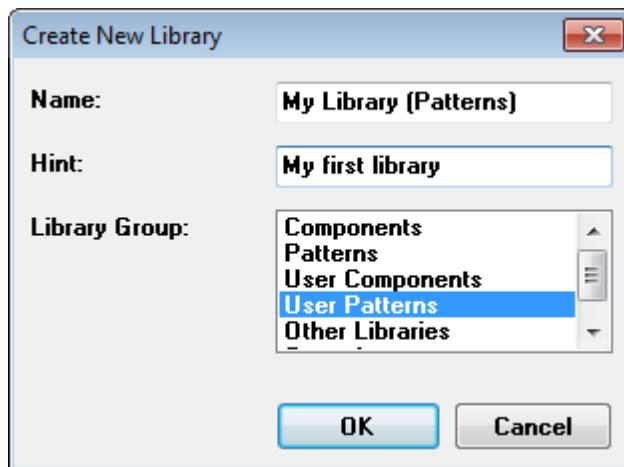


Use "+" and "-" or mouse wheel for Zoom In and Zoom Out in Component and Pattern Editors or change scale in the scale box on the instruments toolbar.

3.1.2 Create/Save library

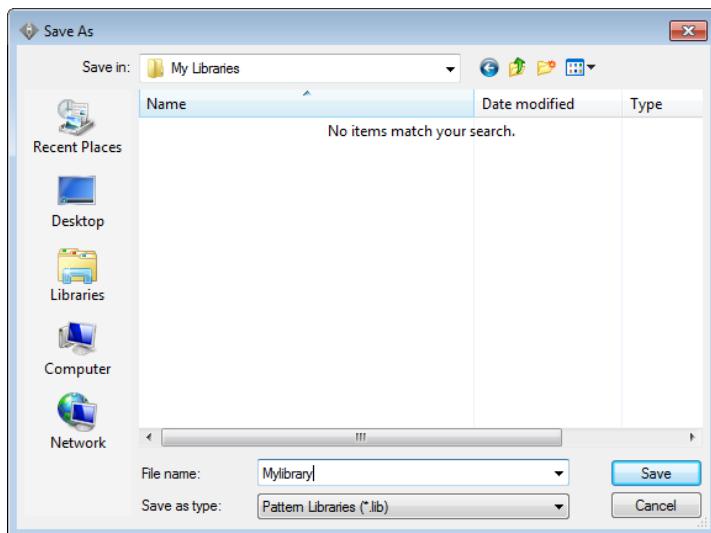
Create Library

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



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

Saving Library



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

DipTrace does not allow to save user libraries in the folder with standard libraries.

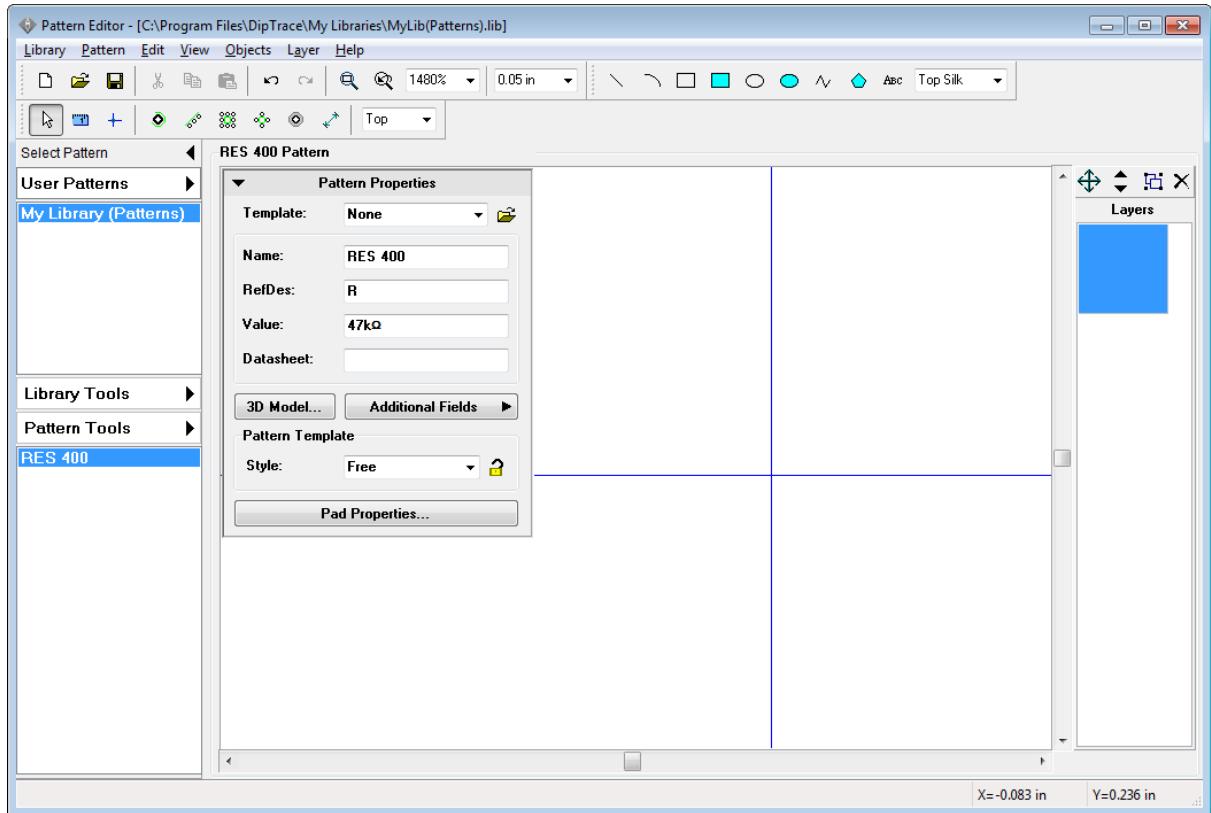
Type in the file name (not shown in DipTrace) and press "Save".

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

3.1.3 Designing Resistor (pattern)

We will design first pattern of the library: resistor with 400 mils lead spacing.

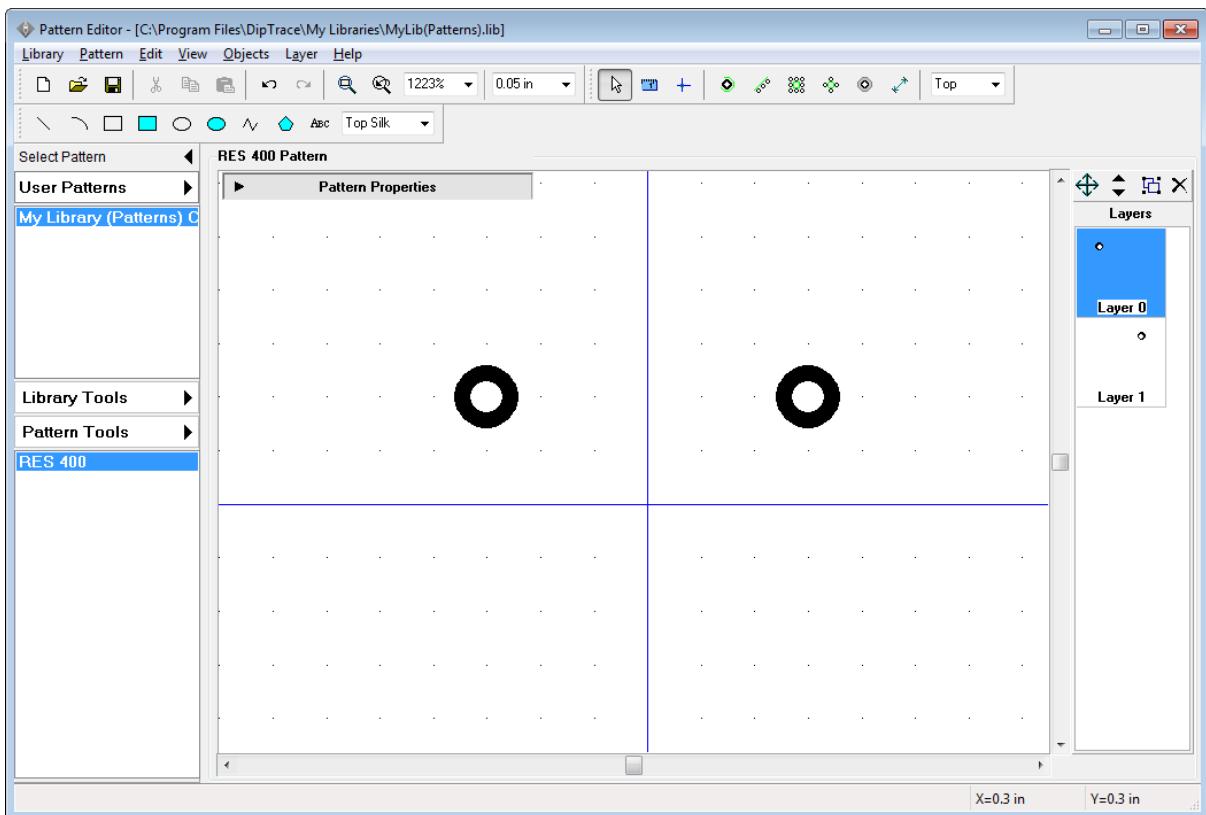
First we need to define name and description. Type "RES 400" into the Name field, "R" - into the RefDes field and "47k " - into Value field of Pattern Properties panel. In DipTrace Pattern and Component Editors you need to define just a basic RefDes (not a number index). For example, when you place several resistors - R1, R2, R3, etc. designators will be assigned automatically. "U" is default value in RefDes field.



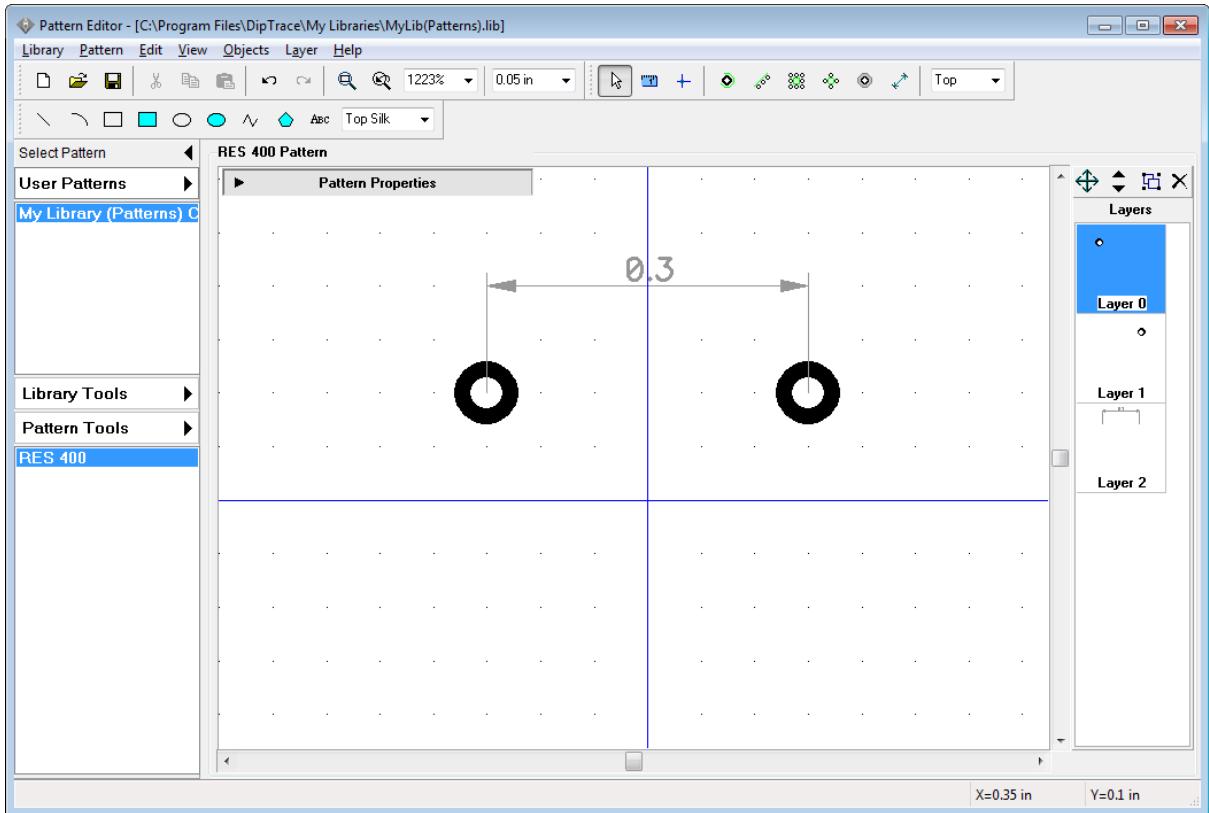
For this pattern we've used "Free" style, but it is faster to use "Lines" instead. You'll see how to do this in one of subsequent patterns.

Placing Pads

Please make sure that "0.05 in" grid is selected (change measurement units in "View \ Units" from main menu) then minimize "Pattern Properties" panel. Select "Place Pad" tool on the "Objects" panel and left click on the design area to place two pads like on the picture below. Right click to exit placement mode.



Such placement is not an accurate method, therefore we should check and correct pad coordinates. First we should place dimensions, this will make editing more simple to understand. Select "Objects \ Place Dimension \ Horizontal" from main menu or Place Dimension \ Horizontal tool on the objects toolbar, left click in the center of first pad, then in the center of second pad, move mouse a bit upwards and click one more time to place dimension. Key points of objects are highlighted when you hover over them. Dimension pointer will be connected to those key points and is recounted automatically, when you move/resize objects.



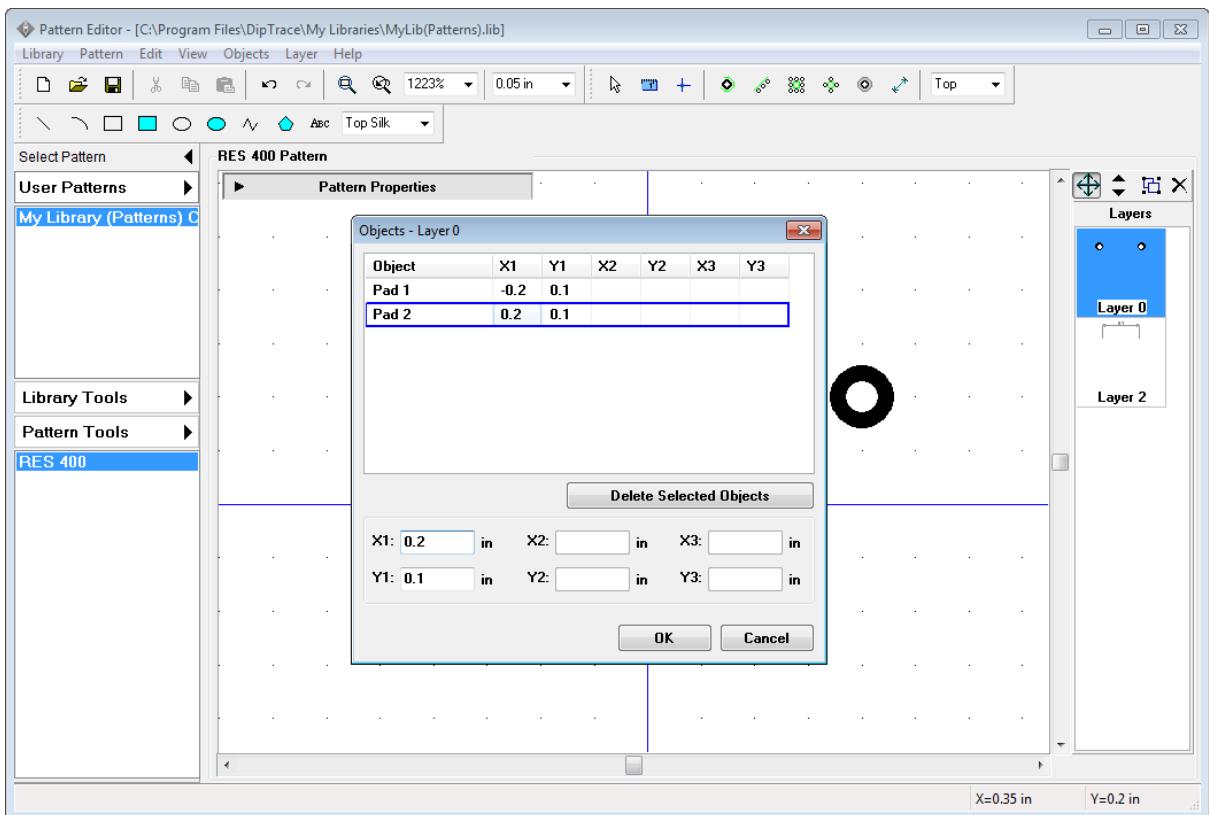
Right click on the dimension line and select Properties from the submenu to change Layer, Units, Arrow Size e.t.c. Drag and drop dimension line like a regular object if needed.

Layers

On the right side of the screen there is a layers list. One layer per object on the design area. Select layers 0 and 1 (hover over the "Layer 0", hold down the left mouse button, move cursor to "Layer 1" and release mouse button). Select "Layer \ Merge Layers" from main menu or press corresponding button in the upper part of layers panel. You have made a single layer with two pads in it; double click it to open "Layer Objects" dialog box.

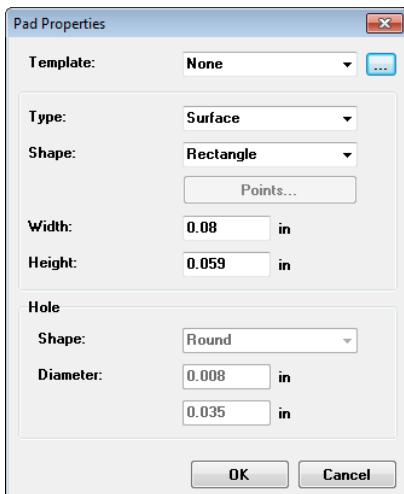
These are only logical layers for editing (not signal or silk layers).

Select pad / pads with incorrect coordinates and change X coordinates to make 400 mils (0.4 inch) distance between pads then click "OK" to close the dialog box.



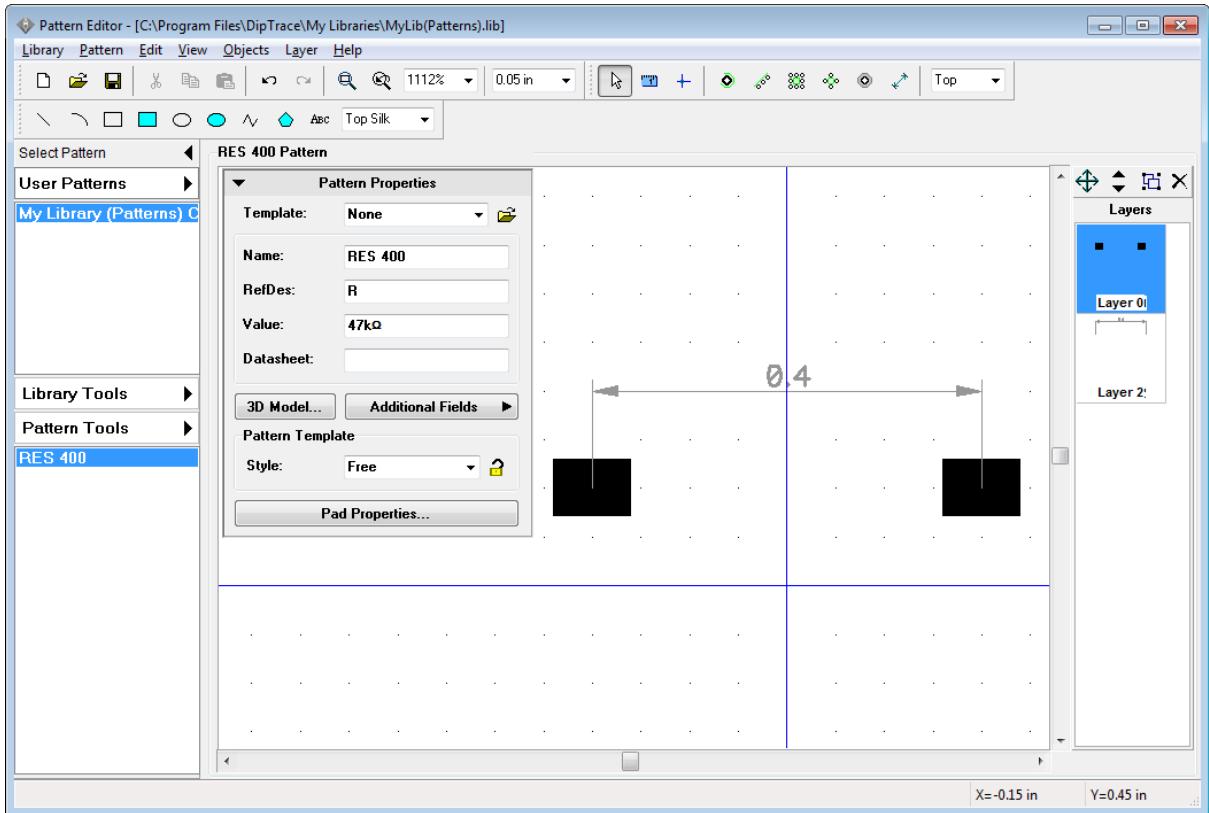
You can see that dimension object was recounted automatically and now displays correct distance value.

Pad Properties



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

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



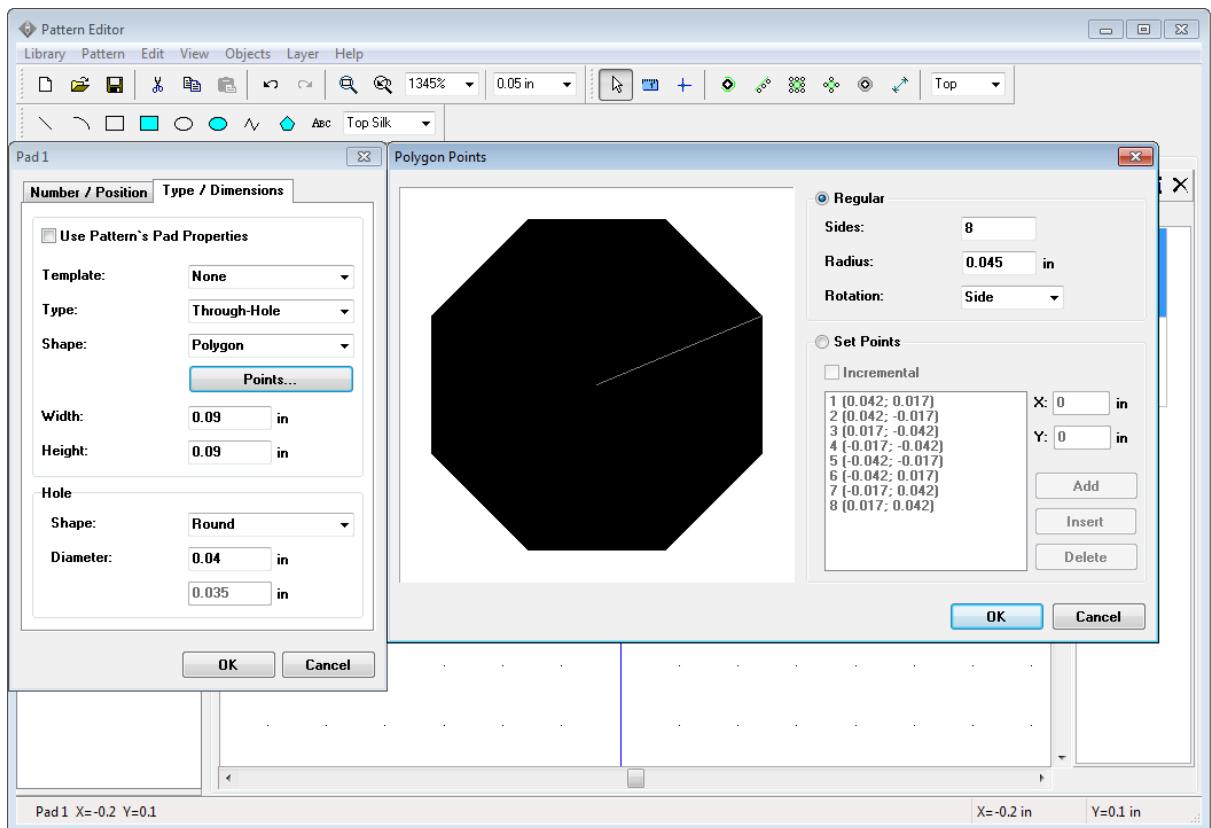
Notice that you can change side for surface pads, i.e. place them on the bottom side of the PCB. Select pad(s), right click one of them and select "Change Side" from the submenu. Current side for placing new pads and shapes can be selected on objects panel (drop-down box with "Top" text).

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

Polygon Pads

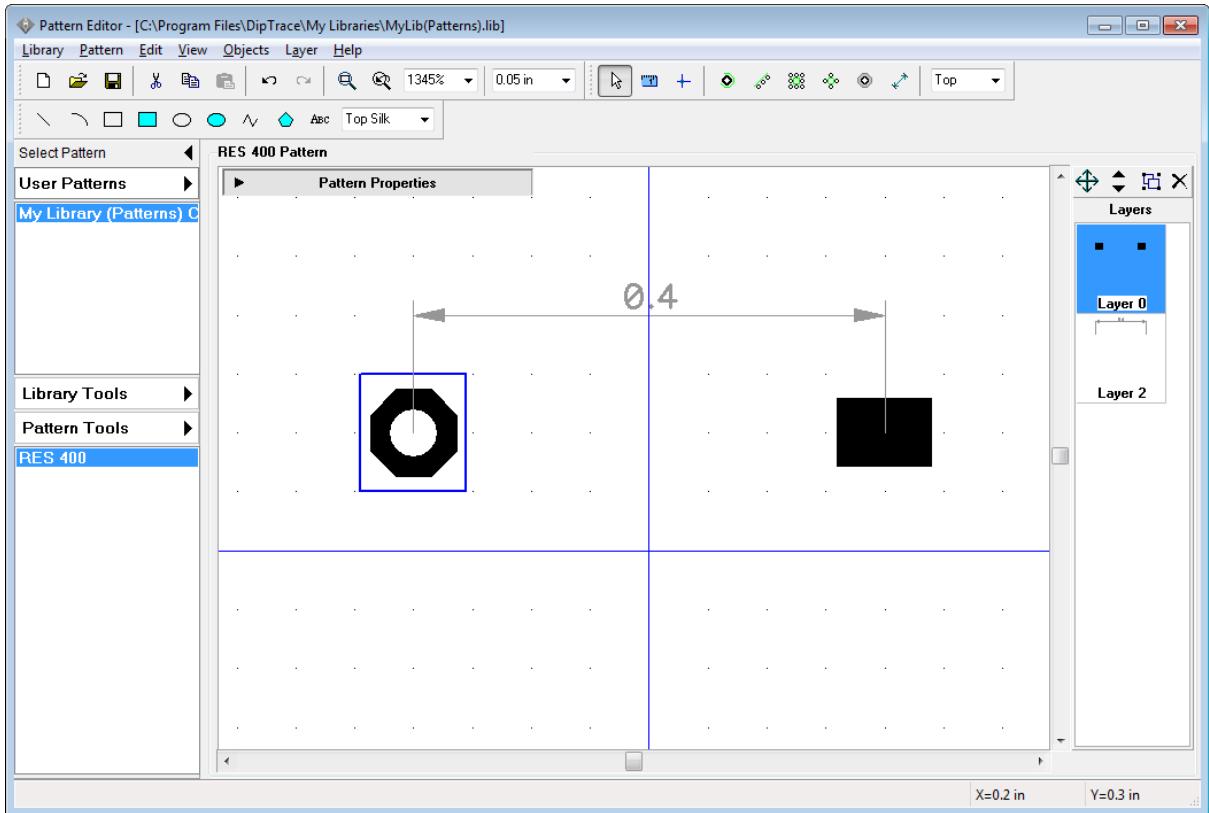
Change type to "Through-Hole", shape to "Polygon", width and height to "0.09 in" and pad hole diameter to "0,04 in" then press "Points" to open Polygon Points dialog box. Here you can create regular or custom polygon shape.

Select: "Regular", "8 Sides" and specify "0.045 in" Radius. Custom shapes are edited with the table below or visually in the preview field.



Press "OK" to apply polygon shape.

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



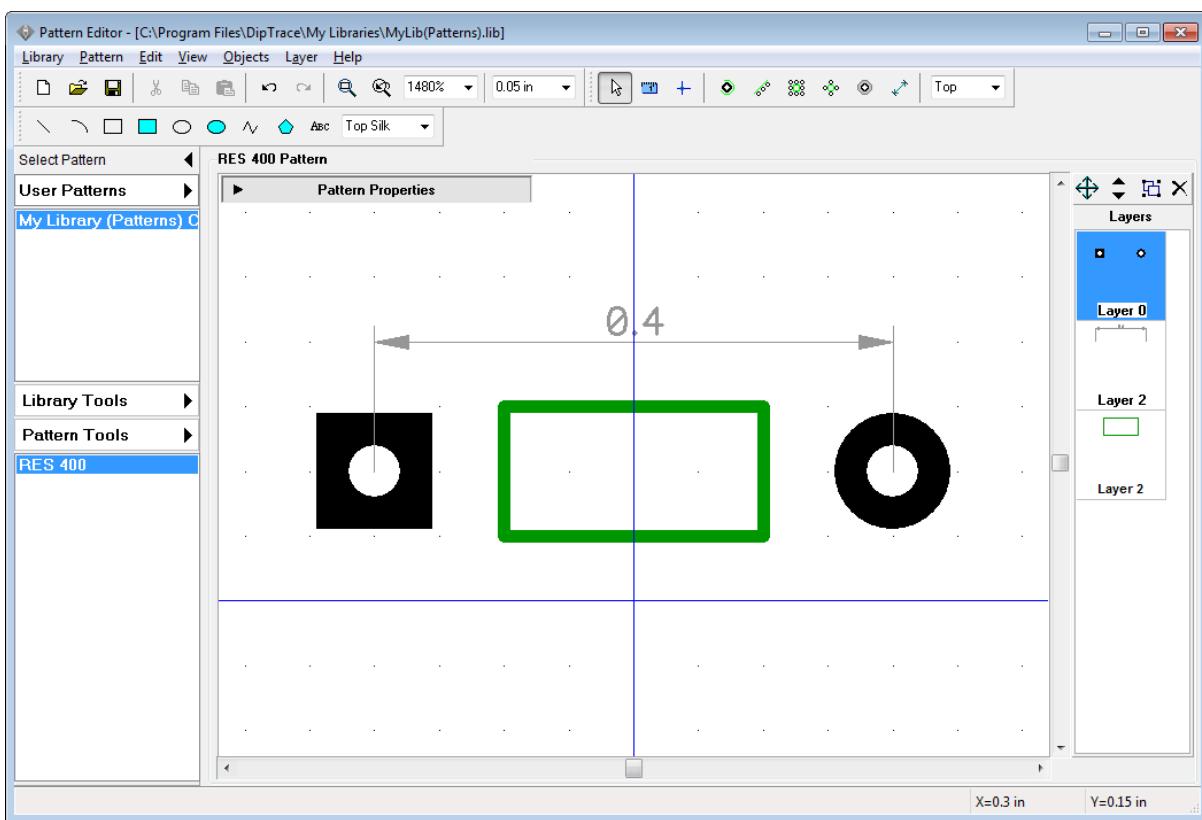
There is also another way to create polygon pads. Select "Polygon" tool on the Drawing toolbar and draw a shape directly on the design area. Make sure that shape is on signal layer (select "Signal" from the drop-down list on the drawing toolbar prior to drawing or change shape layer in shape properties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

Please define the following properties for the pads:

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

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

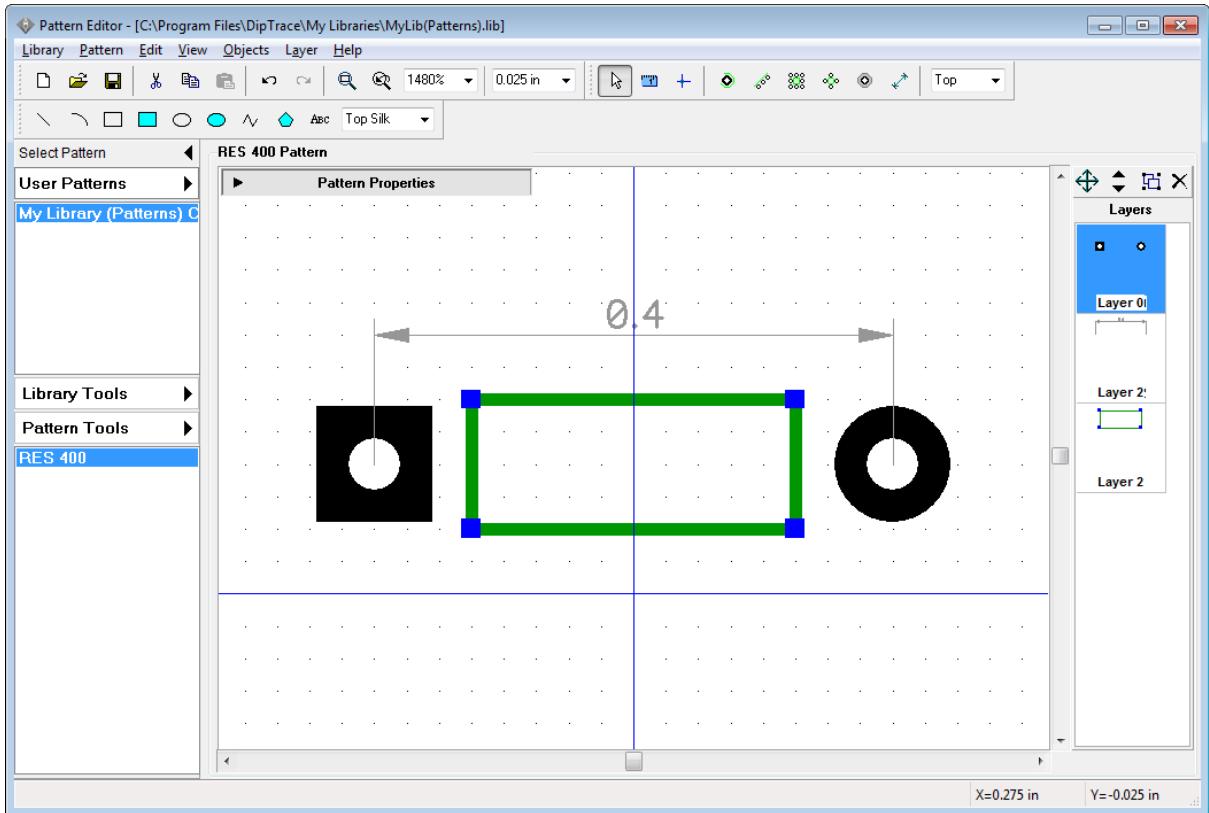
Now let's place silkscreen for this resistor. Select "Rectangle" button on the Drawing panel, make sure "Top Silk" layer is selected. Place rectangle by clicking on two key points.



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

Silk shape looks too small for this pattern. There are several ways to change it: 1) Select "Layer \ Objects" from main menu and enter new coordinate values; 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 drag-and-drop method, but first change grid size to "0.025" inch with "Ctrl-" hot keys or grid box on the instruments toolbar. Then hover over rectangle key points and resize shape (mouse cursor shows possible directions).



Center pattern by selecting "Edit \ Center Pattern" from 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.

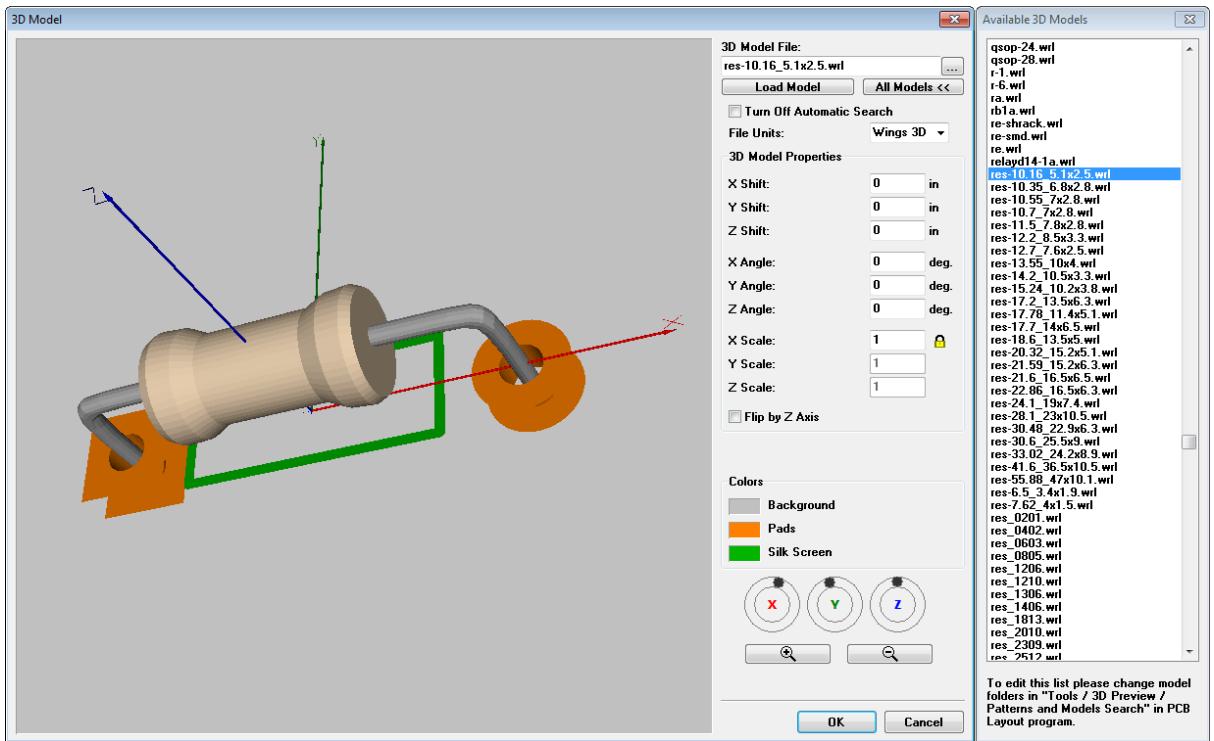
Press "Save" button on standard toolbar. We will attach 3D model for this resistor in the next topic of this tutorial.

3.1.4 Attaching 3D Model

*Please make sure that you have downloaded and installed free **3D Models package** from [DipTrace official website](#), which contains more than 3500 3D models for various components.*

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

In the pop-up dialog box press "All Models" button and the list of all available 3D models will pop up. Scroll it down to find "res-10.16_5.1x2.5.wrl" model, click it and it will appear on the design area on the footprint. You can rotate model in three axes, zoom in and out and pan 3D Model preview with mouse and buttons at the bottom of the panel.

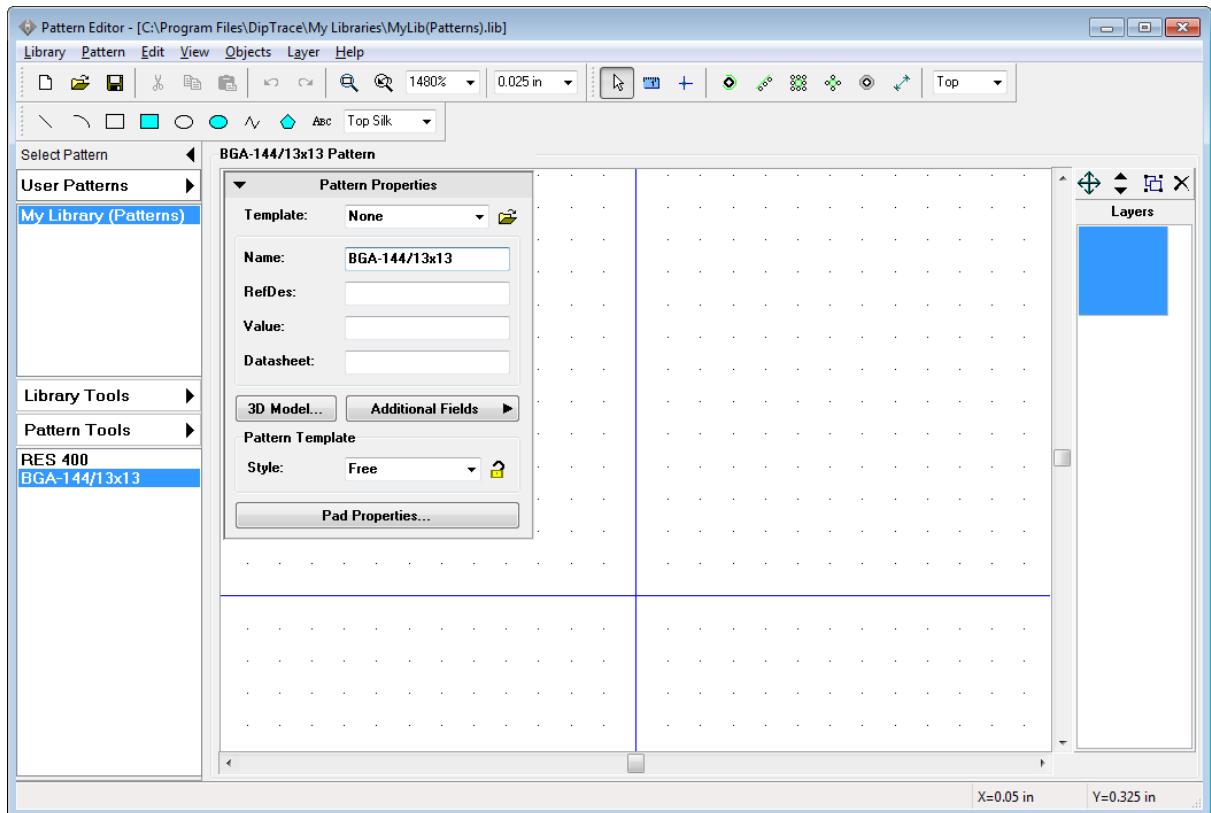


DipTrace automatically places model to fit pattern's drawing, however, sometimes you may need to adjust 3D model location. Just enter new values into corresponding fields in 3D Model Properties section (shift, angle and scale for each axis). More details about [DipTrace 3D Module](#) [206] later in this tutorial and in DipTrace Help ("Help \ Pattern Editor Help" from main menu).

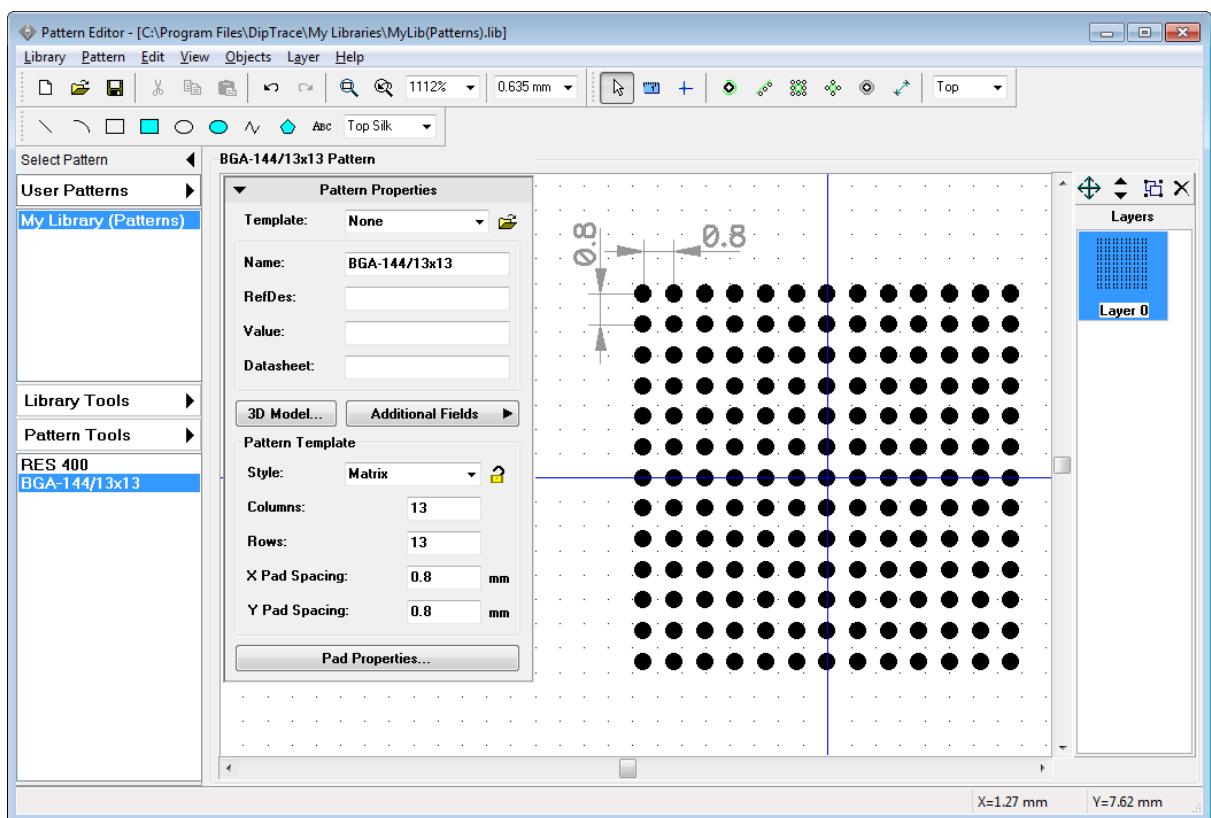
Press "OK" to attach 3D model and then save pattern library.

3.1.5 Designing BGA-144/13x13

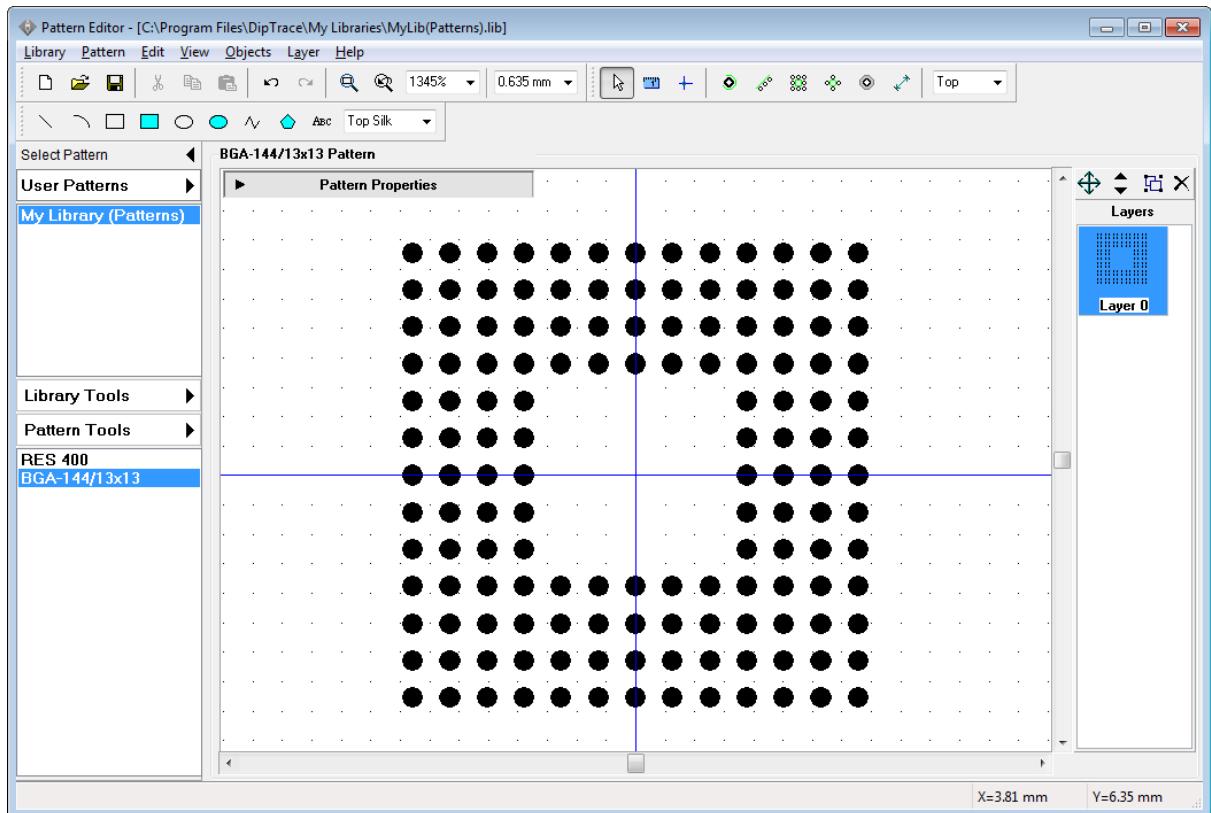
Press "Pattern Tools \ Add New Pattern to My Library (Patterns) Library" from main menu to add new empty pattern. It has appeared in the list of library patterns on the Library manager panel. New pattern is automatically selected. We will create BGA-144/13x13 (x0.8_10x10) pattern using pattern types and automatic pad numeration. Maximize pattern properties panel and type in pattern name.



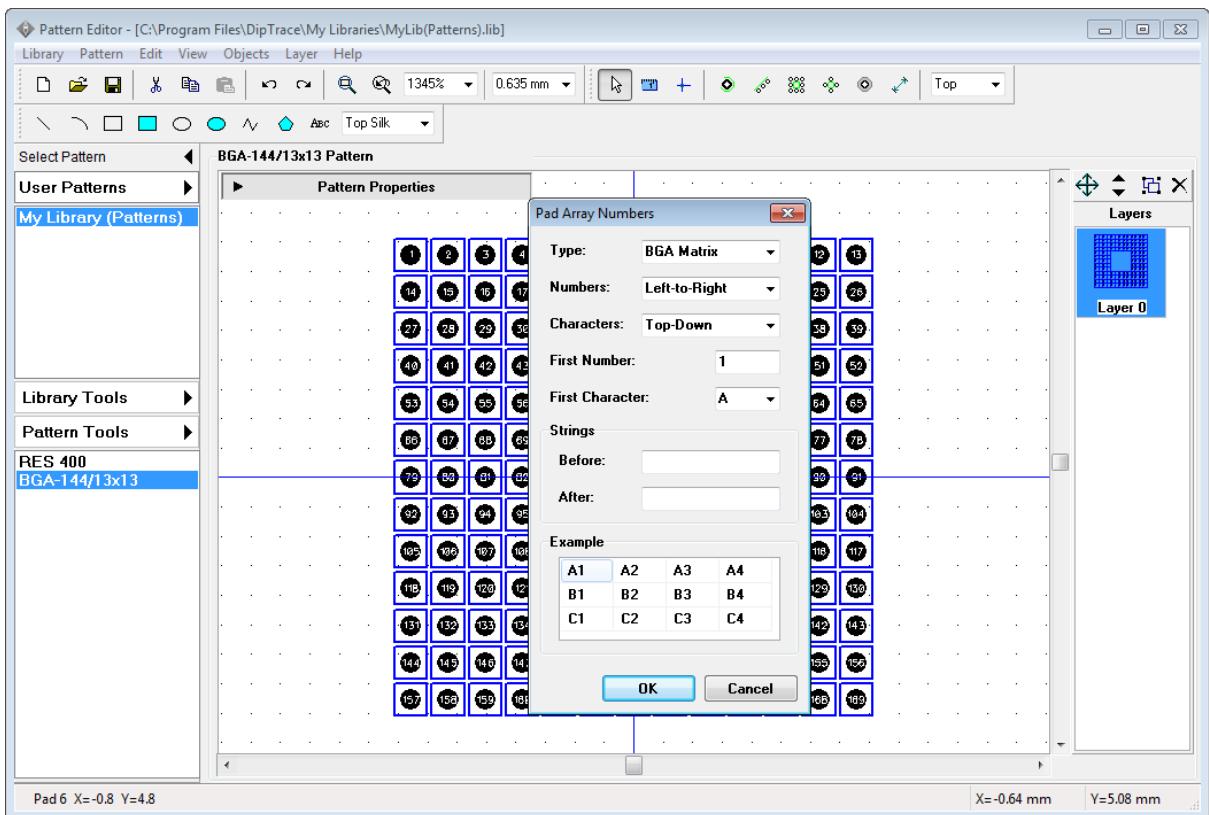
Change units to "mm", select "View \ Units \ mm" from main menu. Then go to "Pattern \ Pad Properties" from main menu and define: "Type: Surface", "Shape: Ellipse", "Width: 0.45 mm", "Height: 0.45 mm". Press "OK" to apply changes to default pad properties. On the Pattern Properties panel set: "Style: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



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

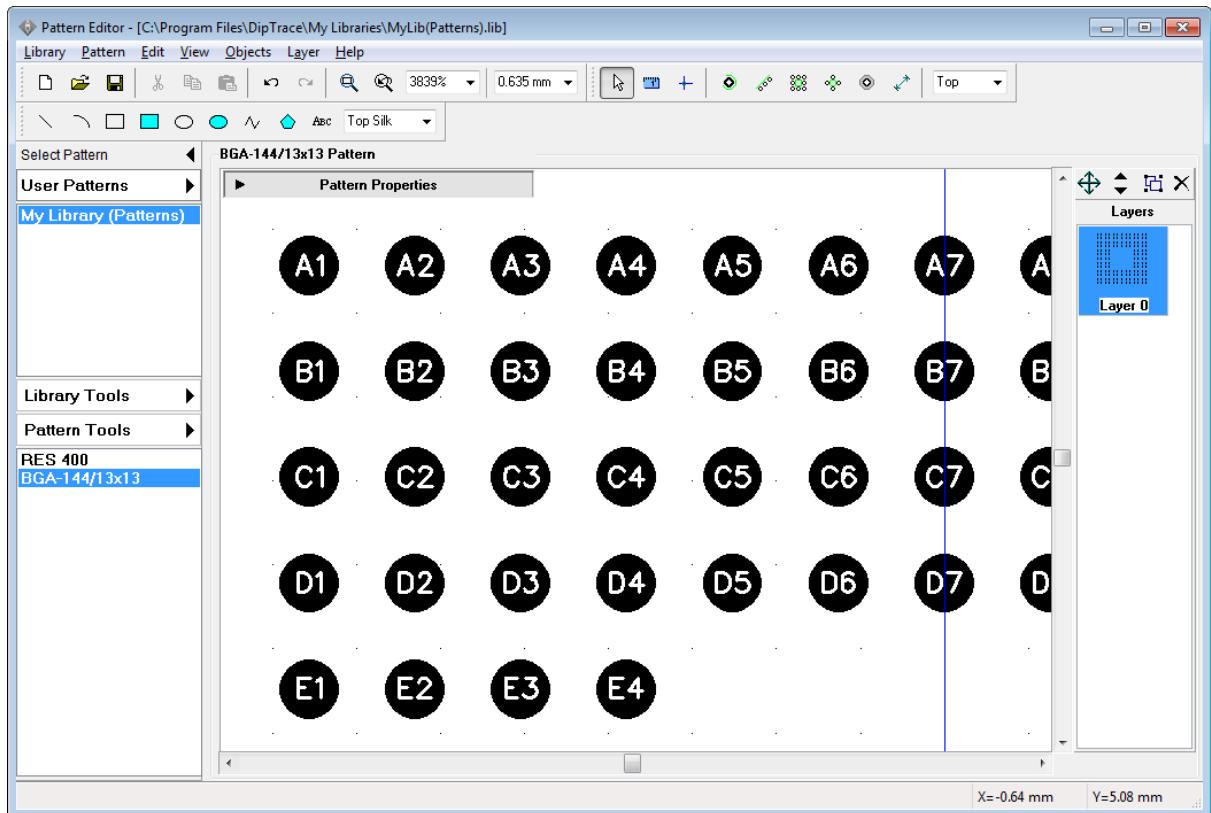


Select "View \ Pad Numbers \ Show" from main menu to display pad numbers. Notice that our matrix has "1" - "169" numbers, but BGA pads should be A1,A2, A3 etc. Select all pads ("Ctrl+A" or box selection), right click on one of the pads and choose "Pad Array Numbers" from the submenu. In the pop-up dialog box select "Type: BGA Matrix", keep other settings and press "OK" button.

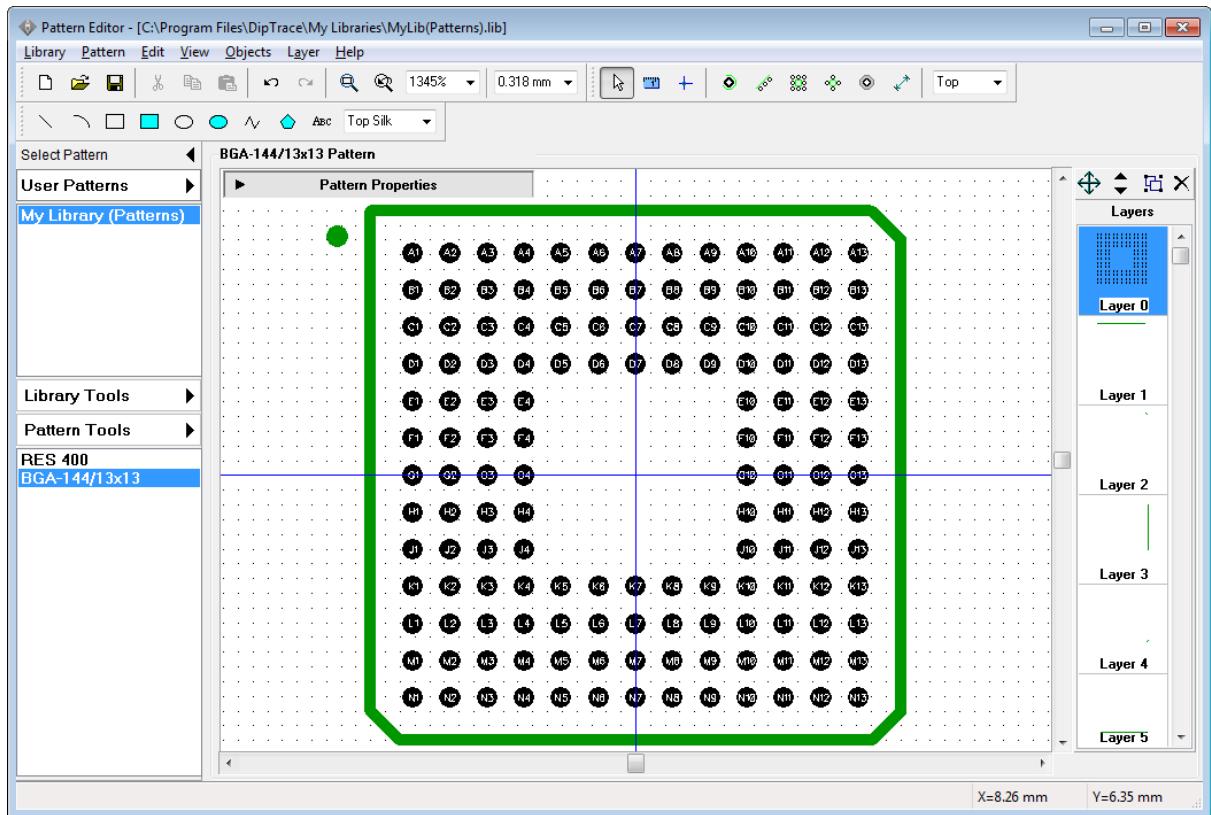


Now pad numeration is correct.

Notice that for "Contour" type of numeration first pad will be the one you clicked on. Hence you can numerate contours (QUAD patterns) starting from the upper-left, center or any other pad of the pattern.



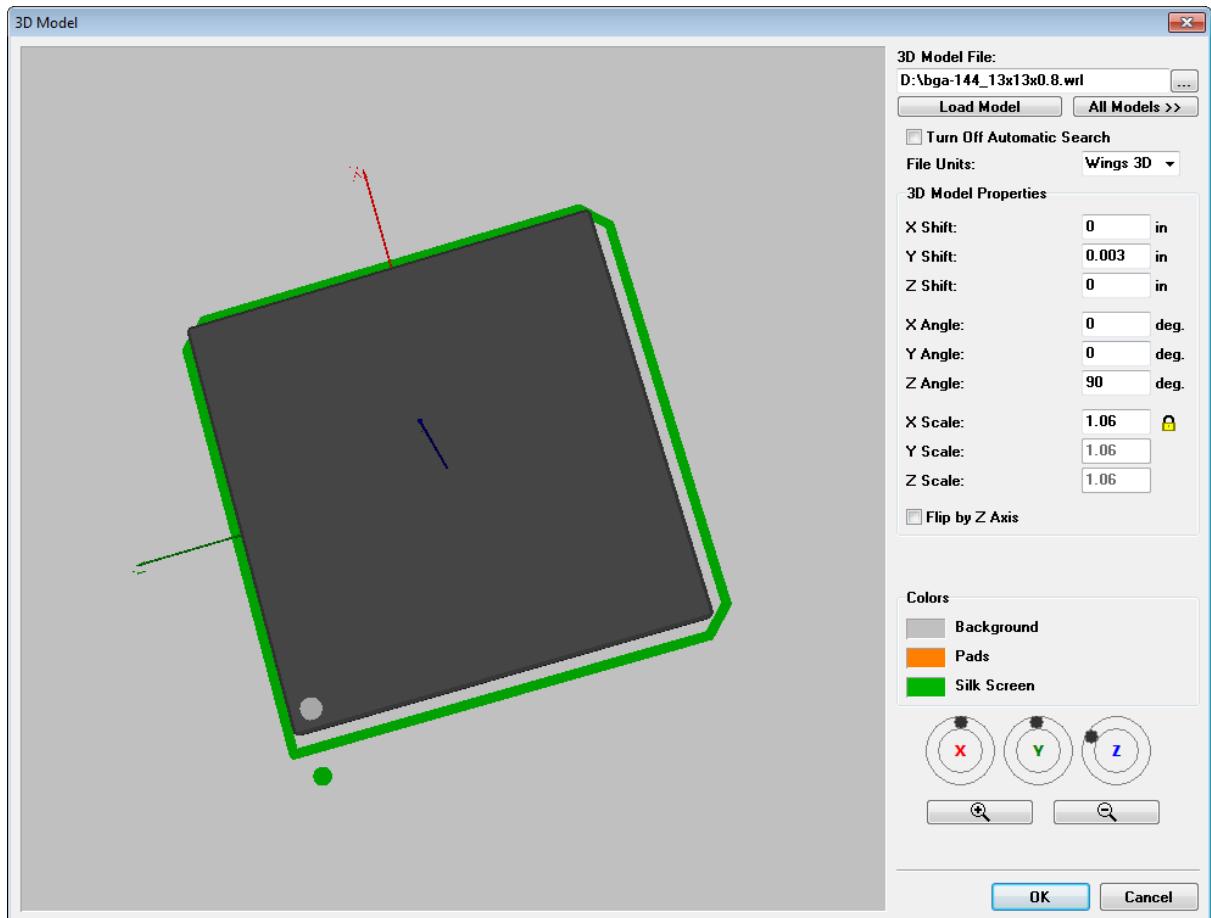
Now draw silkscreen for the pattern (like on the picture below), using tools on the drawing toolbar. Grid can be changed with "Ctrl+", "Ctrl-" or turned OFF/ON with "F11" hot key. Objects can be moved with drag and drop or by "Move Layer" feature (select layer in the list in the right side and drag-and-drop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



BGA pattern is ready. You can attach 3D model now. Unfortunately, there is no 3D model in DipTrace 3D models package that fits this footprint precisely. DipTrace allows to attach 3D models in *.3ds, *.wrl, *.step, *.iges formats. You can download models from component manufacturer's websites or create them in any 3D CAD.

3D Model Properties (Location)

Press "3D Model" button on the Pattern Properties panel. Then click "..." button and select 3D model file on your computer. Press "Open" (or enter 3D model's HDD address and press "Load Model" button) - 3D model will appear on the preview area. Change 3D Model Properties in case model does not correspond to the footprint. For example, we have entered new values into Y Shift, Z Angle and X Scale fields for precise location (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". Then save pattern library ("Ctrl+S" or "Save" button on the toolbar).

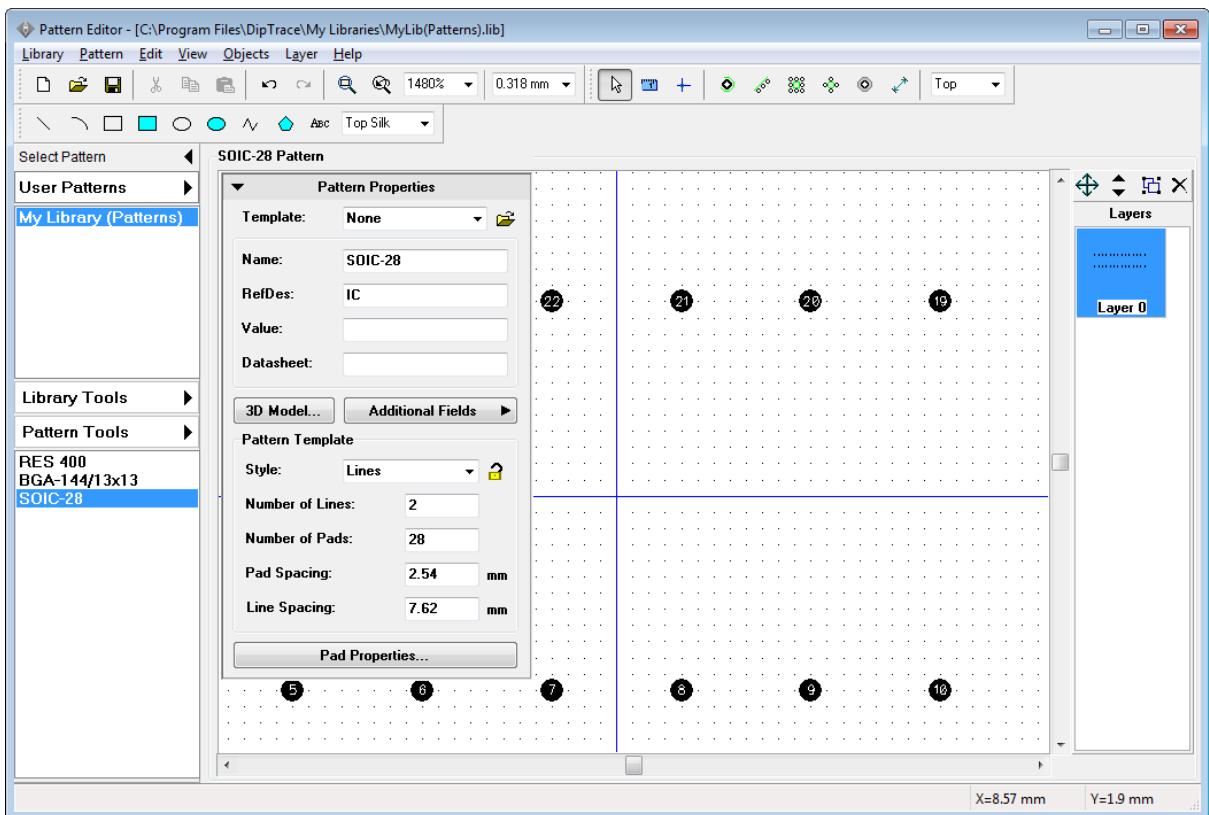
3.1.6 Real component Design. SOIC-28 pattern

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

When you start creating component and you don't have an appropriate pattern for it - always start from creating pattern and then create component's symbol.

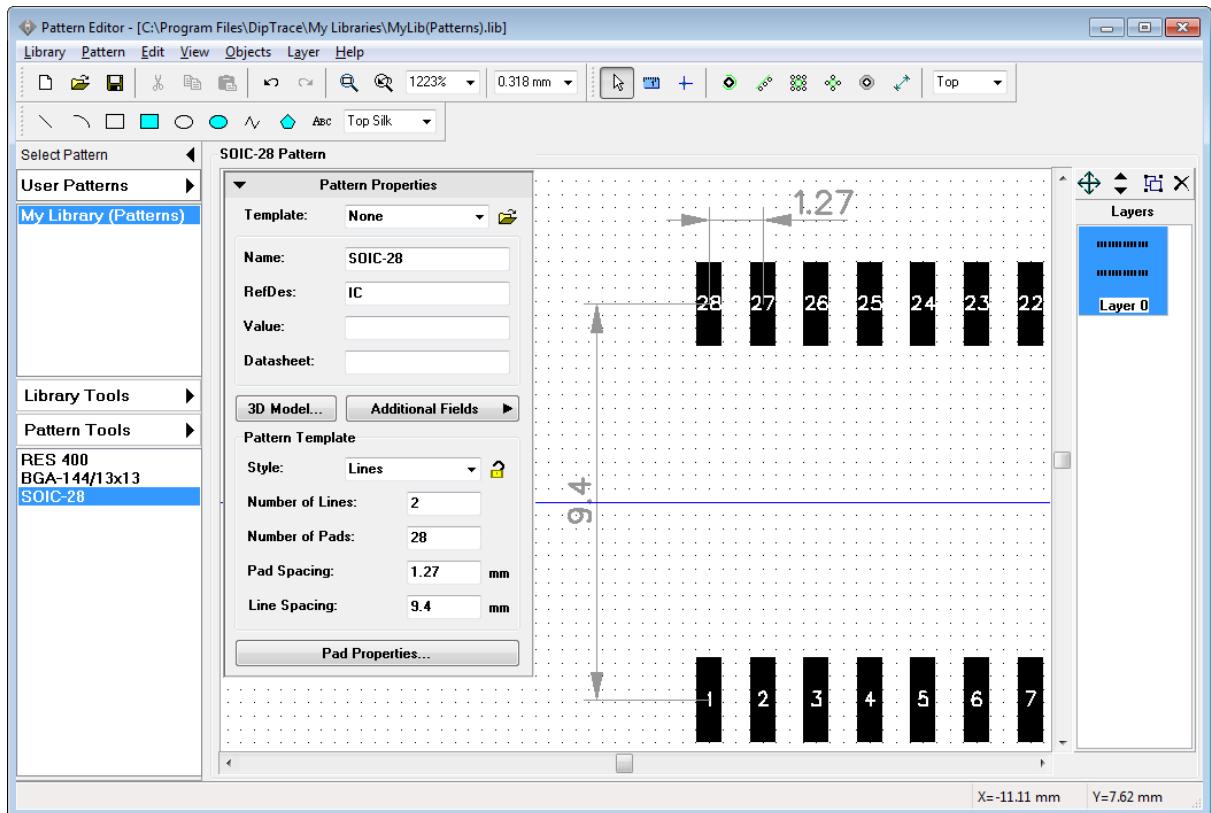
We start from making pattern. Add new pattern to the library ("Pattern \ Add New Pattern To 'My Library (Patterns)' Library" from main menu) then enter name "SOIC-28" and RefDes "IC".

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.

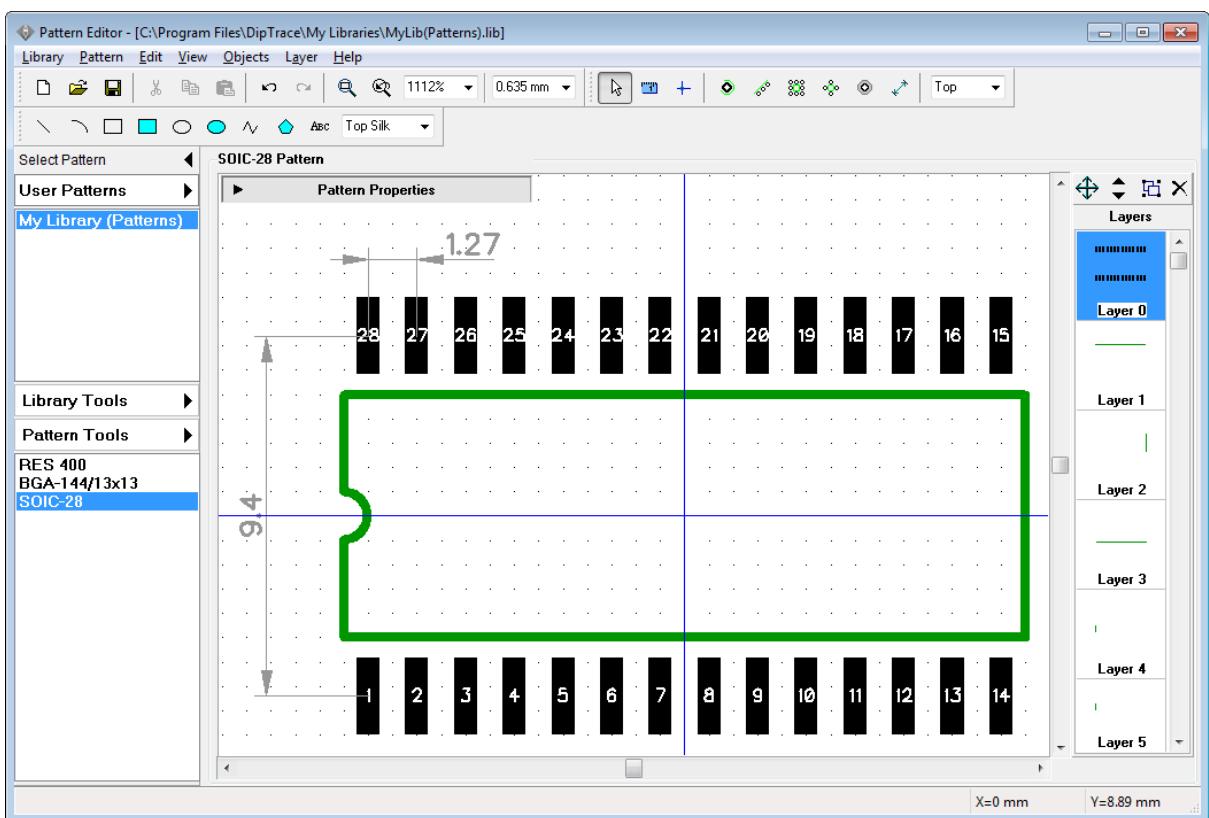


We should define correct pad spacing, line spacing and pad settings for the pattern. SOIC-28 (7.50 mm) footprint dimensions can be found in [Microchip package specifications PDF](#) at Microchip website (page 197 in the latest revision at the moment of writing this tutorial) or you can take SOIC-28 pattern from standard DipTrace libraries as an example.

Define default pad settings ("Pad Properties" button): "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 Pattern Properties panel.



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



Datasheet requires pattern to be rotated 90 degrees - select "Edit \ Rotate Pattern" or "Ctrl+Alt+R". Attach "soic-28_300mil.wrl" 3D model from standard 3D Models package. We will attach this pattern to "PIC18F24K20" component which we will create in the [Component Editor](#) [136] later in this tutorial.

Save library and close Pattern Editor.

3.2 Designing a component library

Open DipTrace Component Editor, i.e. go to Start → All Programs → DipTrace → Component Editor.

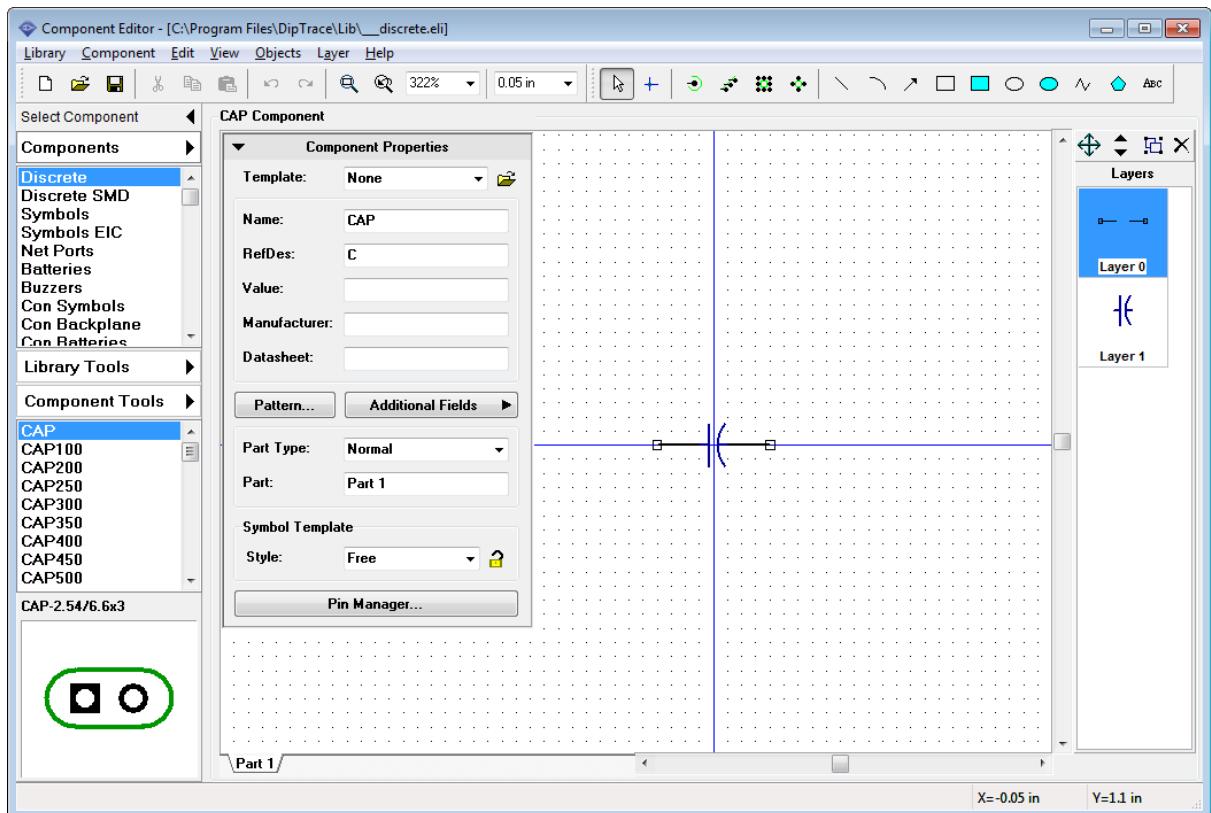
Component editor allows to create / edit and manage components and libraries in DipTrace. Component Editor allows to attach component's pattern to symbol, but it doesn't allow to edit patterns - [Pattern Editor](#) [90] should be used.

3.2.1 Customizing Component Editor

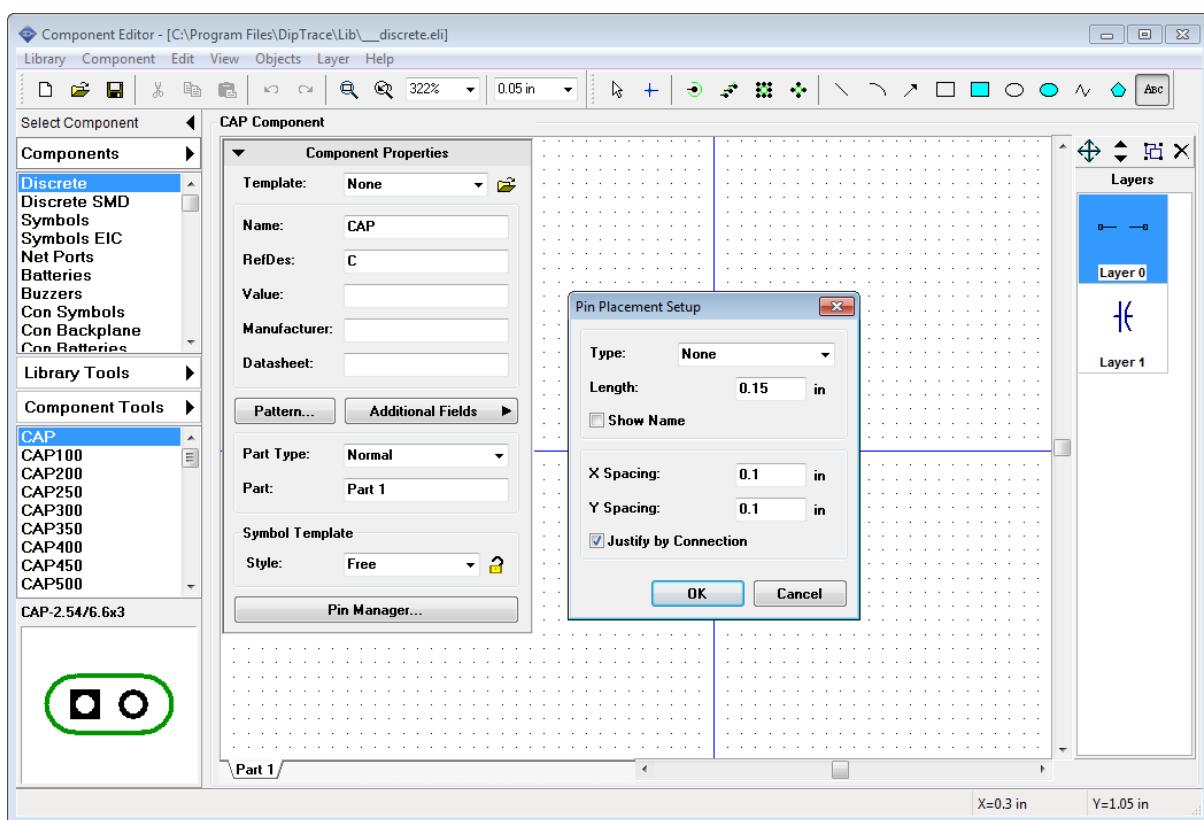
Customizing Component Editor is almost the same as customizing Pattern Editor. Select "View \ Display Origin" from main menu to show zero point and X, Y axes (or press "F1") if it is not displayed yet. Component Properties panel in the upper-left side of design area can be minimized or hidden using buttons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

In Component Properties panel you can define 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 components select "Objects \ Pin Placement Setup" from main menu. We will not change these properties now, but notice that length and X,Y Spacing should be divisible by grid step to create all key points on the grid points. We recommend to use 0.1 inch grid.

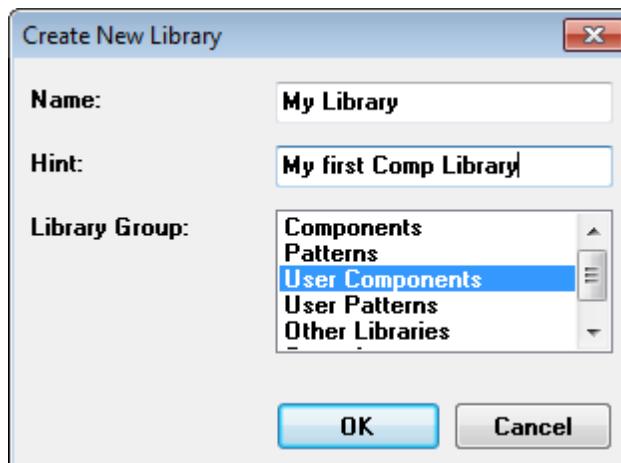


3.2.2 Designing Resistor (component)

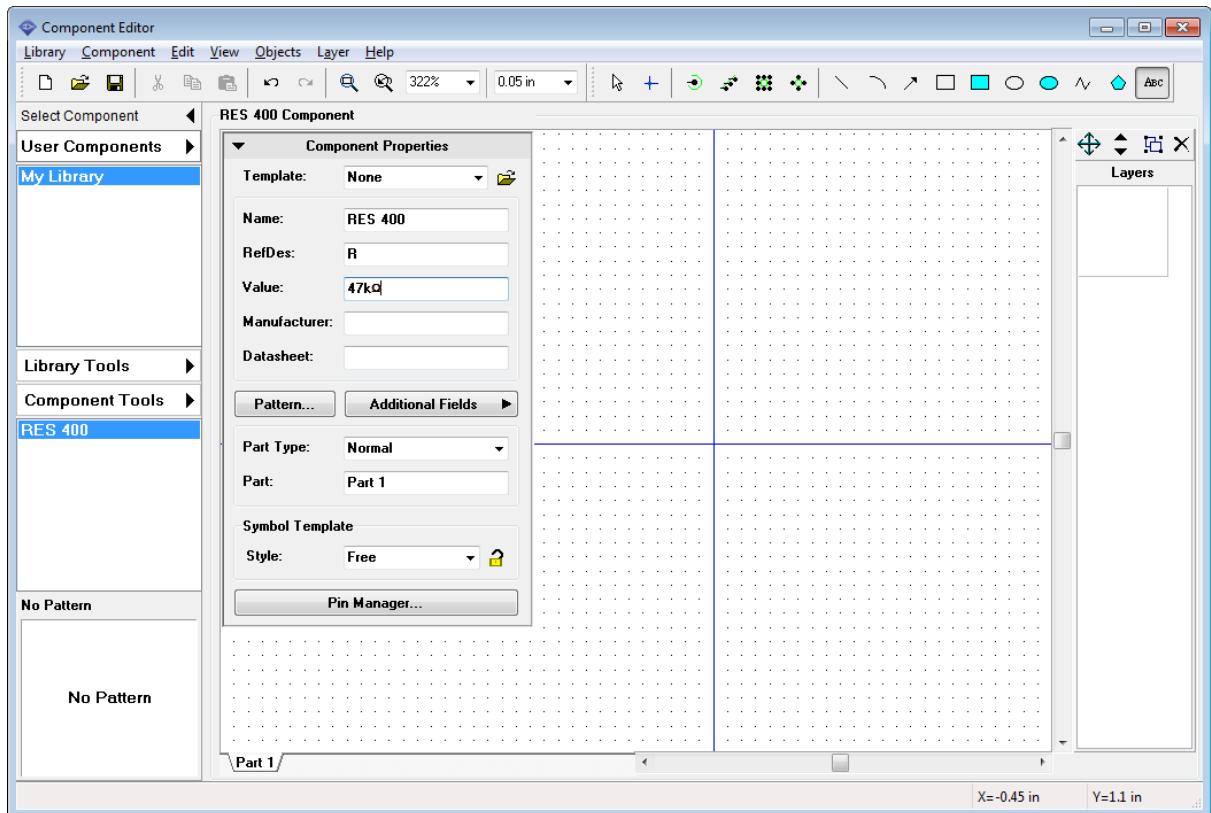
Like in Pattern Editor, we need to create new library first, because DipTrace won't let you to add new components to standard libraries.

Create Library

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

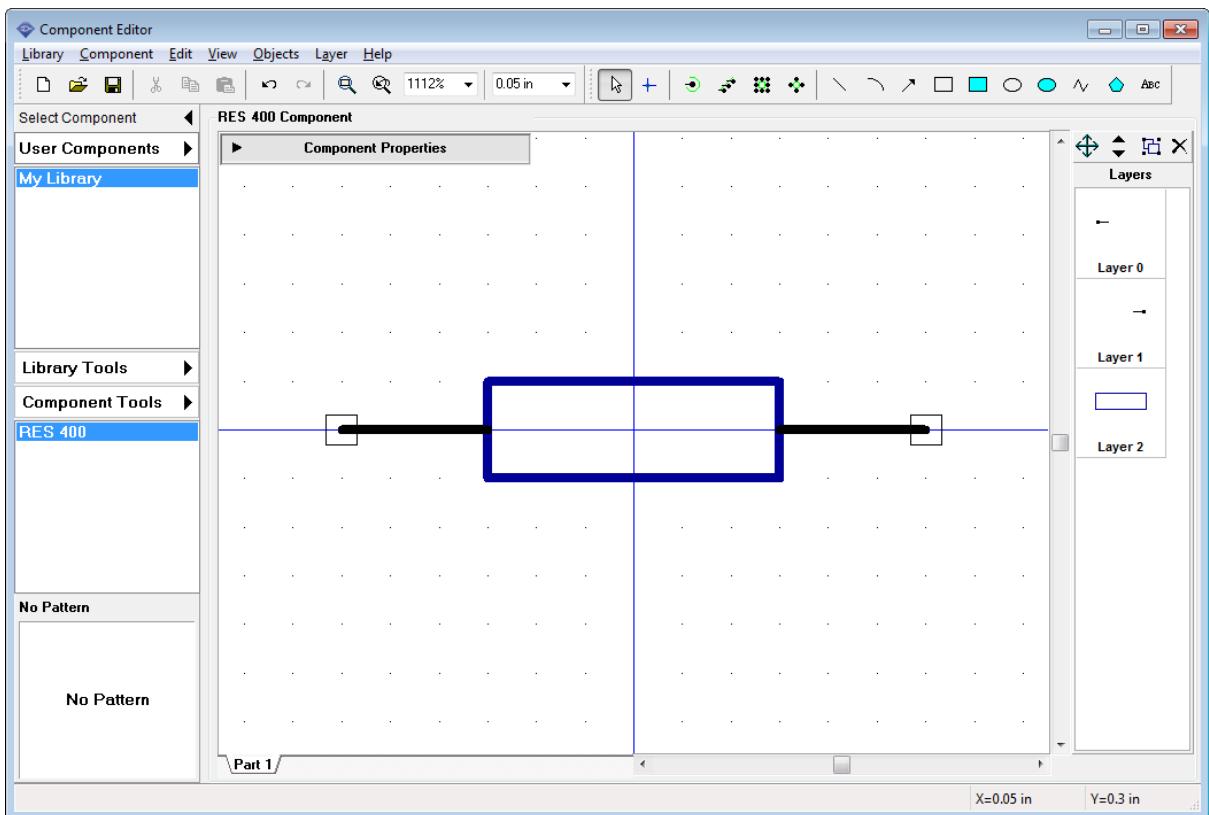


We will design resistor using "Free" style and visual pin placement. Please define component name, RefDes and value "47k ", use corresponding fields on the component properties panel. After specifying these attributes please minimize panel, using arrow in its upper-left corner.



Place Pins

Select "Place Pin" tool on the objects toolbar then move mouse arrow to design area and place two pins by left clicking. Rotate one pin 180 degrees, select it and press "Ctrl+R" twice. Then select rectangle tool and place graphics for the resistor. Pins should be placed by 0.1 grid and rectangle by 0.05 grid (use "Ctrl+", "Ctrl-" to quickly change grid).



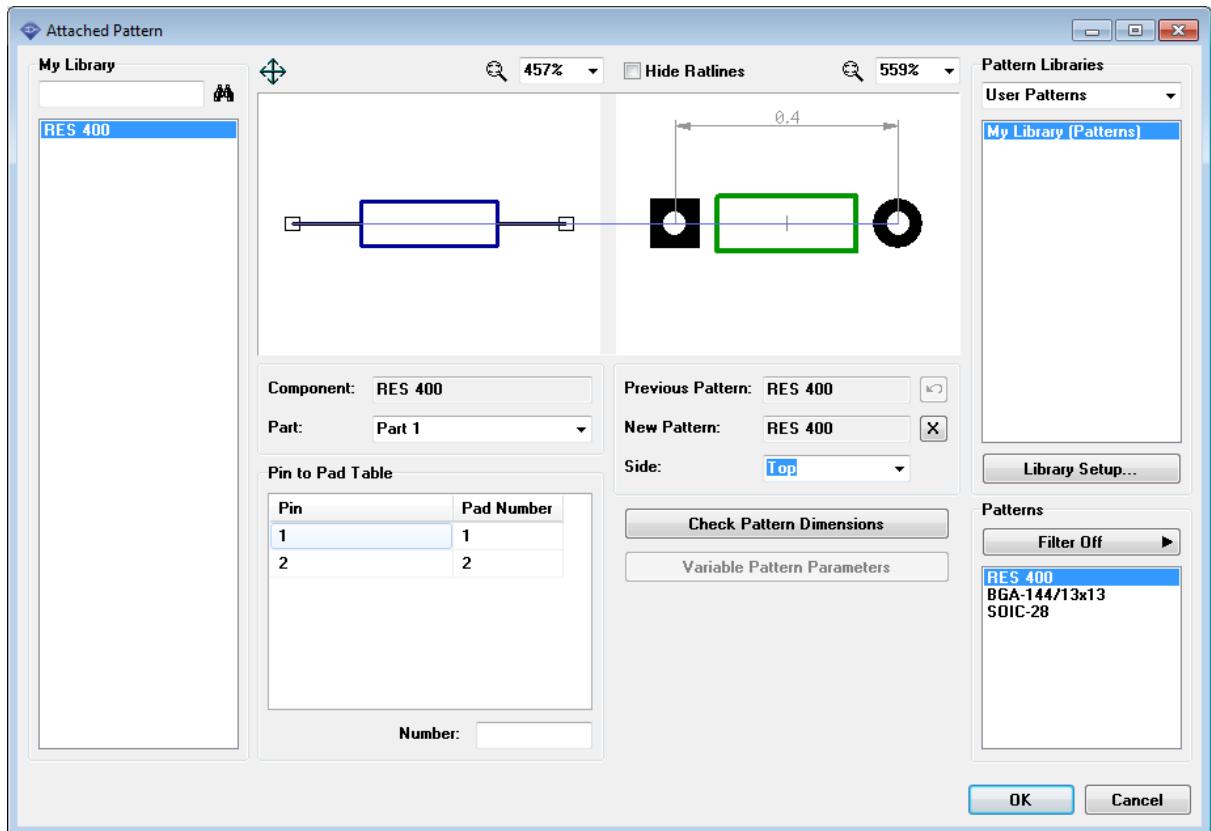
Notice that you can move pin(s) using drag-and-drop method. If you want to move or rotate several pins, please select them first.

Attach Pattern

The symbol of the resistor is ready, but the component is not ready yet. We should attach pattern to this symbol in order to create PCB directly from the Schematic with this resistor. Select "Component \ Attached Pattern" from main menu or press "Pattern" on the Component Properties panel. We need to connect this symbol drawing with pattern drawing created before in Pattern Editor. Select "User Patterns" library group (because, as you remember, we've saved pattern library there (see "[Designing Resistor](#)" [91] 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 DipTrace automatically creates pin to pad connections. You can review and reassign them if necessary. 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 corresponding pad and left click it to connect them. To delete connection, right click on the pin or pad. 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 (related pads) can be changed not only in Attach Pattern dialog box, but with Pin Manager (select "Component \ Pin Manager" from main menu) or in 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 current library can be seen in the left part of the dialog box, this allows 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 Attached Pattern dialog box. Resistor is ready and contains both schematic part and PCB pattern with 3D Model.

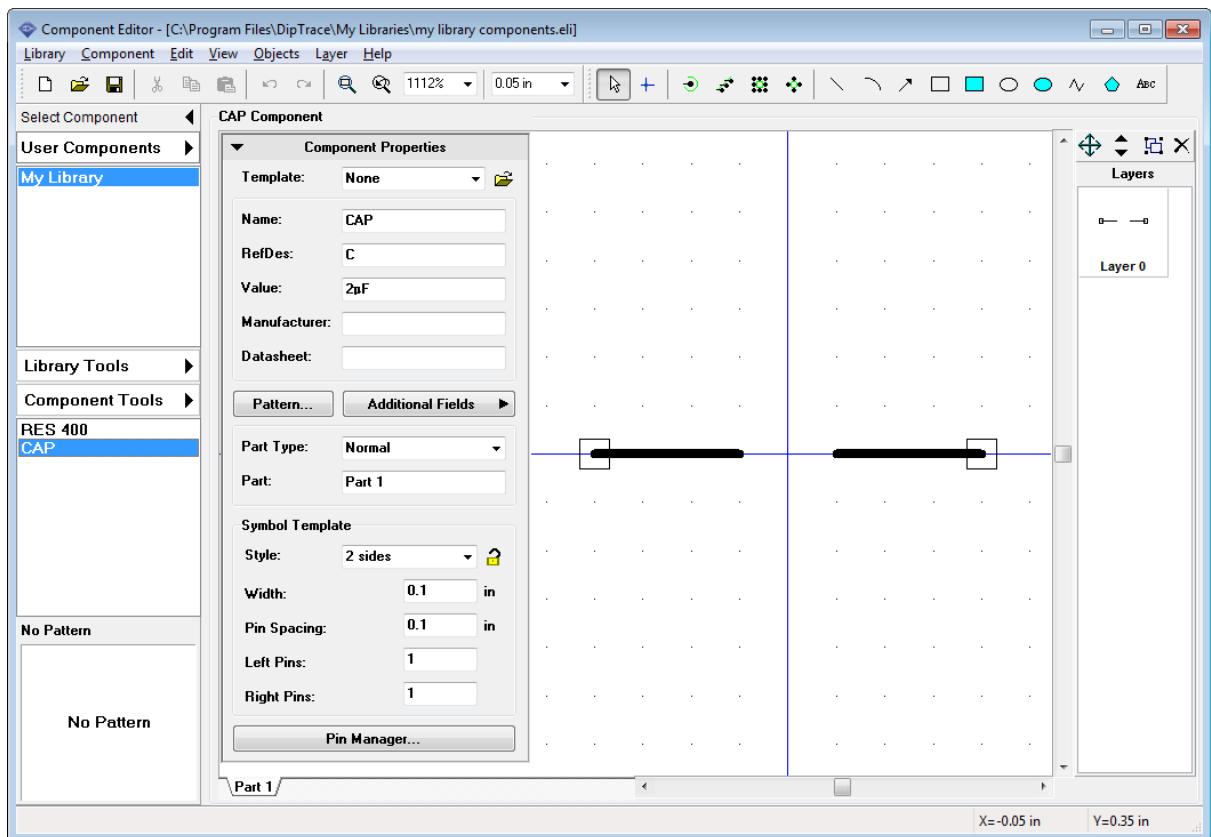
Save component library. Press "Save" button on the standard toolbar, select location (not in the folder with standard libraries), enter file name and press "Save".

Notice that this is the file with ".el1" extension, this means that this is component library file.

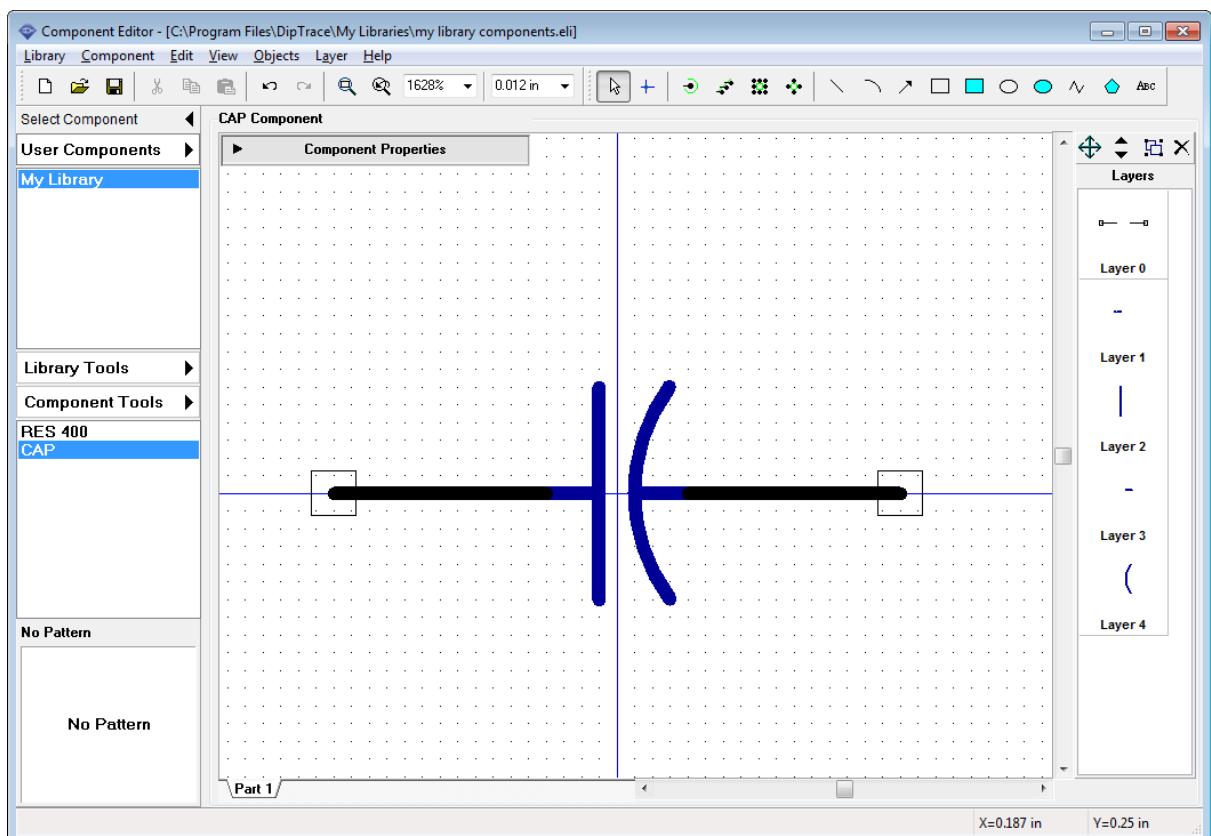
3.2.3 Designing Capacitor

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

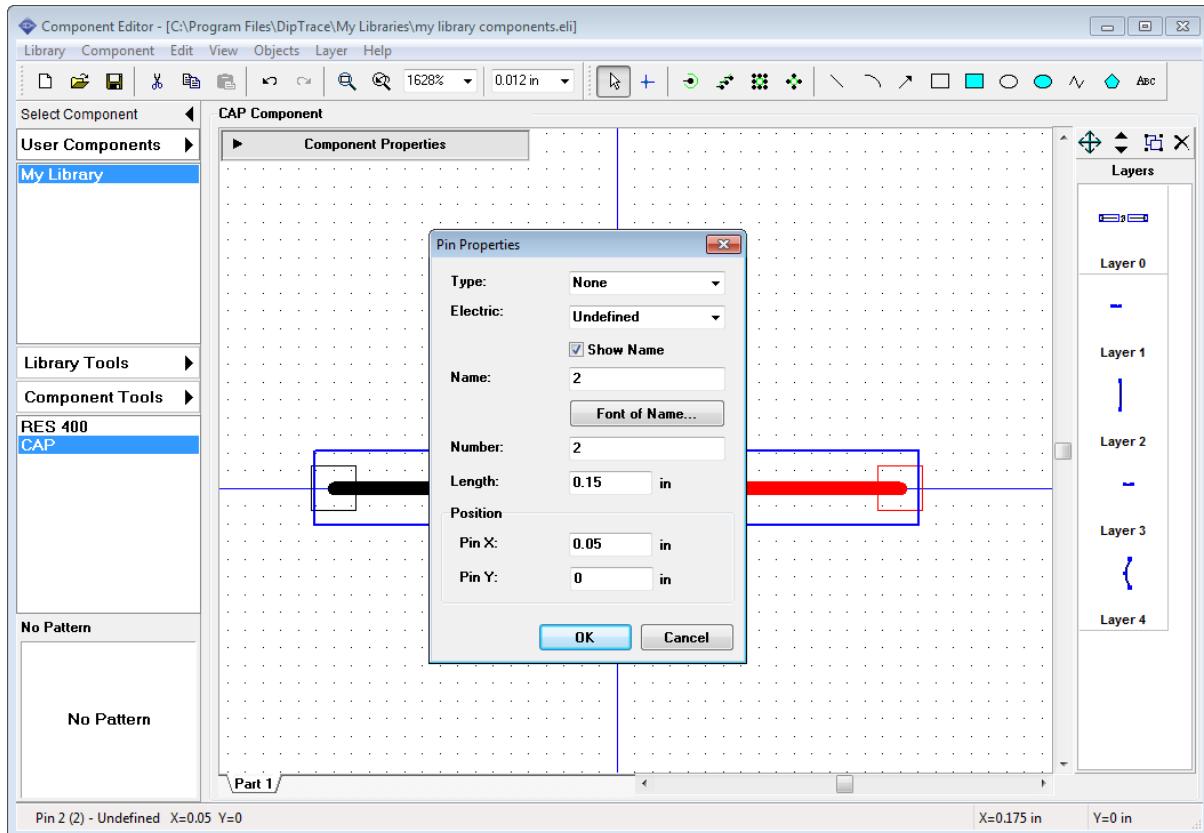
We will design capacitor using "2 sides" component style (Style box on the component properties panel); type in component name "CAP", RefDes - C, Value - $2\mu\text{F}$. Change component width to "0.1", left and right pins to "1".



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



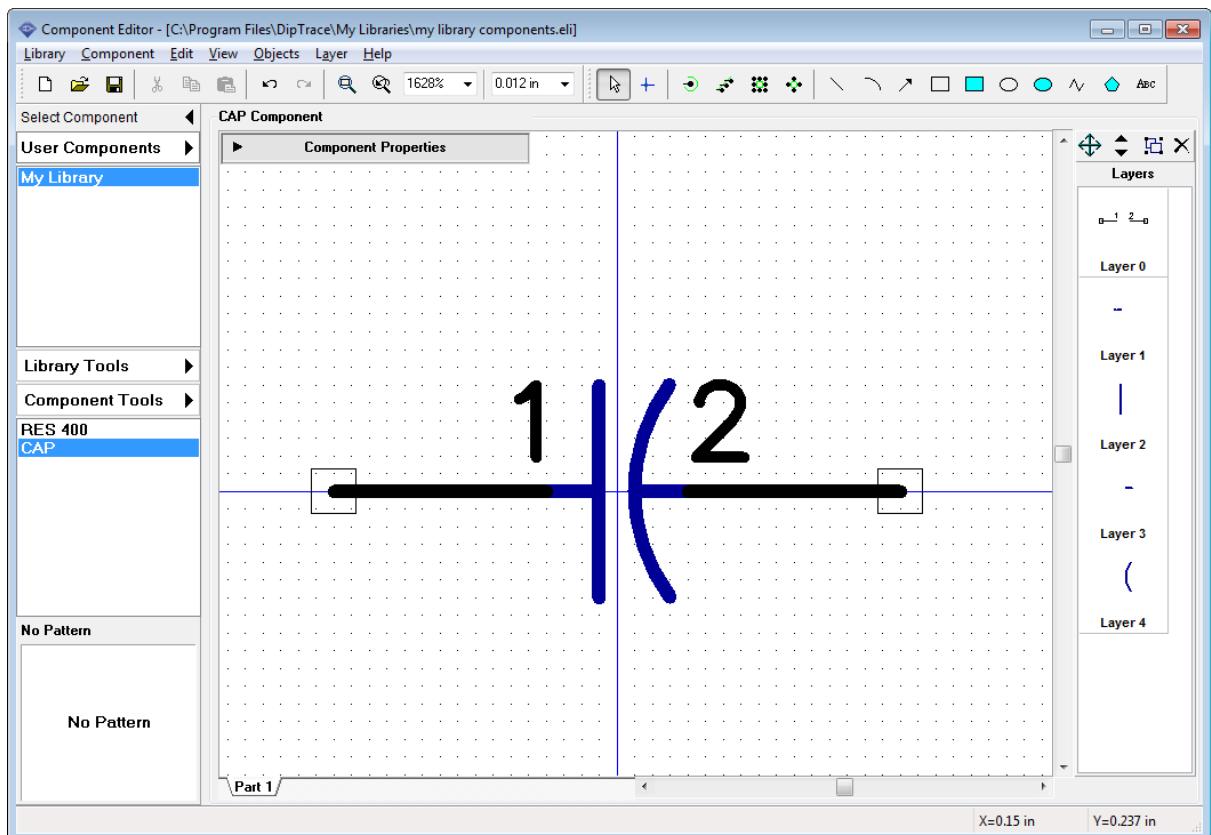
Show pin names for the component symbol, select pins (or select all using "Ctrl+A"), right click on one of them and choose "Pin Properties" from submenu. In the pin properties dialog box check "Show Name" and press "OK".



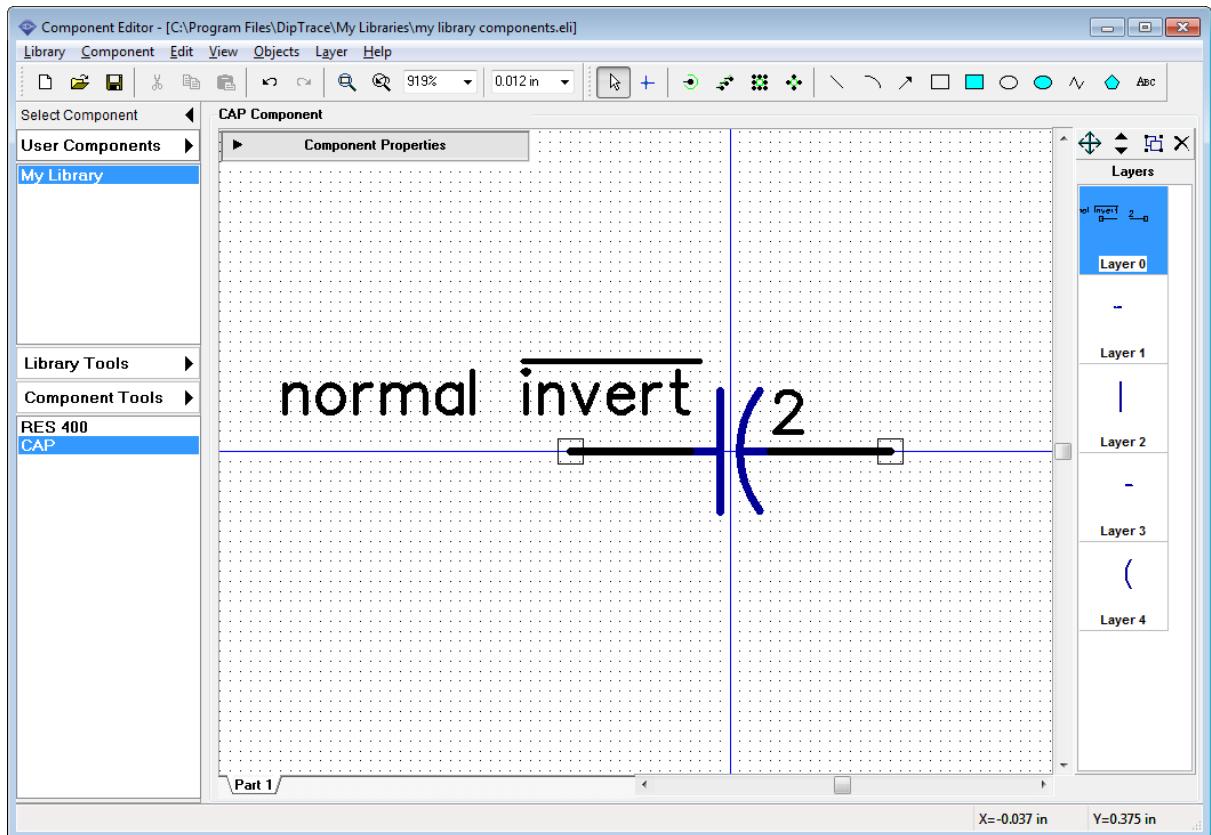
Notice that all new pins have "Undefined" electric type. It can be changed with pin properties dialog box or pin manager (see below). Electric type is used for ERC verification. "Type" property is used for pin graphics, you can try different types to see what it draws (or see Component Editor Help).

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 hover over pin names and drag them to new positions then right click to return to default mode.

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



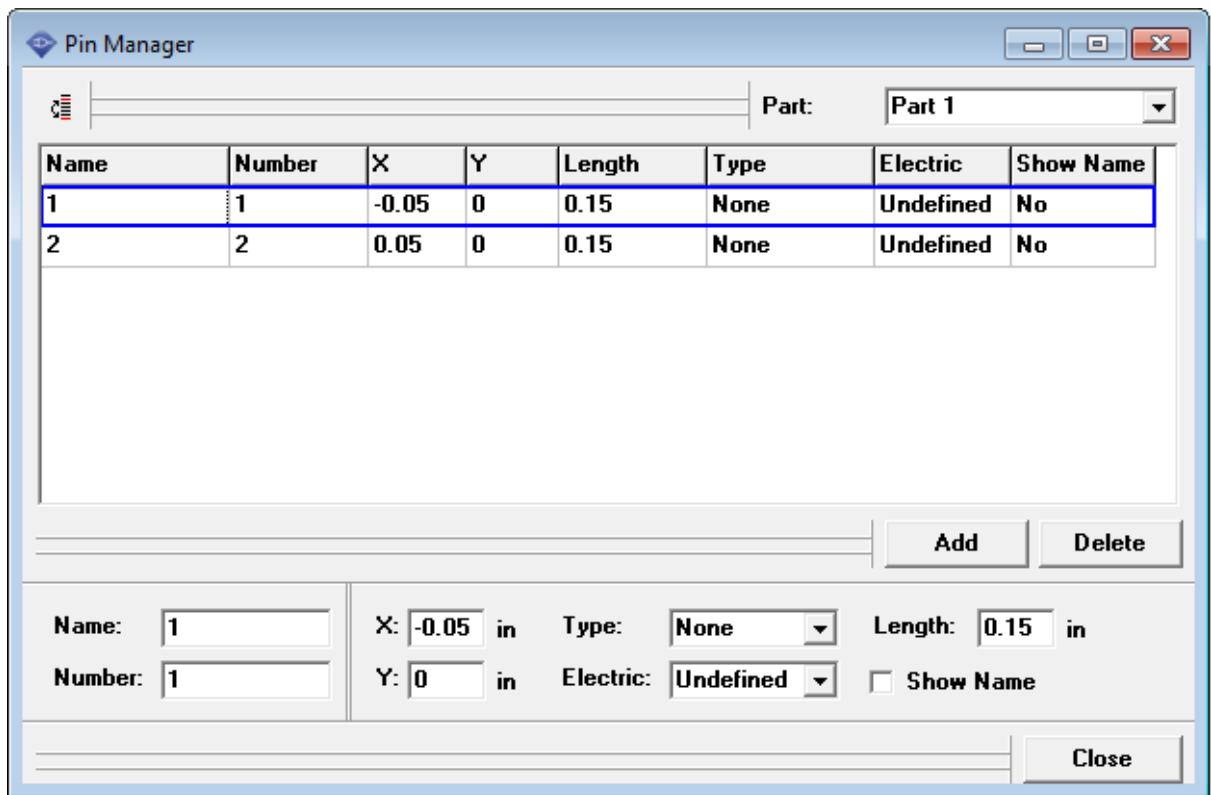
You can show inversion line in the pin name, move mouse arrow over the pin, right click and select first (top) item from the submenu, type in "normal ~invert" text and press "OK", then move pin name using move tool ("F10"). "~" symbol in the pin name starts and ends inversion, so using it you can define the inversion for separate parts (signals) in the pin's name.



You probably don't need to see pin names for simple components like capacitor, pin numbers are more likely to be displayed.

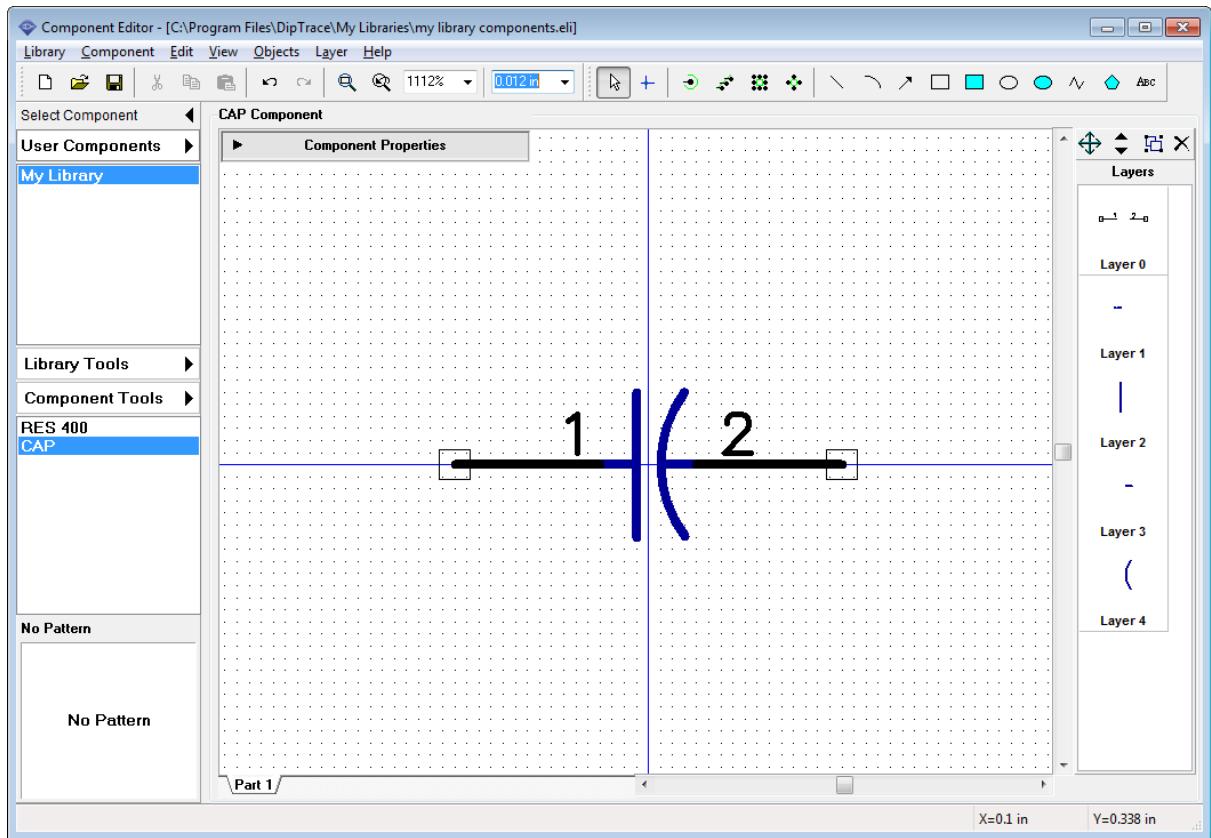
Select "Component \ Pin Manager" from main menu to open Pin Manager dialog box, select pin "2" in the table and change name to "2" (pin #1 should have "1" name), then hide pin names for both pins: select pin row in the table and uncheck "Show Name" box at the bottom of the dialog box. Close Pin Manager.

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



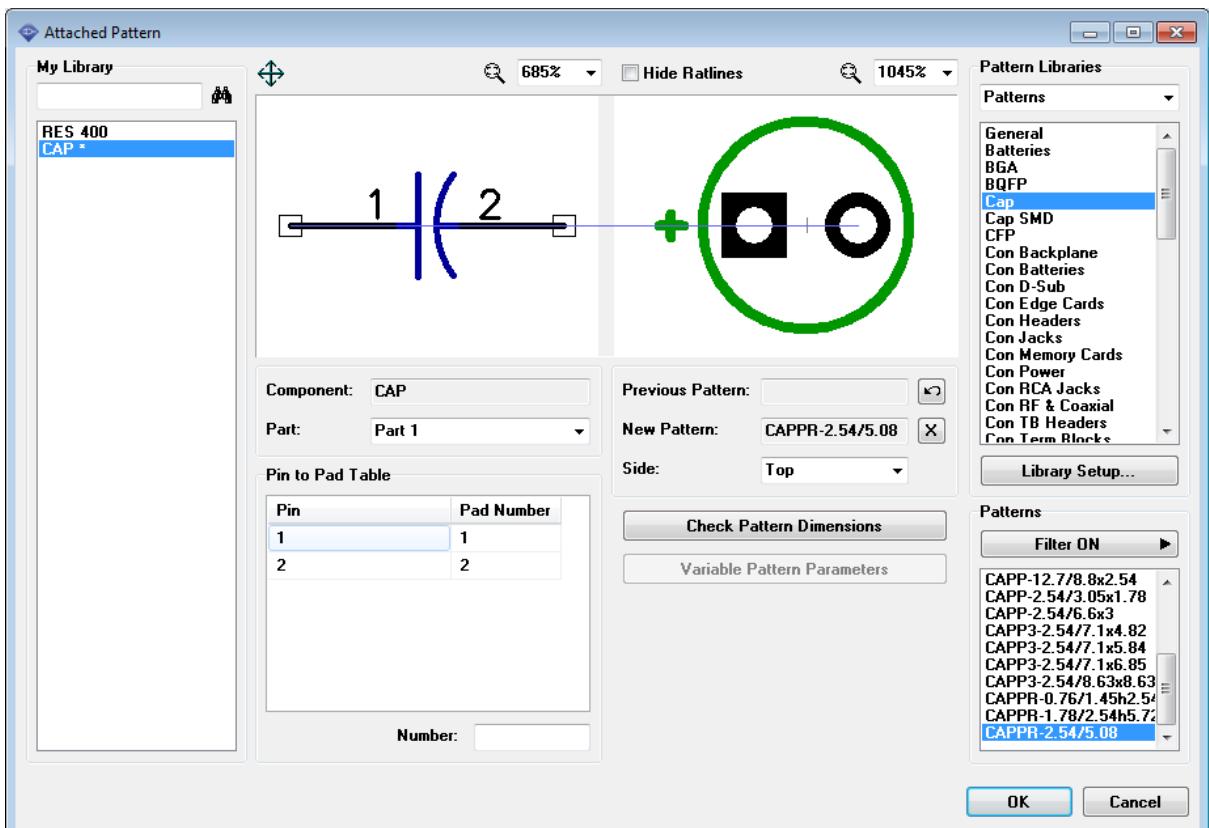
In Component Editor you can set individual show/hide pin numbers settings for current component ("Component \ Pin Numbers" from main menu) and common program settings (the same as in Schematic) in "View \ Pin Numbers \ Show" from main menu.

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 Component Properties panel and press "Pattern". We did not create a pattern drawing for this component, but it is available in standard DipTrace libraries. Select "Patterns" library group then select "CAP" library below and "CAPP-R-2.54/5.08" from the list of patterns. You can use search filters.

Pin to pad connections assigned automatically are good.



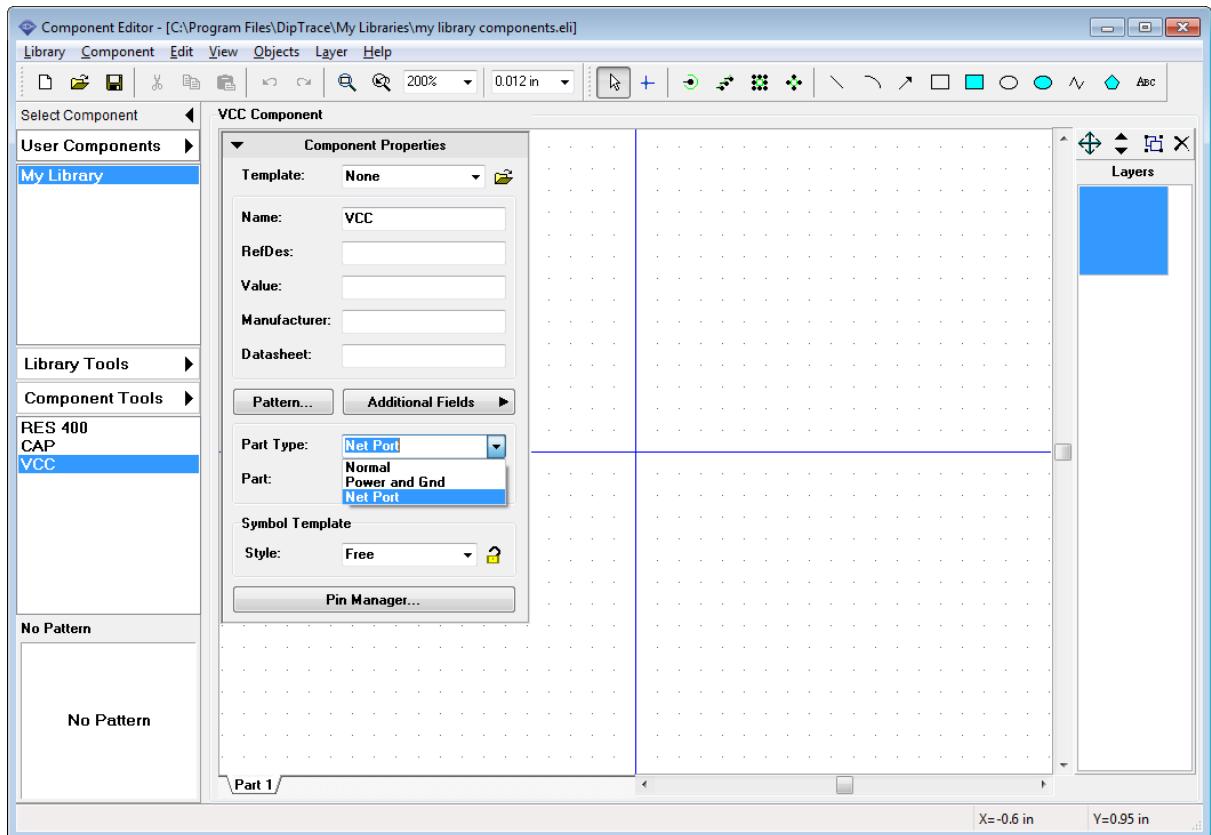
Press "OK". The capacitor is ready. Save changes.

3.2.4 Designing VCC and GND symbols

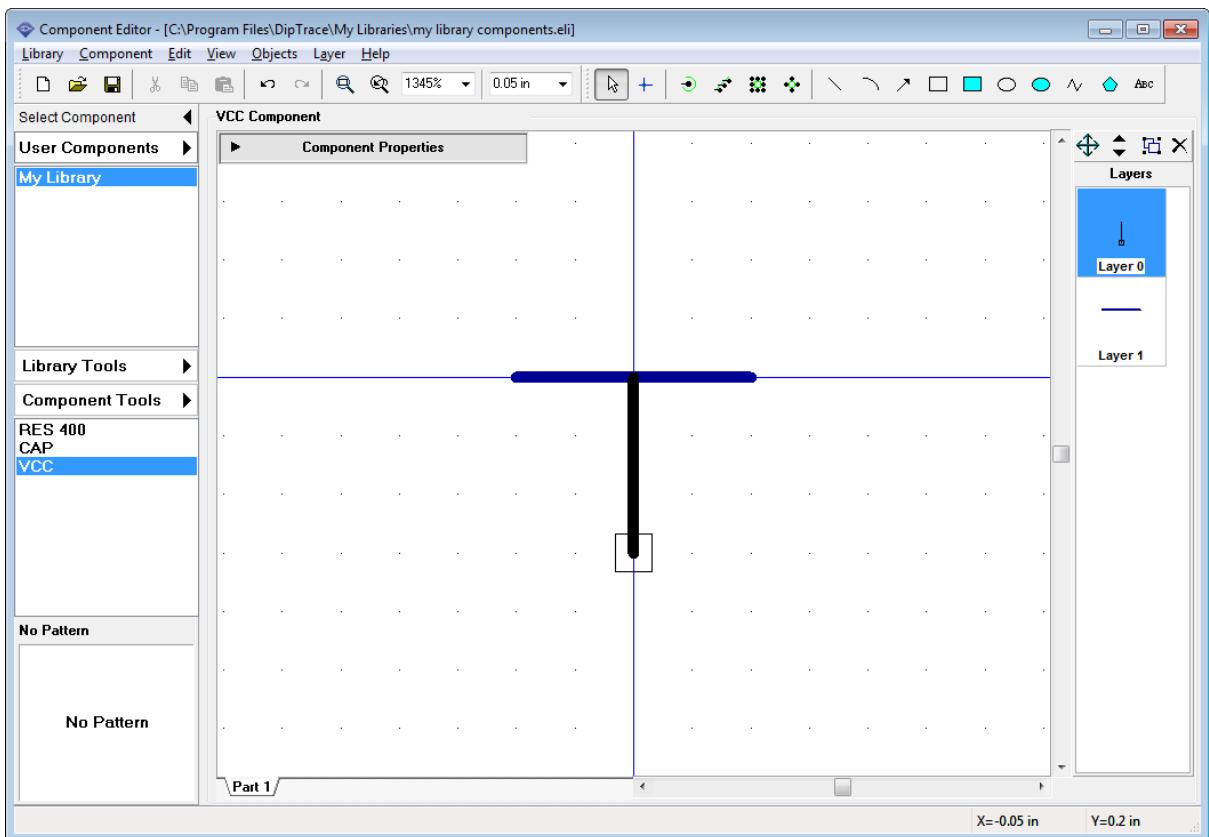
In this lesson we will practice in creating **net ports** by designing VCC and GND symbols.

VCC

Press "Component Tools \ Add New Component to "My Library". 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 component properties panel then select "Pin" tool on the objects toolbar and place single pin, rotate it vertically (select it and press "R" hot key). Select line tool on the drawing panel and place silk line of the symbol like on the picture below, use .05 inch grid.



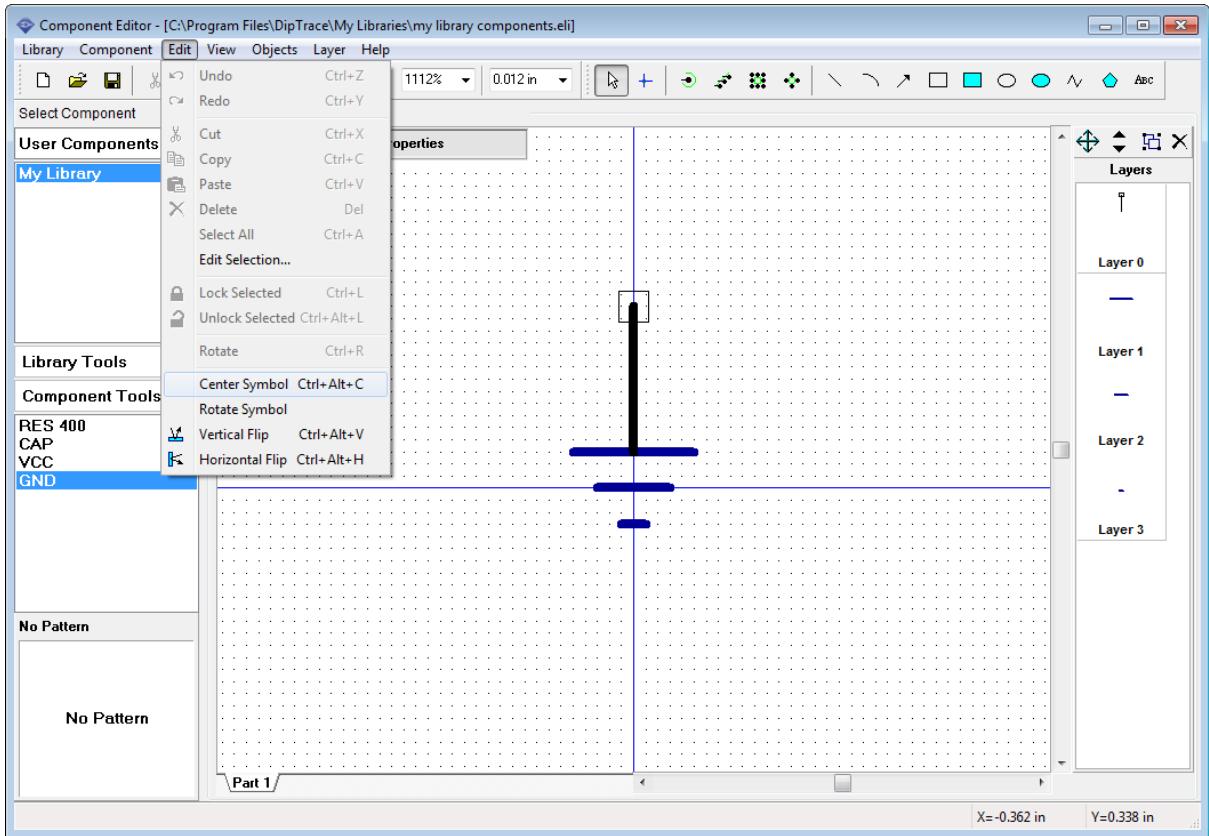
Hide pin number if it is visible, select "Component \ Pin Numbers \ Hide" from main menu. Pin number for single-pin component is non-sense.

VCC symbol is ready.

GND

Now please add another component ("Ctrl+Ins" hot keys) and create GND symbol in the same way like 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 part origin hidden by default in Schematic. Use .012 inch grid to draw 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.

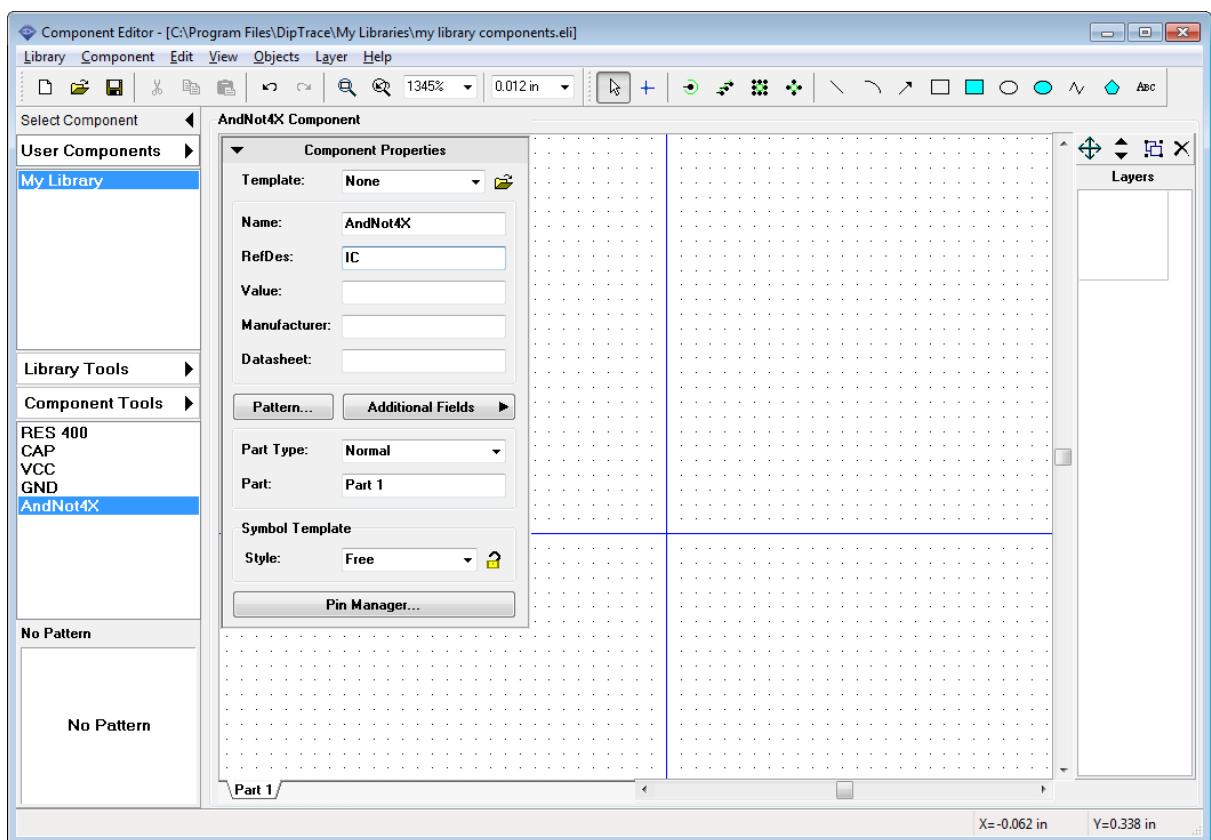
Save library file.

3.2.5 Designing a multi-part component

Creating symbol

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

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



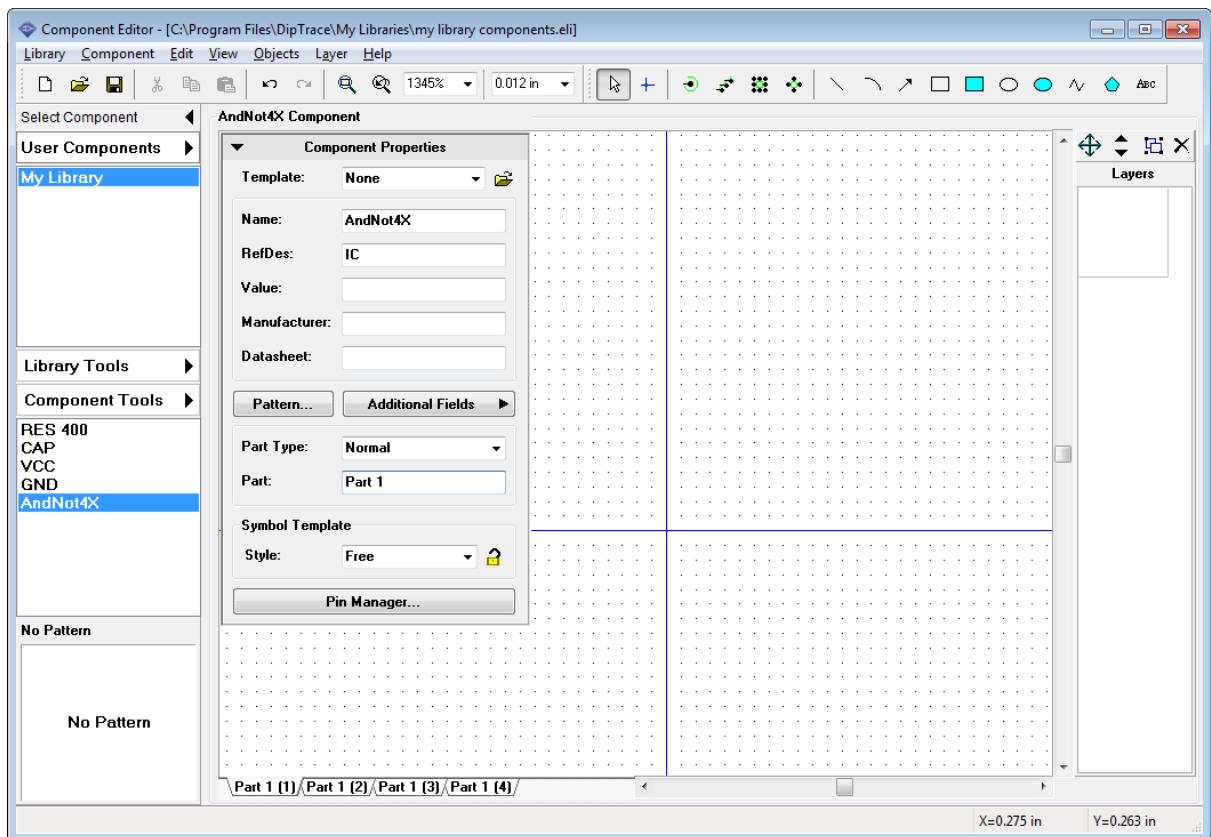
Creating Parts

DipTrace allows to create separate parts and part groups (similar parts) in the component. All parts in the part group have same pins, drawing, except pin numbers (i.e. related pads). Component parts can be Normal, Power and Ground or Net Ports. Power parts and power nets can be hidden in schematic capture, the component may include only one power part.

We will design component with 4 similar AndNot parts and one power part. Select "Component \ Create Similar Parts" from main menu, type "4" in the dialog box and press "OK" to apply.

Notice that similar parts are created based on currently selected part.

Tabs with part names has appeared in the bottom-left of the design area.

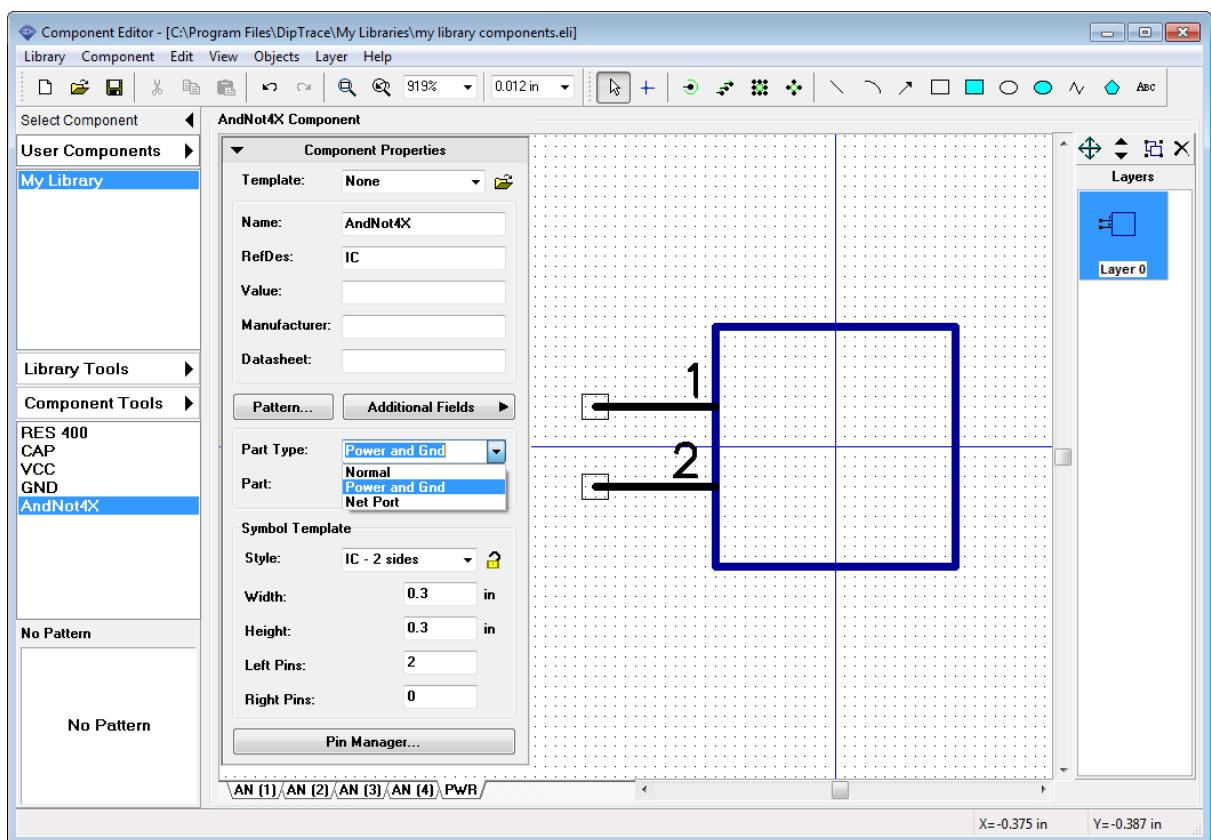


Now you can see the following 4 parts: Part 1 (1), Part 1 (2), Part 1 (3) and Part 1 (4). All similar parts have the same part name. Change it quickly (in "Part" field on the component properties panel) for example to "AN".

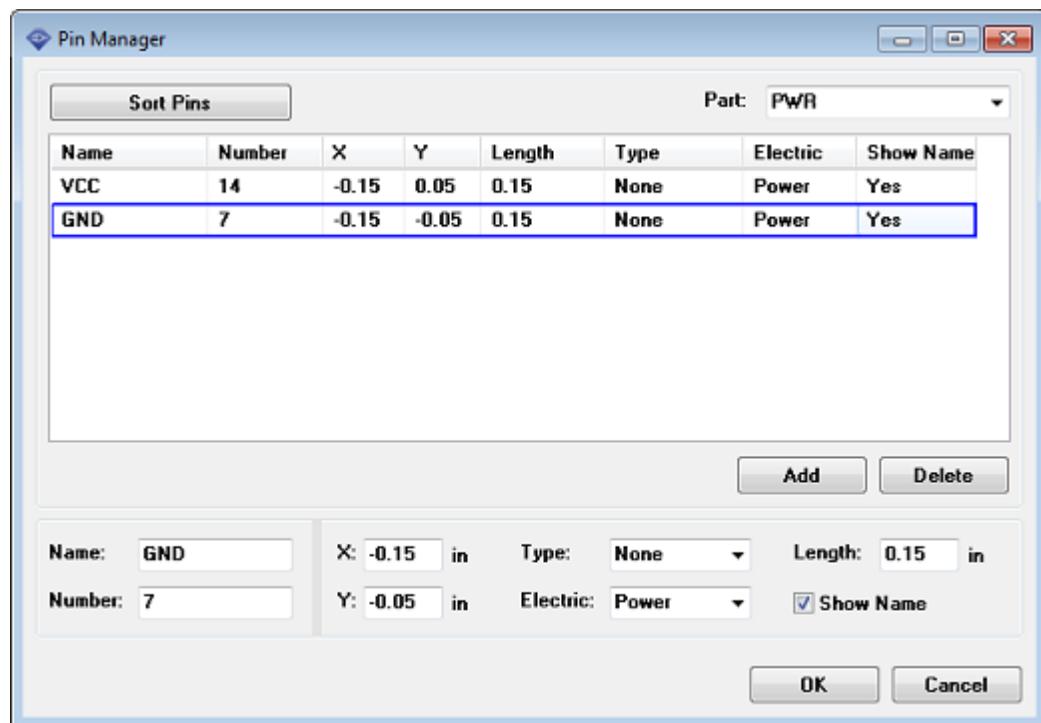
The power part for the component is missing. Select "Component \ Add New Part" from main menu to add single part to component, select new part tab in the bottom-left and rename it to "PWR".

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

Start designing 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 main menu), if they are not visible.



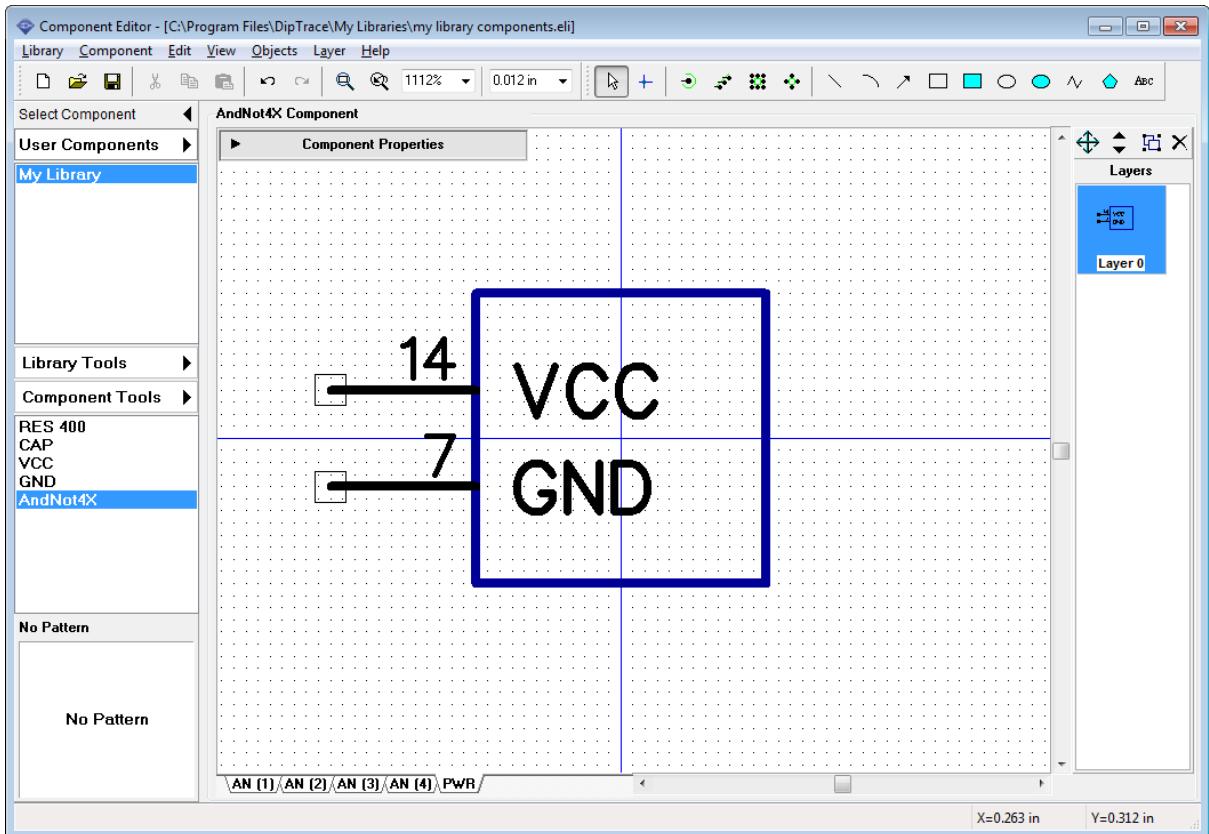
Select "Component \ Pin Manager" from main menu and change pin names to "VCC" and "GND", pin numbers to "14" and "7", electric type to "Power", "Show Name" box should be checked for both pins. Notice that you can change "Type", "Show Name" and "Length" parameters for multiple pins.



Pin Manager dialog box window is resizable and you can change width of rows. These settings are

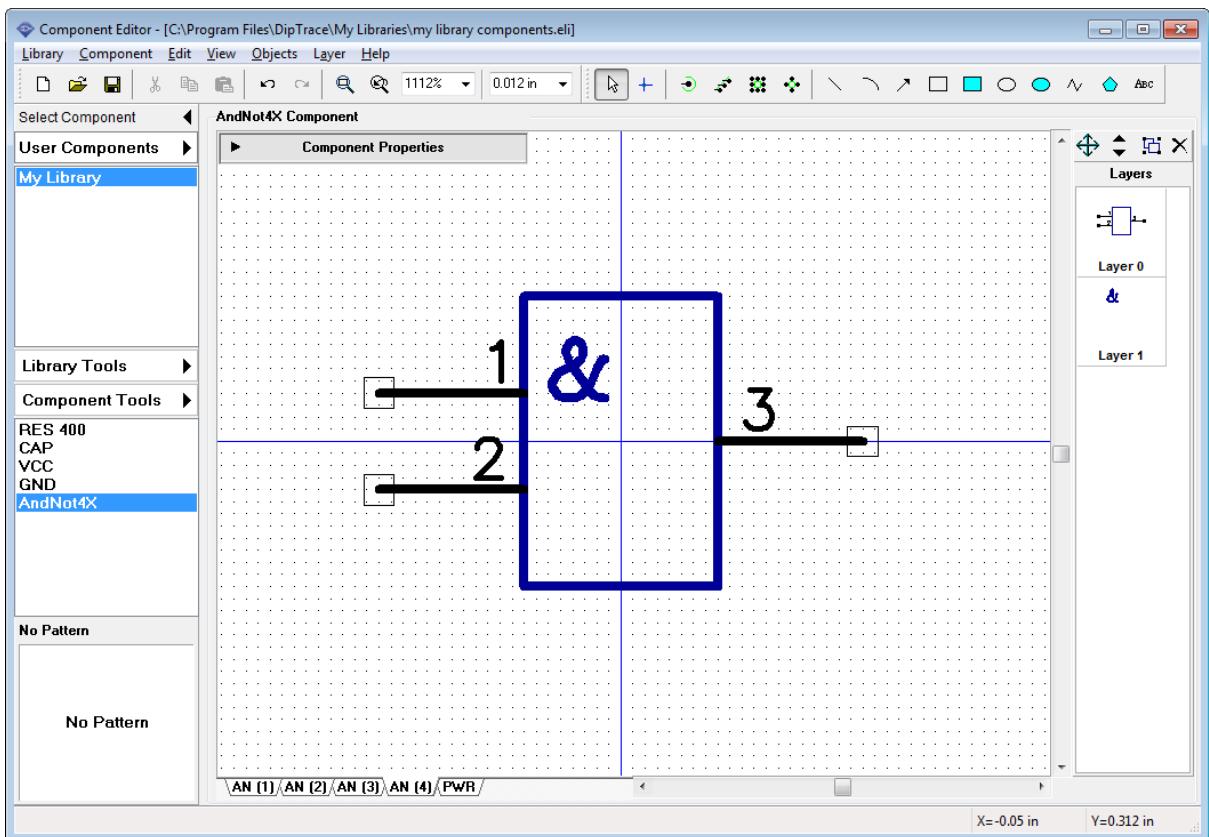
saved when you close program.

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



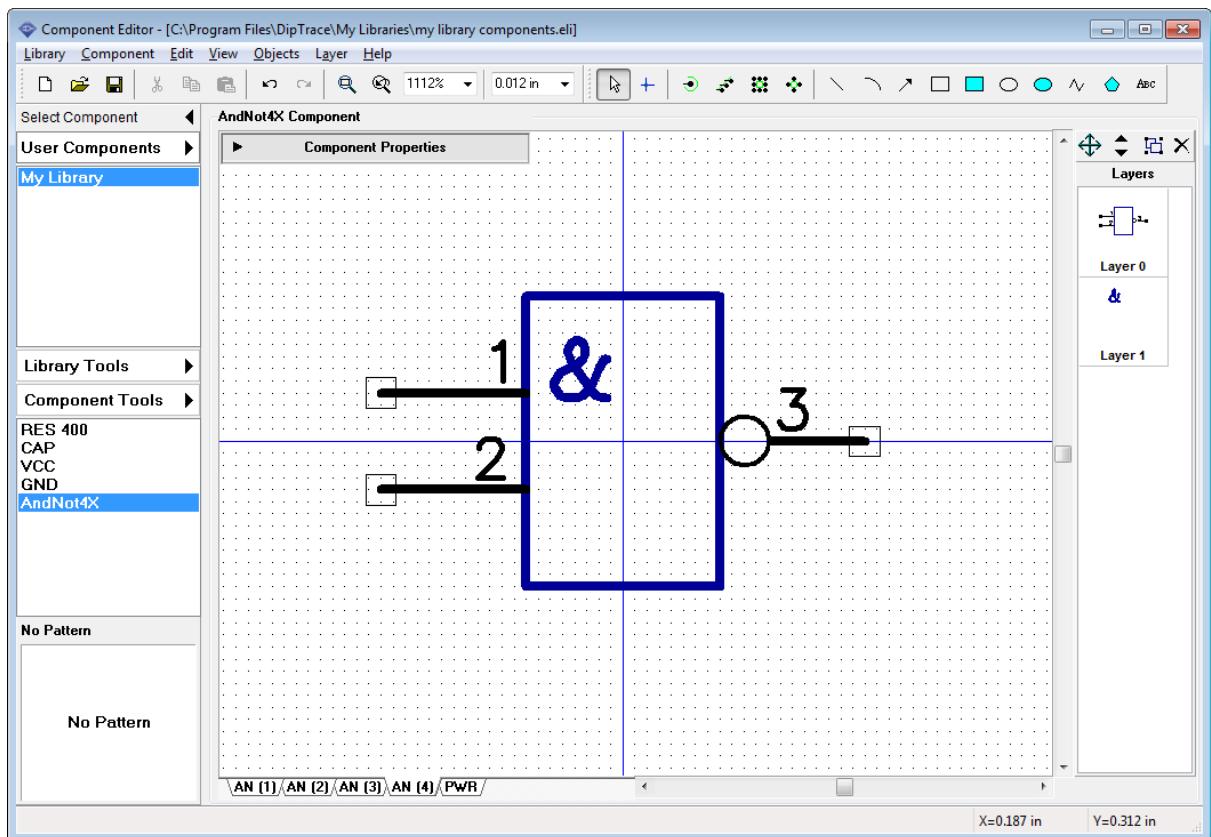
Design other parts of multi-part component: select one of the AN parts and define the following parameters on the component properties panel: "Style – IC-2 sides", "Width: 0.2 in", "Height: 0.25 in", "Left Pins: 2", "Right pins: 1". Then minimize component properties panel.

Select text tool on the drawing toolbar ("ABC" 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 "And - Not" (Not And) parts should be inverted or "Dot" type. Right click on the third pin, select "Pin Properties" from the submenu, in the pop-up dialog box specify "Type: Dot". Click "OK" to apply changes and close dialog box.

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



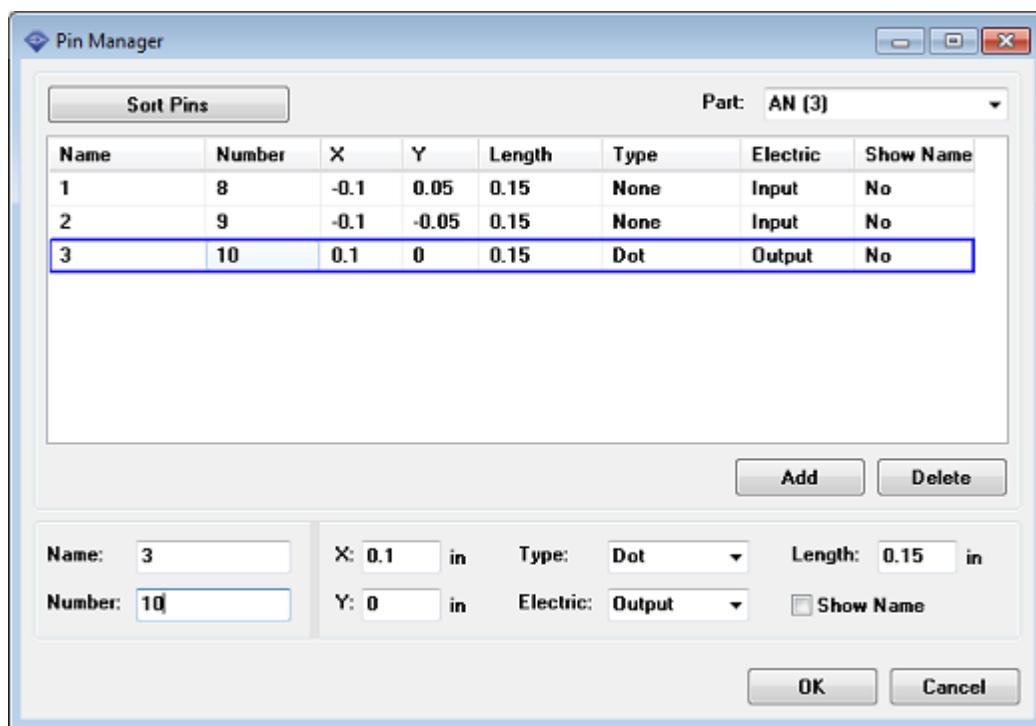
Select AN (3) or AN (4) part just to make sure that segments are the same. All parts in the group are absolutely identical, just like pin numbers, which should be different.

Pin Manager

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

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

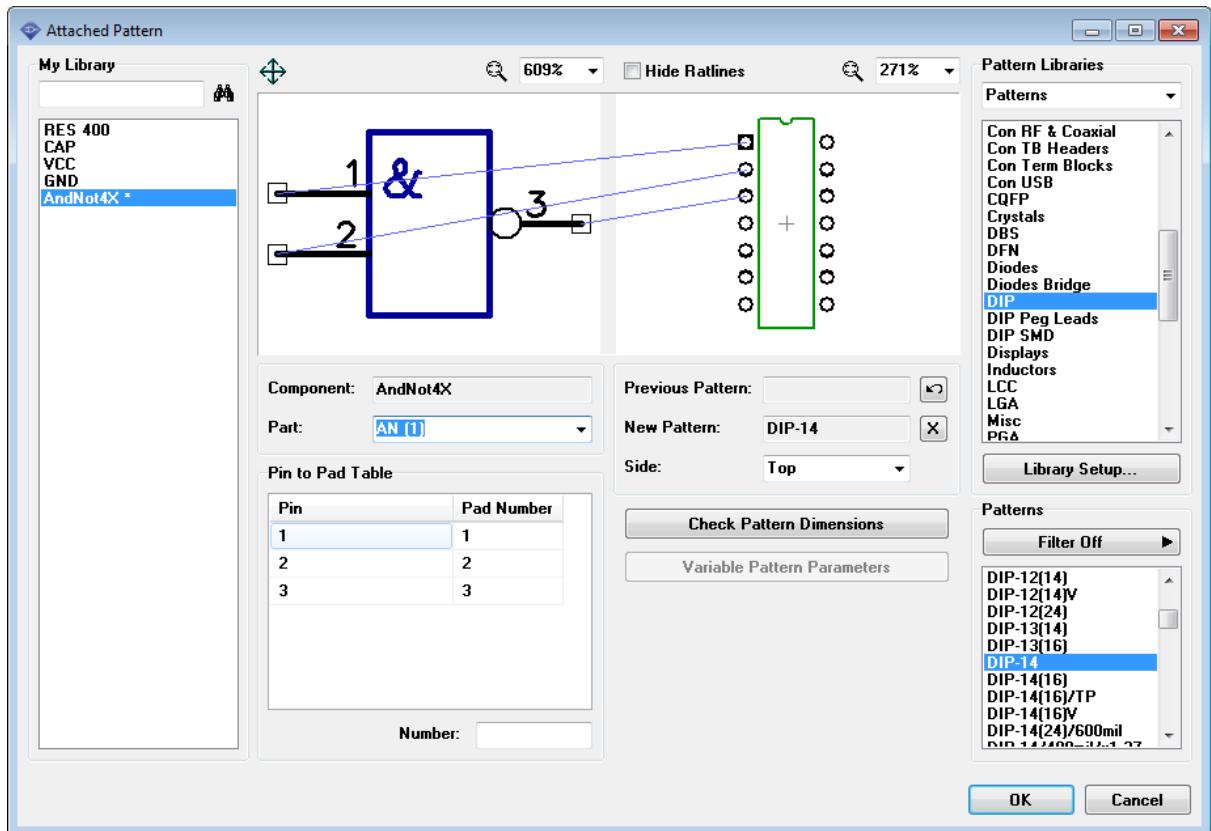
Please, set correct "Electric" type for one of the parts (others will be changed automatically), 2 input pins and one output. Click "OK".



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

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 Pin Manager (this is why pin numbers array was not straight).

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



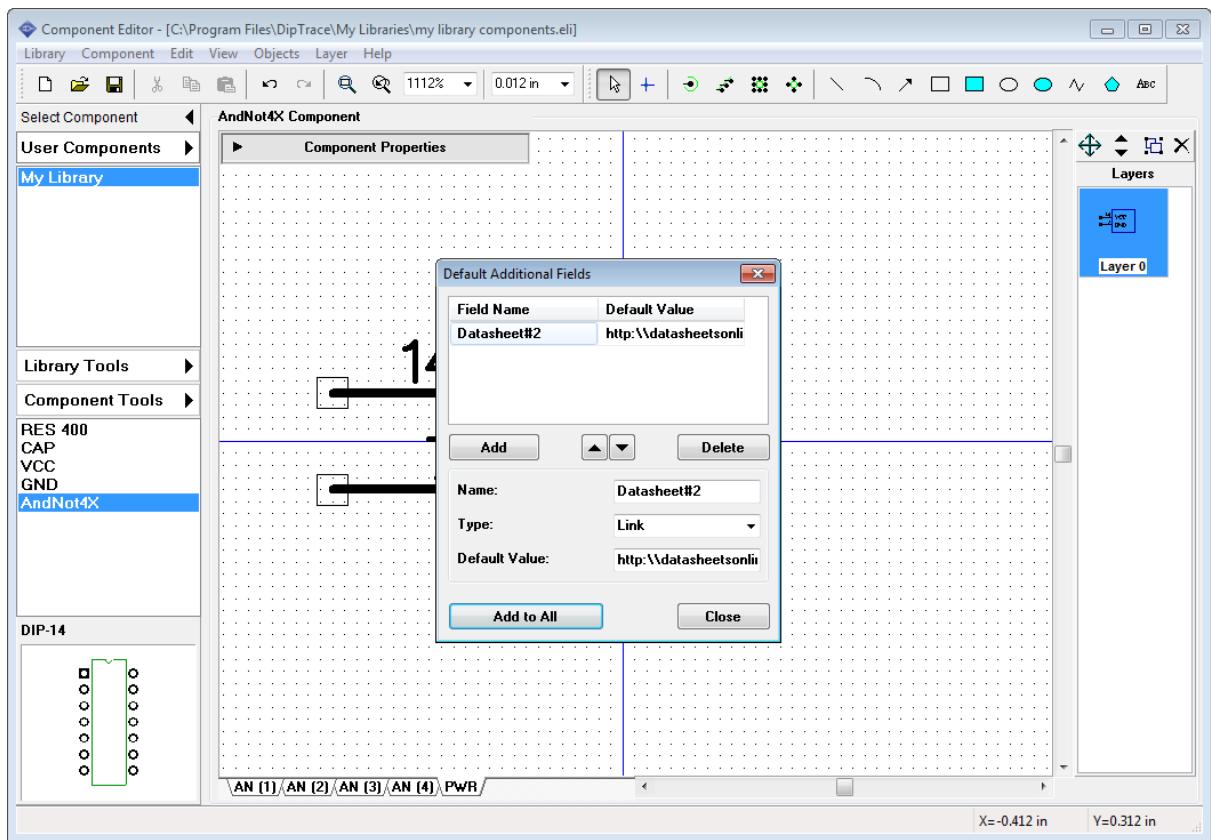
Multi-part component is ready. Save Library file.

3.2.6 Using additional fields

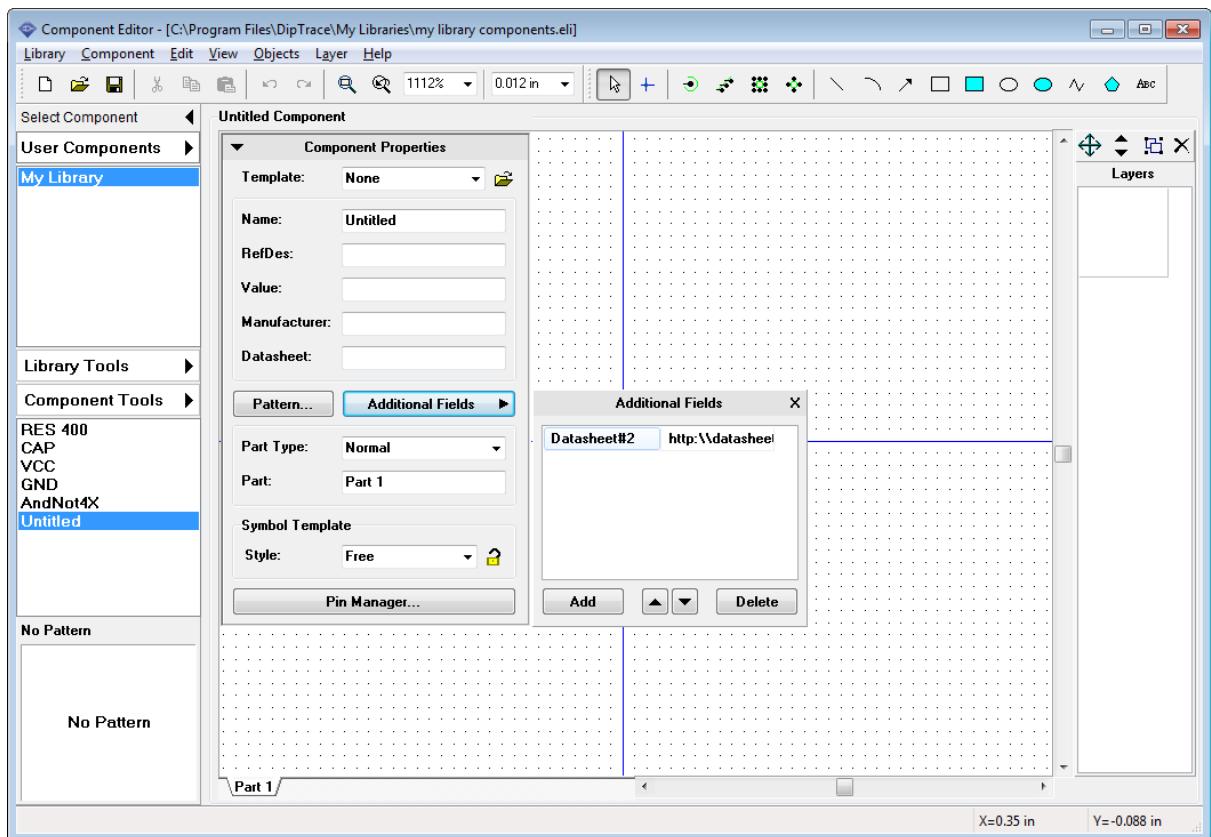
Name, RefDes, Value, Manufacturer and Datasheet are default component fields in DipTrace, usually this is all users need, but sometimes additional description or other information needs to be added. In this case you can use additional fields.

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

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 web-link address and click "Add" button. Press "Add to all".



From now on all new components will have these additional field. Close this dialog box. Select "Component \ Add New Component to "My Library" or press "Ctrl+Ins" to add a new component then select it, maximize Component Properties panel using arrow in its left side and click "Additional fields" button to see the list of additional fields for new component.



Now you can edit, add or delete additional fields to the component, however, we will not do this now. Select component on the Library Manager, 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.

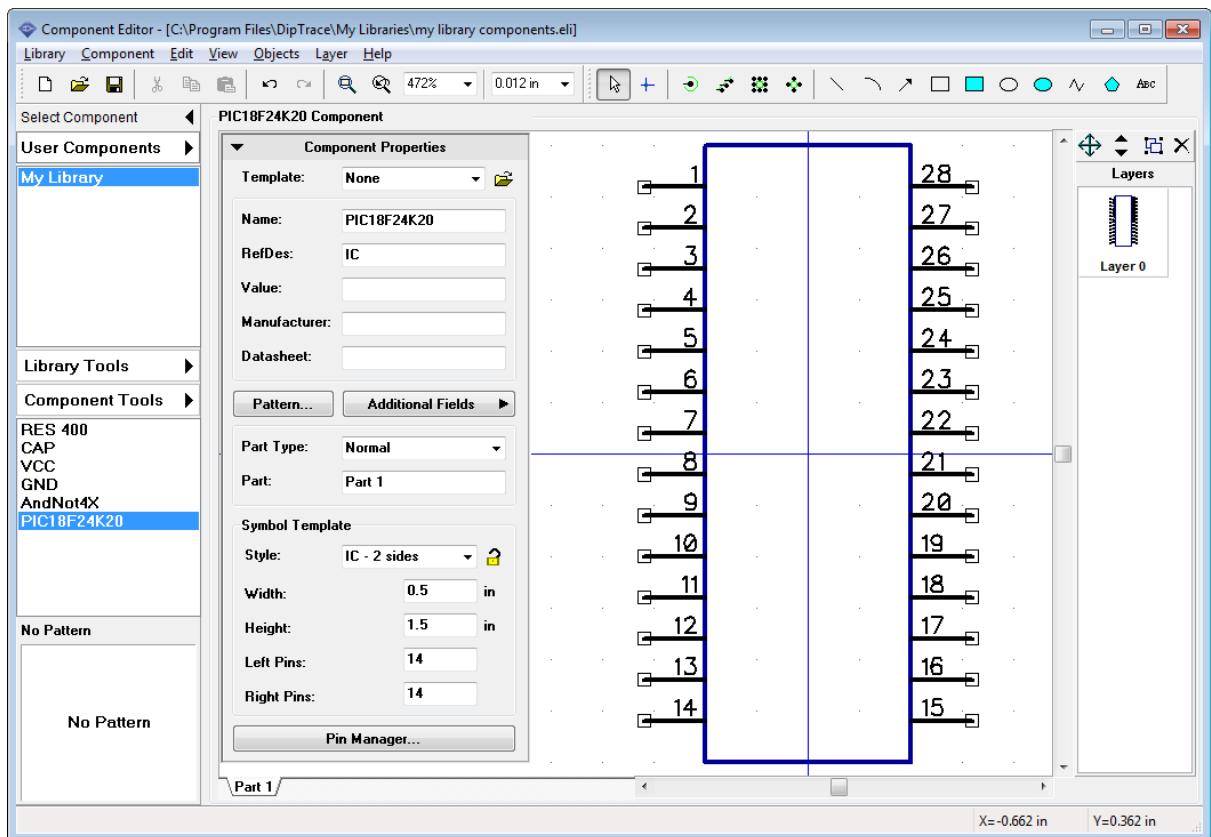
Now select capacitor that we've created before. Notice that it has no additional fields, because we've created it before changes in "Default Additional Field" settings. So we will add several new fields manually. Just press "Additional Fields" button on the component properties panel and practice a bit.

3.2.7 Designing PIC18F24K20

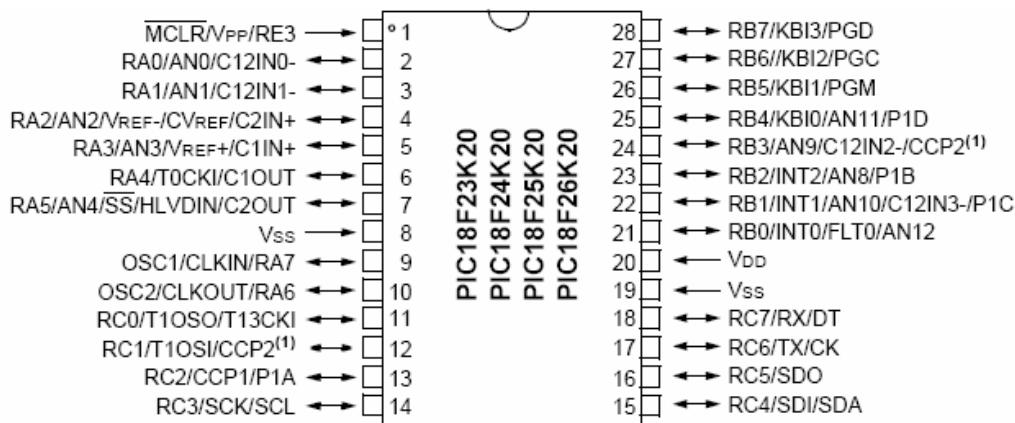
Now we will create "PIC18F24K20" component according to datasheet and attach "SOIC-28" pattern to get a ready real-life component.

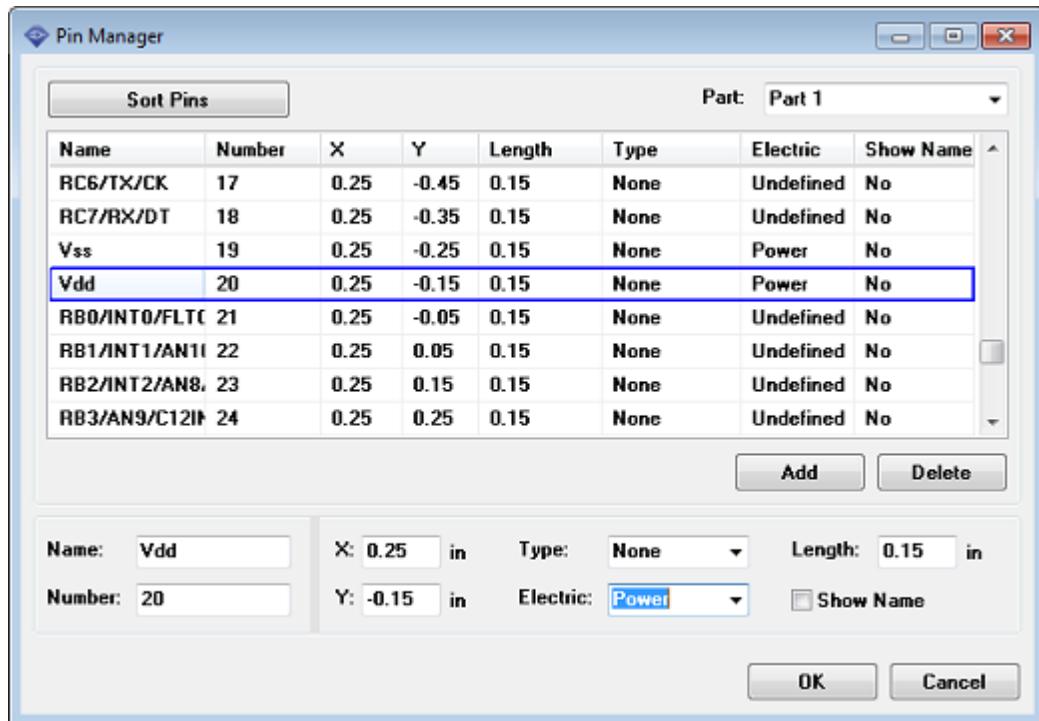
Go to [Microchip web-site](#) and search "PIC18F24K20", then select "Datasheets" on the left. Or use direct [link](#) (however we don't guarantee, that it works at the moment you read this tutorial). Go to "Pin Diagrams", the first diagram is what we need.

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

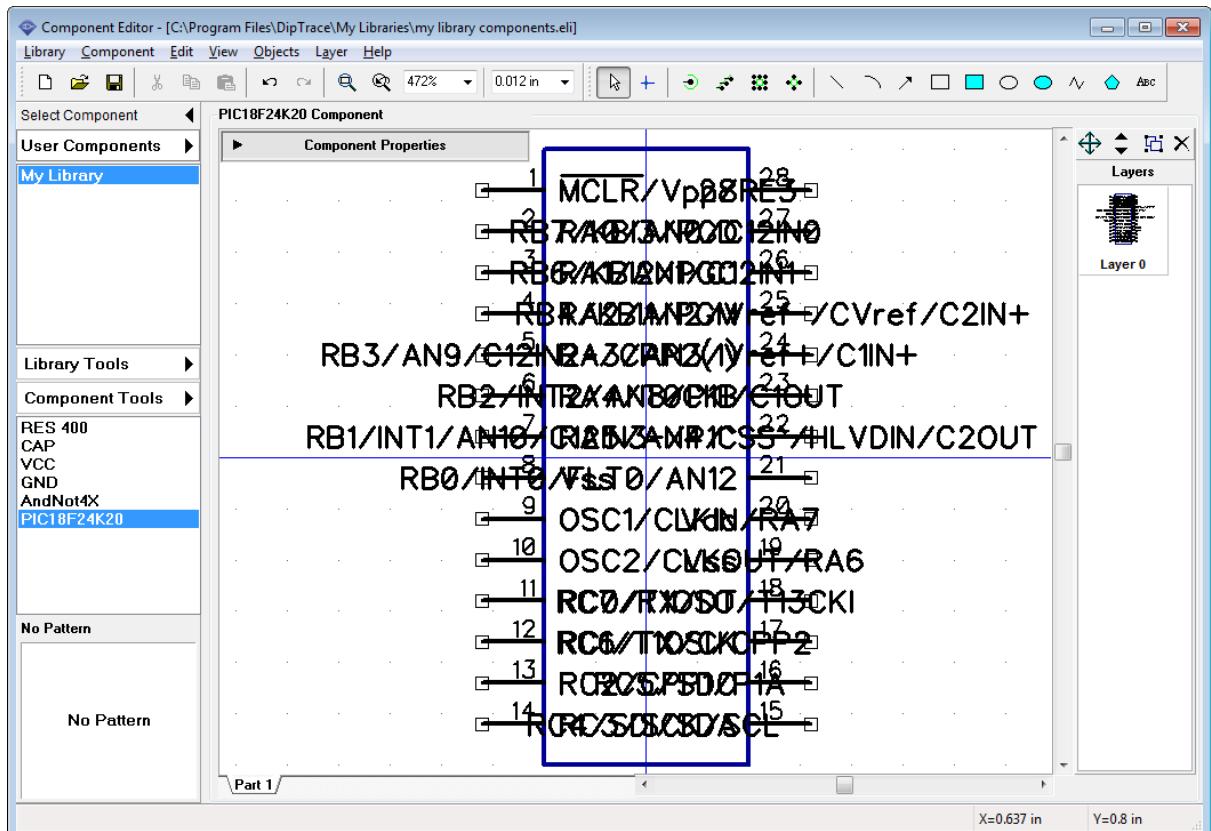


DipTrace allows to enter pin names manually or import them from external BSDM file (select "Library \ Import \ Add BSDL Pinlist" from main menu). We will do this manually. Press "Pin Manager" button on the component properties panel and enter pin names from pin diagram in the datasheet found online.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



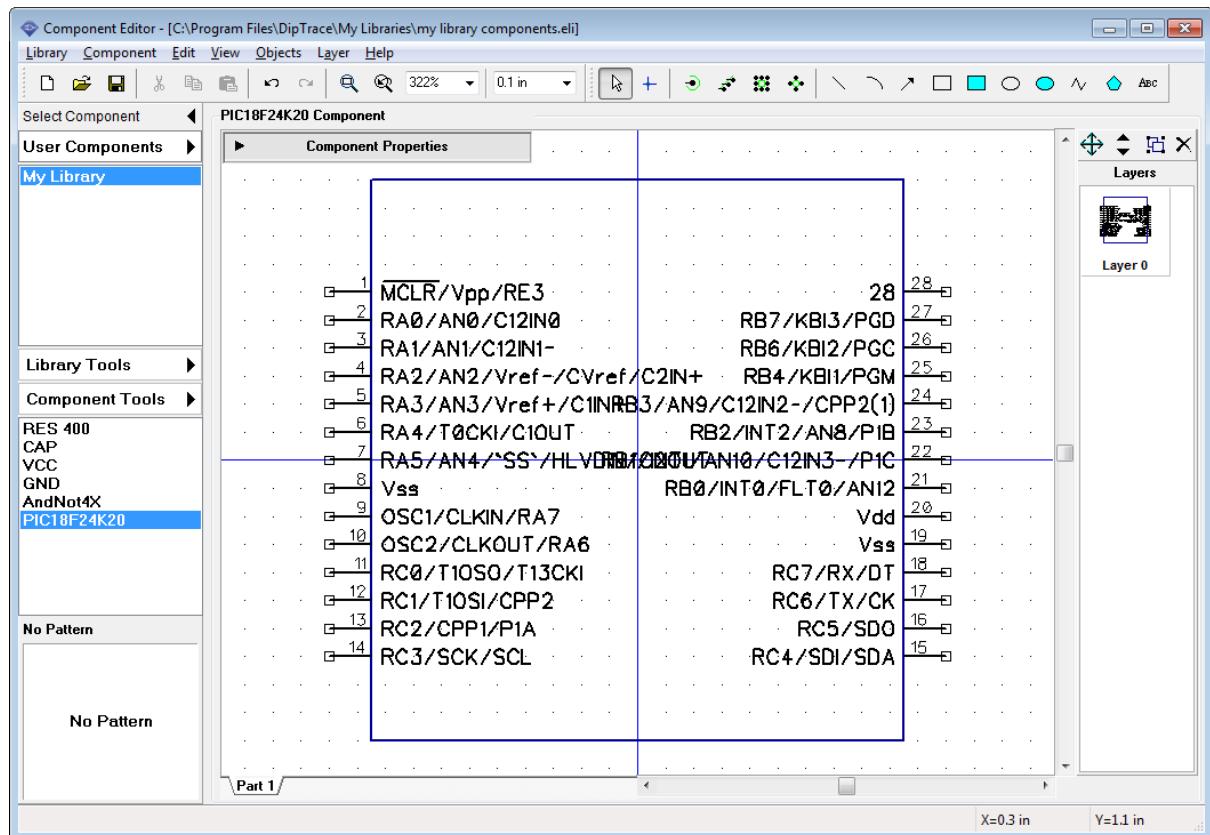


After entering pin names please specify electric type for pins and check "Show Name" for all pins of the component. Notice that you can select as many rows as you want and change properties at a time. Press "OK". Our symbol has inappropriate look, its width is too small, therefore pin names overlay.



On the component properties panel change width to "1.9" and height to "2". Pin names still overlap a

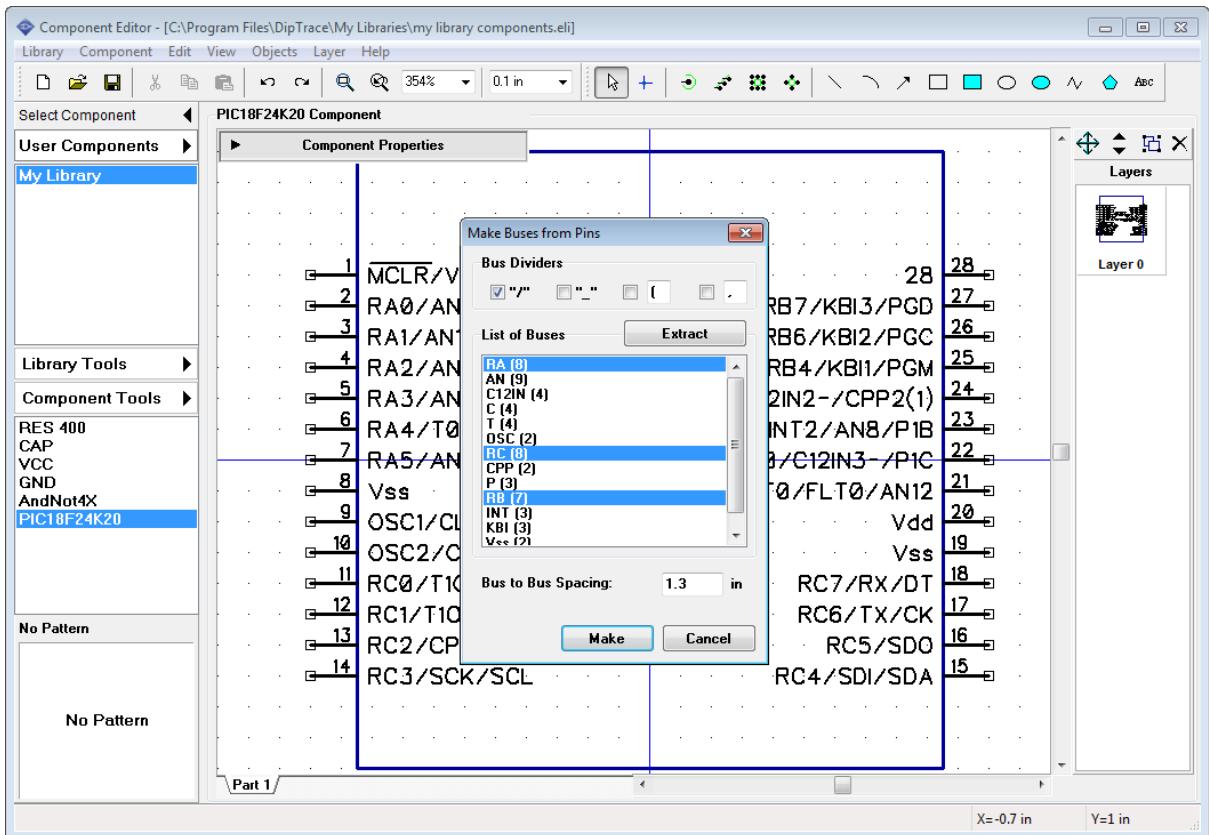
bit, but we will make pin regrouping, which will change the situation. We've made an IC a bit bigger - necessary for regrouping. Change grid to 0.1 inch and place pins by this grid (select all pins, right click and select "Snap to Grid").



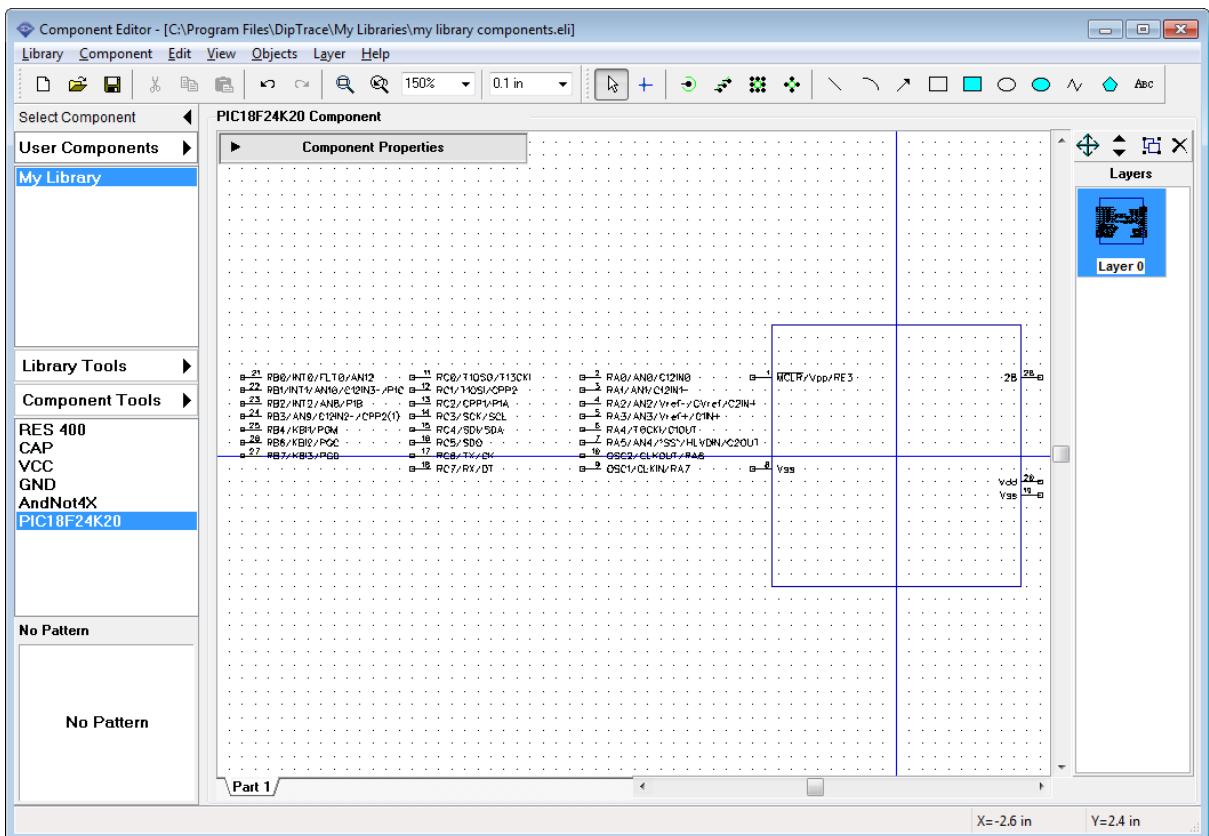
Make Buses - Group Pins

We need to group pins logically. First we will make busses: select "Component \ Make Buses from Pins" from main menu. This feature allows to extract buses from pin names and group pins by buses. 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 may have different dividers.

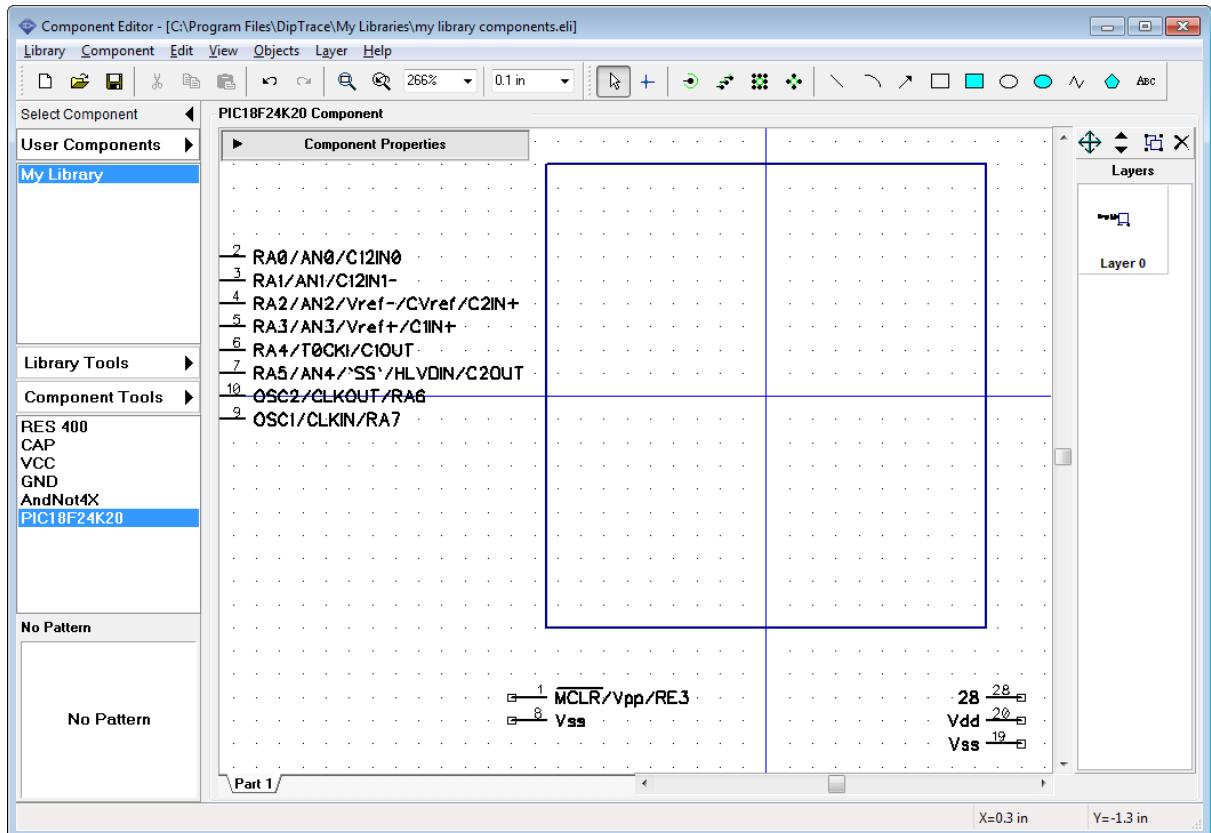
Press 'Extract' button and you will see available busses and number of pins for each. Select RA, RB and RC buses using "Ctrl" key. Change Bus to Bus spacing to 1.3 inches, because pin names are quite long.



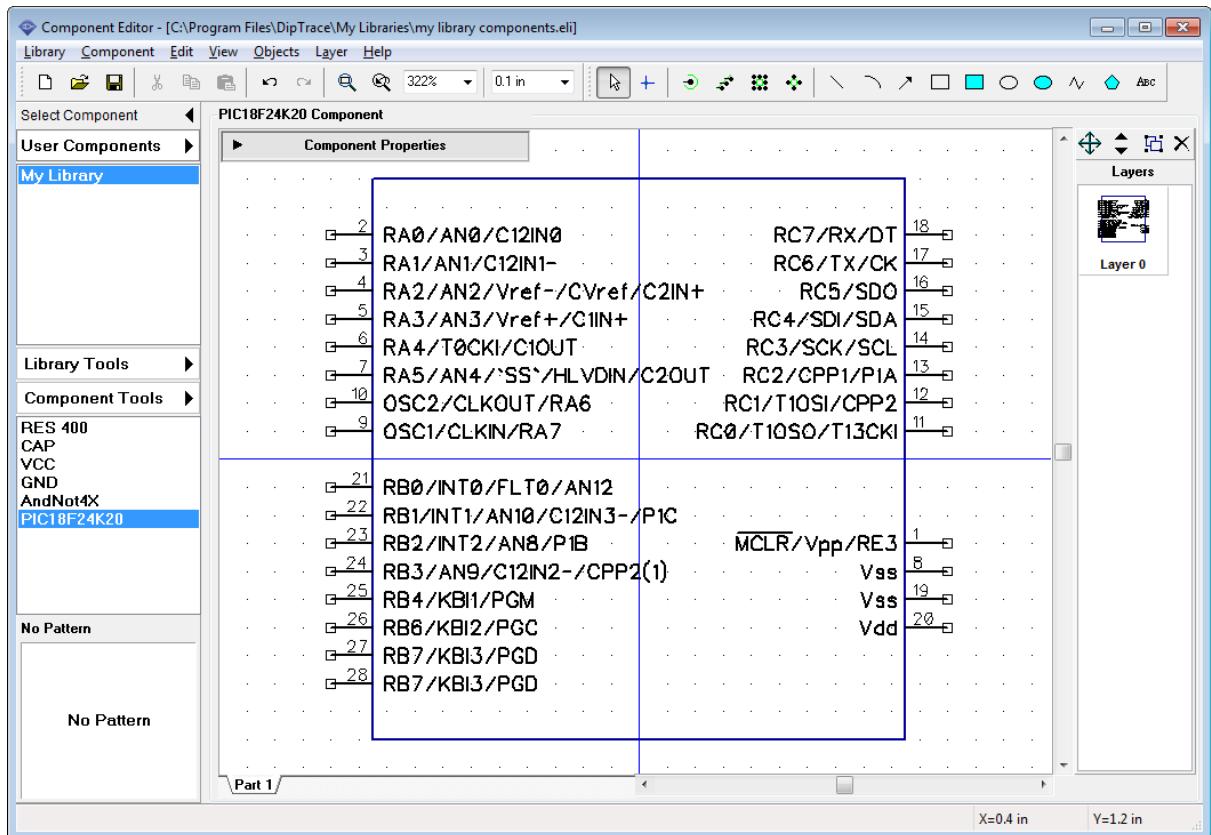
Press "Make" button to make busses and close this dialog box. Buses will be placed to the left from the symbol and sorted by number:



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



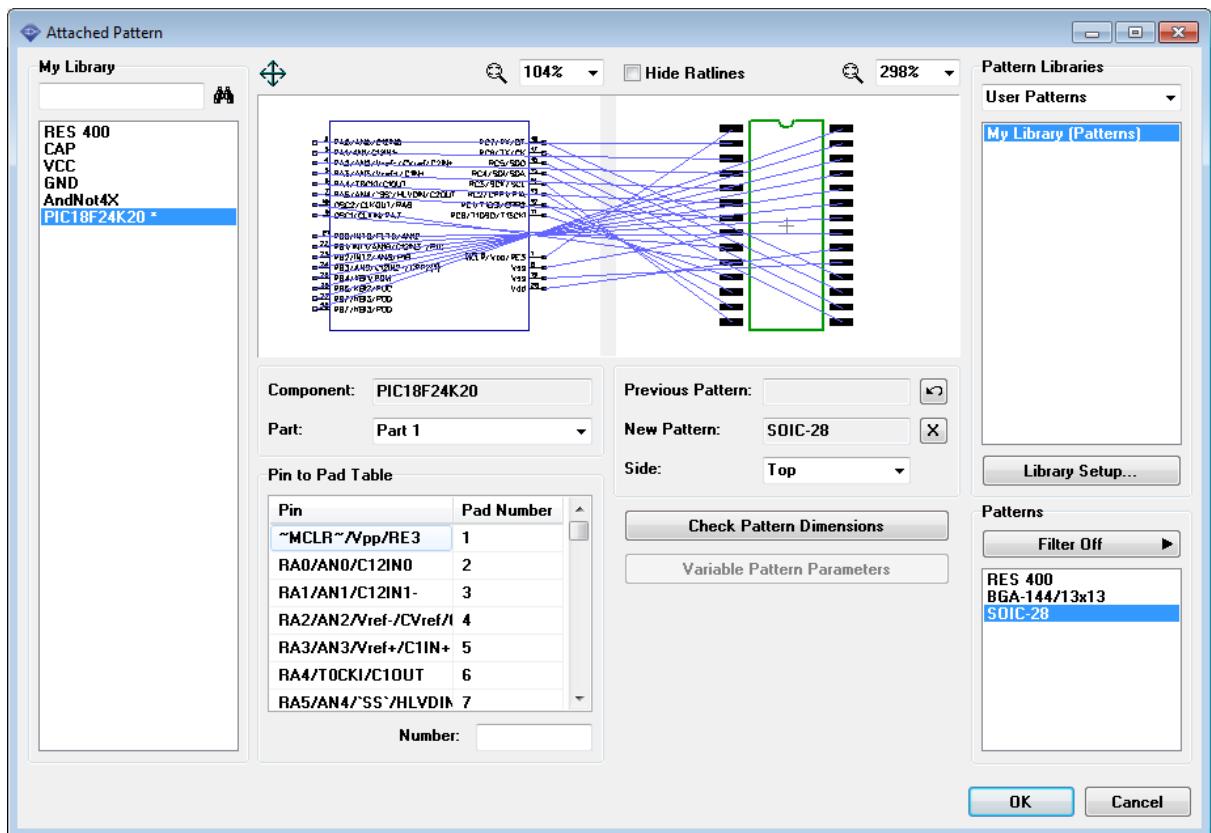
Place buses to the IC rectangle like on the picture below. Use box selection to select bus, then drag it. "Shift+R" can be used to rotate bus and "Shift+F" - to flip pins, these commands can be selected from pin submenu as well (right click on one of bus pins). Then move the rest of pins to the IC rectangle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

Final step is attaching "SOIC-28" pattern to the component. Press "Pattern" button on the component properties panel and select "SOIC-28" pattern from "My library (we've created it [before](#) 108). 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, so you don't need to change anything. Just press "OK".

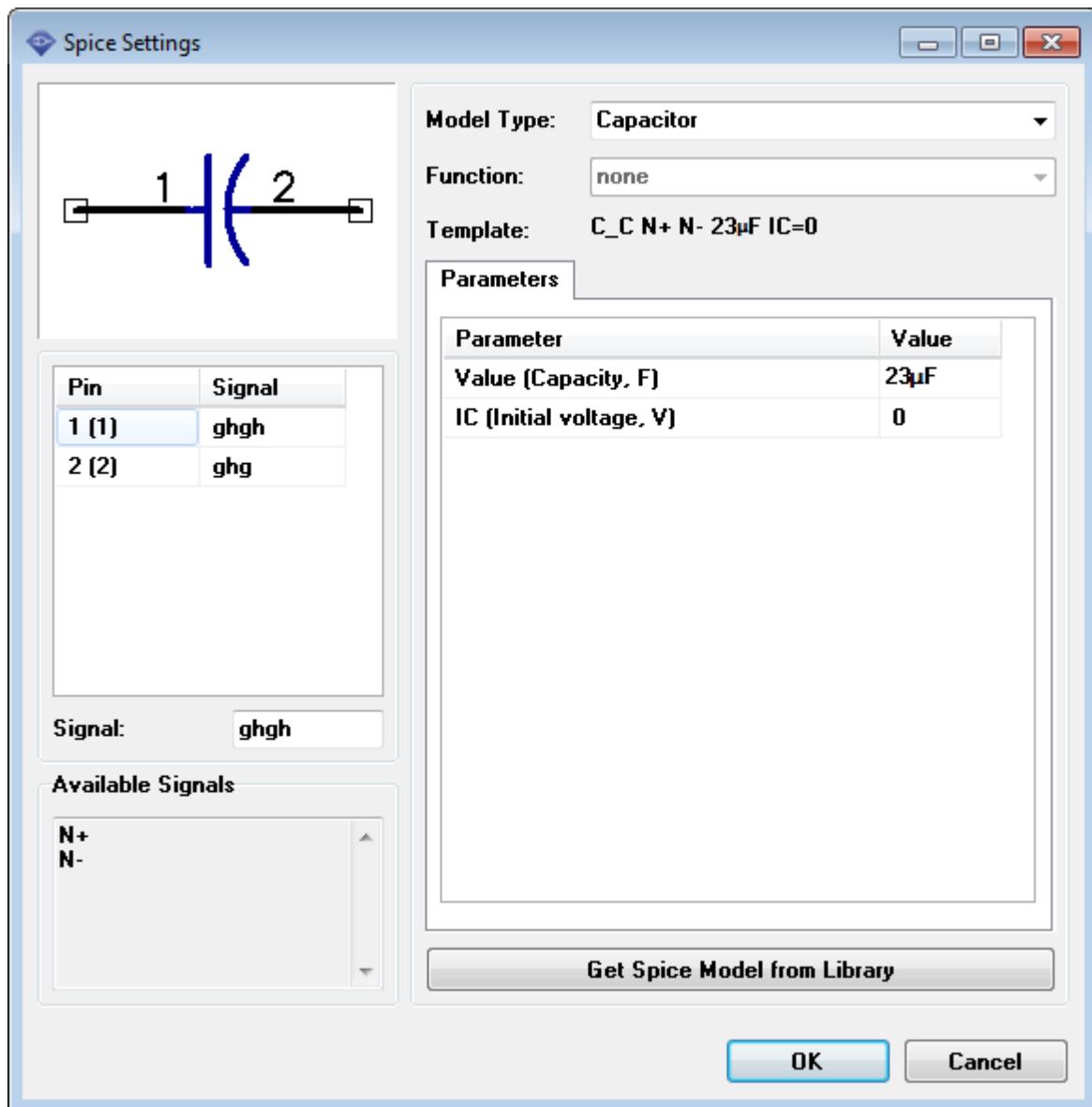


PIC18F24K20 component is ready!

3.2.8 Spice settings

With DipTrace you can export schematics into LT Spice to simulate it and see how it works. We will review this step-by-step later. But now we will specify that our CAP component is a capacitor with some value and it can be added to Spice netlist. Please select CAP in the components table. Then select "Component \ Spice Settings" from main menu. Select: "Model Type: Capacitor", then double click in "Parameters : Value" (cell with "1uF" text) and enter new value - "23 μ F" (All Unicode characters supported). Press "Enter" or just move focus to another field.

In the Template field you can see how this part looks in 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 Spice Settings dialog box. List of available signals (as information) is right below that table.



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

"Get Spice Model from Library" button - load existing Spice settings from another DipTrace component.

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

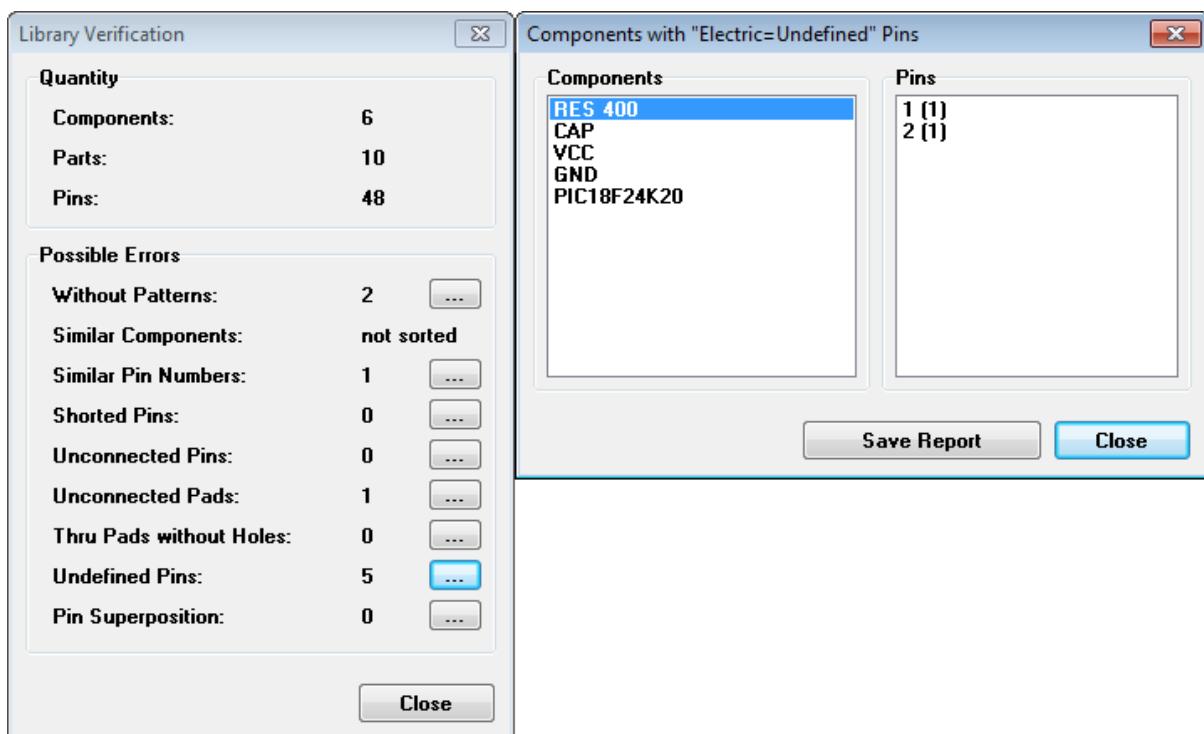
Library design is ready. Click "OK" to apply changes and close Spice settings. Save library file.

3.2.9 Library Verification

It is very important to verify library for any possible errors. We've investigated work of our library

designers and added possible automatic error verifications to Component Editor.

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



The following errors can be found automatically:

1. Components without patterns - keep in mind that some components may have only schematic symbol.
2. Similar components - components with similar names. Notice that library should be sorted ("Library \ Sort Components in 'My Library'") to allow correct verification with this feature.
3. Similar pin numbers - two or more pins have similar numbers (connected to the same pad). In 99% this is a real mistake in your component, please press "..." button and check pin numbers for listed components.
4. Shorted pins - pins shorted by internal pad-to-pad connections.
5. Unconnected pins - pins do not have corresponding pattern pads. Not always a mistake.
6. Unconnected pads - pads of the pattern are not used (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 are really surface type.
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 corresponding mistake. List of errors can be saved as a text file.

Save changes and close Component Editor.

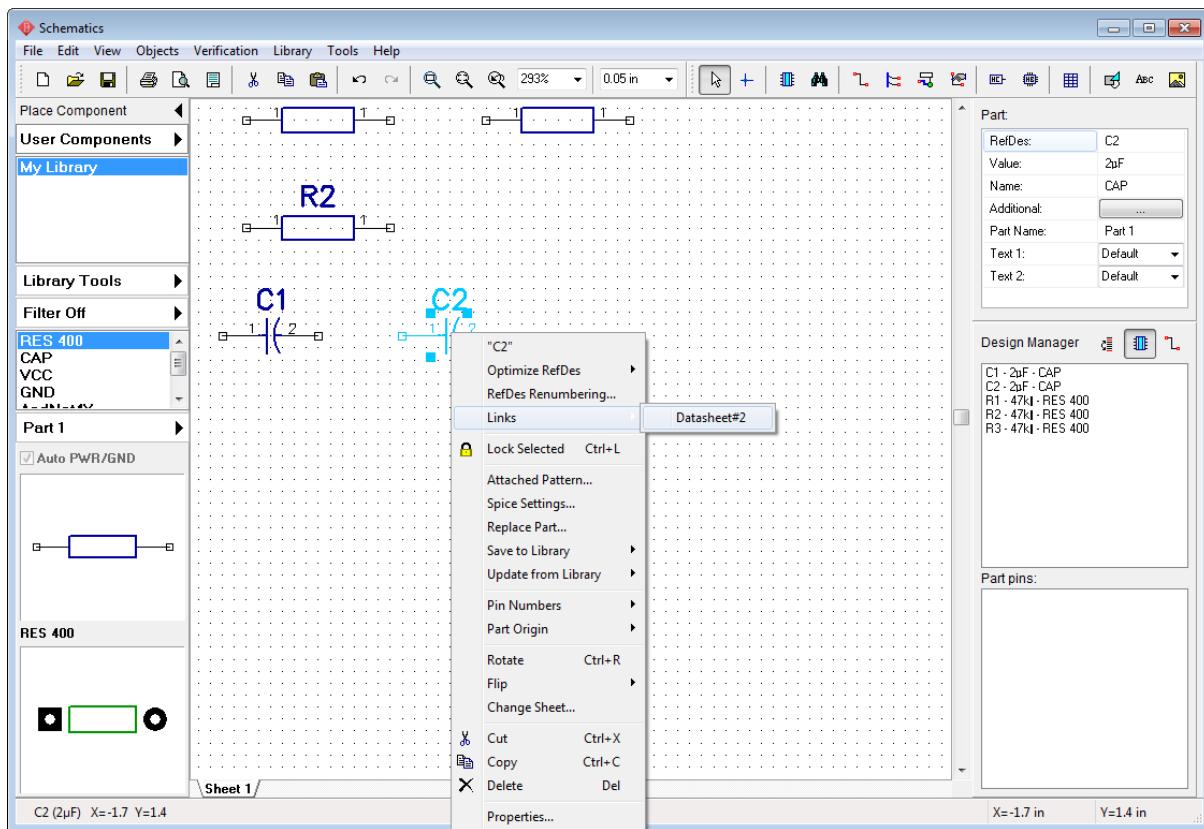
3.2.10 Placing parts

Schematic

Open Schematic module, i.e. go to Start → All Programs → DipTrace → Schematic. Select "My Library" from "User Libraries" library group. Place couple of resistors and capacitors to the design area. If origin is shown, press "F1" to hide it. Usually you don't need origin to design schematics.

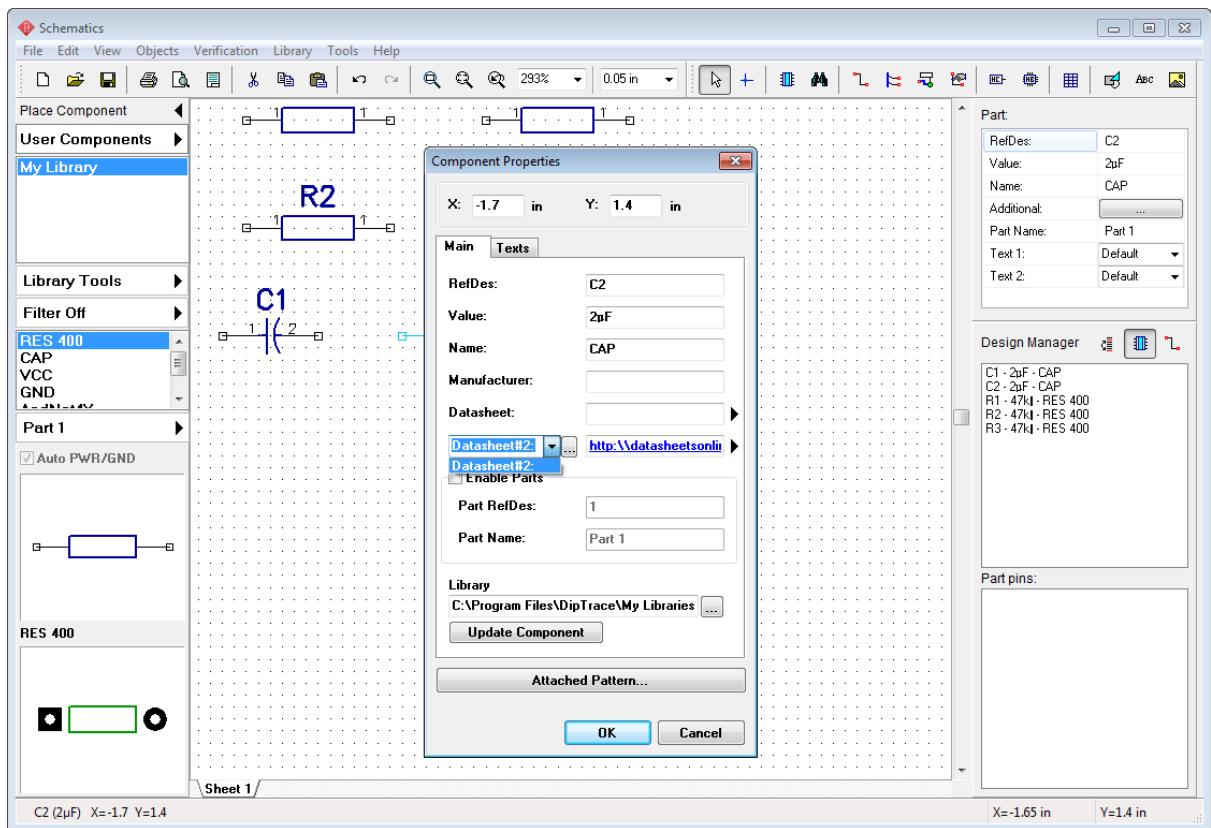
Notice that you can place components using "Objects \ Place Component" panel or corresponding button on the objects toolbar.

Now we'll see how to use additional fields of our capacitor. As you remember, we've created the link to the datasheet "Datasheet#2", while creating CAP component in Component Editor. Right click it and select "Links" from submenu. Now you can easily open web-site you entered.



Notice, that colors of components, selections, e.t.c., depends on the color template and user preferences.

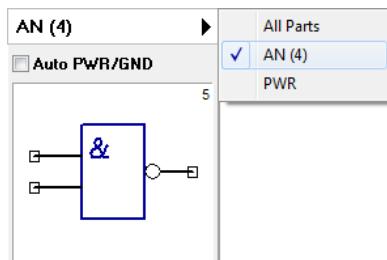
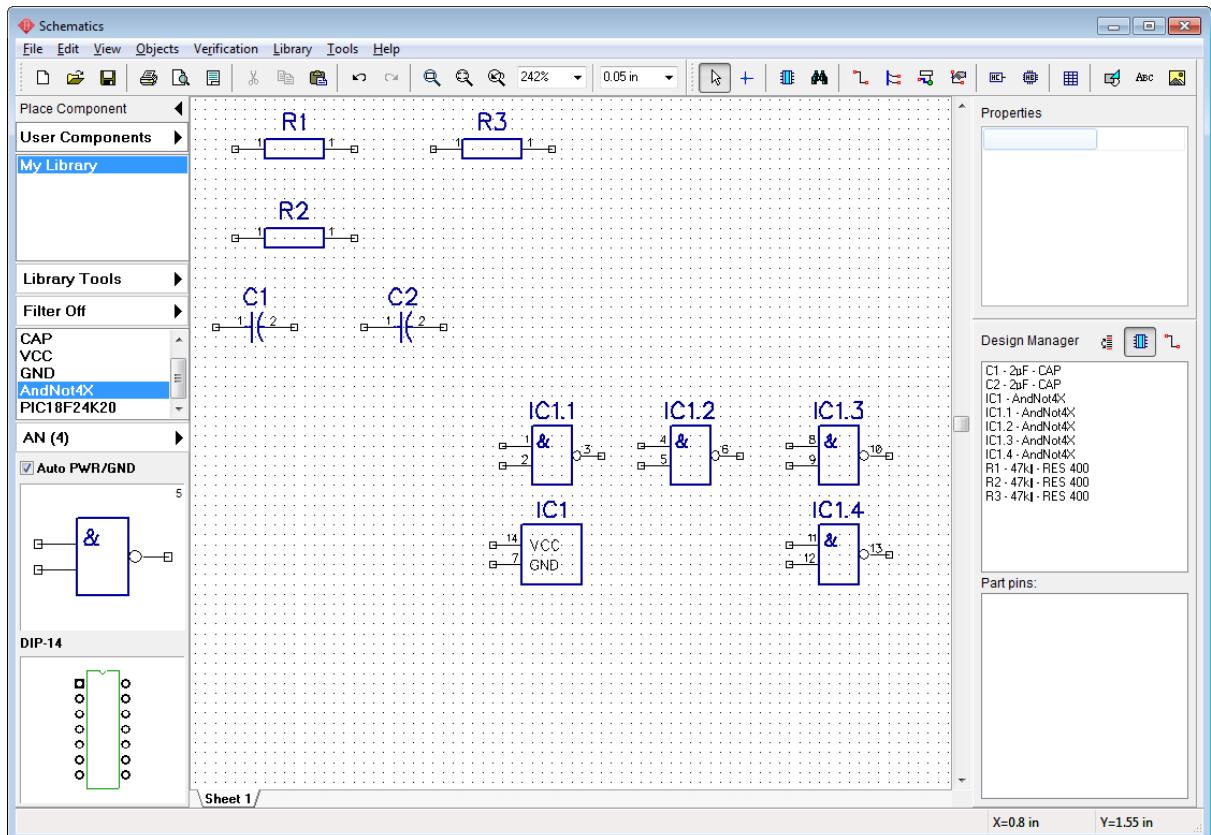
You can display additional fields as Part Markings (for example, "View \ Part Markings \ Main \ Additional" to show additional field as main text), but this is a rare case, since most important attributes (including Manufacturer, Datasheet e.t.c. are available). Select between several options in component properties dialog box (right click on the component and select "Properties" from the submenu).



Multi-Part Component Placement

Select multipart component. We have created "AndNot 4X" with four similar parts and a power part. All parts can be placed as one item or each part separately, use "All Parts" button on the Library Manager panel. Switch to "AN(4)" to place each component part one-by-one or "PWR" to place only power part. If "Auto PWR/GND" item is checked component's power part will be automatically placed on the design area together with first part. Select "AN(4)", leave "Auto PWR/GND" checked and place "AndNot4X" component part by part. DipTrace automatically selects next part from the part group and place power symbol for the component. Number of parts is shown in the symbol preview field on the Library manager.

Notice that program will automatically switch to the next component, when all parts had been placed. Make sure you place parts only with RefDes 1.1 to 1.4.

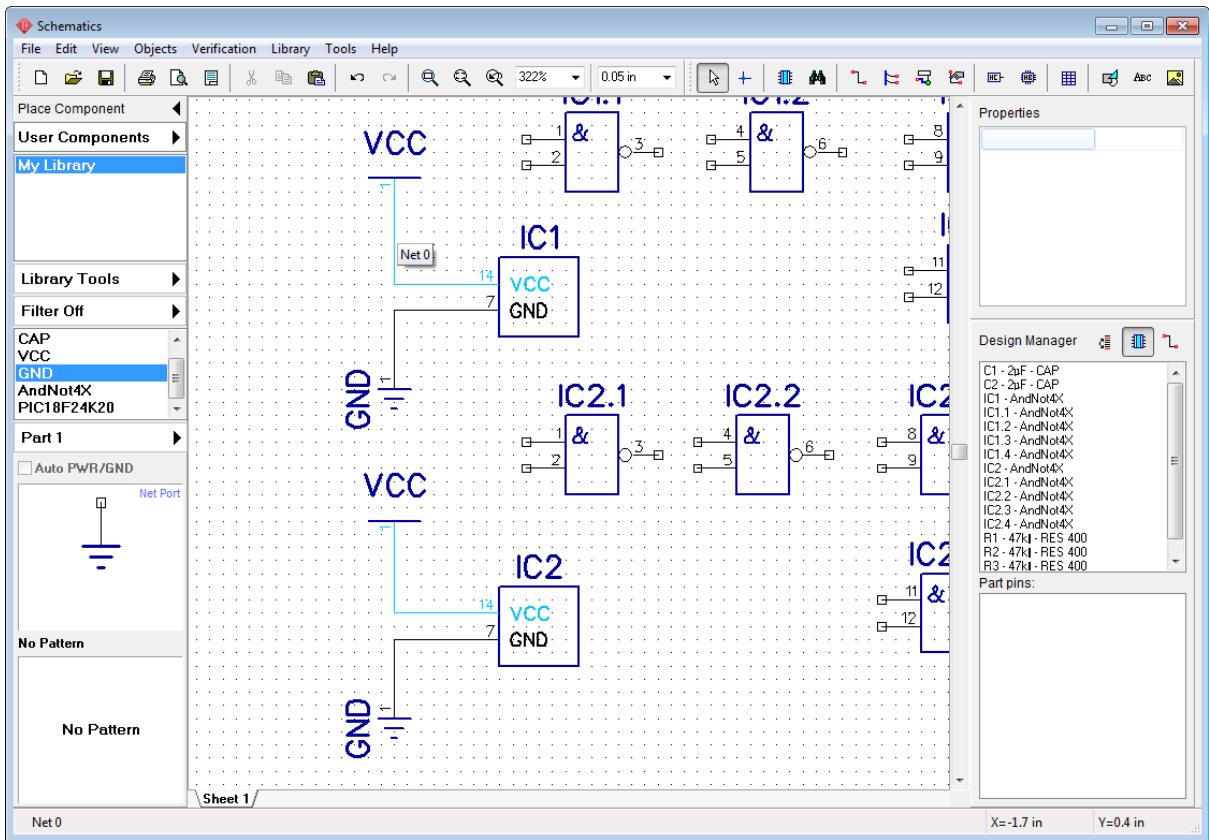


As you can see similar parts of multipart component are already grouped inside part selection drop-down list. Select "View \ Group Parts" from main menu to ungroup parts. If it is checked logic parts of the component ("AN(4)" in this case) will be grouped inside "All Parts" pop-up menu. If unchecked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

Now we'll try to use Net Ports. Place two complete AndNot components on the design area (IC1 and IC2) with two power symbols. Select VCC and GND net port symbols from the library and place two of each net port to the schematic. Connect pins like on the picture below.

DipTrace always shows that current component is multi-part or net port in the symbol preview area.

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



To rename 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 Component Properties).

Schematic allows to:

- 1) Connect pins to nets without wires (right click on the pin, select "Add to Net", then select net, check "Connect without wire" and press "OK");
- 2) Merge nets by name (check "Connect Nets by Name" box in the net properties dialog box);
- 3) Connect pins to the net with similar name automatically (check "Connect Net to Pins by Name" box in net properties dialog box). The last method is the fastest way to connect VCC, GND (if you plan to hide power nets and parts), CLK, etc. More information [later](#) [158] in this tutorial. Close Schematic. Do not save changes.

PCB Layout

Open DipTrace PCB Layout module, i.e., go to Start → All Programs → DipTrace → PCB Layout.

As you already know, correct component always includes at least schematic symbol (for Schematic) and attached pattern (for PCB Layout). Schematic works only with symbols, while PCB Layout allows to select component libraries and place component's patterns on the board or select patterns and place them separately. If you have selected component library and there are components without attached patterns, you can not place them on the board.

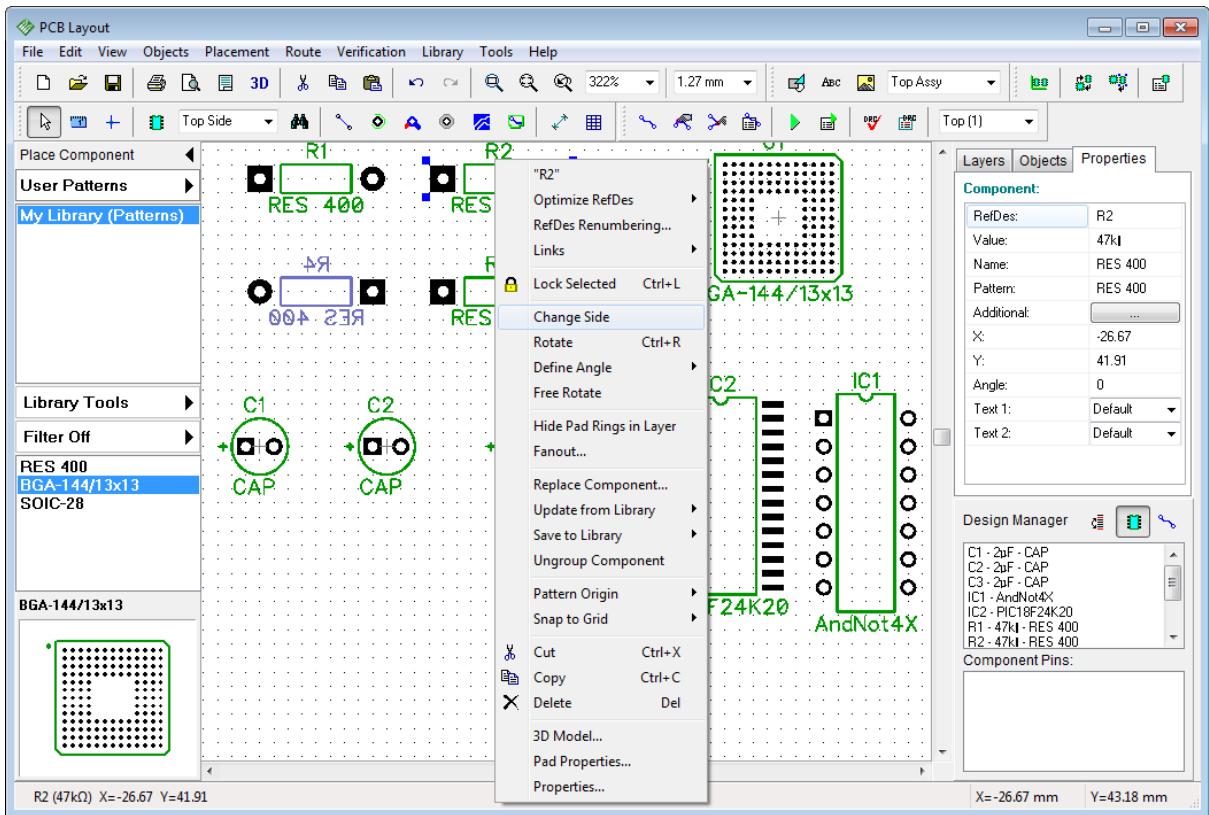
Select "User Patterns" library Group and "My Libraries (Patterns)". You will see that all three

patterns that we've created during lessons of this section of tutorial are available. Now select "My Library" from "User Components" library group. As you remember, we did not attach patterns only to netports, hence we can not place them on the board in DipTrace PCB Layout. All other components are good, because they have attached pattern.

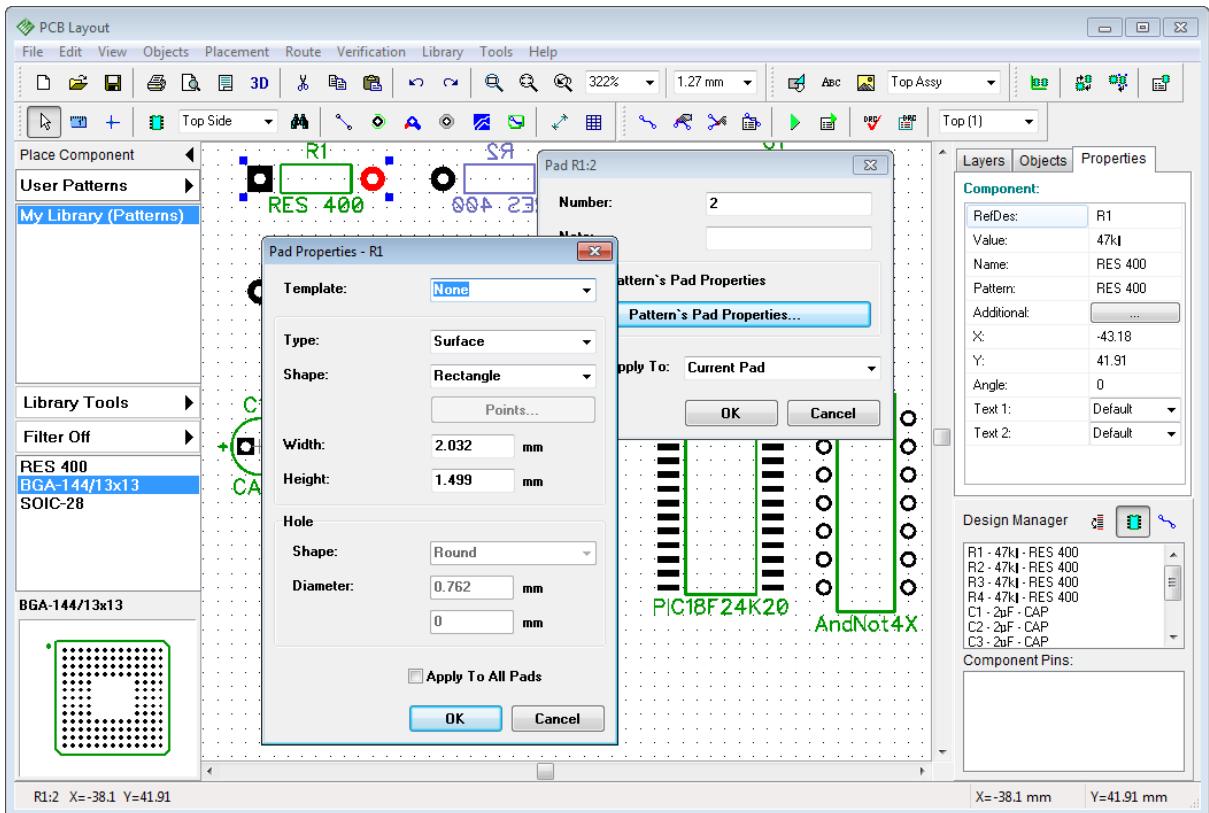
Place all components to the design area, excluding "VCC" and "GND" (select component on the Library Manager panel and left click on the design area to place it), then return to "My Library (Patterns)" from "User Patterns" library group and place BGA-144... pattern, which we did not attach to any of the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

Change common marking settings to show RefDes and Name ("View \ Component Markings \ Main \ RefDes" and "View \ Component Markings \ Additional \ Name"). 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 "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 side with right-click submenu. R4 resistor is on the bottom side (picture below).



Pad properties for pad or component's pattern can be changed directly in 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. Uncheck "Use Pattern's Pad Properties" for custom pad settings or press "Pattern's Pad Properties" button to change default pattern pad settings. To edit pattern pad properties right click on the pattern (not the pad) and select Pad Properties from the submenu.



Notice that if pattern's origin is different from pattern's center position it will be shown while you place 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 the center of rotation. When hovering over patterns you see coordinates of pattern origin.

4 Using different package features

This part of tutorial includes description of important DipTrace features not reviewed above. We consider that reader already knows how to accomplish basic tasks in DipTrace, hence we can move on.

4.1 Connecting

4.1.1 Working with Buses and Bus Connectors

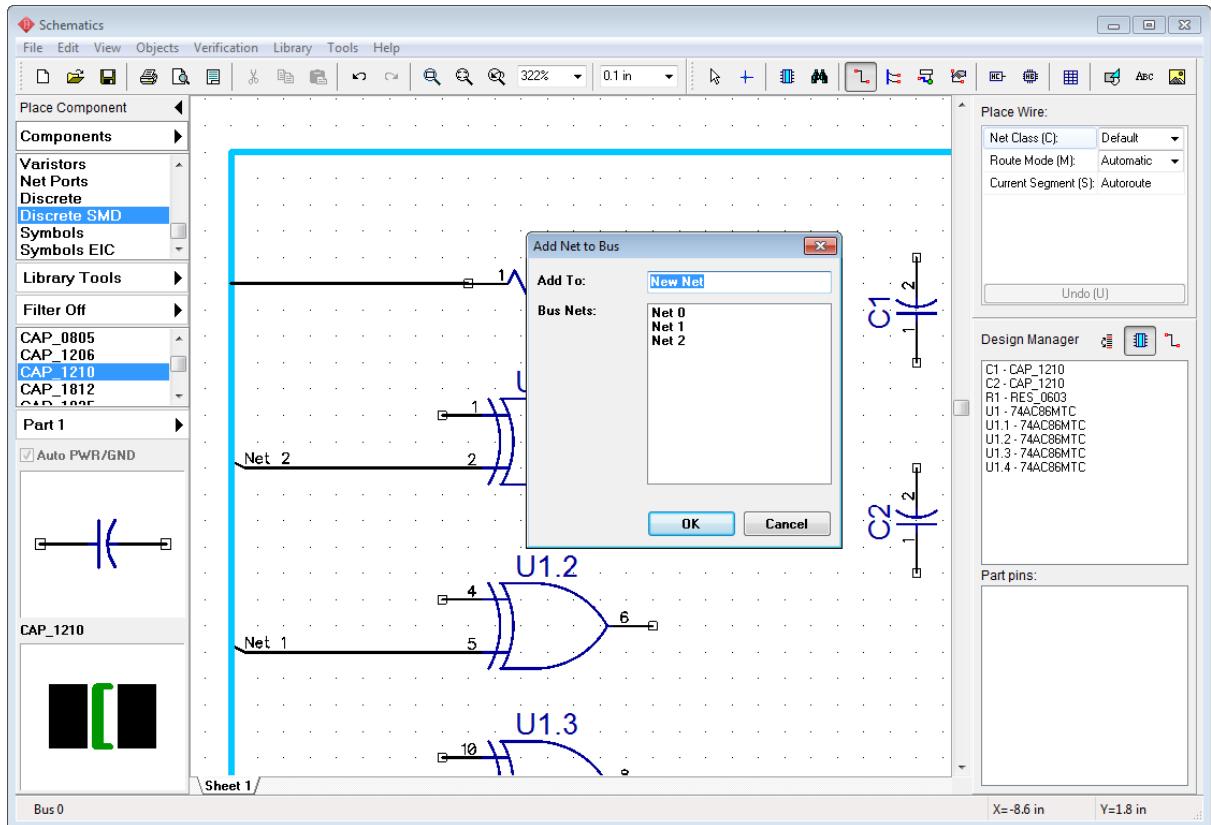
This section of tutorial will show how to use buses and connect sheets with bus connectors in Schematic. You can work with circuit from previous subsection of this tutorial or create a new schematic with random components for practicing with this feature.

Create Bus

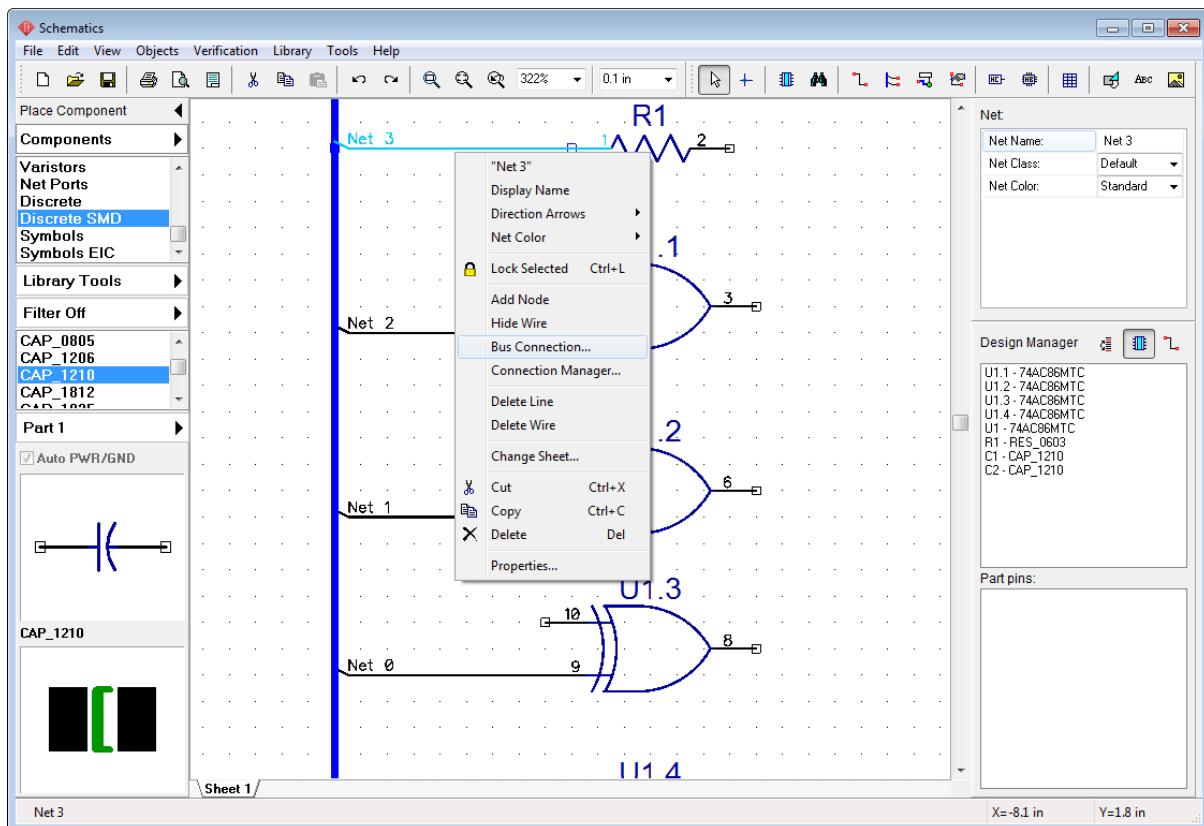
Select "Objects \ Circuit \ Place Bus" from main menu or press corresponding button on the objects toolbar then draw a bus line on the design area by defining its key points. Right click and select "Enter" to finish bus placement. Right click on a free space to switch to default mode. Hover over part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

created.

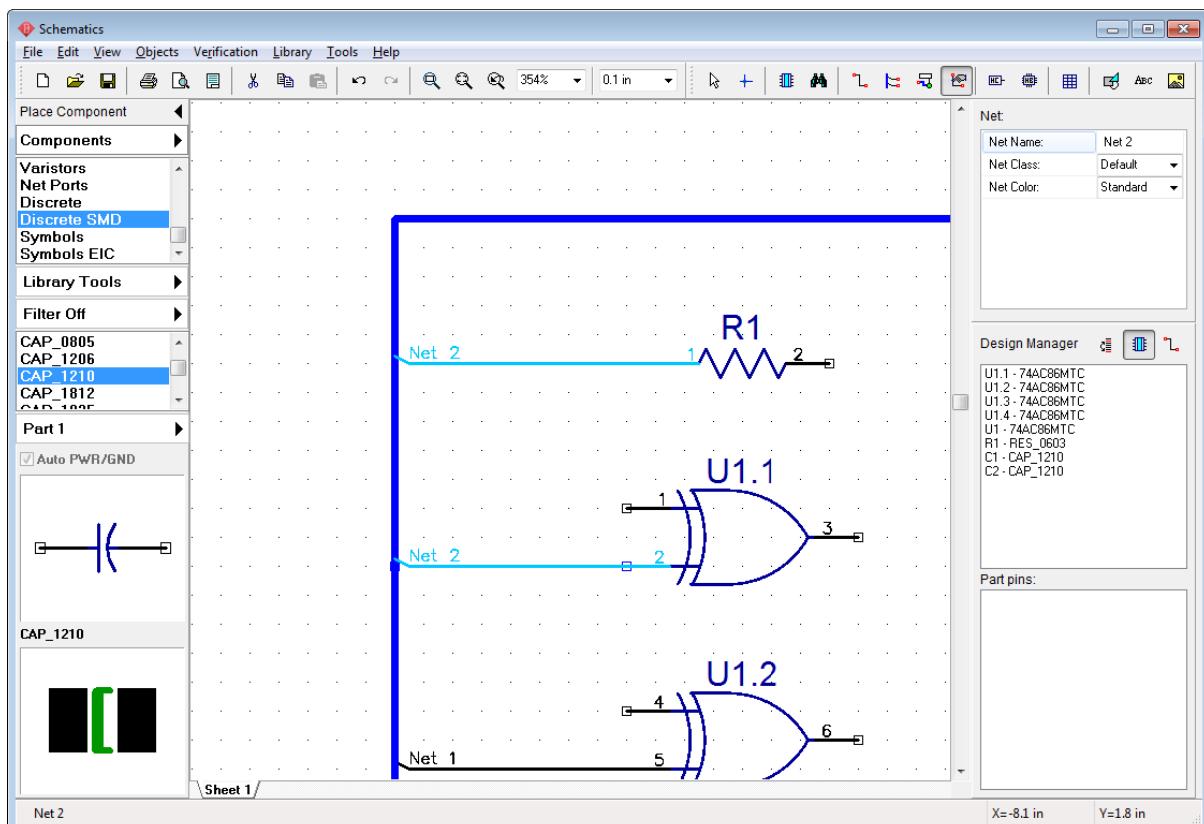
In the pop-up dialog box you can define name of the new net or connect wire to one of existing nets (which are already connected to that bus).



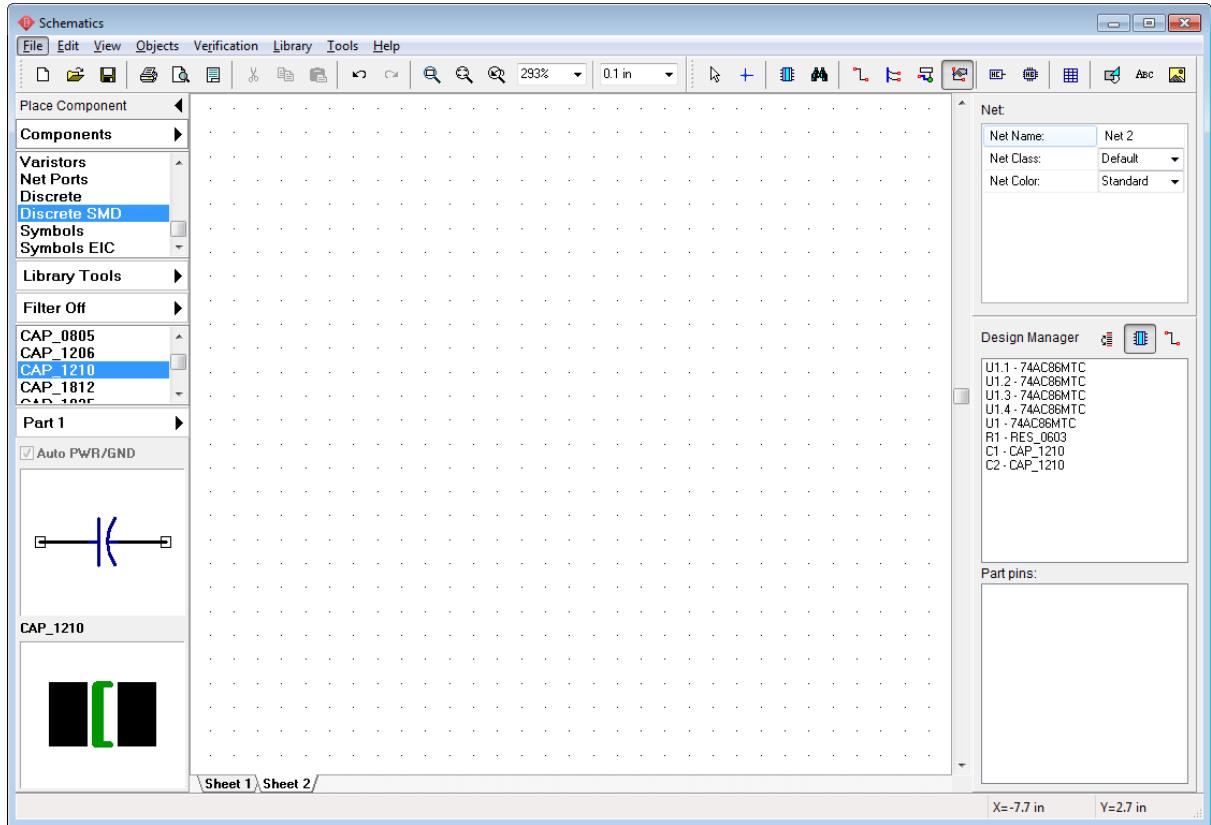
We didn't connect wire to existing nets, hence we have 4 separate wires-nets (Net 0 - Net 3) not connected to each other via bus. Fortunately, wire-to-bus connections can be changed at any given moment - move mouse to wire segment connected to bus, right click and select "Bus Connection" from the submenu.



In the pop-up dialog box connect Net 3 to Net 2 (select Net 2 from the list of bus wires). Now there is no Net 3 anymore. We have a single Net 2, connected via bus.



Please add new sheet to the schematic. Select "Edit \ Add Sheet" from main menu or press "Ctrl+Ins". You can see the list of sheets as tabs at the bottom left corner of design area. Select "Sheet 2". [Multi-sheet and hierarchical structure](#)^[196] will be described later in corresponding section of this tutorial.

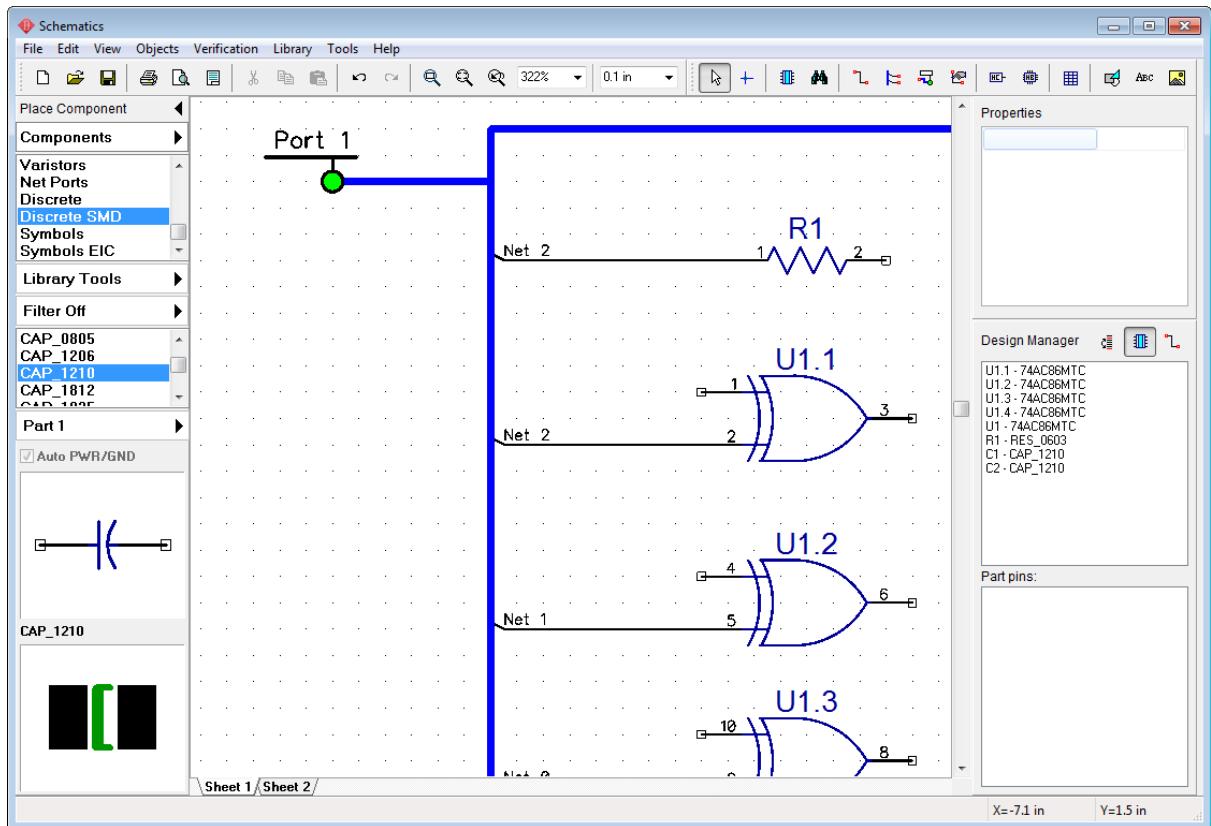


You can rename, move, delete or insert new schematic sheet, right click on the tab in bottom-left and select appropriate item from the submenu.

Bus Connector

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

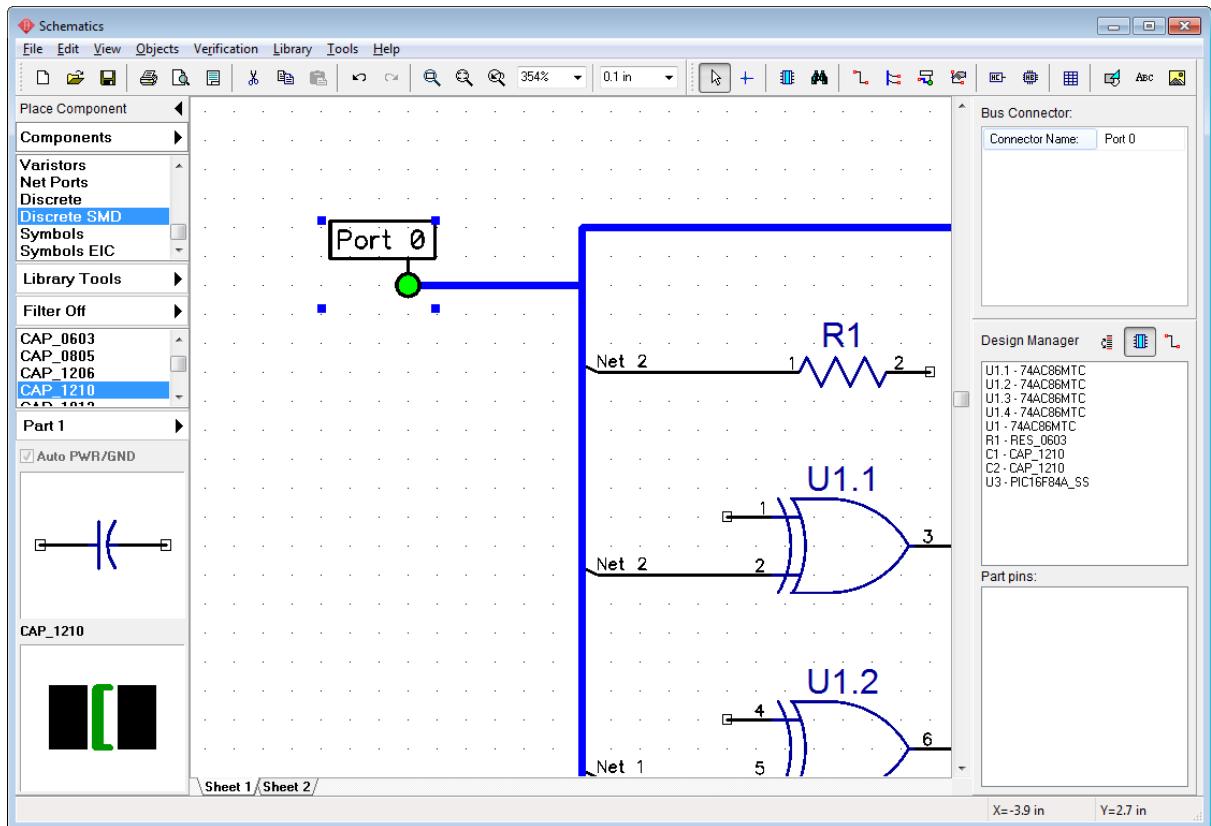
Notice that bus connector glows green if bus is properly connected; blue circle means unconnected to bus, green - connected with bus, like on the picture below.



Notice that bus connectors are still unconnected (should have same names in order to be connected).

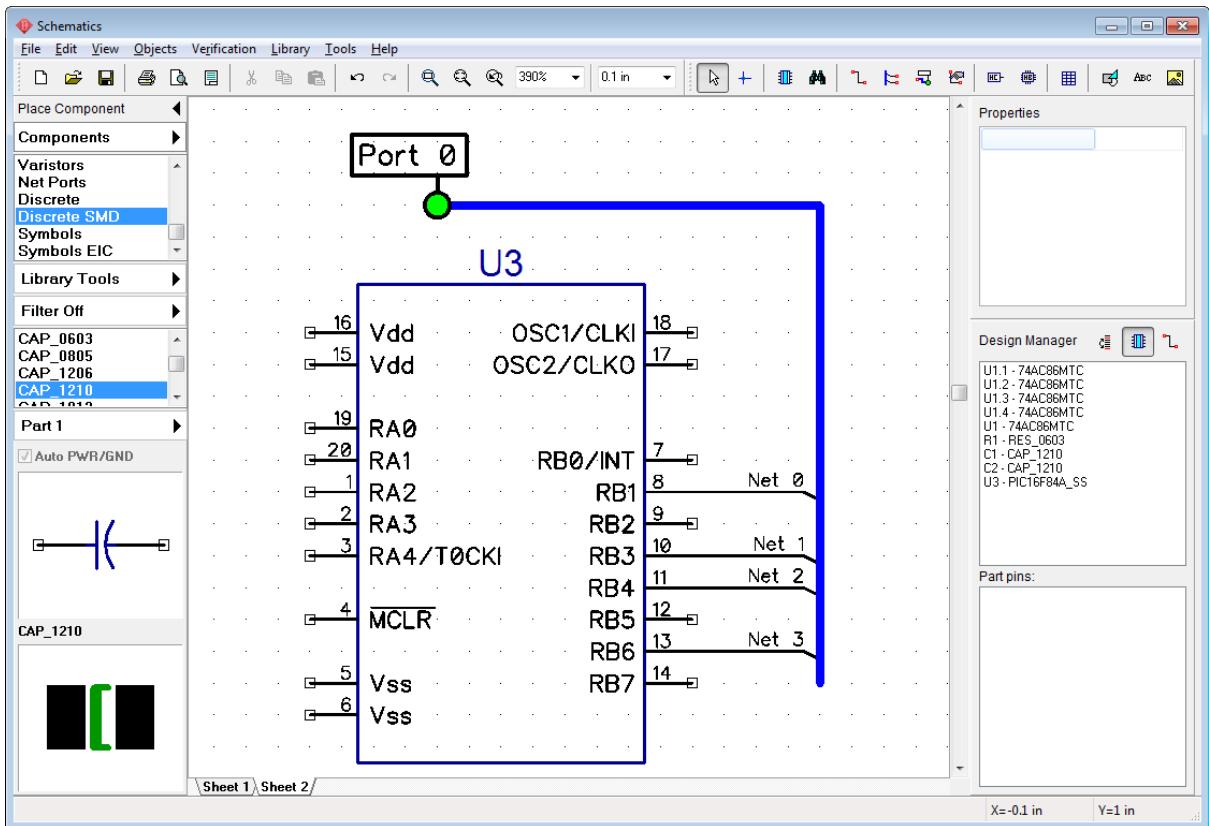
Rename "Port 1" - hover over, right click it, select first item from the submenu and rename bus connector to "Port 0" (as you remember we've placed "Port 0" on the Sheet 2). Press "OK". You can see that box has appeared around port's name. This means that current bus connector is connected. In our case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

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



Select "Sheet 2" and create bus connected to "Port 0" there.

Notice that name of the bus on this sheet is the same as on Sheet 1, i.e. this is common bus. Now you can place random electronic parts on the second sheet ("AD1317" from "Analog Devices" library in this case) and connect their pins to nets which we've connected to bus on the first sheet.

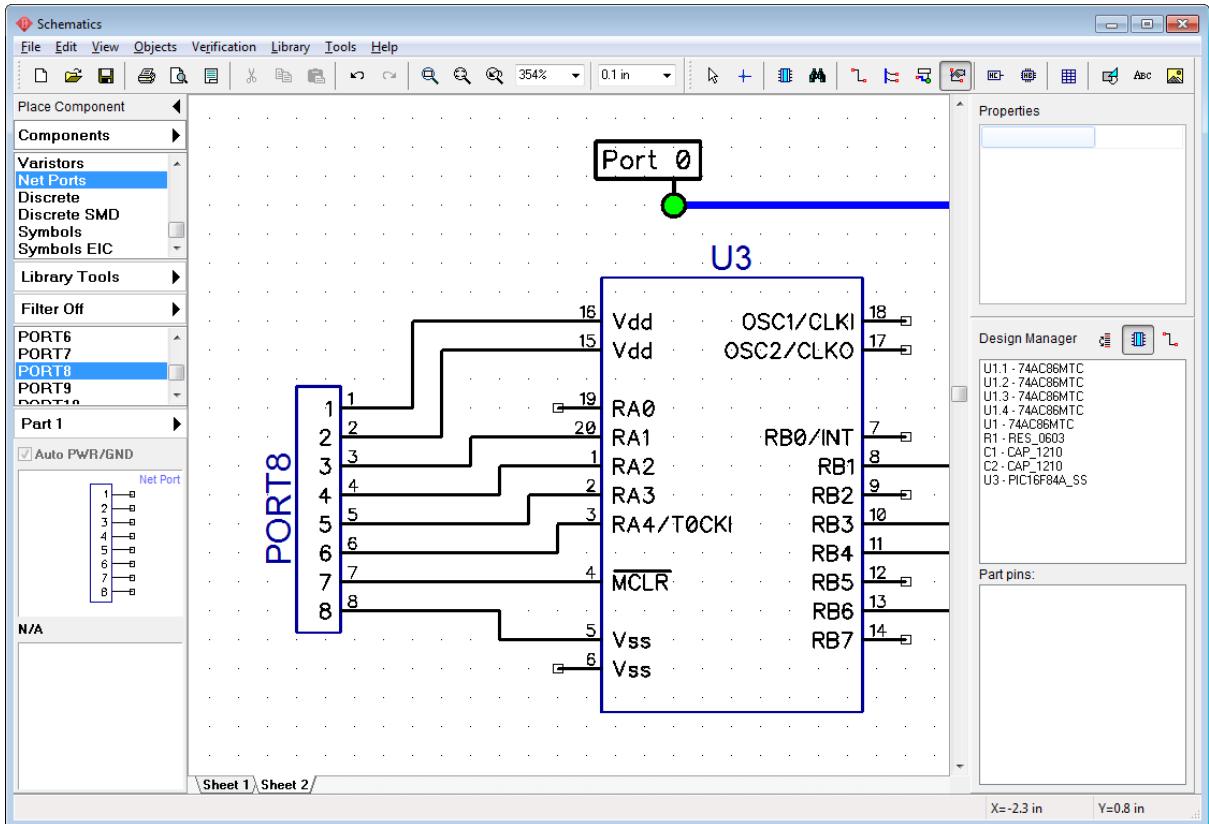


4.1.2 Working with Net Ports

We tried to use net ports [before](#)¹⁴⁶ in order to create VCC and GND connections. Most often netports are used for this purposes, but sometimes designers need several-pin netports.

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

Make connections from the component's pins to pins of Port 8 then place one more Port 8 component to Sheet 1 and connect components to it. 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 Port 8 are connected to the single net, the same with other pins. You can connect or disconnect ports (i.e. change schematic structure) by renaming them.

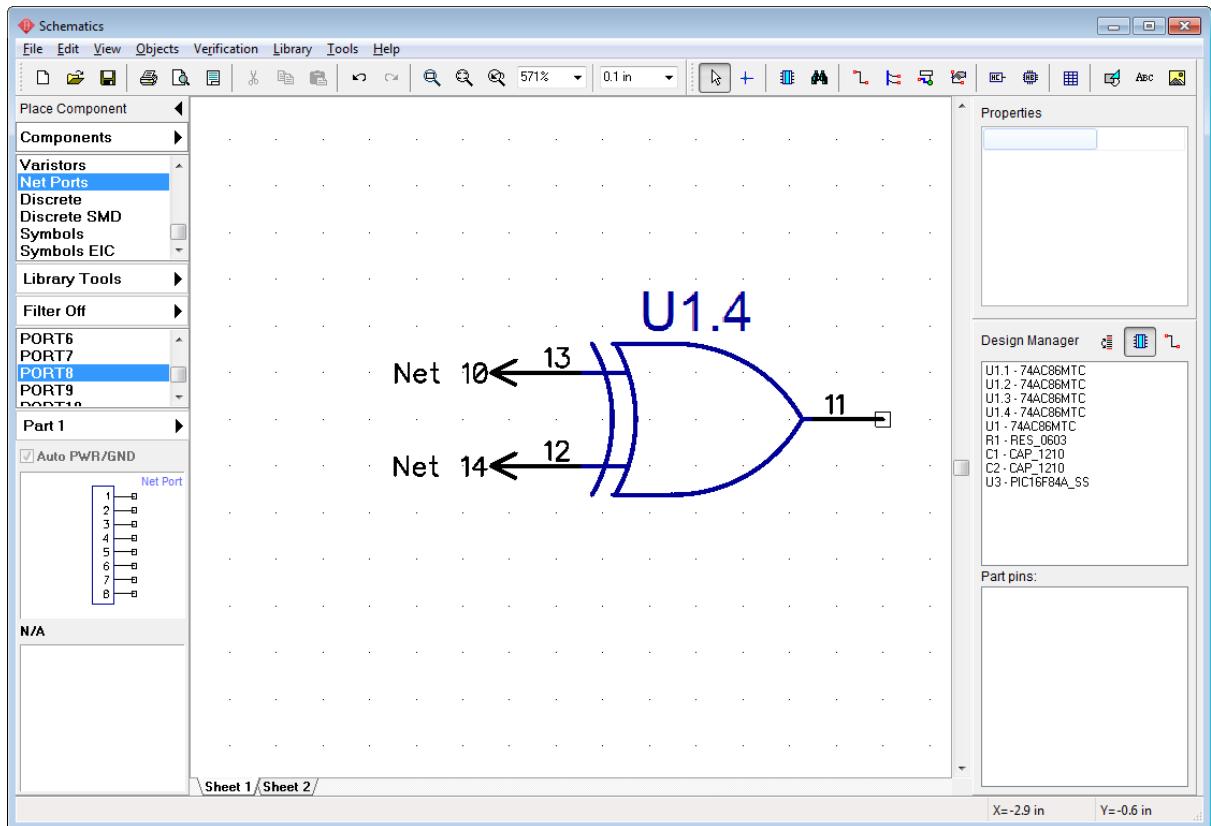


4.1.3 Connecting without wires

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

Connect 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, 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.

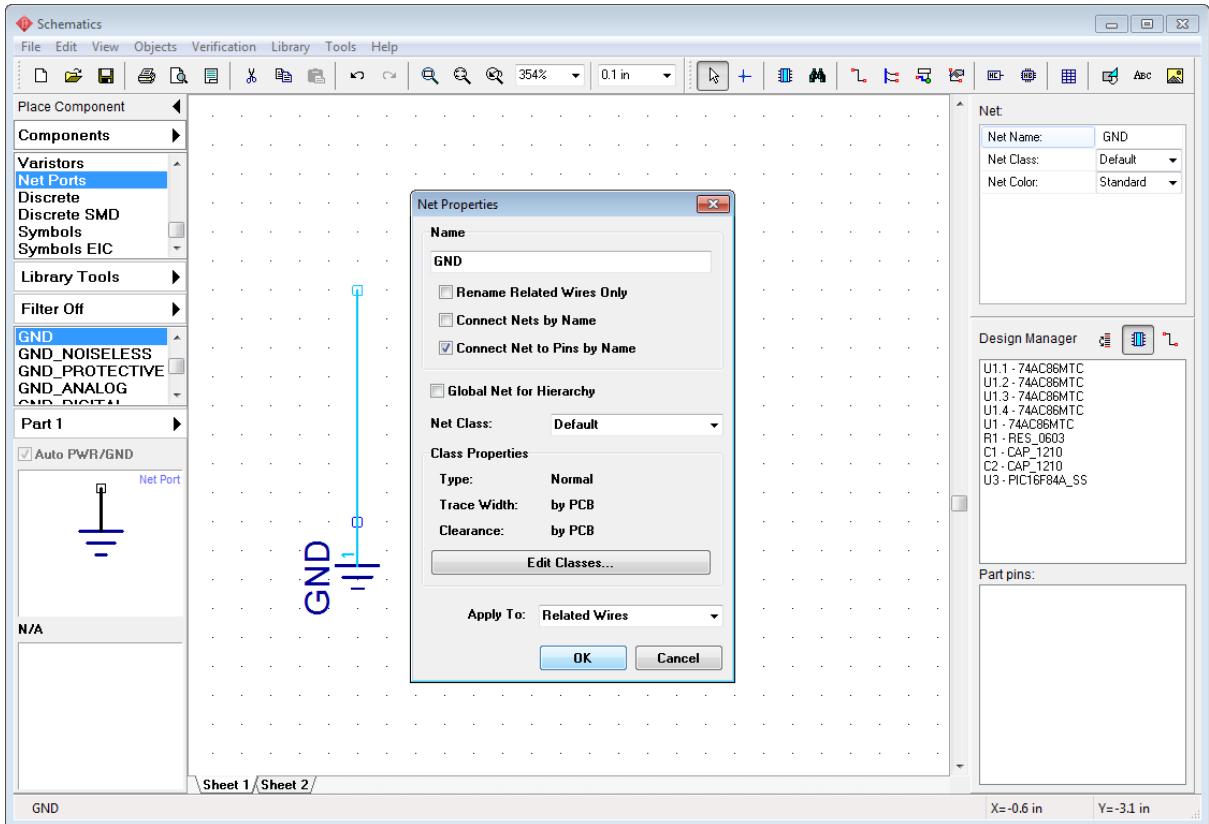


Connect Pins by Name

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

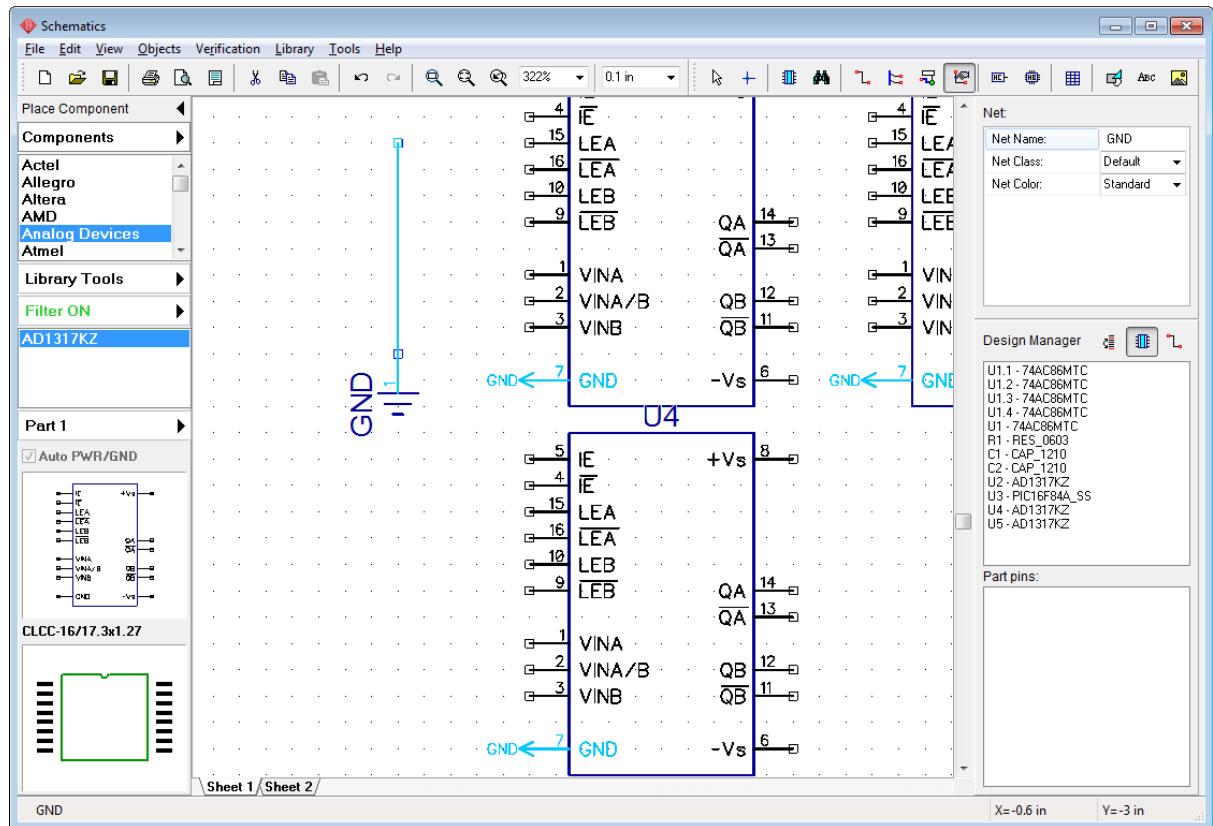
In Net Properties dialog box rename net to "GND" and check "Connect Net to Pins by Name" box. Press "OK" to apply changes.

DipTrace will automatically connect all unconnected pins with corresponding name to this net.



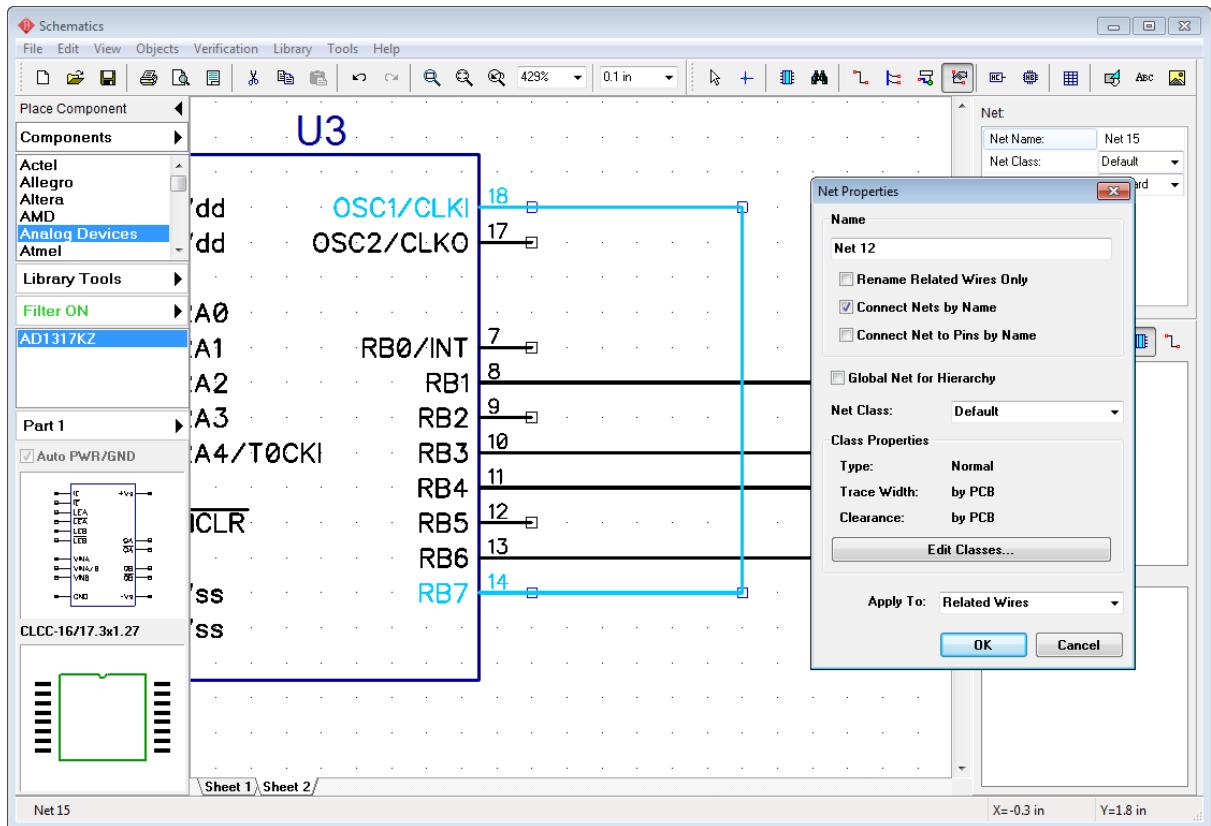
Select "Analog Devices" library and [find](#), for example, "AD1317KZ" component (press "Filter ON/OFF" button, type "AD1317" into name field and press "Apply Filters"). Place several AD1317 components to the design area.

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



Connect Nets by Name

DipTrace allows to connect nets on different sheets without net ports or buses. Remember name of some net on Sheet 1 (in our case it's going to be Net 12). Then go to Sheet 2 and right click on the net you want to connect to net from the first sheet (Net 12), select "Properties", type in the new name ("Net 12") and check "Connect Nets by Name". Press "OK".



As you can see, basically, it works like a regular net renaming. If you change name of certain name to the one which already exists, DipTrace will ask if you want to connect these nets by name.

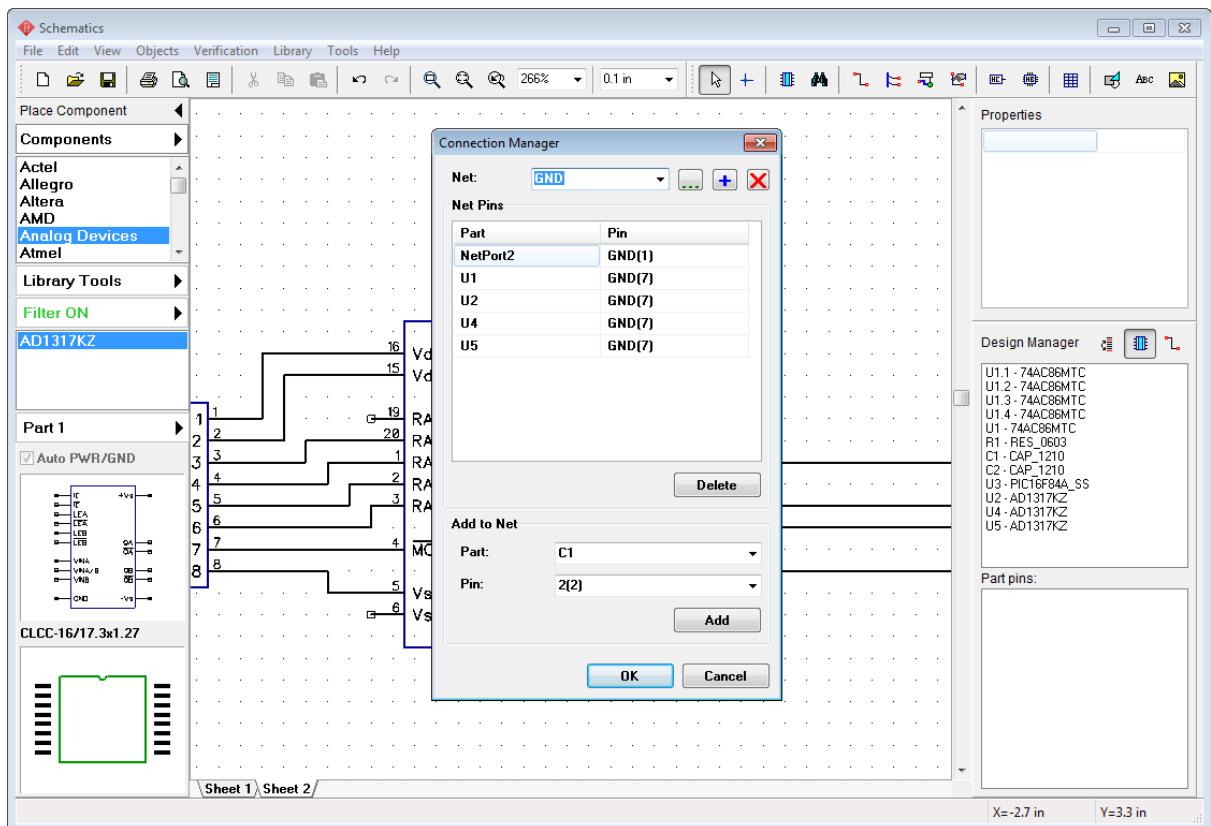
*Notice that you can **not** connect nets by name on different levels of hierarchy.*

For that purpose you can create **global nets**. We will learn how to use them later in [Hierarchical Schematic](#)^[196] subsection of this tutorial.

4.1.4 Connection Manager in Schematic and PCB Layout

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

Open connection manager in Schematic. Select some net from the drop-down list and you will see all net's pins. Add / delete pins to / from the net easily. To add new pin to net, select part and its pin below and press "Add" button. Notice that only free pins are shown in these drop-down menus, so if you can't find a pin that you need, it is probably connected (maybe to another net). Use "+" button to create new net. "..." button renames current net and "X" - deletes it.



Press "OK" to apply changes and close connection manager or press Cancel to close it and recover old net structure.

4.2 Reference Designators

Now we will work with schematic examples located in Documents \ DipTrace \ Examples folder. Open Schematic_2.dch file from Examples folder.

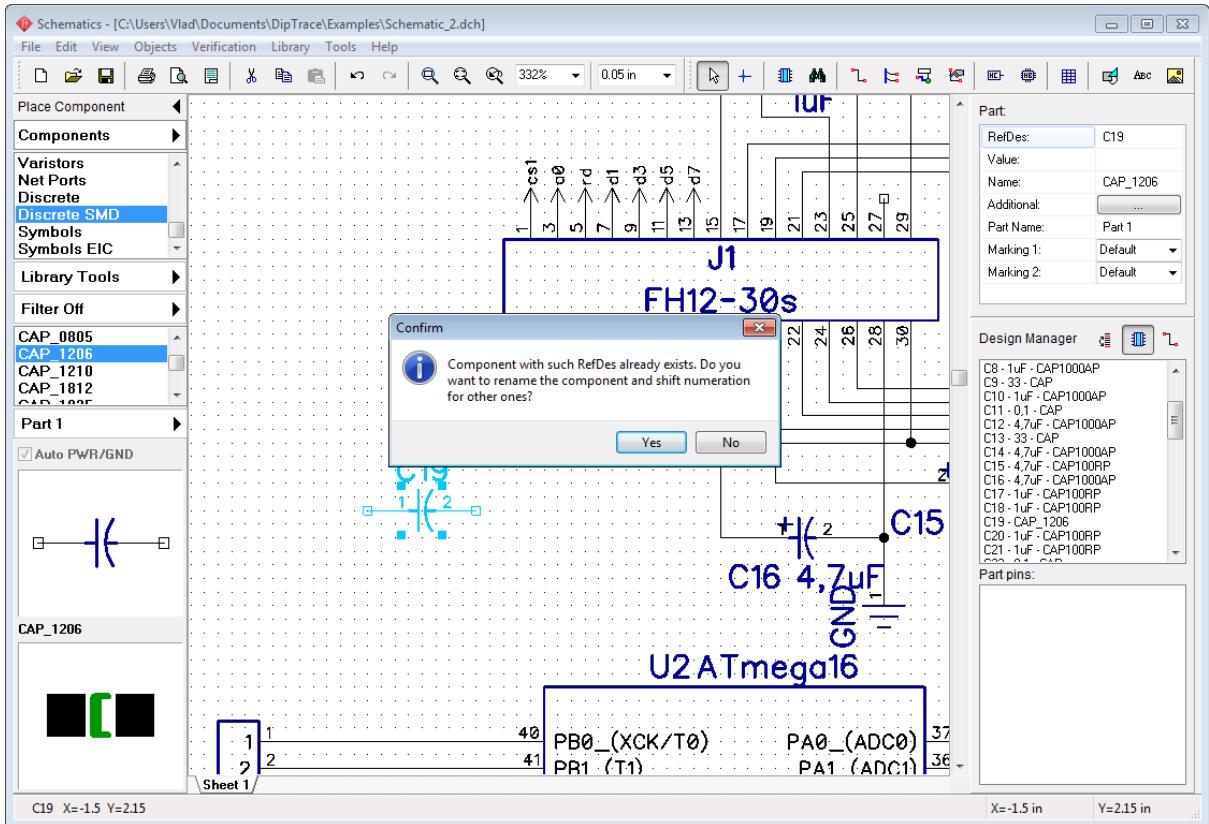
This project demonstrates applications of various DipTrace features. However, we will experiment with some basic principles of working with Reference Designators in Schematic.

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

Optimize RefDes

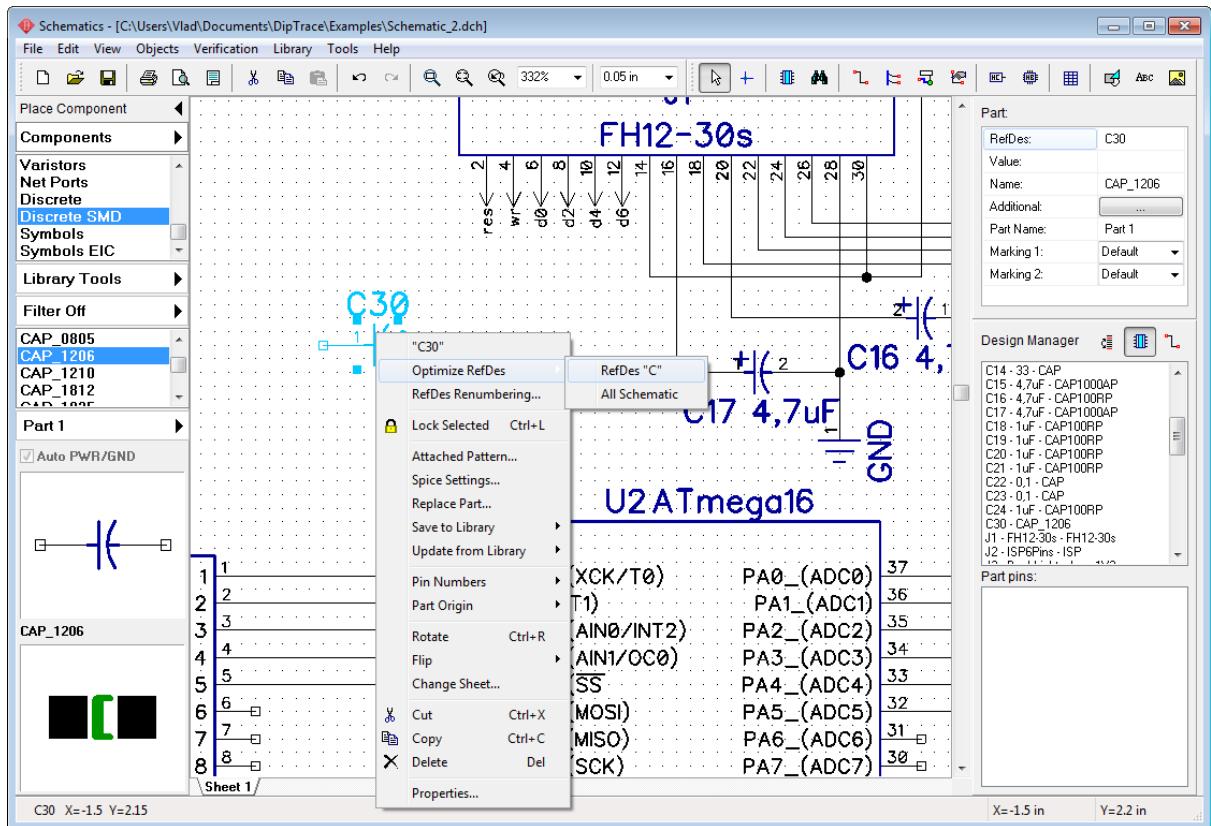
While editing any circuit you will probably need to insert, for example, C5 capacitor. Pick up some capacitor from the library you've created recently ("My Library") or "Discrete SMD" Library and place it to design area. We want "C5" RefDes for this component, but it is "C19" now (because C19 is missing and C5 is already present on the schematic).

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



C19 Capacitor was renamed to C5 and old C5 became C6 and so on till C18 capacitor, which is C19 now. You can see in connection manager that C19 designator is not missing, because you inserted C5 and C5-C18 were shifted.

Now please rename C5 capacitor to "C30" then check the list of capacitor designators on the design manager ("F3" to show/hide design manager, press "Sort components" button) – C5 and capacitors from C25 till C29 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, DipTrace removes all empty places in the designators array, therefore C6-C24 become C5-C23 and C30 becomes C24.

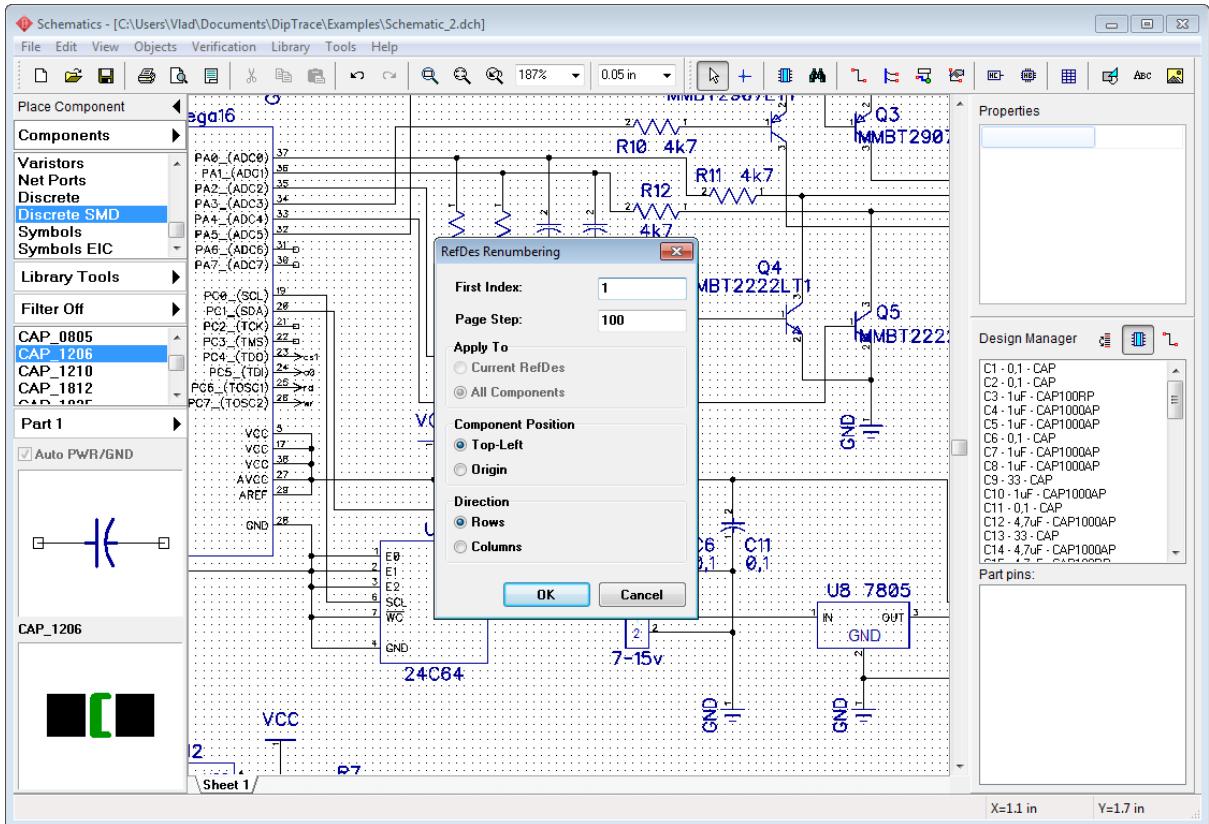


RefDes Renumbering

But what if we need to renumber circuits' Reference Designators in a very easy way that makes it simple to navigate through the design? Select "Tools \ RefDes Renumbering..." from main menu. Do not change 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 DipTrace is going to count components while renumbering. There are components of different sizes and shapes. If we choose "Top-left" in Component Position section of the dialog box, DipTrace will renumber components, based on the position of the top-left corner of each component. If you choose "Origin" it will use components origins.

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

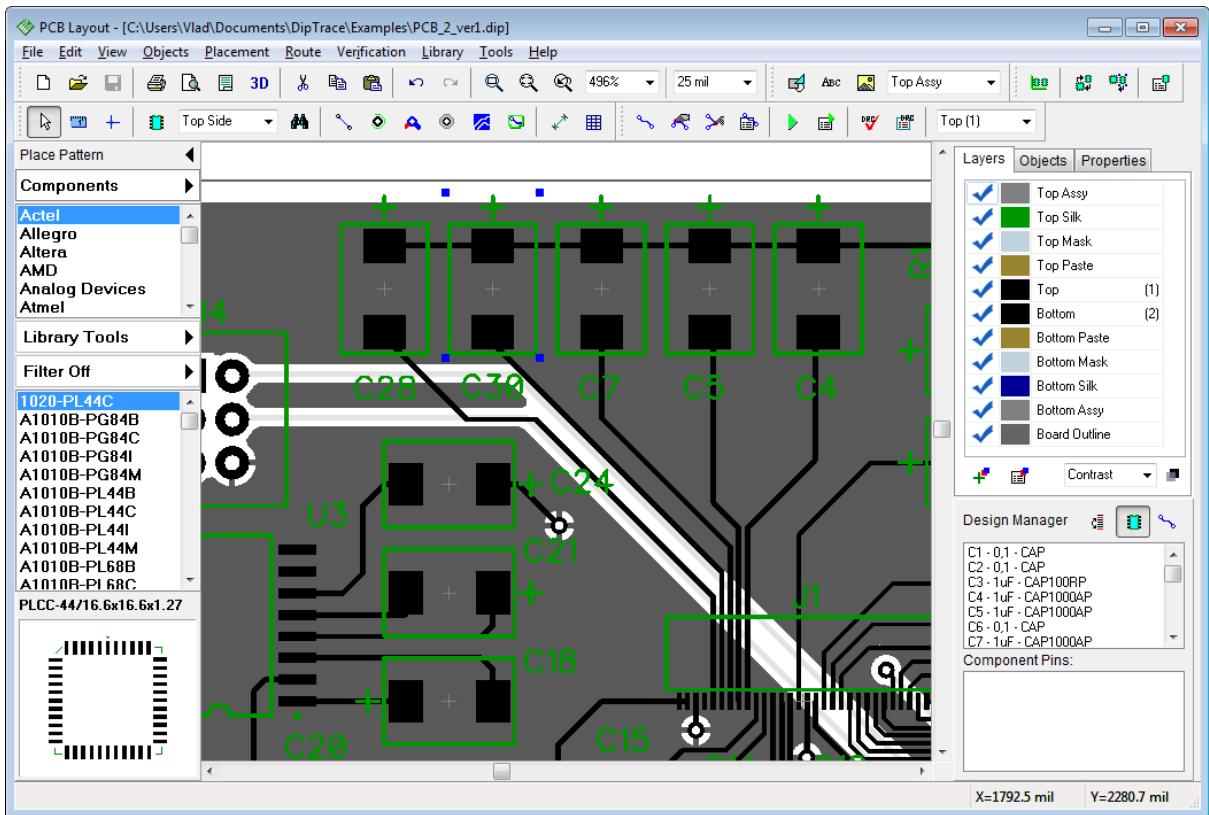
Press "OK" and all components will be renumbered.



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 schematic.

RefDes Renumbering works the same in PCB Layout.

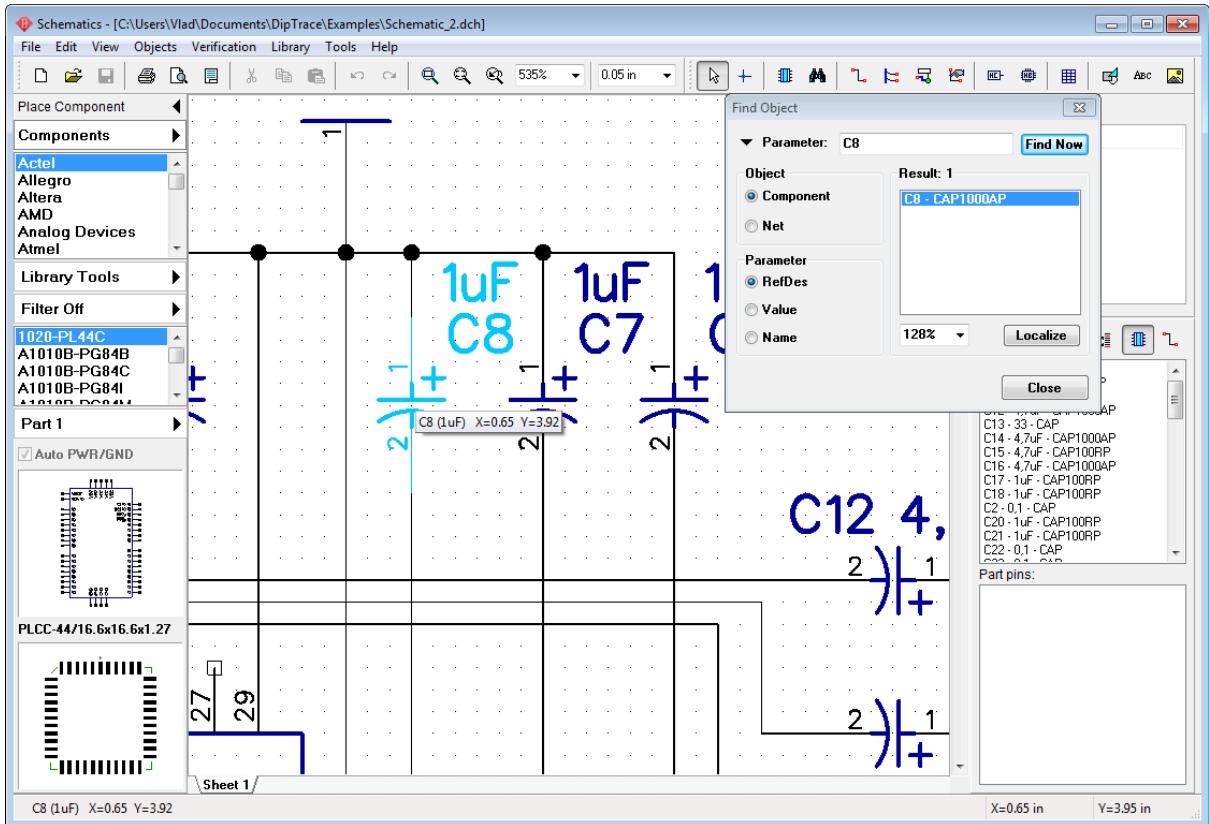
Please close Schematic **without** saving progress and run PCB Layout module. Then open "PCB_2" file from Examples folder. Use Design Manager to find C8 and C10 capacitors - double click on the component name in the list to localize and pan design area to selected component. Rename C8 and C10 capacitors to C28 and C30 (right click on the component and select first item). Select "File \ Save As" and save changed PCB in another file, for example "PCB_2_ver1".



Close PCB Layout and open Schematic Capture again (notice that you can open it directly from PCB Layout by selecting "Tools \ Schematic" from main menu, however we don't recommend to do this on Win 98/ME, if someone is still using these systems).

Back Annotate

Open Schematic_2.dch file and find C8 and C10, use Design Manager or press "Ctrl+F" (or select "Edit \ Find Object" from main menu). Type "C8" and press Enter to find it, C8 will be placed in the center of design area and highlighted.

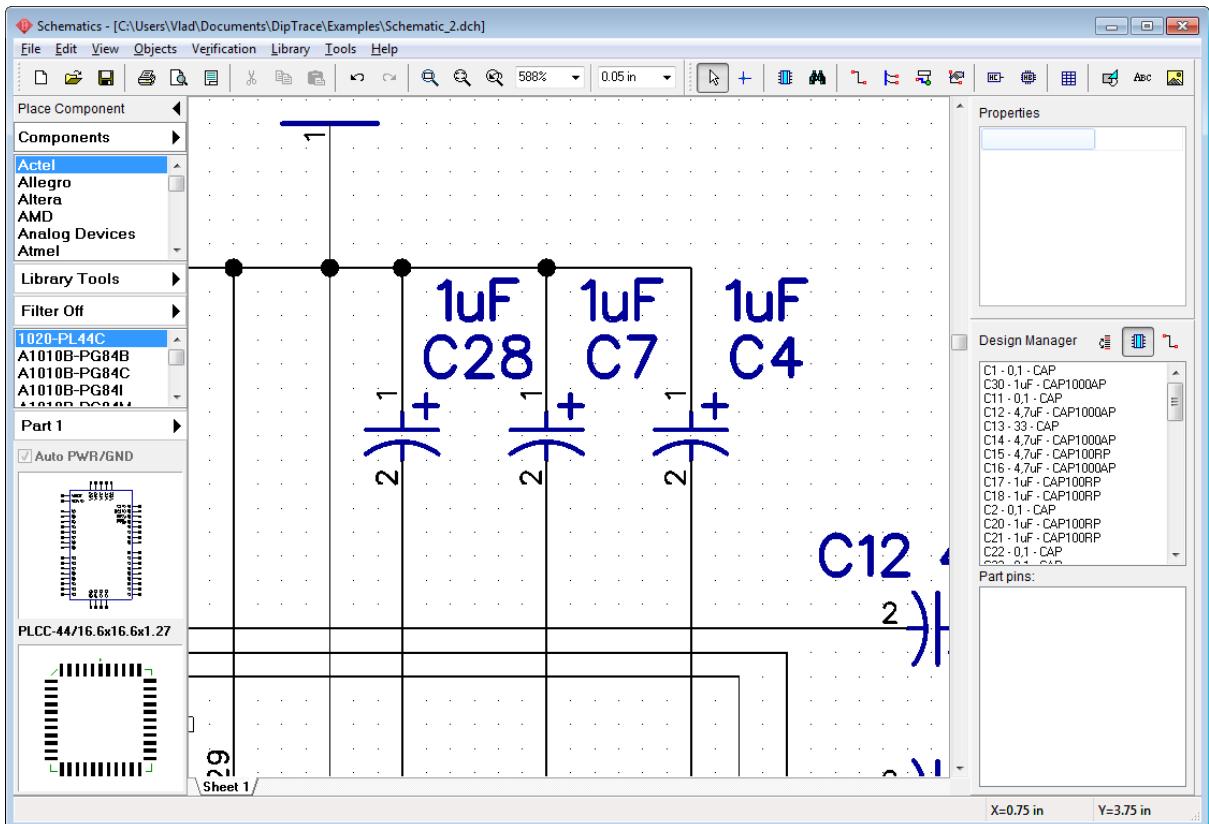


Notice that you can minimize "Find Object" window by clicking the arrow in its upper-left and use it while editing your design without showing all parameters.

Zoom In the schematic to see C8 and C10 better. PCB_2 is 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 main menu and select PCB file, where modified PCB_2 board was saved ("PCB_2_ver1" in this case). Press Open. Now you can see that all designators in Schematic (in our case C28 and C30) are changed according to PCB.

Notice that net names and net classes are also back annotated from PCB.

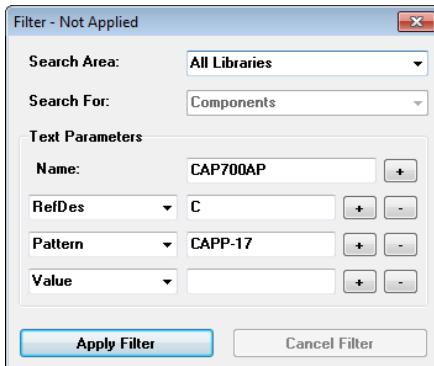


4.3 How to find components in libraries

DipTrace 2.4 includes 120,000+ components in standard libraries and we continuously expand them. Components are sorted to libraries 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 or press "Filter ON/OFF" button on the Library Manager panel to customize search filters.



In the pop-up dialog box specify search area: all libraries, current library group or current library and type in the name or part of the name of the component. DipTrace 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 drop-down list to select 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 dialog box. You will see that only filtered components, which correspond to search filters are visible in the component list of Library Manager Panel.

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

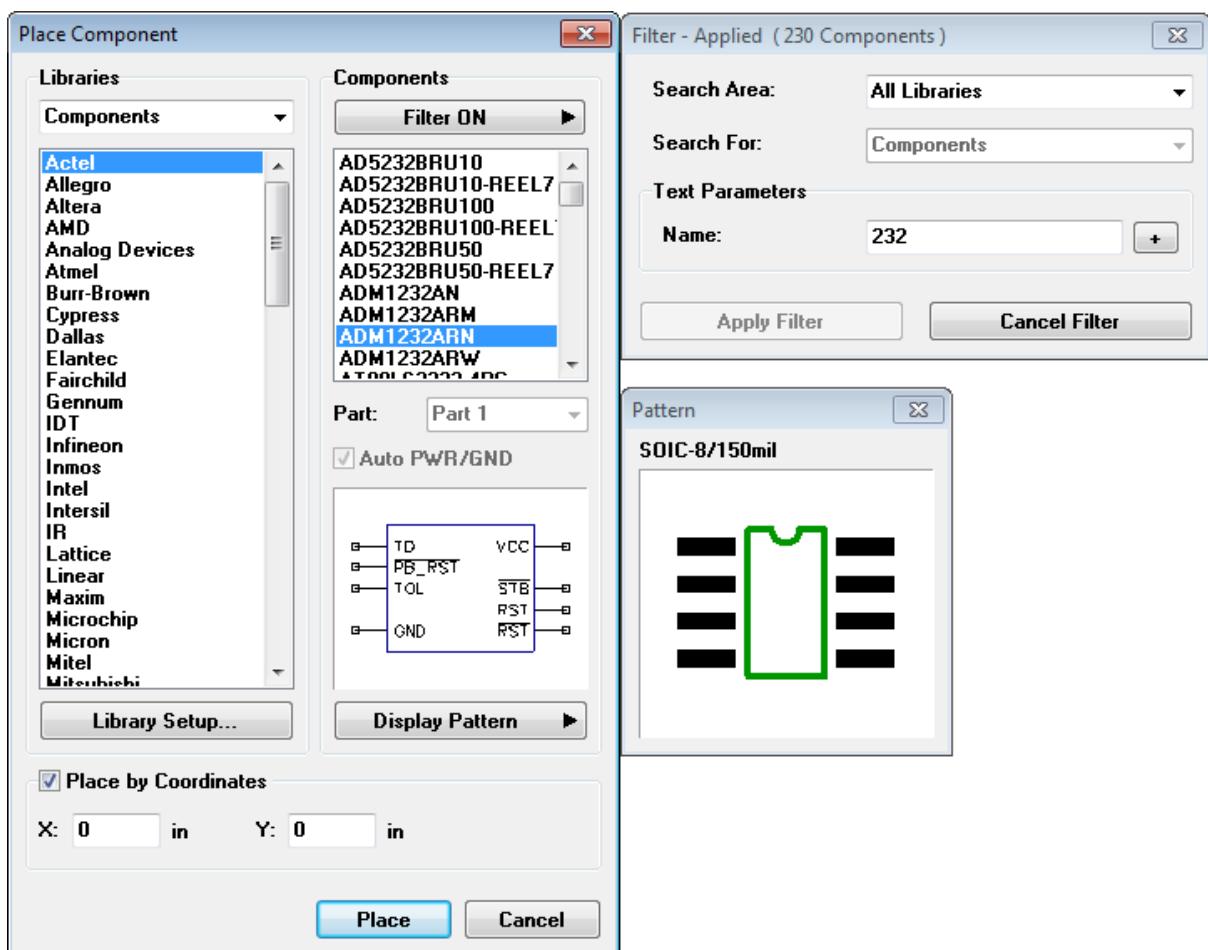
Place Component

DipTrace offers to find and place components with special Place Component dialog box ("Objects \ Place Component" from main menu). 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.

For example, we need component that contains "232" in its name, but we don't remember other characters, letters or even library that it belongs to. You can try to find it manually, select "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 of filters dialog box and press "Apply Filter". DipTrace has found 230 components with "232" in their name available in the list right below. Select component and you will see its 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" 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.



4.4 Electrical Rule Check

Electrical Rule Check (ERC) feature is one of the main verification features in DipTrace. 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 Examples folder in DipTrace Schematic. Define electrical rules, select "Verification \ Electrical Rule Setup" from 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 selected color) by clicking in the grid cells with green, yellow and red squares.

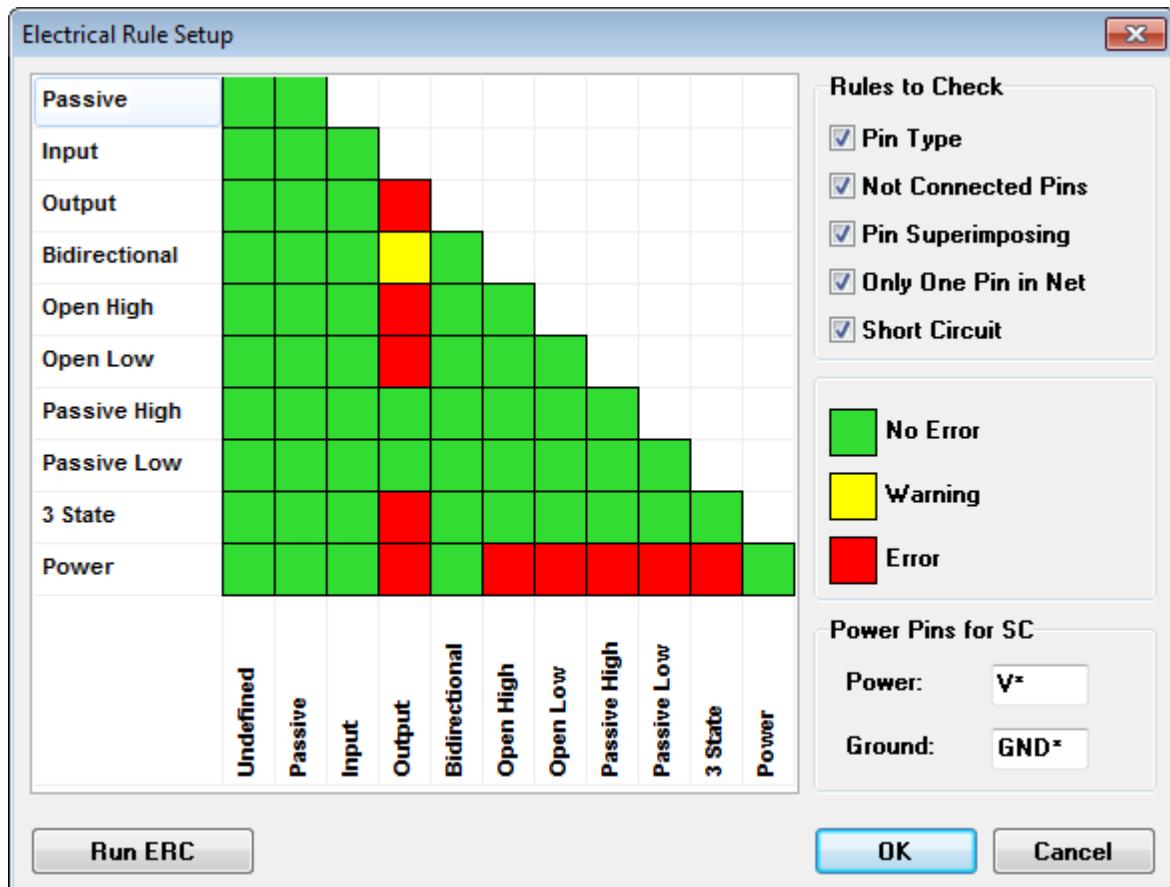
"Pin Type" item in "Rules to Check" box means checking pin-to-pin connections defined in the grid;

"Not Connected Pins" - program searches for unconnected pins;

"Pin Superimposing" - 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 net structure;

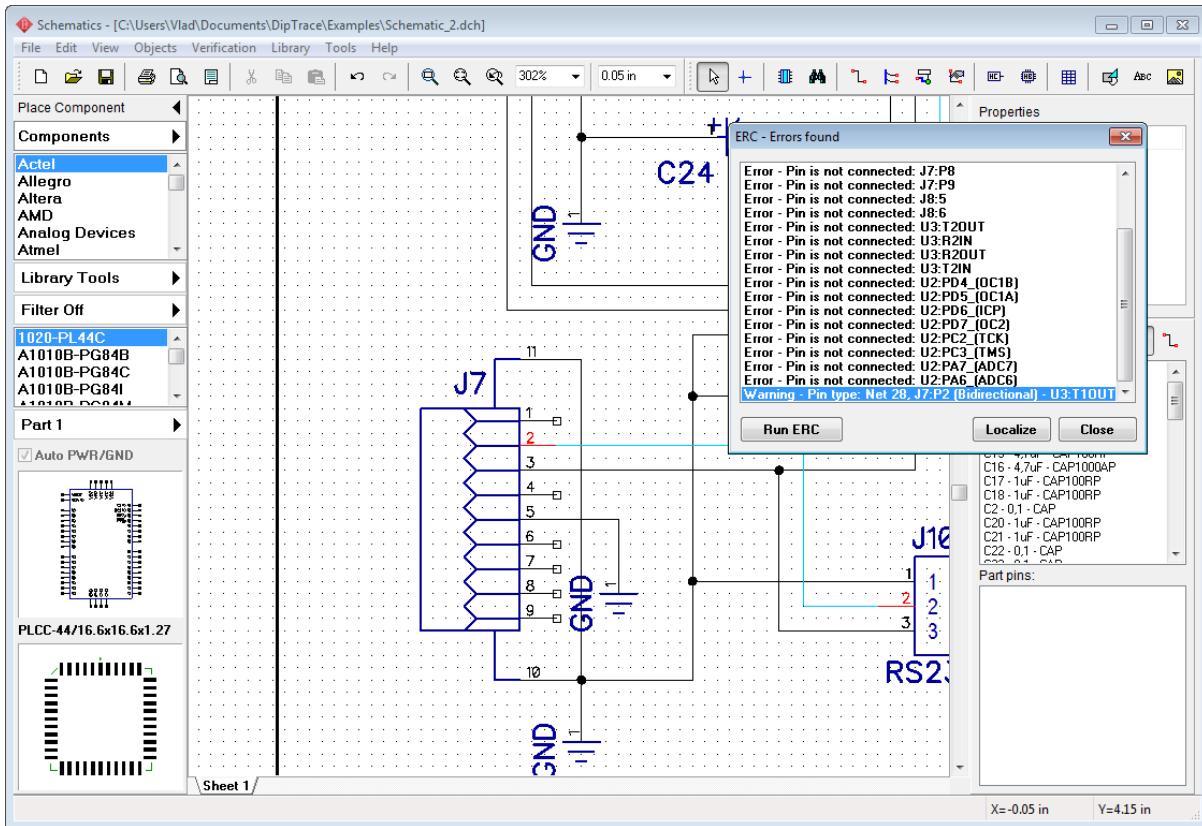
"Short Circuit" - program reports Power to GND connections. Define names for PWR and GND nets of the project in "Power Pins for SC" section below.



Press "OK" to close the dialog box.

Now select "Verification \ Electrical Rule Check (ERC)" from 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 schematic, double click it in the list (or press "Localize"). Errors can be corrected without closing ERC results window. 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 good.

4.5 Bill of Materials (BOM)

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

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

"Include Power/GND parts" option allows to include corresponding parts to BOM file;

"Include Net Ports" checkbox allows to include net ports into BOM table;

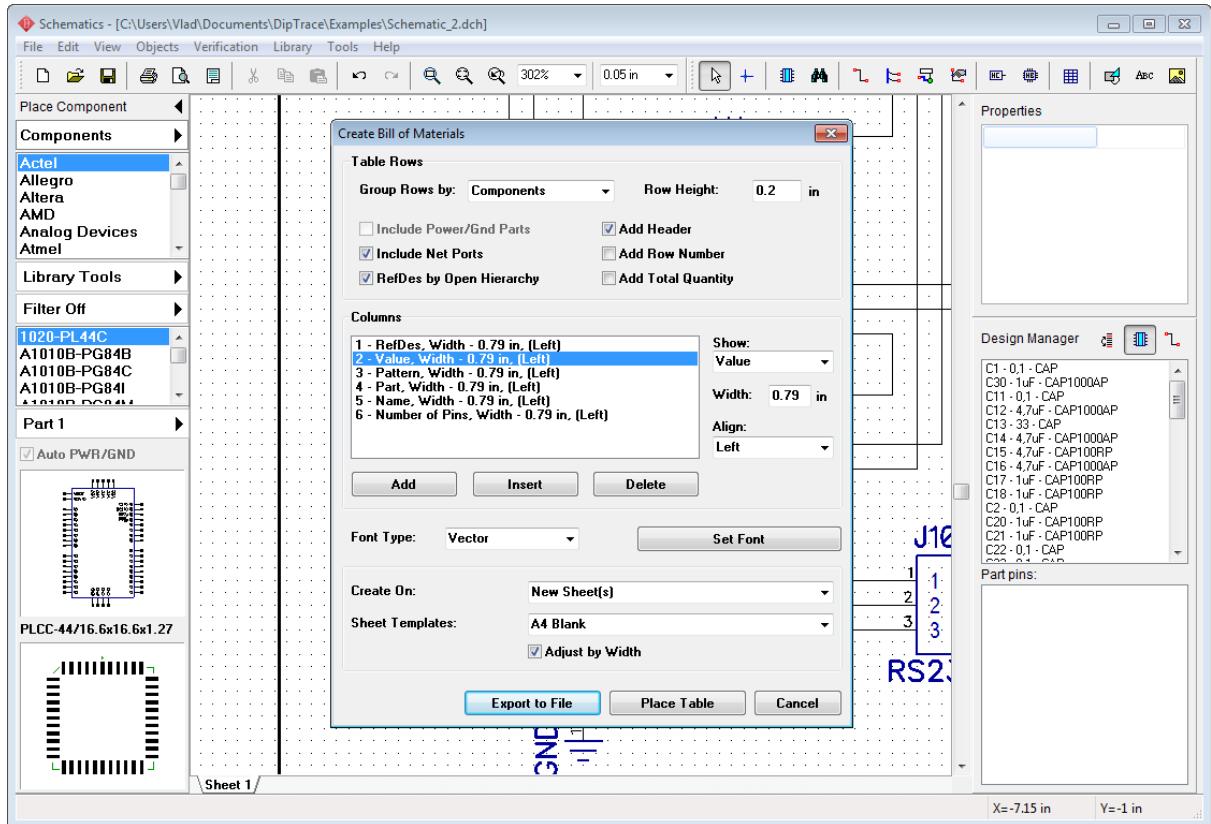
"RefDes by Open Hierarchy" allows to show RefDes with hierarchy block prefixes;

other check boxes allow to add header, row number and total quantity. Check them if you need.

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

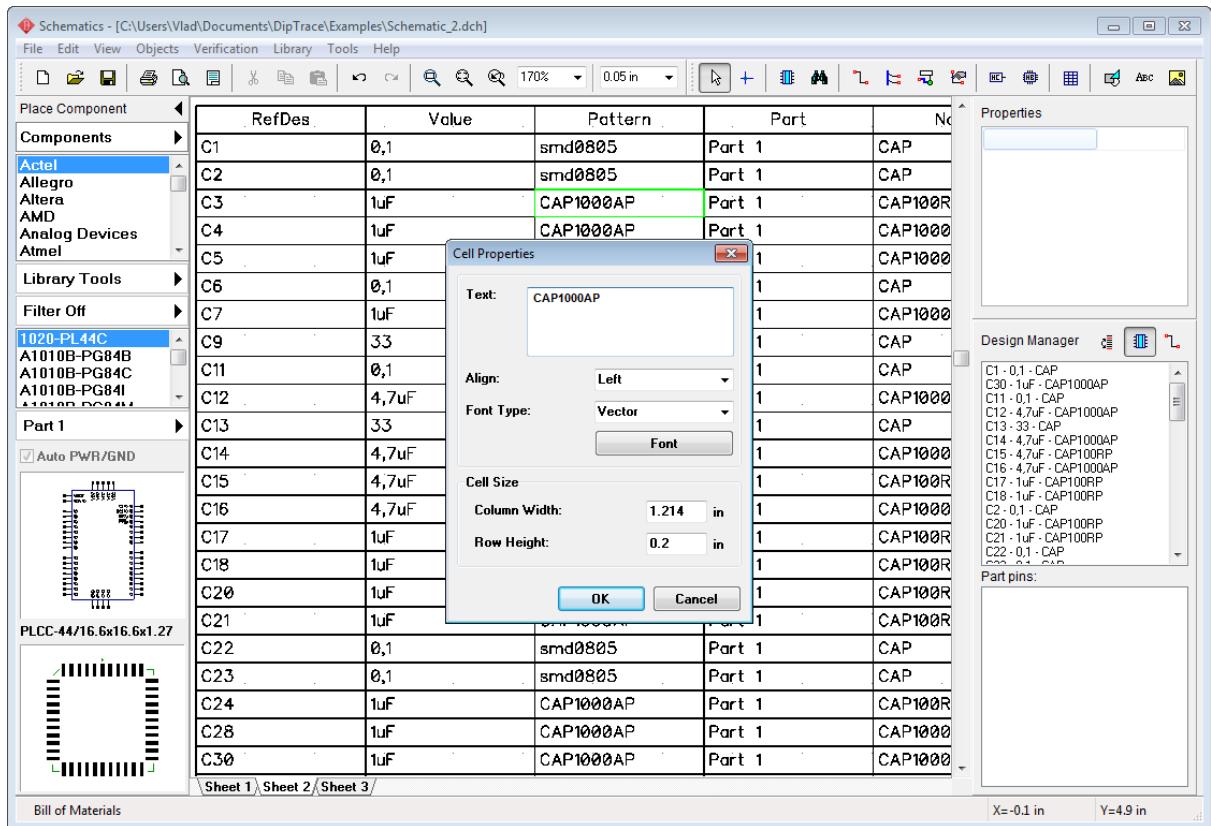
from the "Show:" drop-down list and press "Add" button. Rows can be deleted and edited. Press "Set Font" in order to customize font settings. Notice that TrueType font should be used for Unicode characters.

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



Press "Place Table" button to add new A4 sheet(s) with ISO title and BOM table to current project (several sheets for big projects or detailed BOMs). Do not forget to check "Display Titles" and "Display Sheet" in "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, e.t.c., 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 dialog box (left click on design area). If you have multi-sheet schematic with many components separate BOM tables for each schematic sheet are possible.

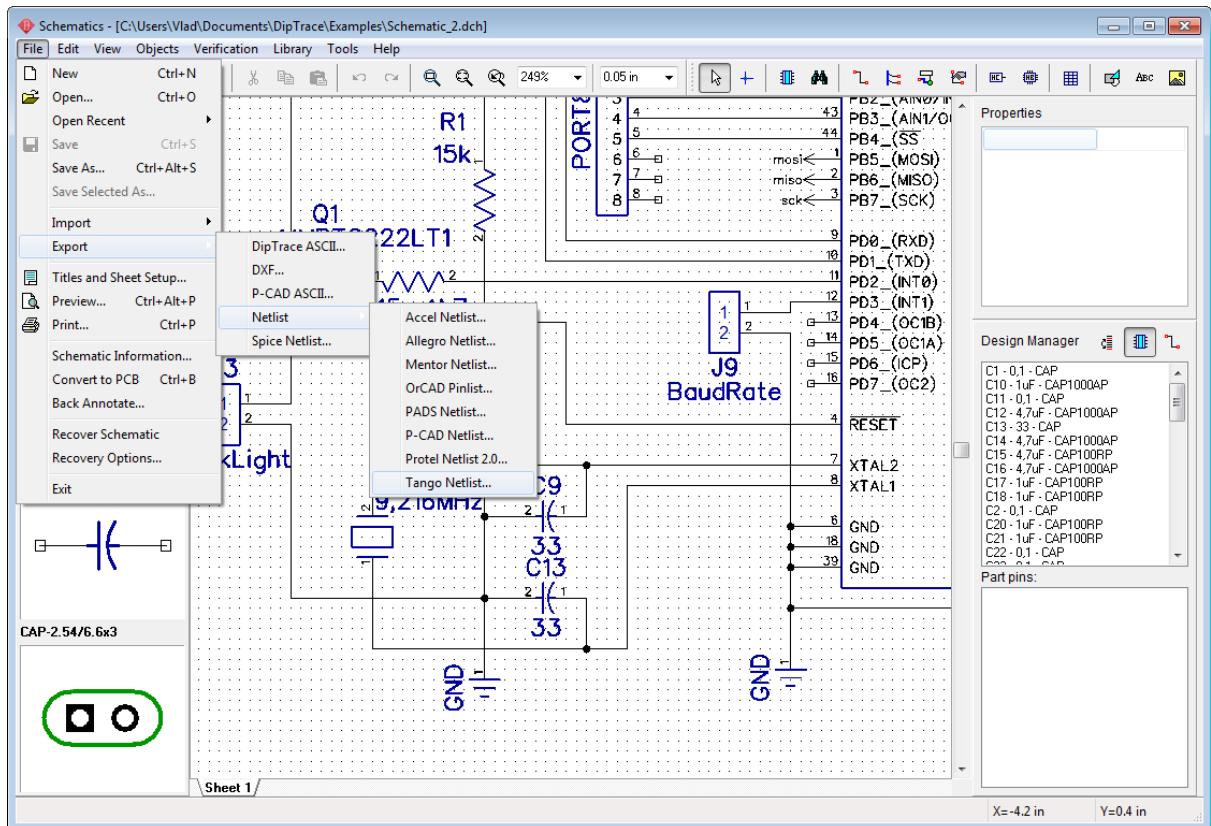
You can export BOM file directly from BOM dialog box or from placed table after editing it (for example, you need some columns that can not be inserted directly in BOM dialog box). Tables in Schematic and PCB Layout can be easily saved to CSV-file or text with formatting: right click on the existing table and select "Save to File" from the submenu.

4.6 Importing/Exporting netlists

DipTrace allows 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.

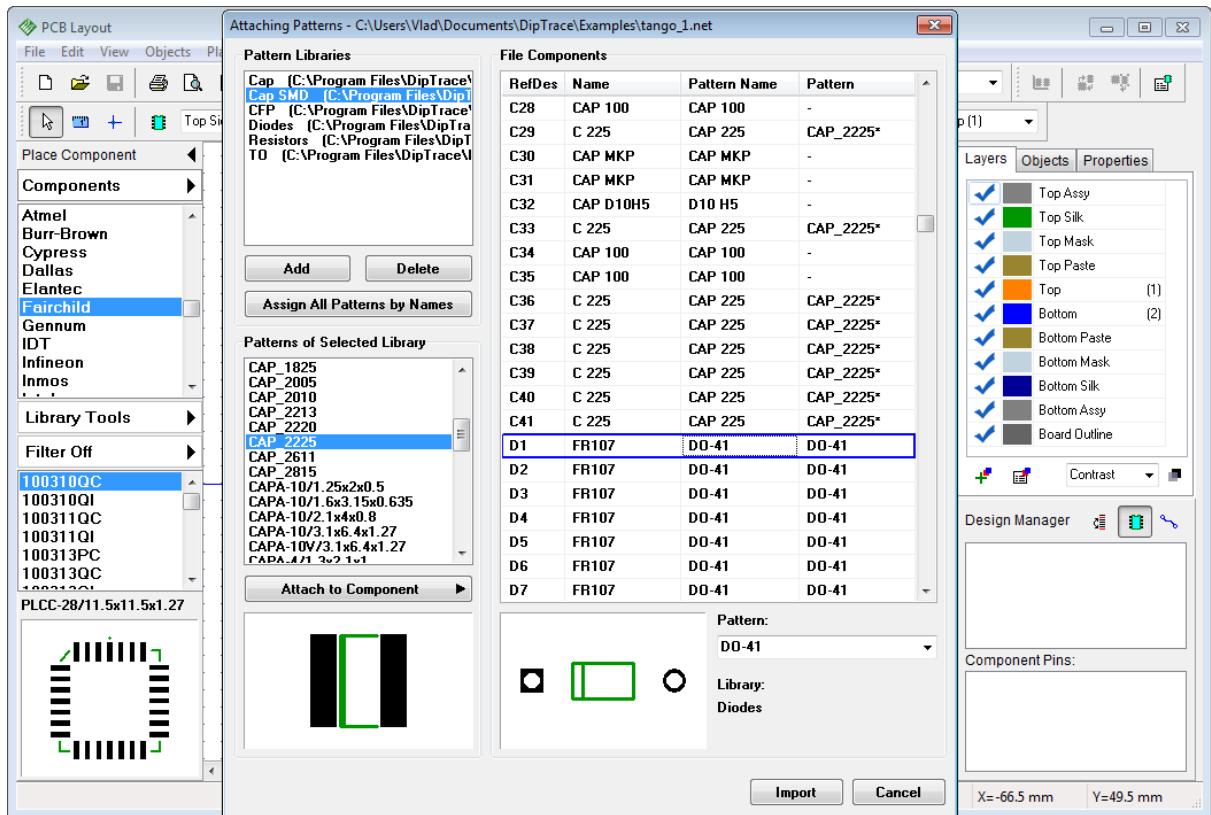
Export Netlist

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



Import Netlist

DipTrace PCB Layout allows to import netlists, created in other software. We will import Tango netlist as an example. Create new project and select "File \ Import \ Netlist \ Tango" from main menu then select "tango_1.net" file from "C:\Users\<UserName>\Documents\DiptTrace\Examples" folder and open it.



As you already know, all components are represented by patterns on the board.

The first step while importing netlist is to **make sure that each component has its pattern**. Check all patterns in File Components list. RefDes column shows component RefDes in the netlist, Name - component's name, Pattern Name - pattern's name from the netlist, Pattern - attached pattern in DipTrace. If it is blank - component does not have pattern. We should fix this.

Press "Add" button to add new pattern libraries, where we will get patterns that we need. For example "CAP_SMD" library for "C225" components. Add several libraries at a time with "Shift" and "Ctrl" buttons. DipTrace standard libraries are in "C:\Program Files\DiptTrace\Lib" folder and user libraries - in "My Libraries" folder by default.

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

Sometimes DipTrace is not able to find corresponding patterns, because of partially or completely different pattern names in DipTrace libraries and 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 respective lists. Press "Attach to Component" button which allows to attach patterns to component/s according to their RefDes, name or pattern name. For example, we have added "CAP_2225" pattern to all C225 components.

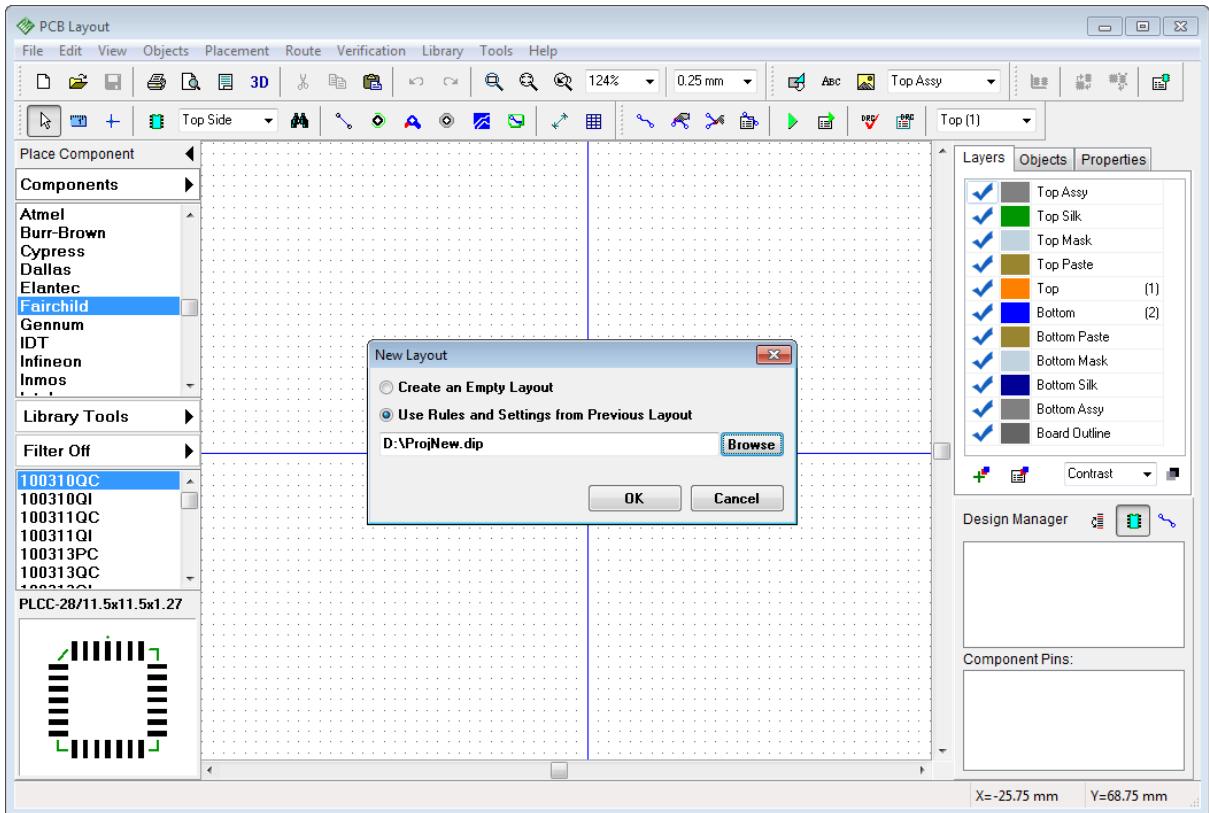
"*" symbol after pattern name in the list means that this pattern has been connected manually.

Click "Import" button when all components have correct patterns.

4.7 Saving/Loading Design Rules

In "Converting to PCB"²⁷ section of this tutorial we mentioned that you can use Schematic rules or load rules from any PCB layout project, while converting circuit to PCB. No need to specify all layers, net classes, via styles and verification settings.

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



Check "Use Settings from Previous Layout", press "Browse" and select *.dip file of the project, which contains layers, via styles, net classes and design rule settings that you may need. Press "OK". In our case we have selected *.dip file of tutorial project, hence only layer colors and some DRC settings had been changed.

In DipTrace you can save settings separately from PCB layout project in special file. Just go to "Route \ Save Rules", enter file name and press "Save". Rules and settings from this file can be used while creating new project or loaded later - go to "Route \ Load Rules" and choose *.dip or *.rul file.

Please add one via style and net class with random parameters and save it as *.rul file. We will use it later.

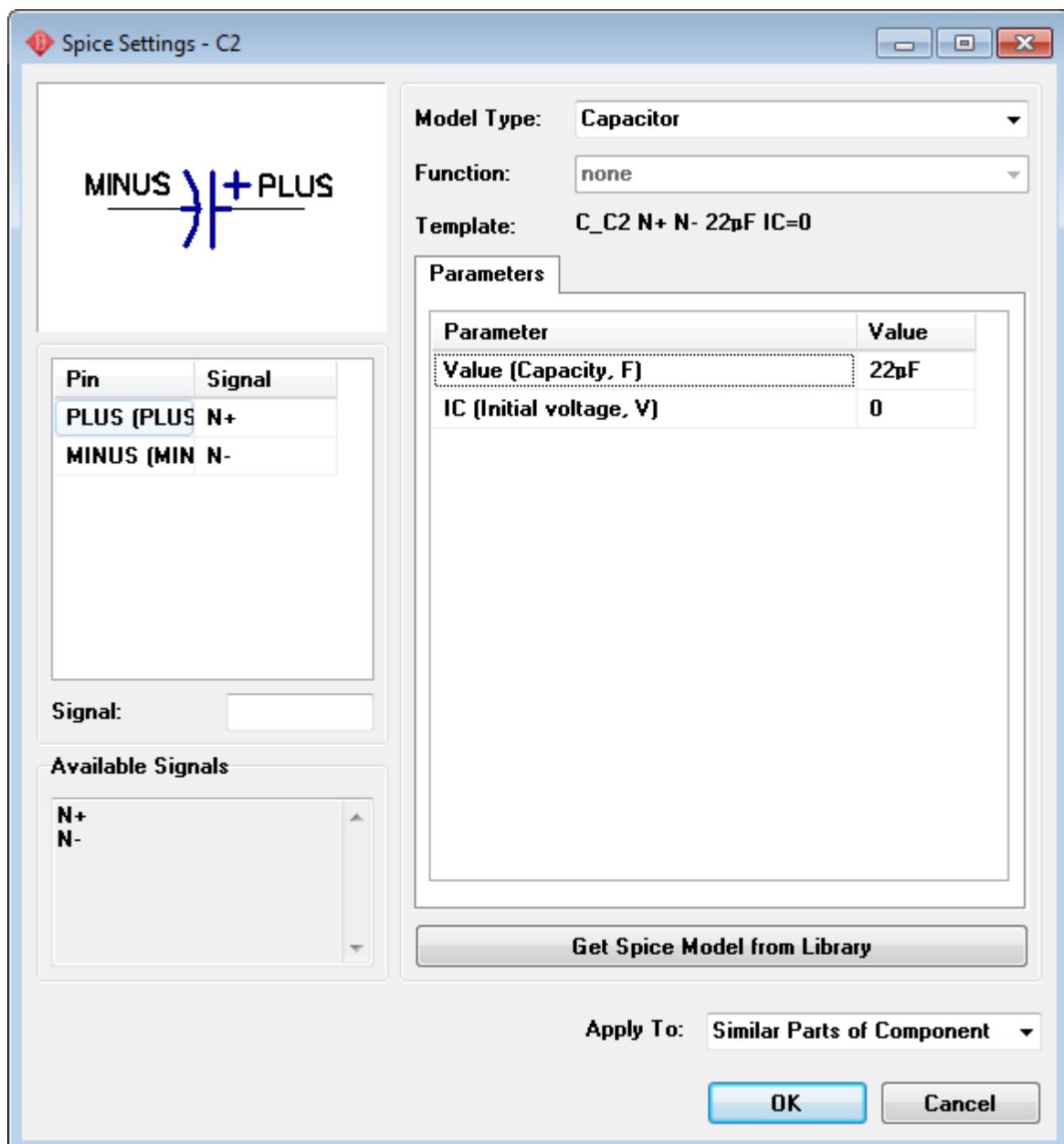
4.8 Spice simulation

There is no built-in Spice simulator in DipTrace, but program allows to define spice settings and export spice netlist to any third-party simulation software. We will try to simulate astable flip-flop schematic⁶, using LT Spice, which is free and of a good quality. However, if you have another program, you can use it.

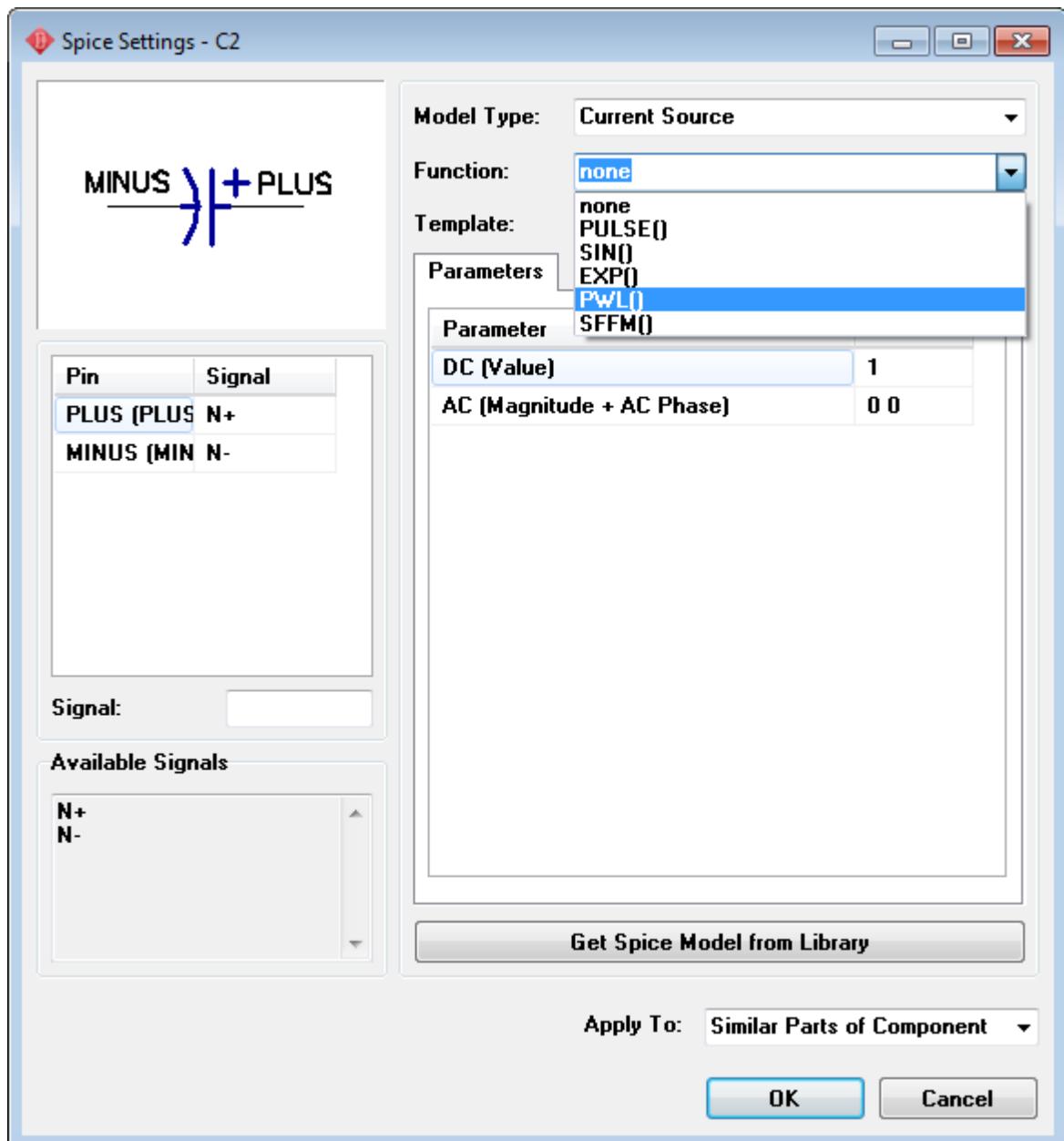
Edit Spice settings

Launch Schematic and open "C:

`\Users\<UserName>\Documents\DiptTrace\Examples\Spice\Astable_Flip_Flop_Spice.dch`". We have already defined all spice settings for this circuit. However, let's review couple of parts just to learn how to work with it. Right click on C2 capacitor and select "Spice Settings" from the submenu. Spice settings for capacitor are pretty simple: you should select "Model Type : Capacitor", enter values into parameters table (in our case "22 μ F") and specify positive and negative pins (enter values into pin-to-signal table on the left side of the dialog box, list of available signals is right below). Notice that you can enter parameters directly into table cells. Template field shows how component is represented in Spice netlist. You can scroll that field to the right if it is long. Make sure settings are like on the picture below.



Select another model type (for example, "Current Source"). For this model type function can be specified, use "Function" drop-down list and select "PWL".

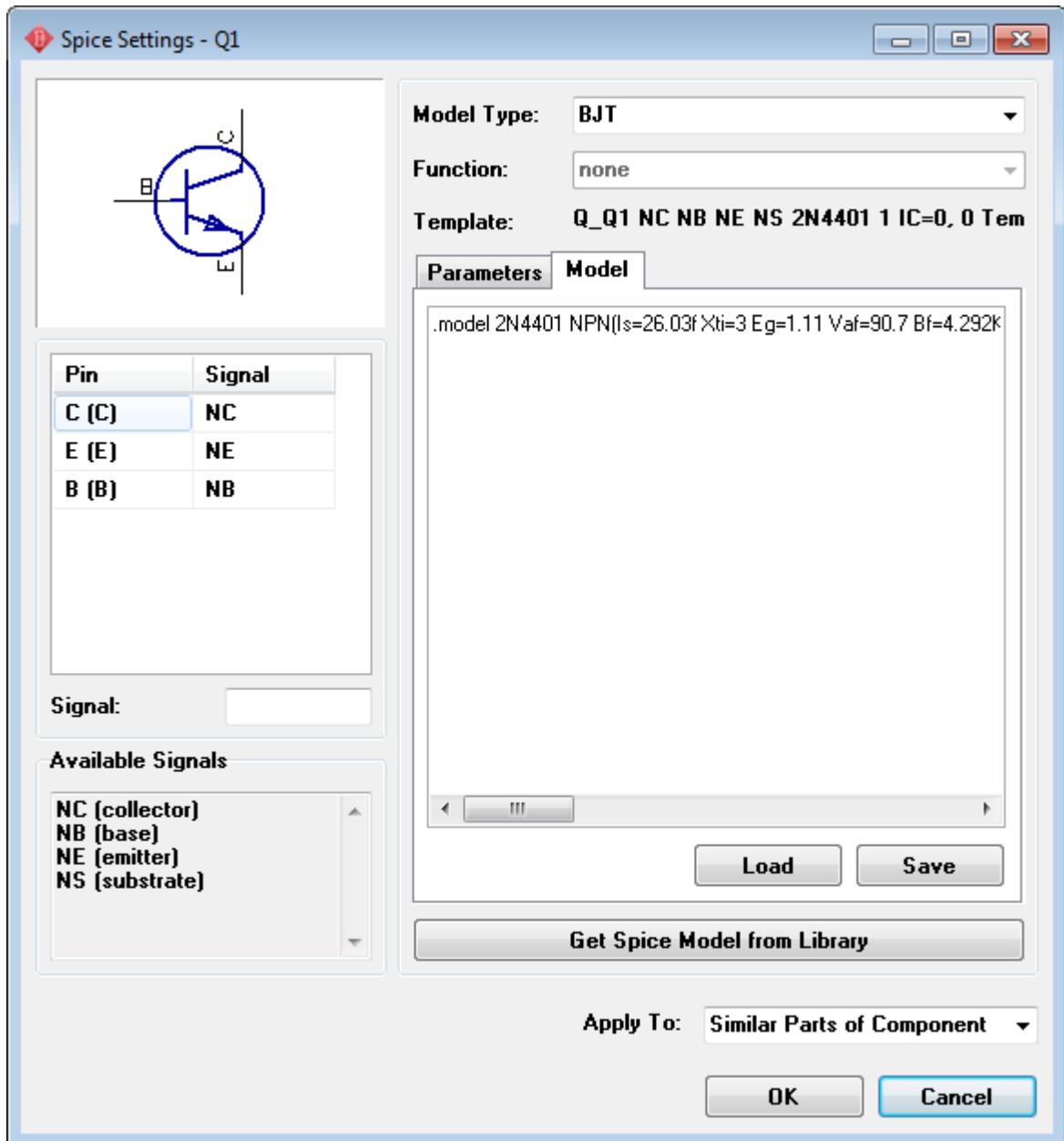


In the pop-up dialog box enter number of points for PWL function and click "OK". Now you can enter values for each point in Parameters table. Different functions require different parameters (amplitude, phase, e.t.c.). See detailed description in Spice language documentation.

Now return back to Capacitor model type, define its value (**discard any changes**) and click "OK".

Spice Model

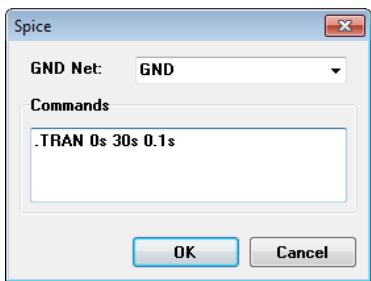
Capacitors do not require additional model description. Hence, right click on Q1 transistor and select Spice settings. In the pop-up dialog box you can see that there is a "Model" tab near "Parameters", select it. Now you can enter model text or load model from external file ("Load" and "Save" buttons).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



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

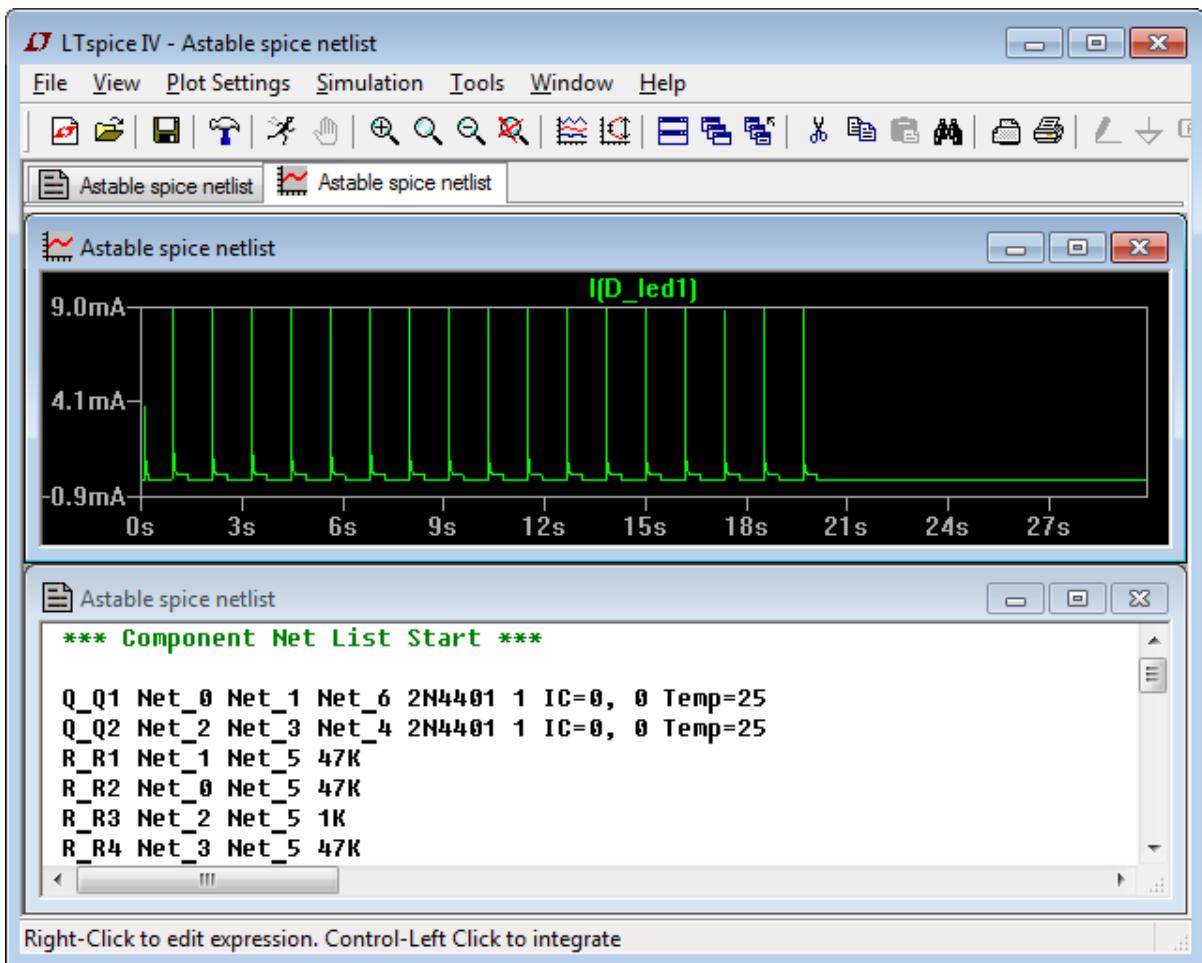
There is no valid spice model for power source (battery) in `Astable_Flip_Flop_Spice.dch`, we should define it. Right click on B1 component and select Spice Settings from the submenu. You can see that component has Voltage Source model type but no valid function. Please select: 'Function : Pulse' specify following parameters in the table below: "Pulse V2=5", "Pulse PW=20s", "Pulse PER=30s". Click "OK". Now we have voltage source that produces 5V during first 20s, then 10s interval. Now everything is ready for simulation.

Export Spice Netlist



Select "File \ Export \ Spice Netlist" from 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"- this means simulate circuit from 0s to 30s with 0.1s step. Notice that you can define/change commands directly in simulation software (for example, LT Spice). Click "OK" and save *.cir file. Launch Spice Simulator that you have. We will use LTspice as an example (download at [Linear Technologies](#) website).

Select "File \ Open" and open *.cir netlist you just saved (notice that you should select correct "Files of Type"). You can see netlist in text format. Select "Simulate \ Run" and close error log window. Select "Plot Settings \ Visible Traces" and choose I(D_led1) to see electrical current on LED1. This component works during first 20 seconds then has 10 sec interval. Select other signals to see how they work.



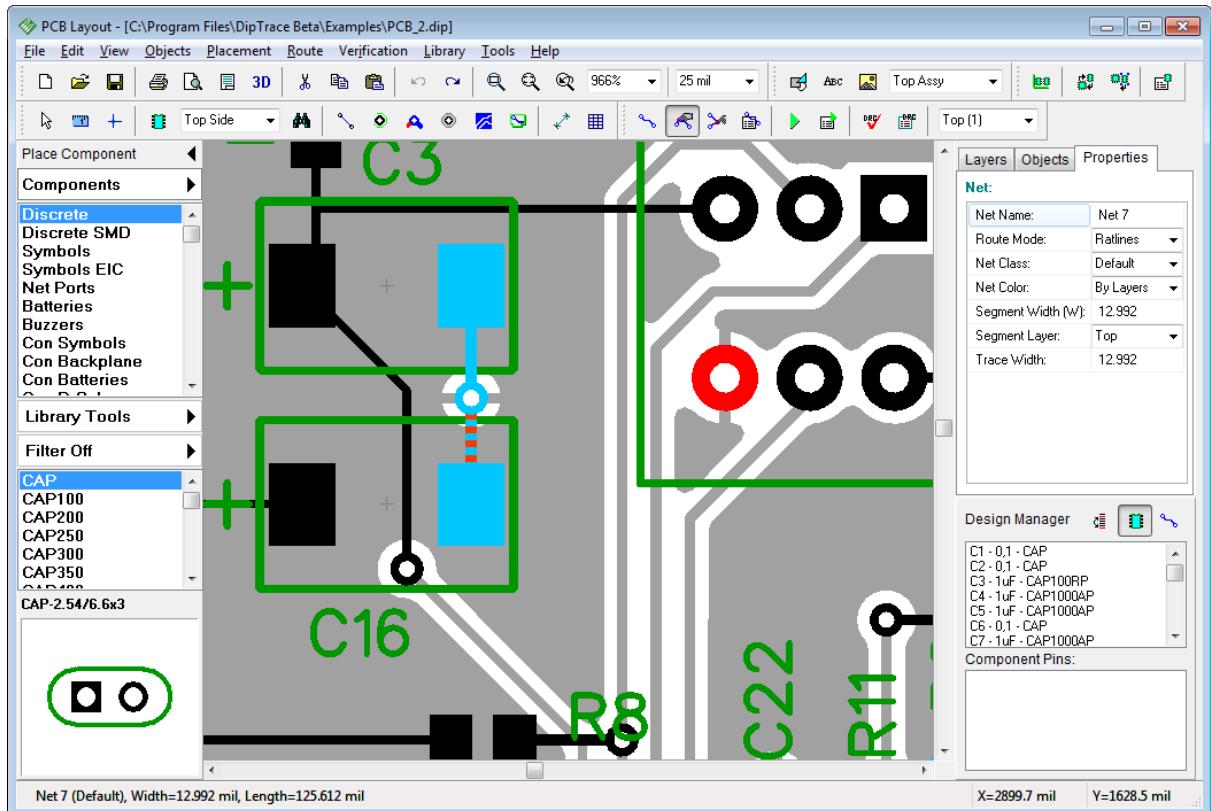
4.9 Checking net connectivity

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

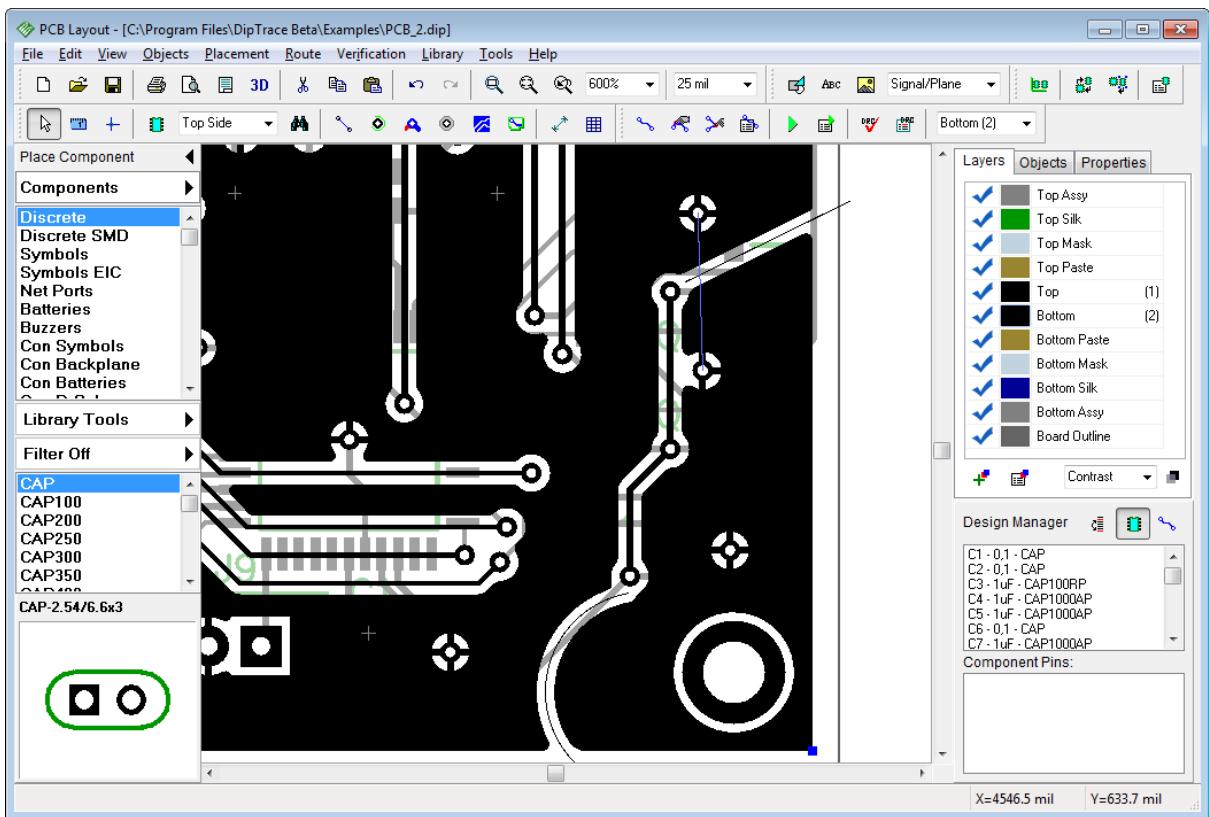
Launch PCB Layout module and open "PCB_2.dip file" from "C:\Users\<UserName>\Documents\DiptTrace\Examples". Select "Verification \ Check Net Connectivity" from main menu. In the pop-up dialog box you define objects that will be considered as connectors by connectivity verification, typically it is recommended to keep all boxes checked. Press "OK".

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

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

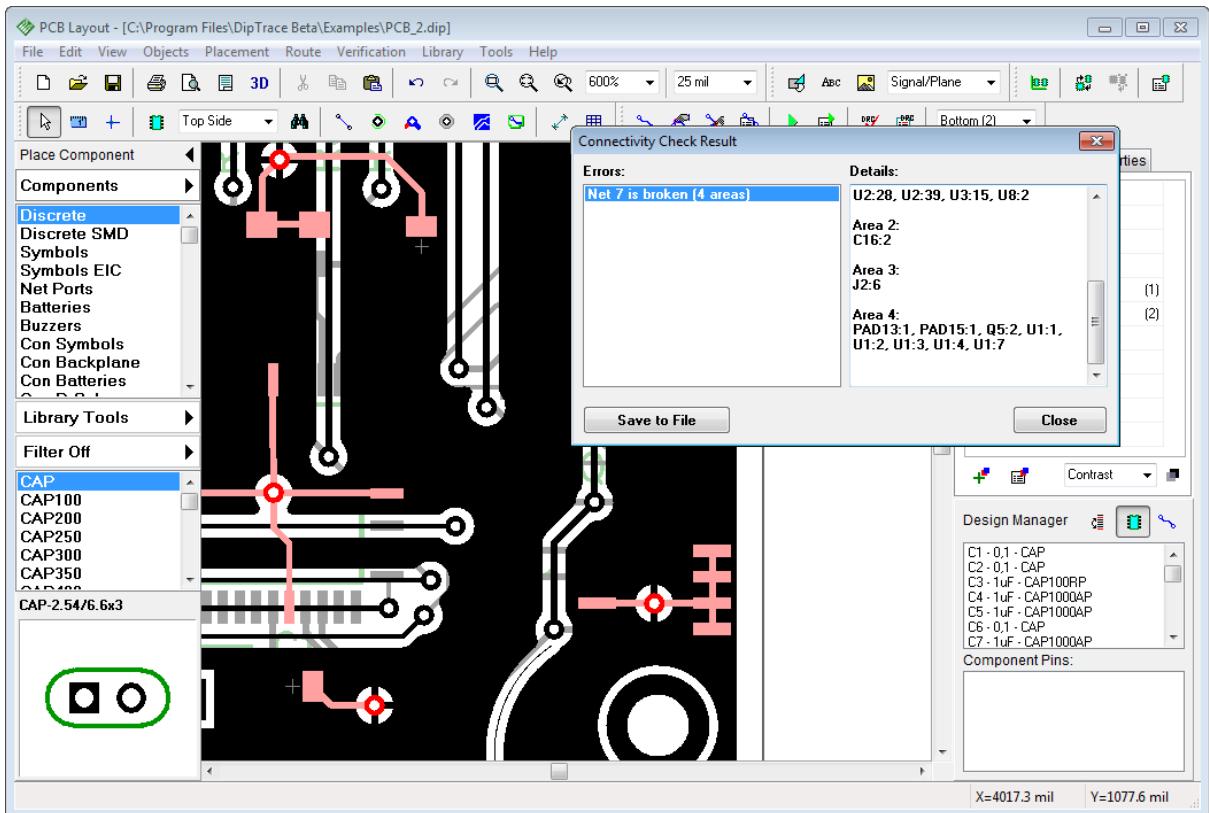


Isolated copper pour area is the second error that we are about to commit. Double click on Bottom layer, pan to bottom-right corner of the design. Place a couple of shapes (arcs or lines) on signal layer to isolate one of the vias and don't forget to update copper pour (right click on copper pour edge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



This is a simple situation when a mistake can be found visually. But very often 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 Net 7 as a broken to 4 unconnected areas. The first area is copper pour and all pins connected, second and third appeared as result of unrouting trace from pad 2 of C16 capacitor (our first error) and the fourth is isolated copper pour at the bottom-right of the board.



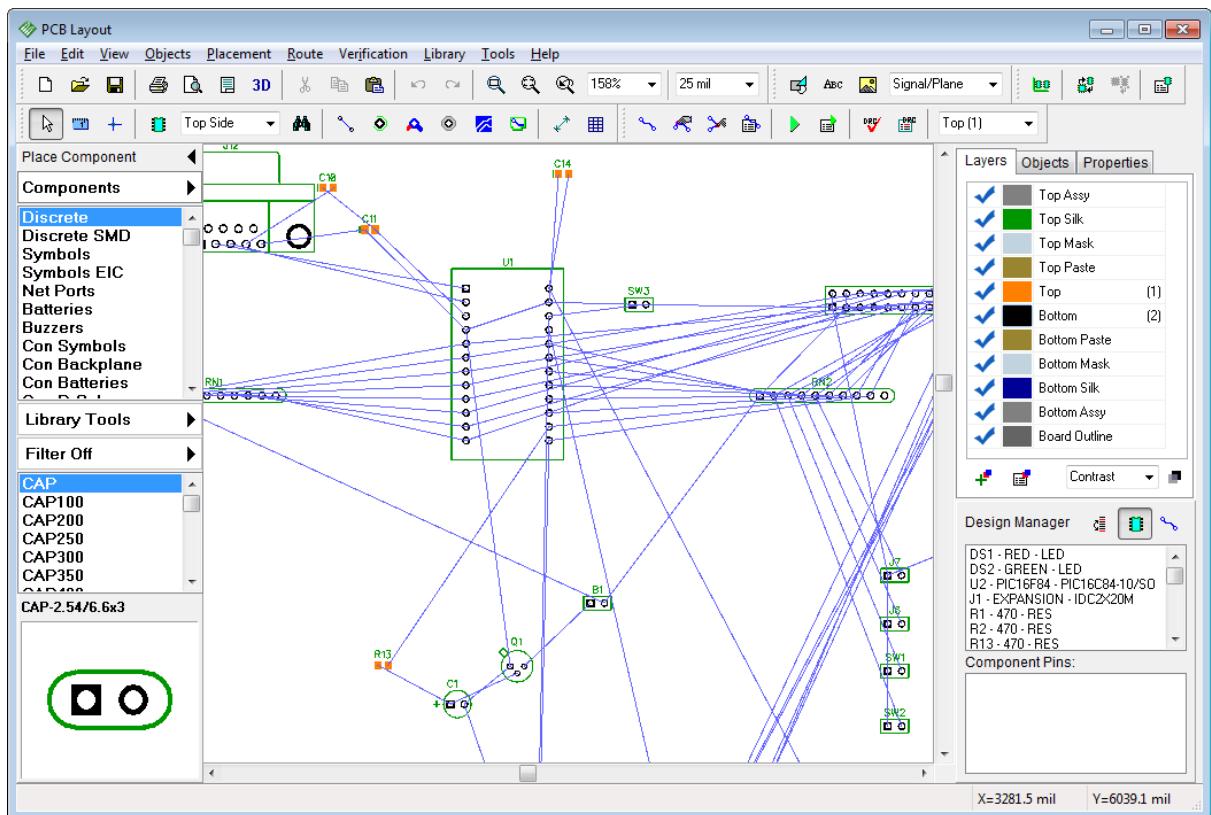
To make further corrections you don't have to close error report panel. Verification results can be saved into text file.

4.10 Placement and Autorouting

DipTrace has advanced placement features and integrated auto-placer. This makes placement and layout optimization much easier.

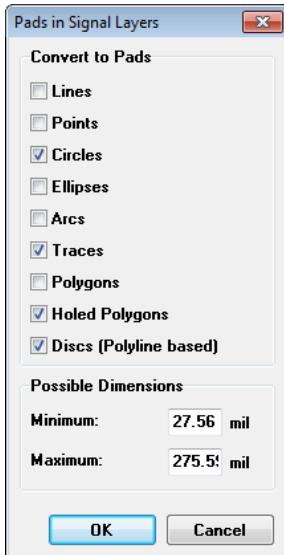
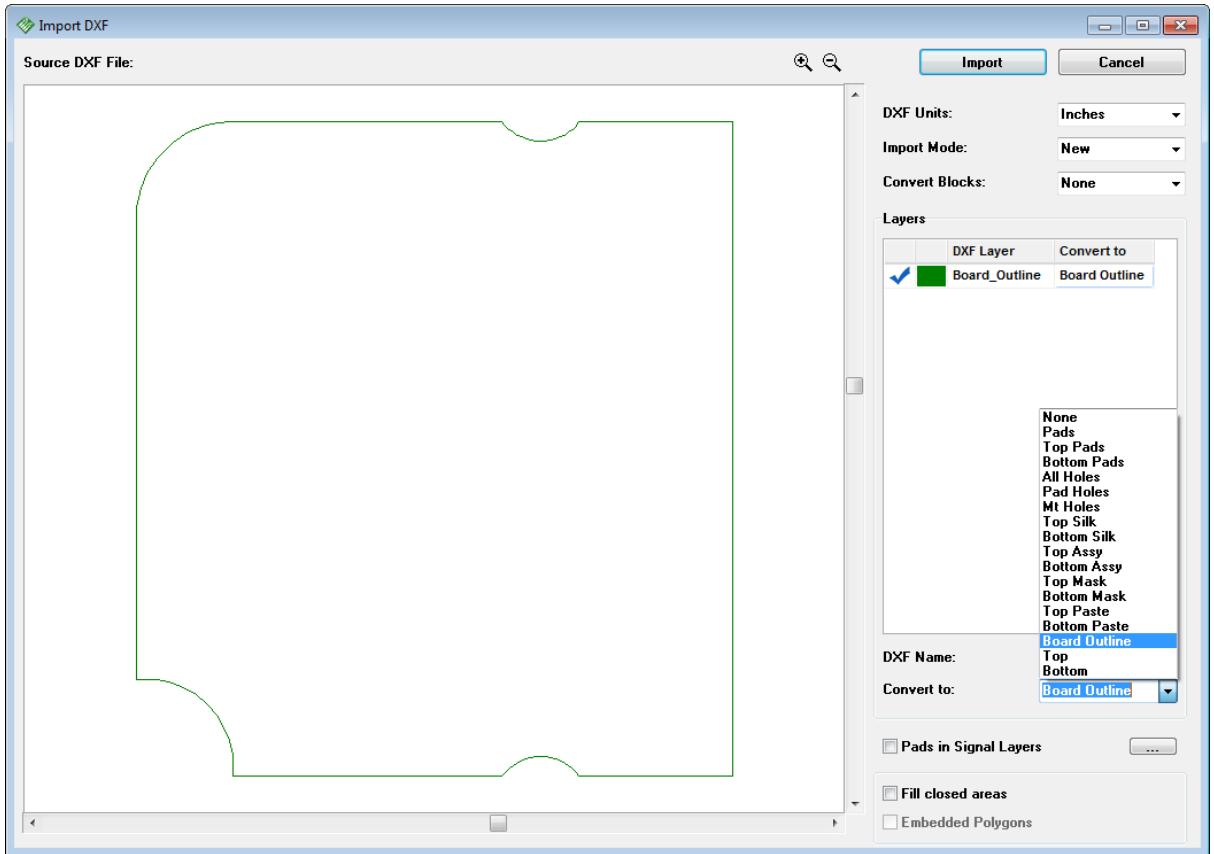
Launch PCB Layout module, select 'File \ Open' and select "C:

\Users\<UserName>\Documents\DiptTrace\Examples\Schematic_4.dch" and use schematic rules for this layout. You will get something like on the picture below. Layout is chaotic, picking components manually would be a time-waste. Automatic arrangement should be applied.



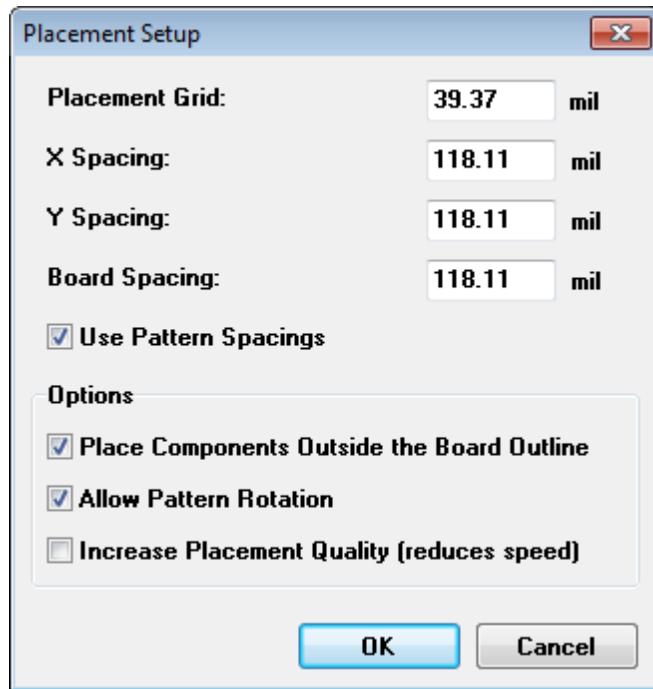
DXF Import

We will import board outline from DXF file. Select "File \ Import \ DXF" from main menu and open "C:\Users\<UserName>\Documents\DiptTrace\Examples\outline.dxf" file. In the pop-up dialog box you can see DXF file that will become board outline. Select "Board Outline" DXF layer and specify "Convert to: Board Outline" in a drop-down list below.

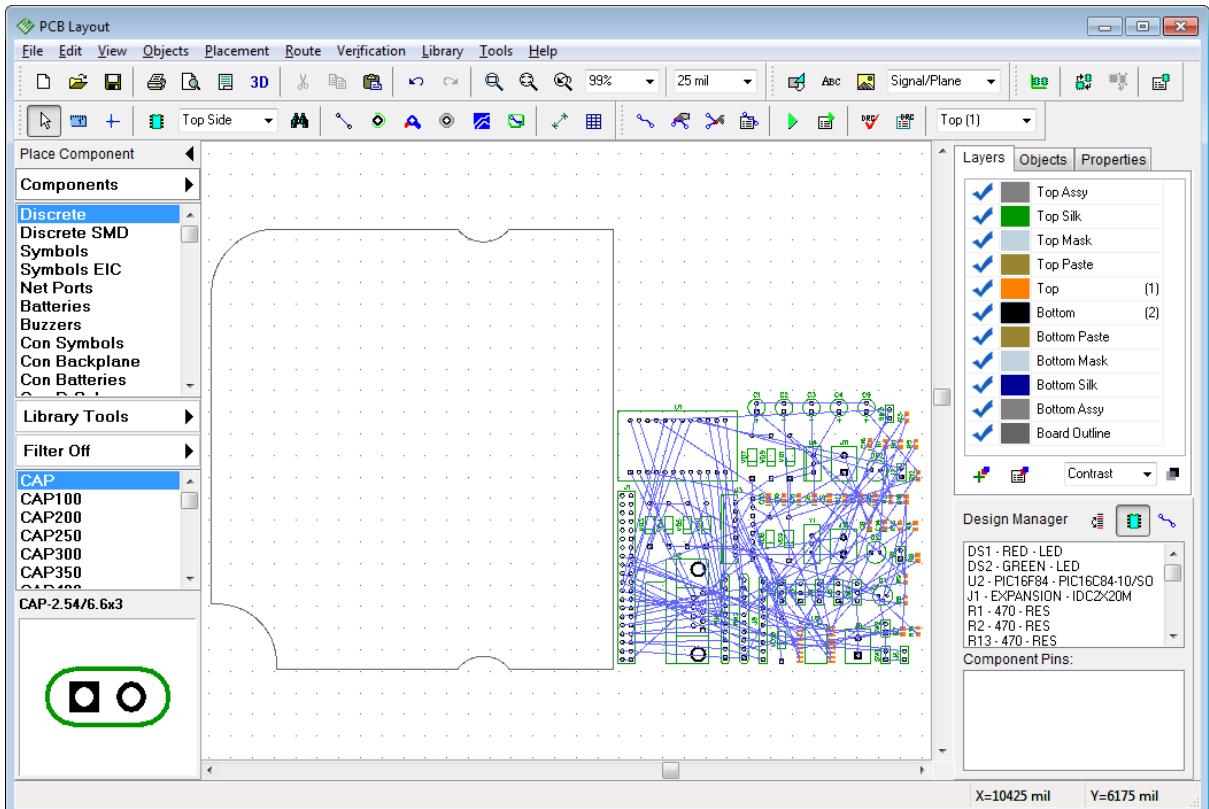


When importing component's drawing or design from DXF you can "Fill closed areas" and cut holes there using "Embedded polygons" (usually DXF designs are made from outlines without fills). This feature works for copper and mask/paste layers only. If you're importing pads in signal layer, you can check 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 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 main menu). Click "OK" to apply changes and press "Arrange Components" button on the placement toolbar or select "Placement \ Arrange Components" from main menu.



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

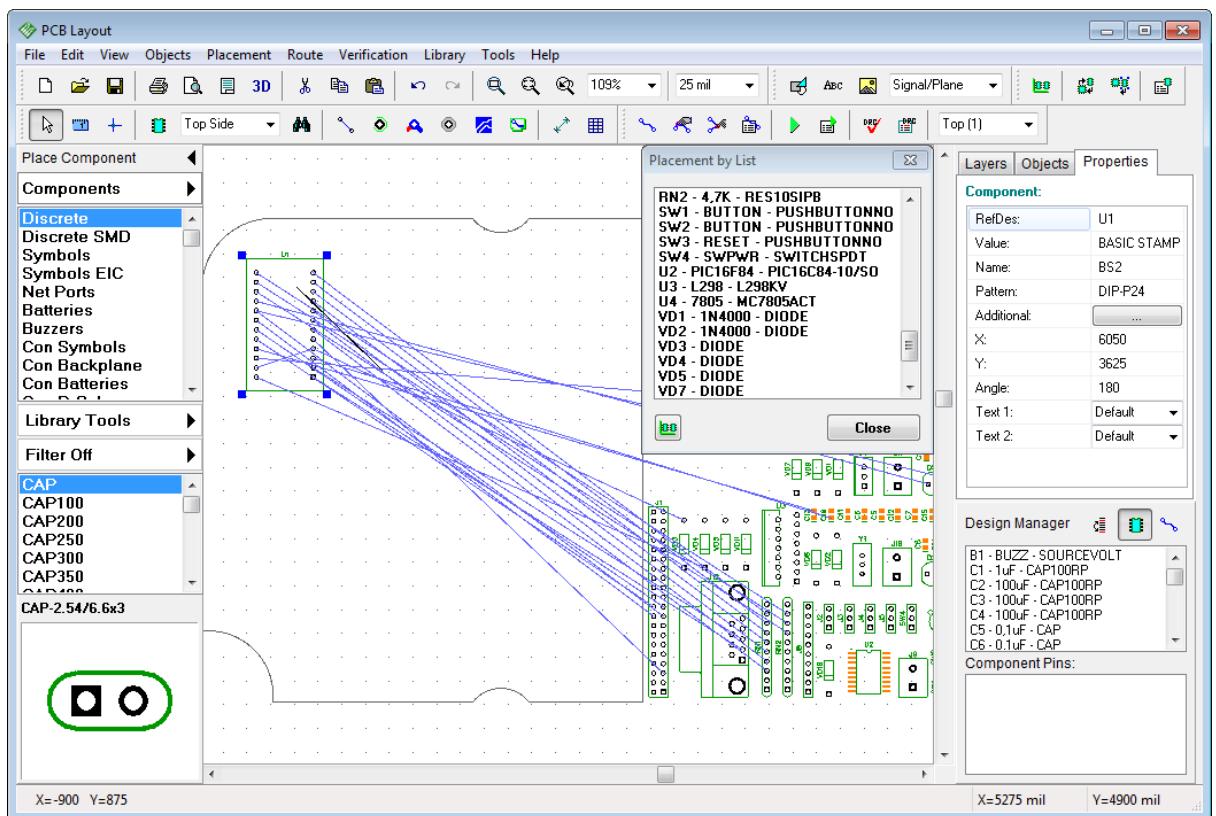
Notice that "Arrange components" feature isn't the same as "Auto-placement".

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

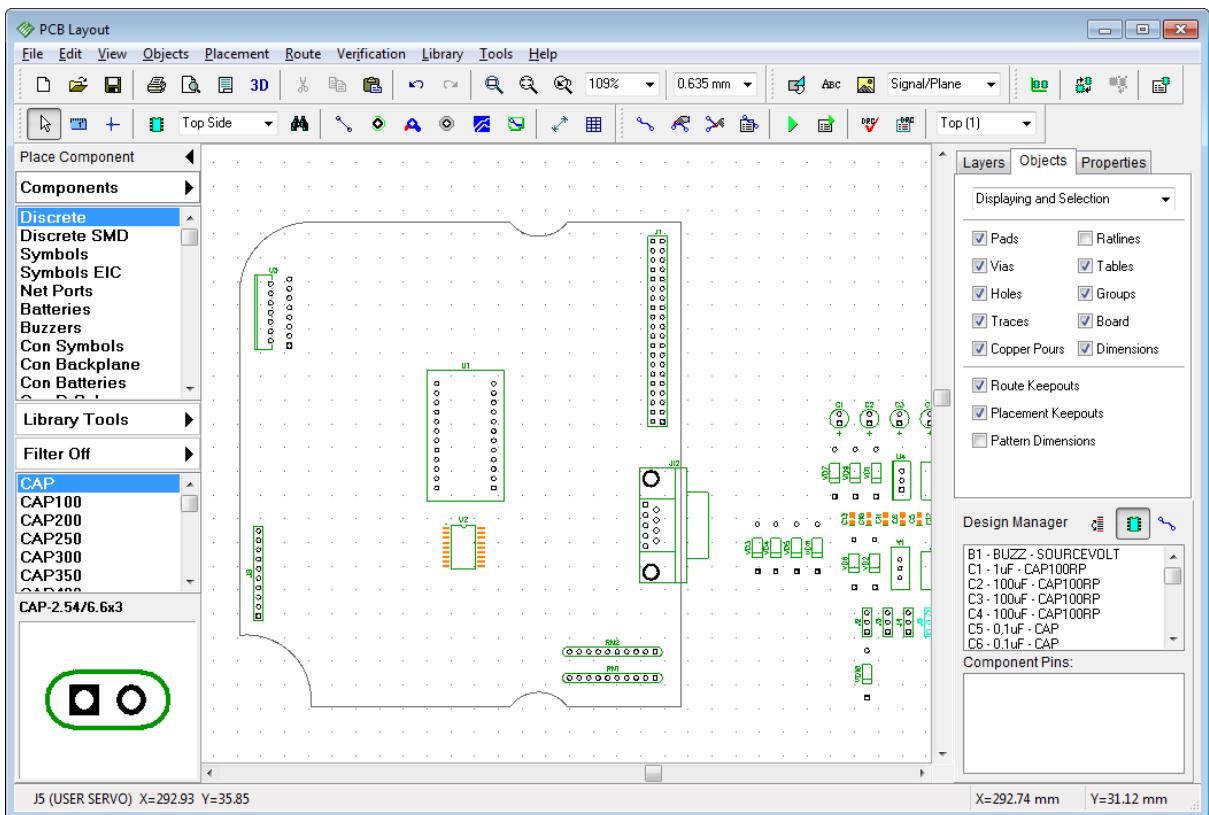
However, in real life manual placement is widely used, because not all components can be placed anywhere on the board. DipTrace allows to combine automatic and manual placement opportunities.

Placement by list

Select "Placement \ Placement by List" from 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.

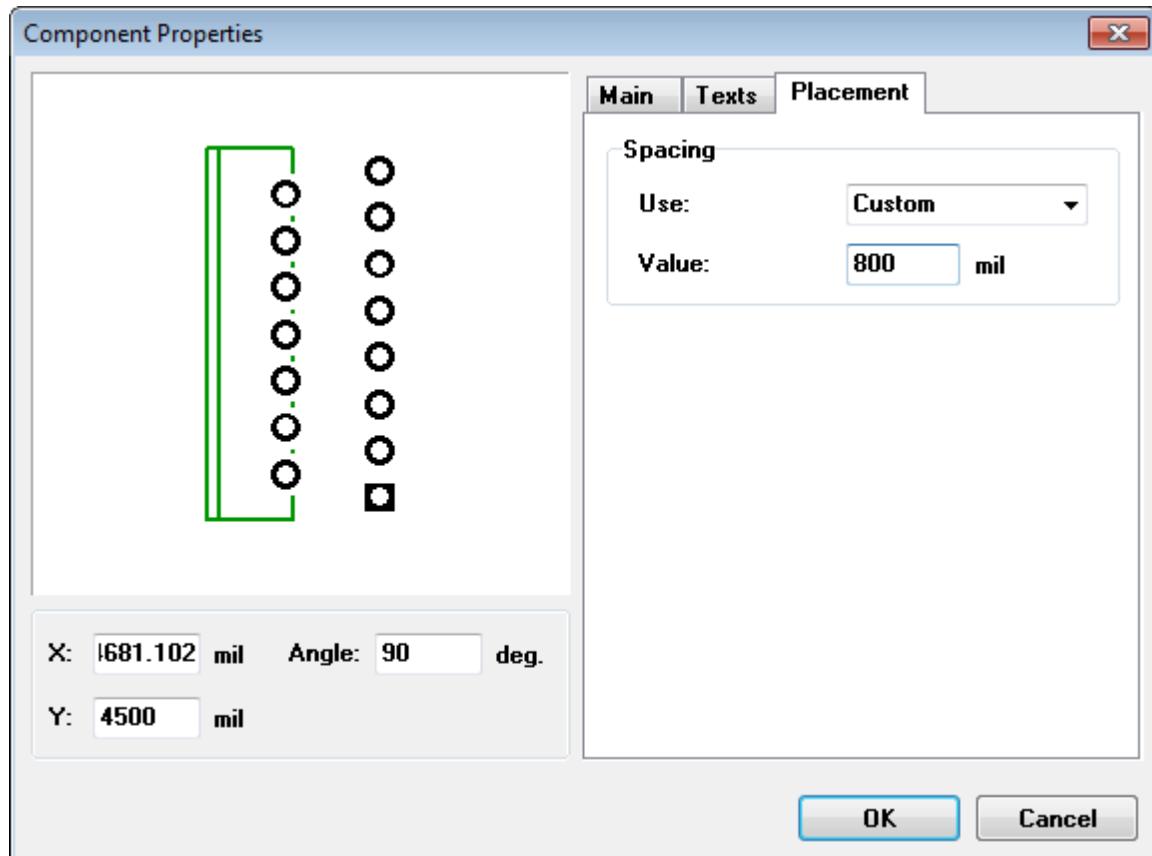


Component disappears from the list after placing it (list shows only components outside the board outline). Place U1, U2, U3, J1, J8, J12, RN1 and RN2 components manually, like on the picture below (you can optimize connection lines with "F12" button or hide them, uncheck "Ratlines" item on Objects tab of Design Manager). Close "Placement by List" dialog box when done.



Custom Component Clearance

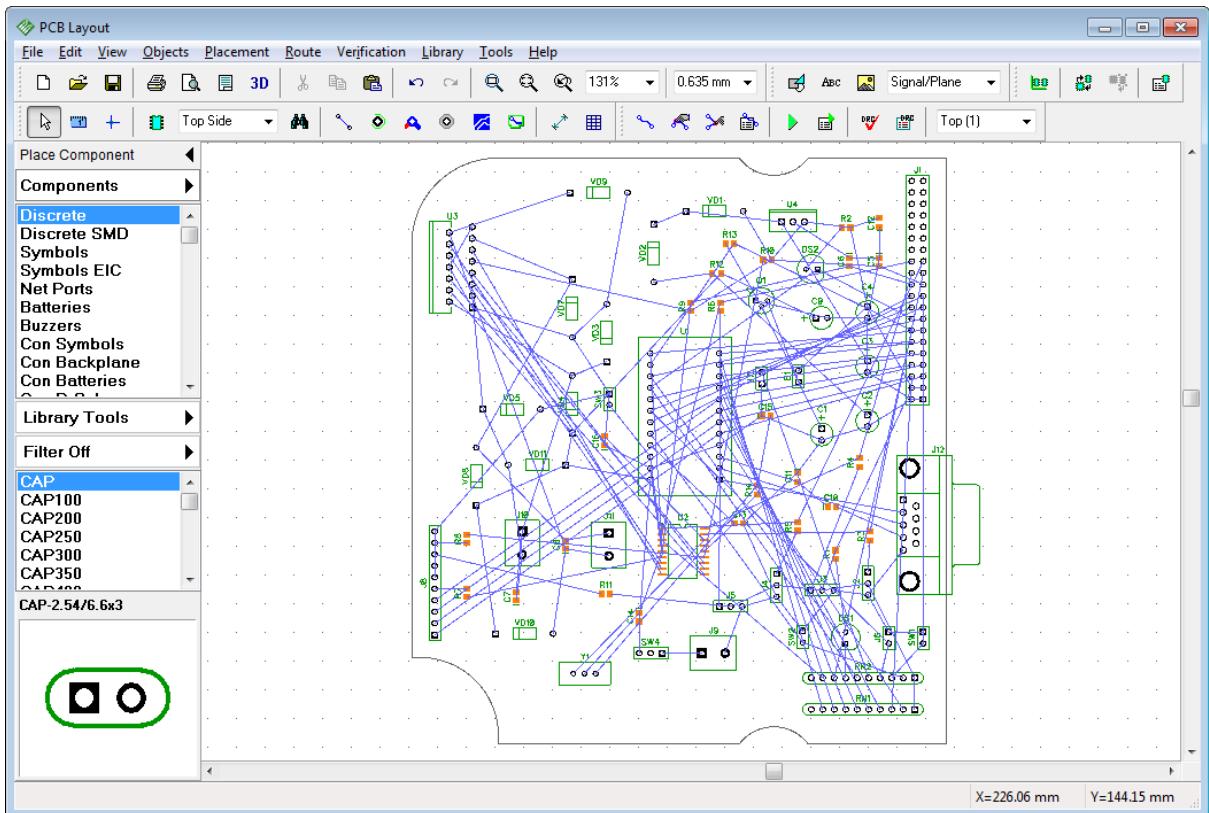
Placed components should have fixed positions, except U3. Select and lock them ("Ctrl+L"). Now right click on the U3 component and select Properties from the submenu. Go to "Placement" tab, Spacing section. Define: "Use: Custom" and "Value: 800 mil" (approx. 20 mm). This means that all other components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



Auto-Placement

We do not have special requirements about other components, therefore we can place them automatically with 5 mm spacings. Change measurement units ("View \ Units \ mm" from main menu). Select "Placement \ Placement Setup", change X Spacing and Y Spacing to 5 mm, and 3mm 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 Patterns Outside the Board Outline" and make sure "Use Pattern Spacings" item is checked, this will allow program to use 20 mm custom clearance of U3 component. We do not recommend to check "Increase Placement Quality" (you can try it later).

Press "OK" to apply changes and "Run Auto-placement" button on the placement toolbar or select "Placement \ Run Auto-placement" from main menu. DipTrace looks for the best location for each component. You'll get something like on the picture. Notice that Design Manager is hidden ("F3" hot key).

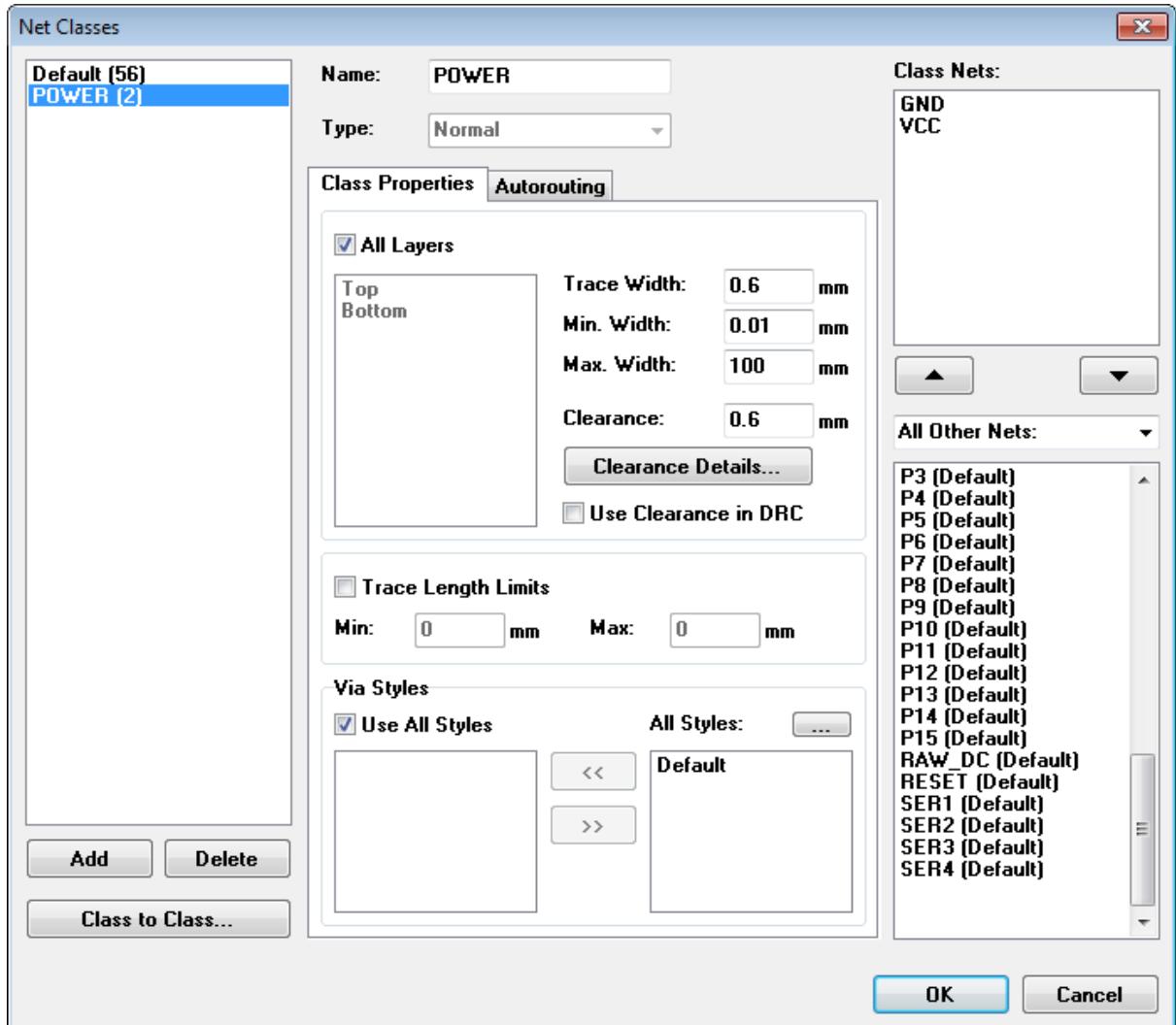


Some connections are not optimal, because we have placed large components manually. If you auto-place entire board you can get better results, but usually this is not the option in real life.

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

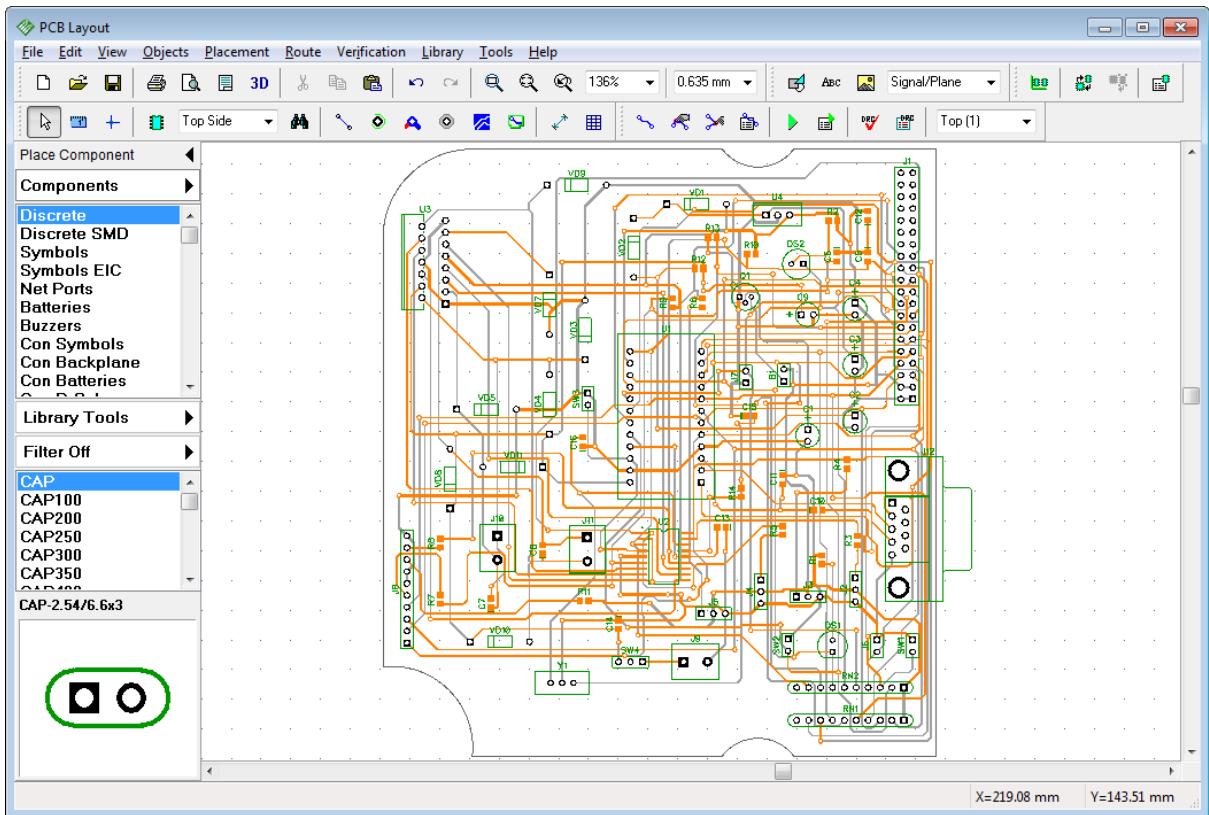
Autorouting with Net Classes

Check via properties in "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 separate net class for Power and Ground nets, because traces of these nets should be a bit wider. Select "Route \ Net Classes" from main menu. All nets belong to Default 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 (press up arrow button right above the list).



Now select Default net class and specify 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 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 Settings tab of Autorouter setup dialog box.

Now press "Ctrl+F9" or green arrow on the route toolbar to launch autorouter. In a few second you get the results:



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

4.11 Fanout

Fanout allows 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 PCB Layout module or if it is already open select "File \ New" from main menu or press "New" button on standard toolbar. Load rules from the *.rul file we created at the end of "[Saving/Loading Design Rules](#)"^[177] topic of this tutorial with some via styles, net classes and layers.

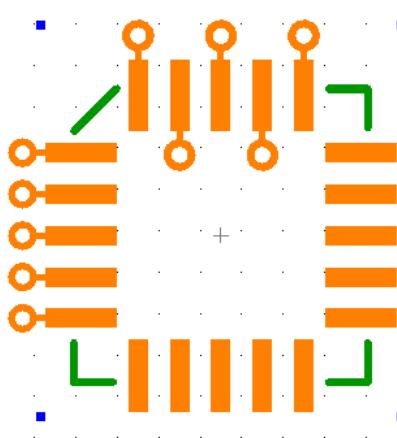
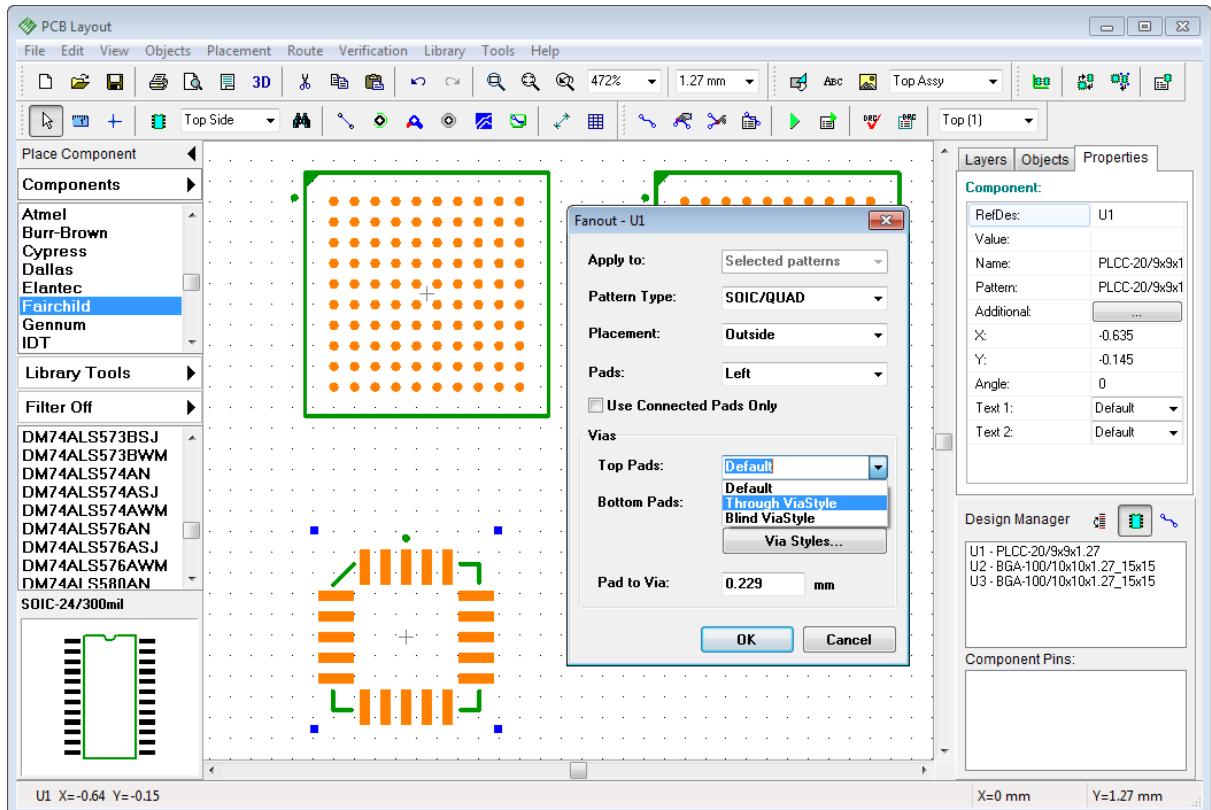
Fanout Component

Now select "Patterns" library group, it contains all patterns available in standard DipTrace libraries. Notice that these are just patterns without schematic symbols. Place one "PLCC-20/9x9x1.27" from "General" library and two "BGA-100/10x10x1.27_15x15" from BGA library. We will use this patterns for demonstration, but you can select another patterns/components for practicing fanout.

Right click on PLCC and select Fanout from the submenu. In the pop-up dialog box specify settings: "Pattern Type: SOIC/QUAD", "Placement: Outside", "Pads: Left" (this means that vias will be created only for the left pad line of the PLCC pattern) and make sure "Use Connected Pads Only" box is unchecked (vias will be created for all pads, not only for connected to some nets).

Select different Via Styles for pads on the top and bottom side of the board (inactive if there are no pads of component on selected side). Preview 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 Default via style. Select the one which fits current component size.



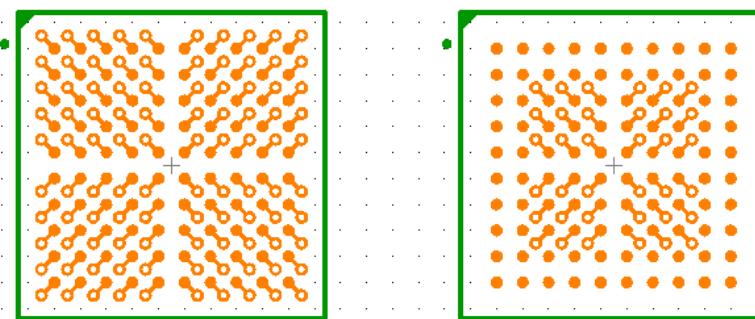
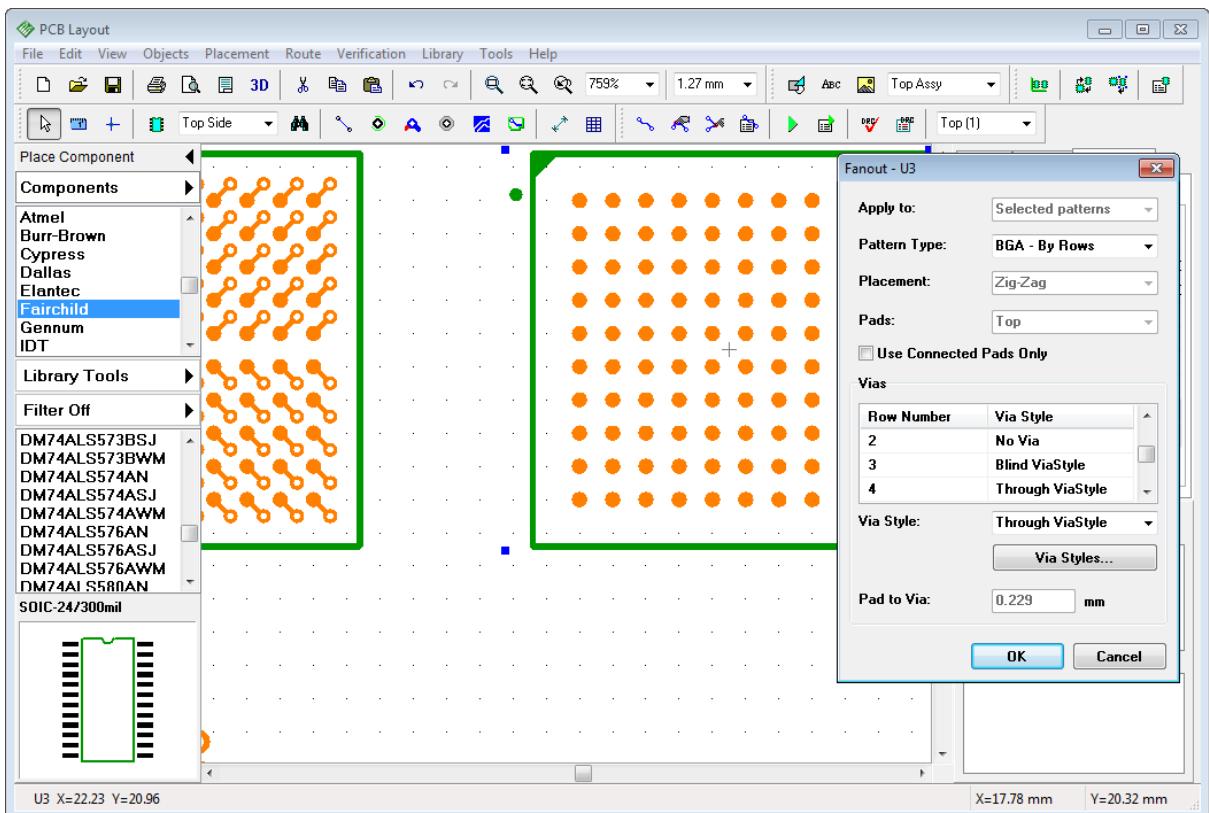
Press "OK" - vias will appear outside the left pad line. Right click on the same pattern and select Fanout again. Now we will place zig-zag vias for the top pads. Select: "Placement: Zig-zag" and "Pads: Top", keep other settings and click "OK".

Now we will make through-hole vias for one of the BGA patterns and blind/buried vias for another. Please, 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 via style with through-hole vias. Press "OK".

Now select second BGA package. Right click it and choose "Fanout". Select: "Pattern Type: BGA - By rows". This allows to apply different via styles to different pad rows of the same pattern or leave some rows without vias. 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".

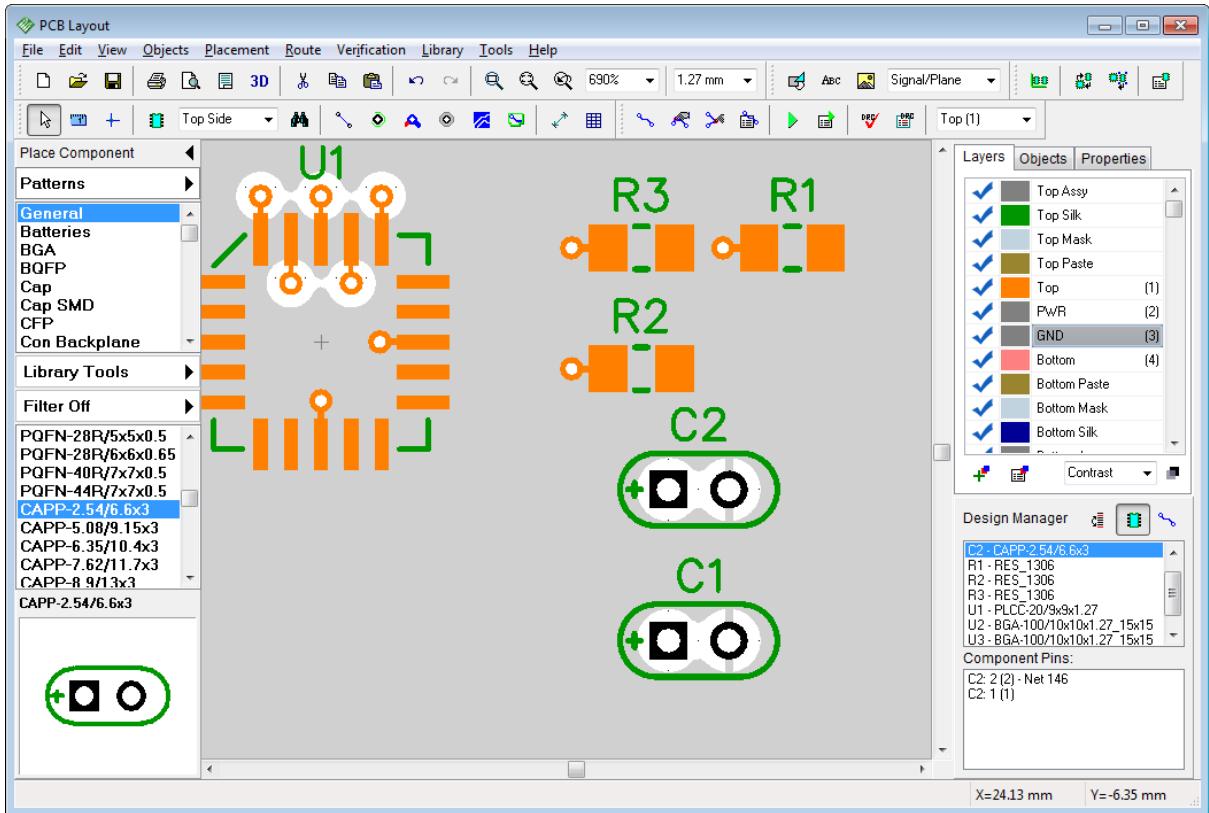
We can see, that for the first BGA pattern all pads are connected to vias, for the second one - two rows are

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

Fanout Net

We will connect several SMD pads to GND plane layer using fanout feature. Place several SMD and couple of through-hole patterns on the design area. Create net that connects some pins of these packages (we suppose this is our GND net that we should connect to GND plane layer). Select "Objects \ Place Ratline" from main menu or press corresponding button on the objects toolbar to create ratlines (connections). Rename net to "GND" if you want.

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 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 pads to GND inner plane layer, where we've placed a copper pour connected to GND net.



4.12 Hierarchical Schematic

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

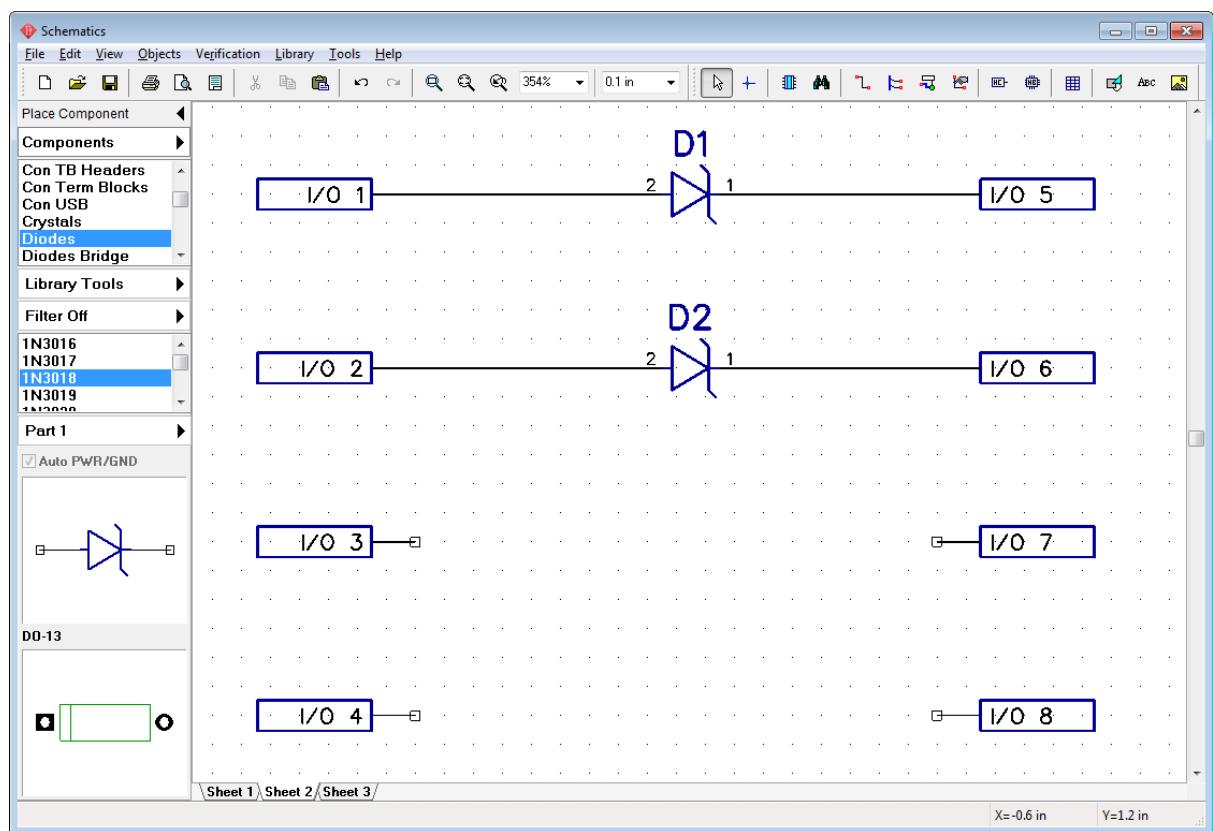
Hierarchy Blocks

Open Schematic module. In DipTrace hierarchy blocks are associated with sheets, so first of all we have to add two sheets to blank schematic, select "Edit \ Add Sheet" from main menu twice. Then specify that additional sheets are hierarchical blocks, not just regular schematic sheets. Select the second sheet in the bottom-left corner of design area and go to "Edit \ Sheet Type \ Hierarchy Block" from main menu. Do the same for the third sheet.

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

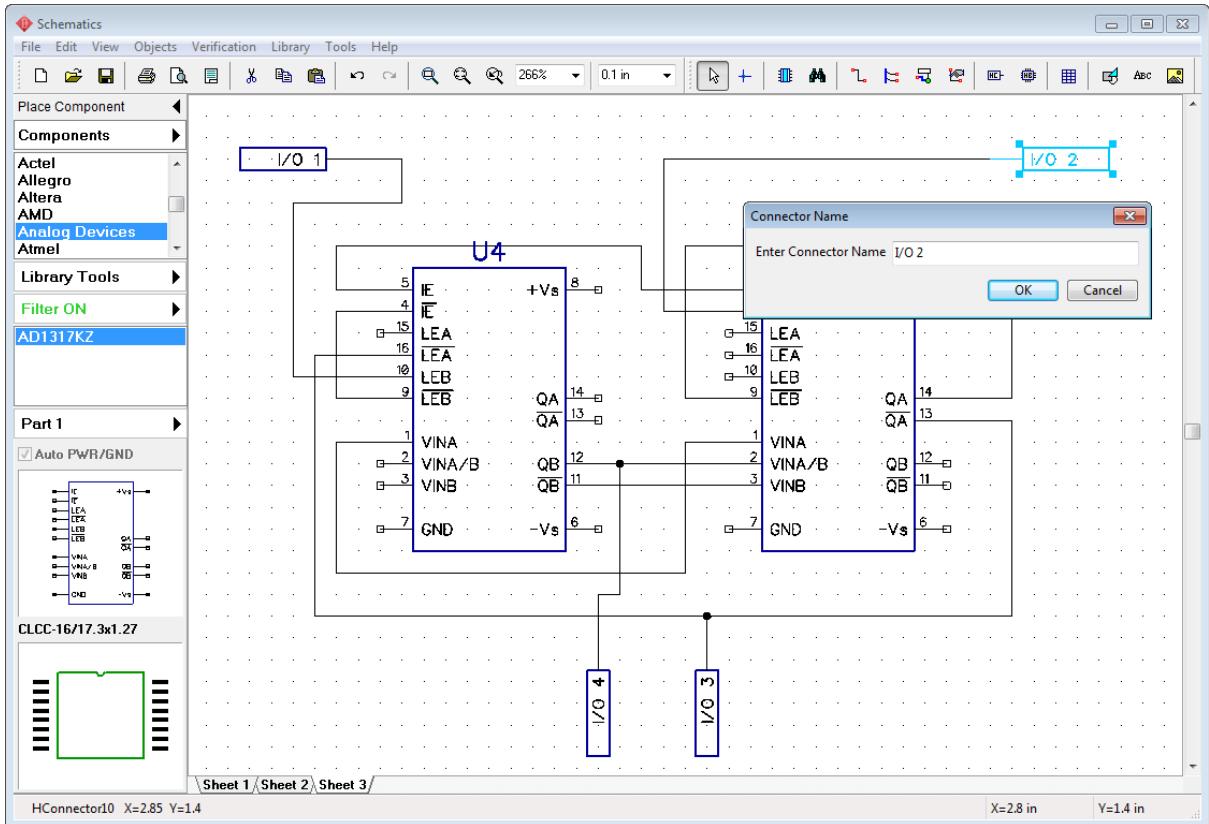
Select second sheet. Choose "Object \ Hierarchy \ Place Connector" from main menu or press button with connector and "HC" text on the objects toolbar. Place several hierarchy connectors to the second sheet (notice that you can not place hierarchy connectors to non-hierarchical sheets). These connectors are inputs/outputs of 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 "Diodes" library, connect them to connectors and leave free space for upcoming hierarchy block of second level. Use "R" to rotate hierarchy connectors.



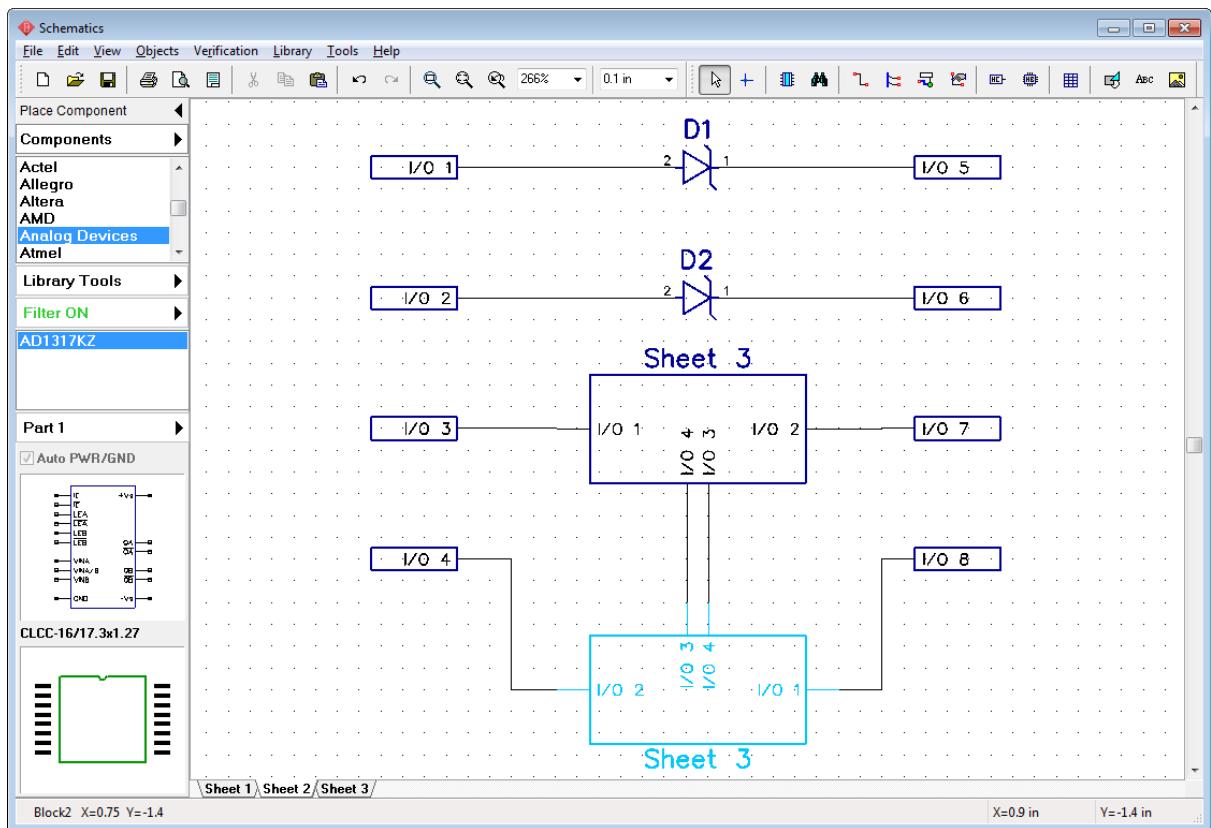
Select Sheet 3 and create second hierarchy block there. Place several hierarchy connectors (for example two on sides and two at the bottom) and couple of components (for example, two AD1317KZ components from "Analog Devices" library) and connect them.

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



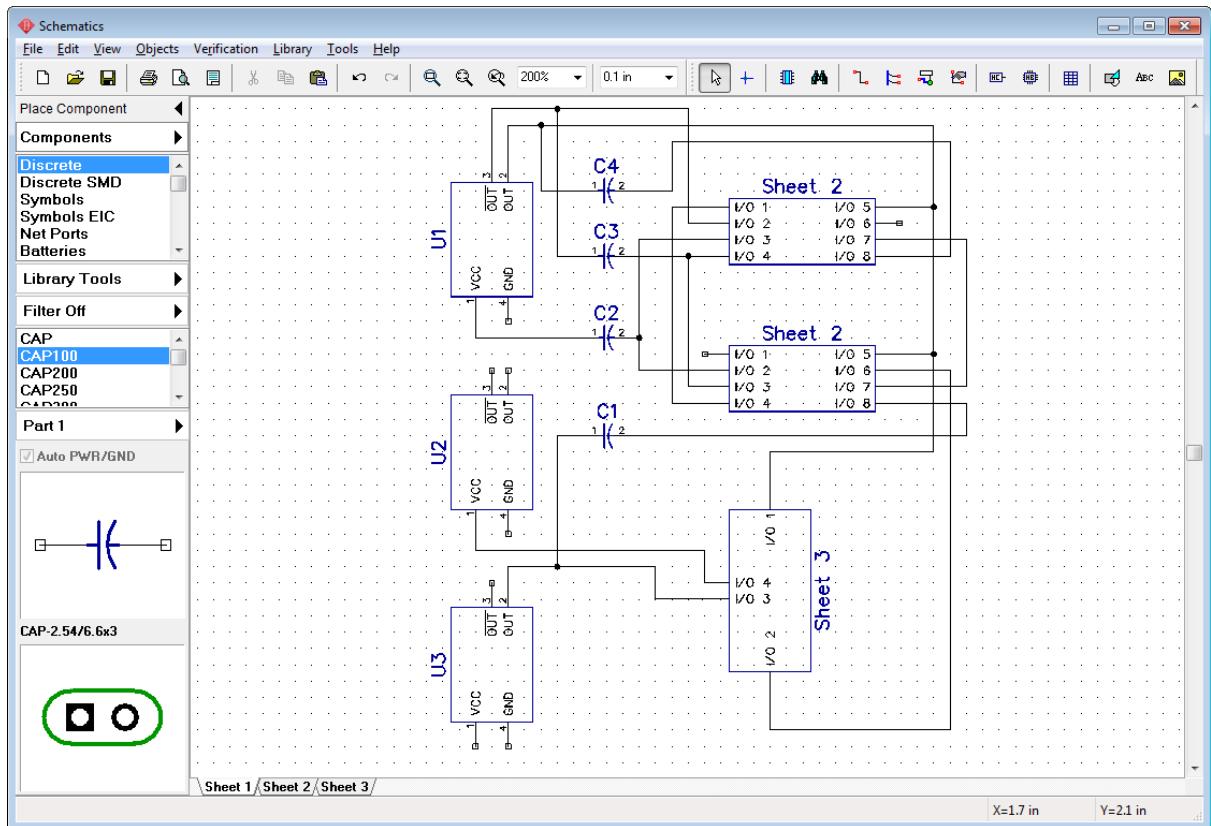
As you know DipTrace supports multi-level hierarchy, i.e. hierarchy blocks can be inserted into main circuit and into each other as many times as needed.

Select Sheet 2 then go to "Objects \ Hierarchy \ Place Block" or press button with HB text 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" to rotate blocks. Notice that you can place Sheet 2 inside itself or make a closed loop of hierarchy blocks, it is an error. To avoid this situation, use "Verification \ Check Hierarchy" from main menu. PCB Layout also checks hierarchy for closed loops and displays warning message when you open schematic with hierarchical errors. We will not make loop right now, just place two Sheet 3 blocks into Sheet 2 and connect them to connectors like on the picture below.



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

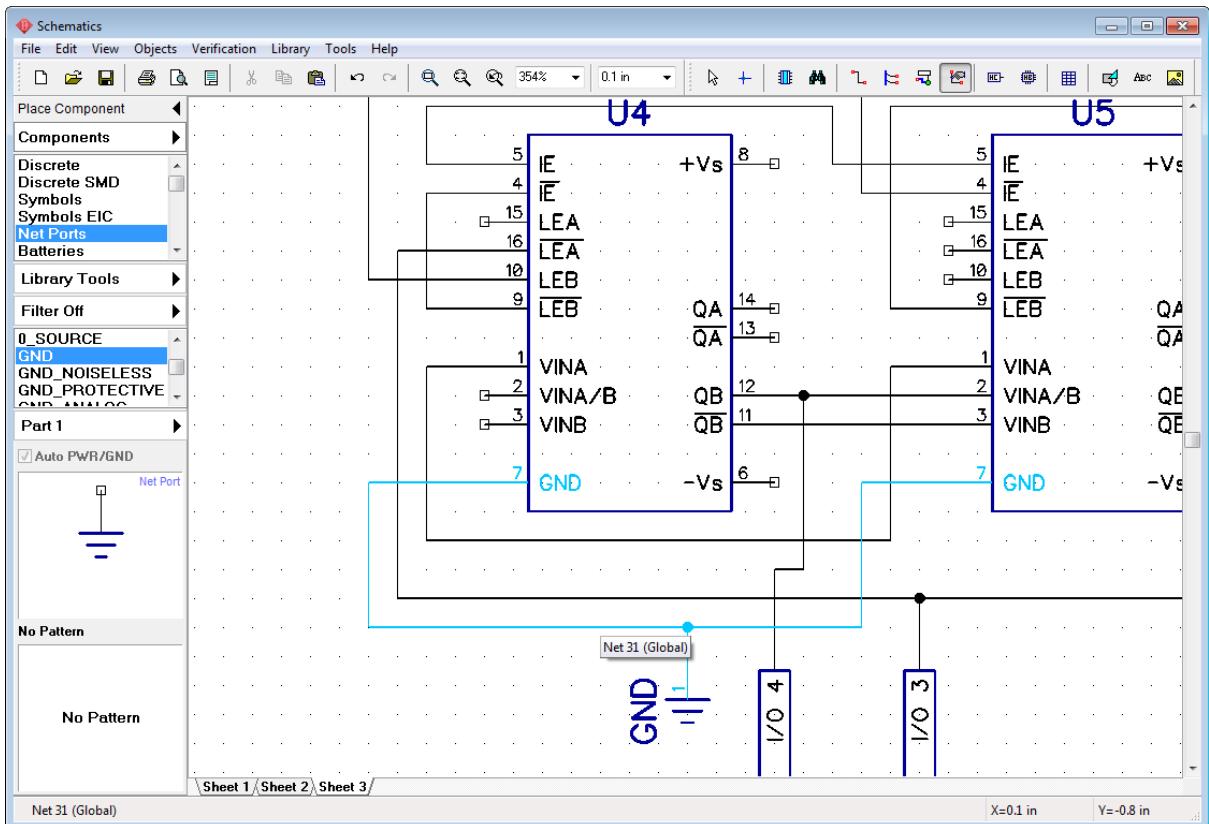
Select main sheet and place hierarchy blocks to the main circuit (for example two Sheet 2 blocks and one Sheet 3, like on the picture below). Connect hierarchy blocks with other components of the schematic. Notice that hierarchy blocks are similar to regular components, they have pins and can be easily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



Global Nets

As you already know 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 hierarchy structure of schematic.

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



Select Sheet 1 (main circuit) and place GND net port in there then connect it (create wire from net port to some free GND pin/s. You'll notice that this net has also became "Net 31 (Global)". Now we have single global net on two hierarchical levels. We can continue this net to Sheet 1. Please rename it to "GND".

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

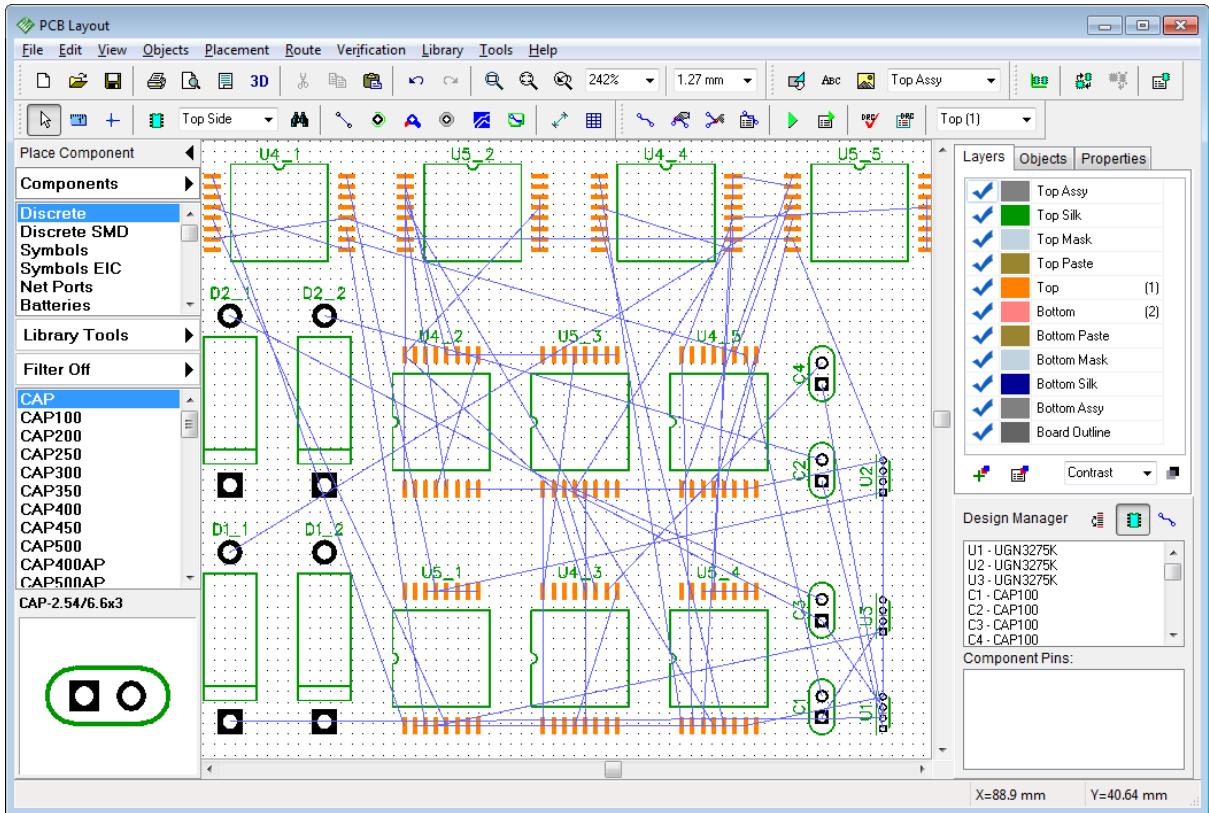
Global nets can be created very similar to [connecting nets by name](#)^[158]. Right click on a random net and select "Properties" from the submenu. In the pop-up dialog box check "Global Net for Hierarchy" and "Connect Nets by Name" boxes. Type in the name of global net that already exists and press "OK".

Hierarchy in PCB Layout

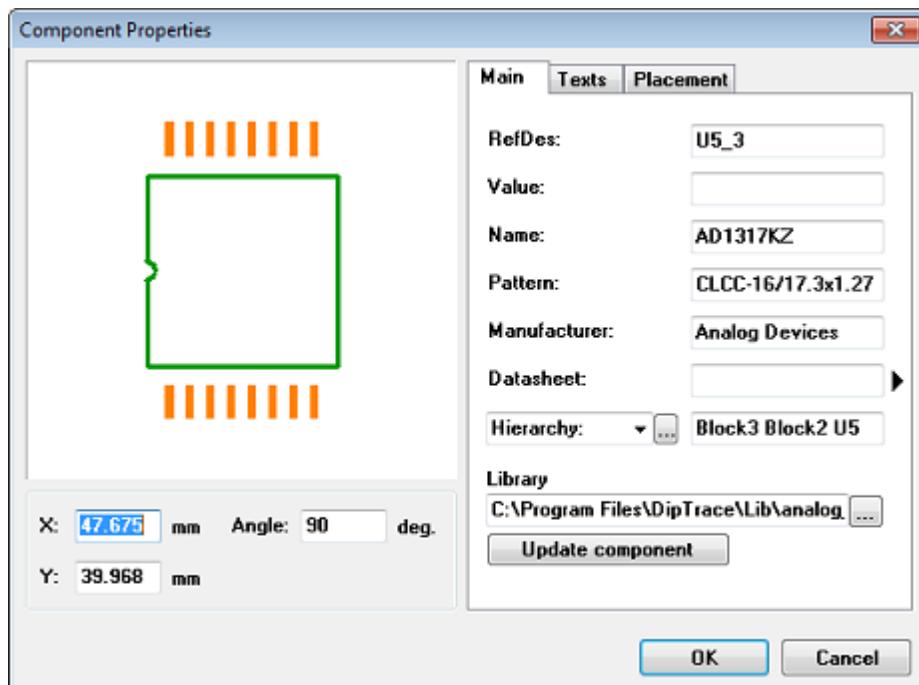
Convert this non-real hierarchical schematic to PCB. Press "Ctrl+B" and select "Use Schematic Rules". In PCB Layout module components that were in hierarchy blocks are superimposing each other, [arrange](#)^[184] them (first button on the placement toolbar).

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

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



Right click on one of 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 main menu).

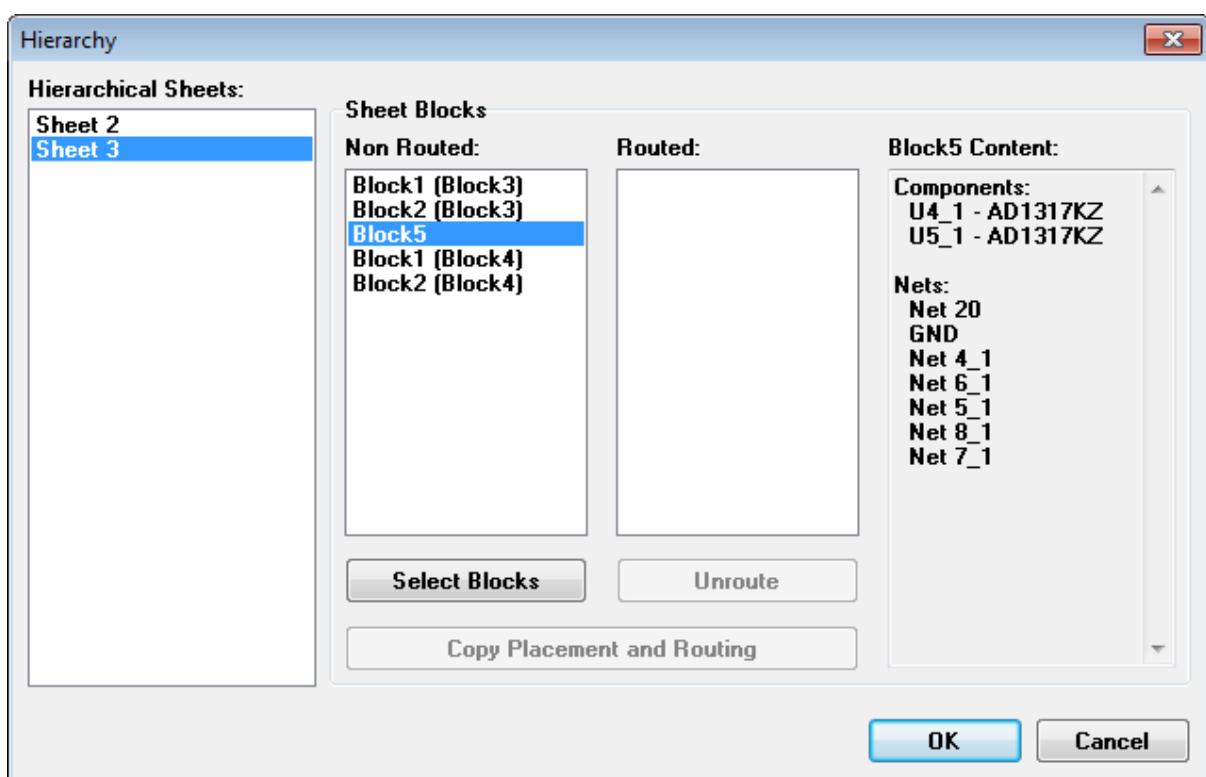


DipTrace works with hierarchy circuits in PCB Layout module - components can be automatically arranged by hierarchy blocks, routing from one block can be applied to another similar

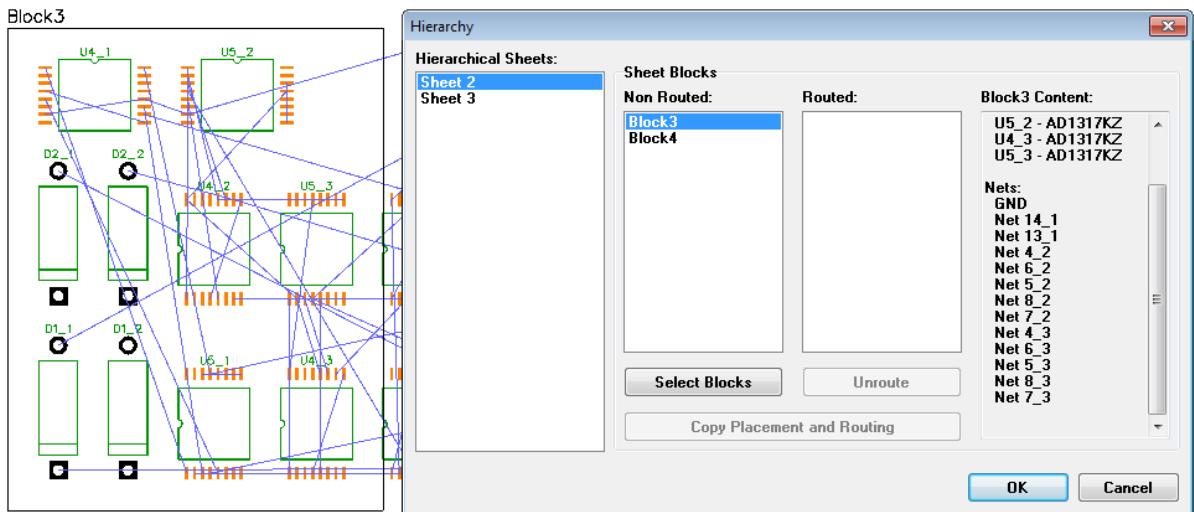
block in a blink of an eye. On the printed circuit board all components, regardless of their hierarchy level are on the same surface.

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

None of the blocks is routed at the time.



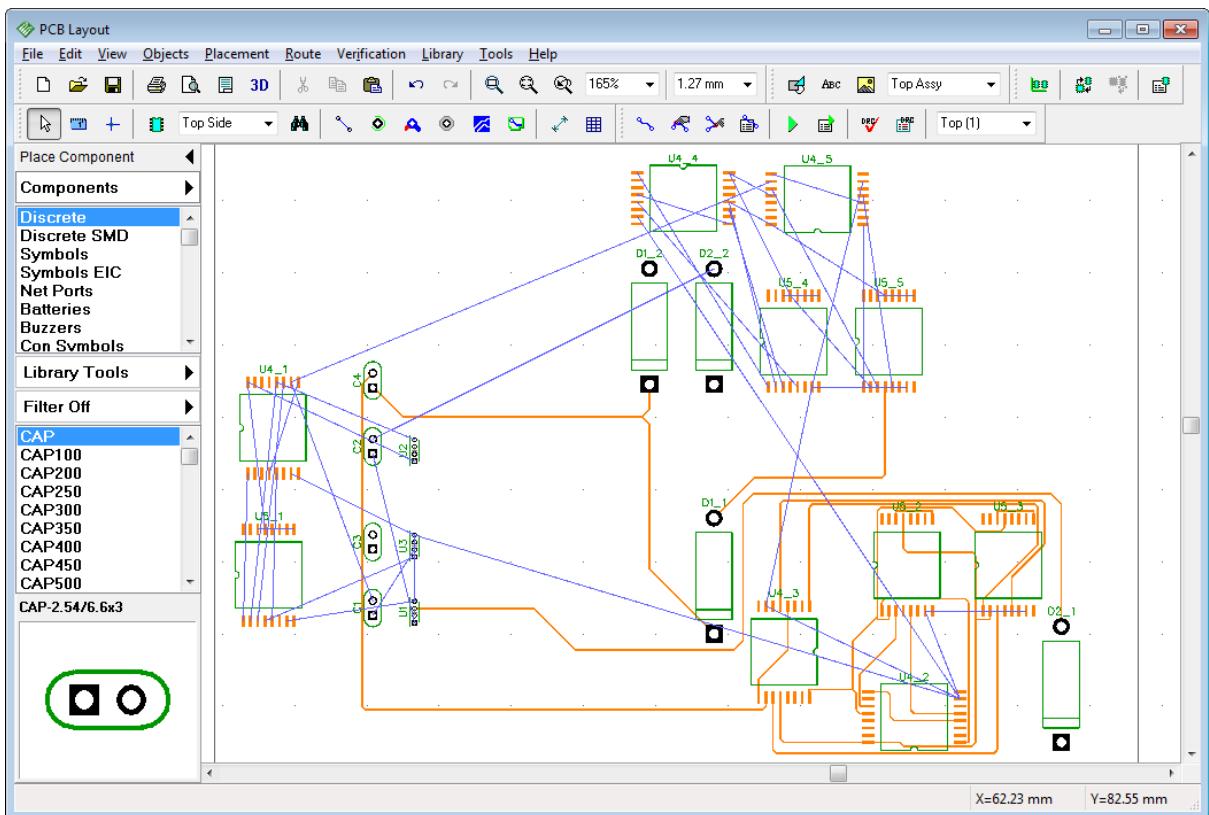
First we need to arrange components by blocks on the board. Select block in the list and its components will be highlighted on the design area. Press "Select Block" button to arrange components by selected blocks.



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

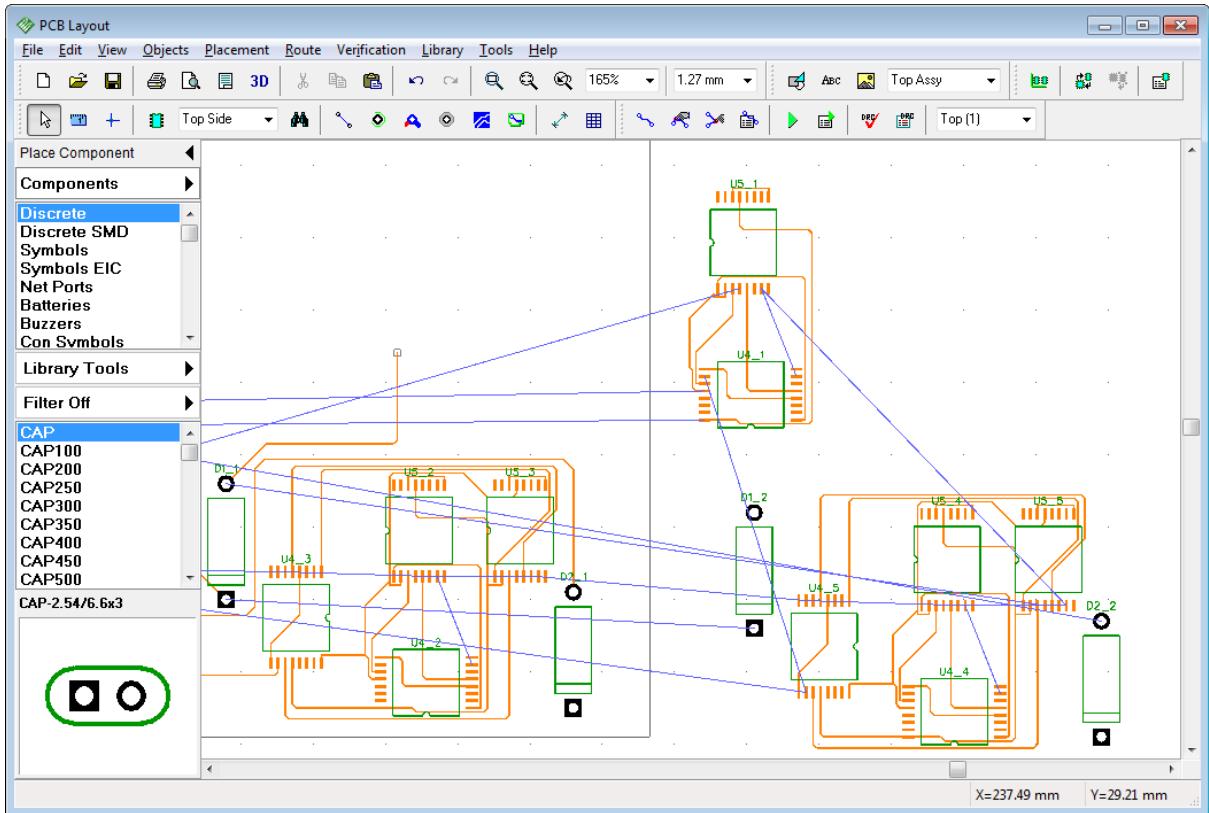
We will exclude GND net from autorouting. Right click on any pad that belongs to GND net, select "Net Properties" from the submenu and in the pop-up dialog box specify "Route Mode: Don't Route". Now select other components right click on component pad and select "Route Net" from the submenu or right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



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

We will apply placement and routing to Block 5 as well without closing 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 and change dialog box.

Routed blocks are located next to the board outline, use box selection to drag them to a correct position and finish project. Notice that traces from one hierarchy block to another or to main circuit have not been copied.



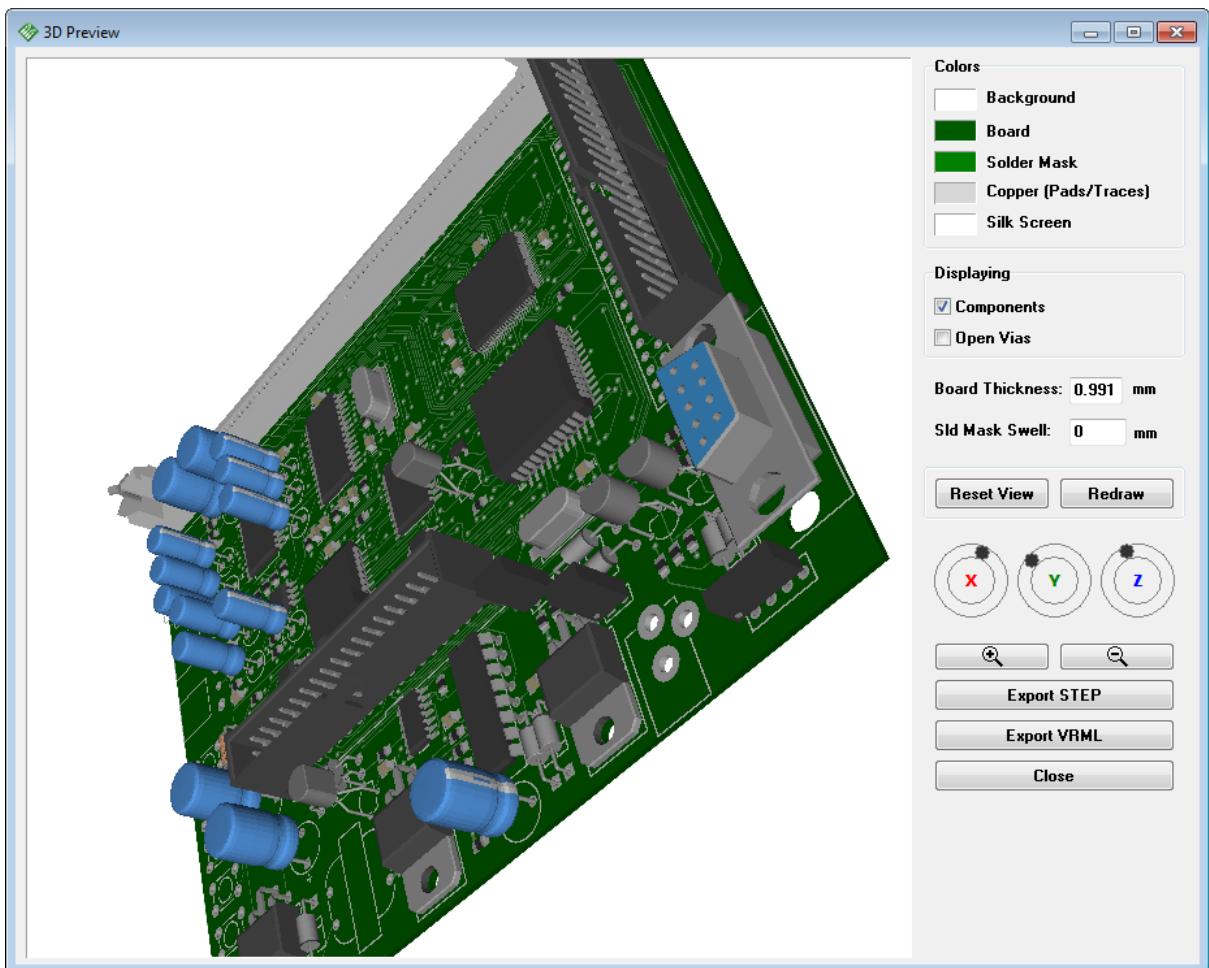
4.13 3D Preview and Export

DipTrace features built-in real-time 3D visualization with STEP and VRML export. This tool allows to see how PCB is going to look after manufacturing with all components installed. 3D models pack should be downloaded from DipTrace [web-site](#) and installed, because components without 3D models will be shown only as footprints.

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

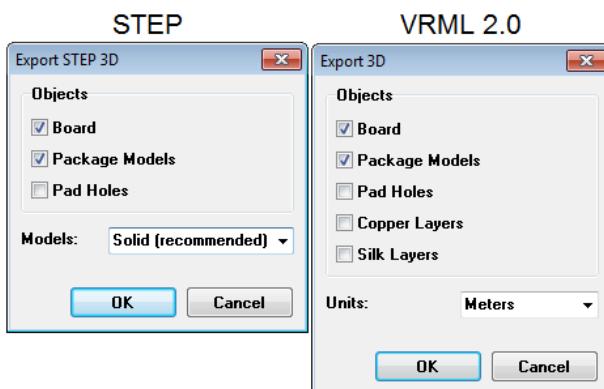
\Users\<UserName>\Documents\DiptTrace\Examples\PCB_6.dip". Then press "3D" button on standard panel or "Tools \ 3D Preview \ 3D Visualization" from main menu. "Attached 3D Models" dialog box will pop up. Press "OK" and you'll see PCB 3D model. You can rotate board in three axes, move it with mouse, zoom in and out with a mouse wheel.

Colors of the background, board, solder mask as well as component and via display options can be changed (press "Redraw" button to implement certain changes). Press corresponding buttons to export board model to STEP and VRML format which are standard for mechanical CAD systems.



You've probably noticed that some components are missing.

3D Export



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

When in DipTrace 3D Preview & Export module press "Export STEP" button. Specify export objects and select model export mode in the pop-up dialog box. Press "OK" to specify filename and folder.

Notice that exporting holes dramatically slows export process.

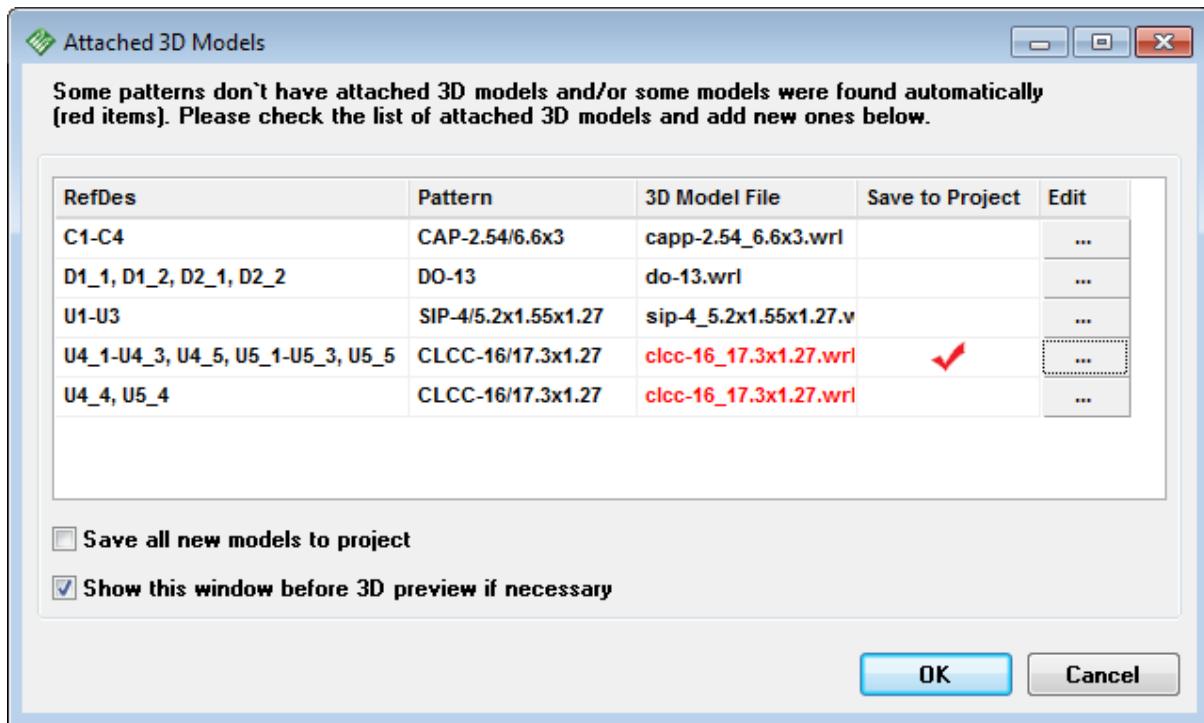
We recommend to export project as a solid body in STEP format.

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.

Close DipTrace 3D Preview module. Press "3D" button on the standard panel in PCB Layout again. We've returned to "Attached 3D Models" dialog box. DipTrace checks if all components have 3D models and tries to find correct model in libraries for component without them. Red items mean that model has been found automatically or manually attached, black - components with pre attached models. If you don't want to see this panel before 3D preview, uncheck "Show this window..." item. To save all found and / or edited models, check "Save all new models to project" item or check corresponding field in the table, otherwise new models will be used only for existing 3D preview session.



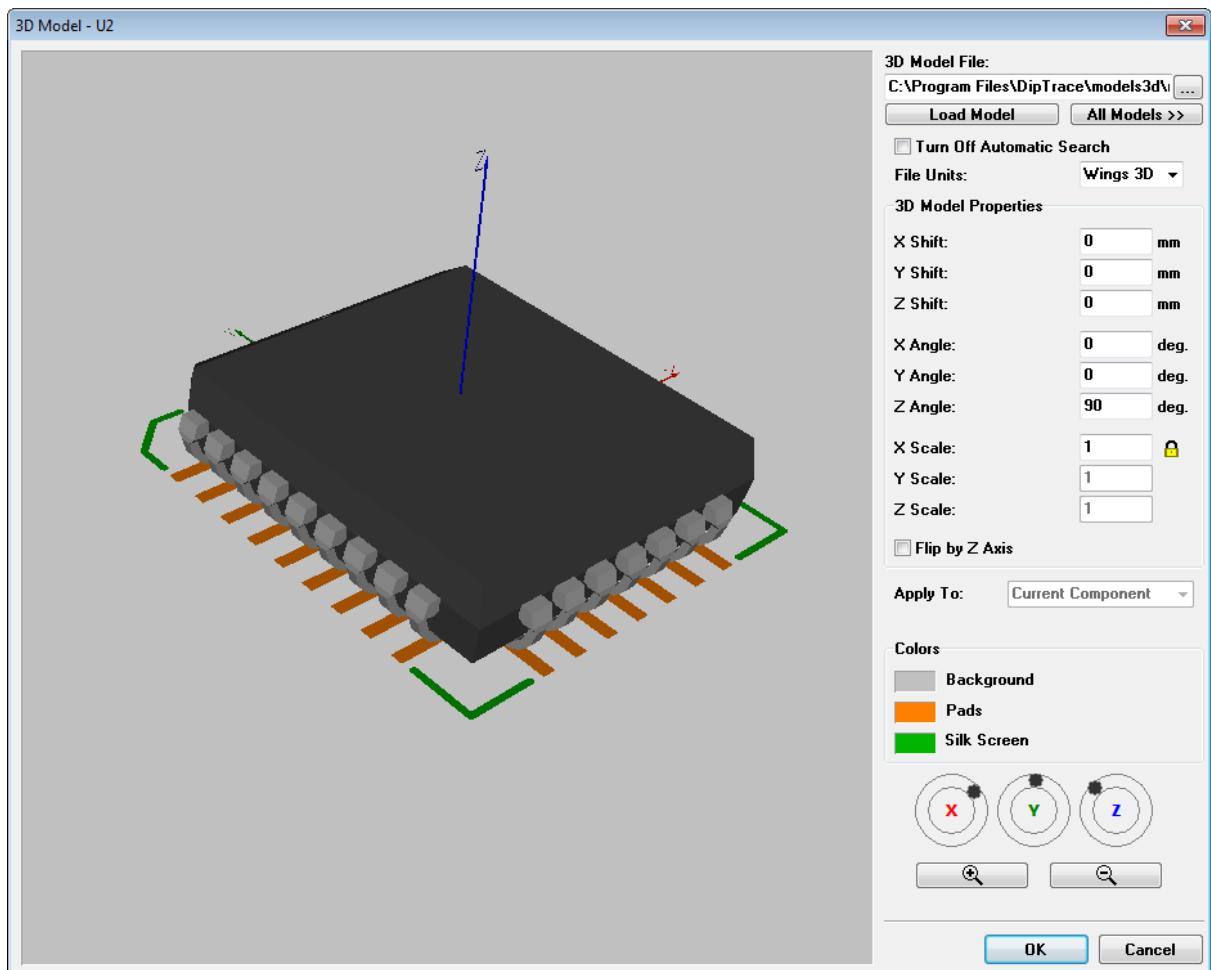
Attach 3D Model

Press "..." button in "Edit" field of the corresponding component and you'll see 3D model attachment dialog box. Rotate model in three axes, zoom in and out, move model by holding right mouse button and change colors.

If you need to change 3D model press "All Models" button and select 3D model from the list of all available models in active folder. Press "..." button to change active folder and "Load Model" button - if models are not in default 3D models folder.

DipTrace supports *.3ds, *.wrl, *.step, *.iges 3D model files, which can be designed in 3D CAD software or download from manufacturer's website and easily attached to component in DipTrace.

If you see that attached model does not lie directly on the footprint, enter new values in corresponding fields of 3D Model Properties section: rotation angle by different axes, shift and scale. Just practice a bit by entering different values and you will quickly understand that attaching model is very easy.

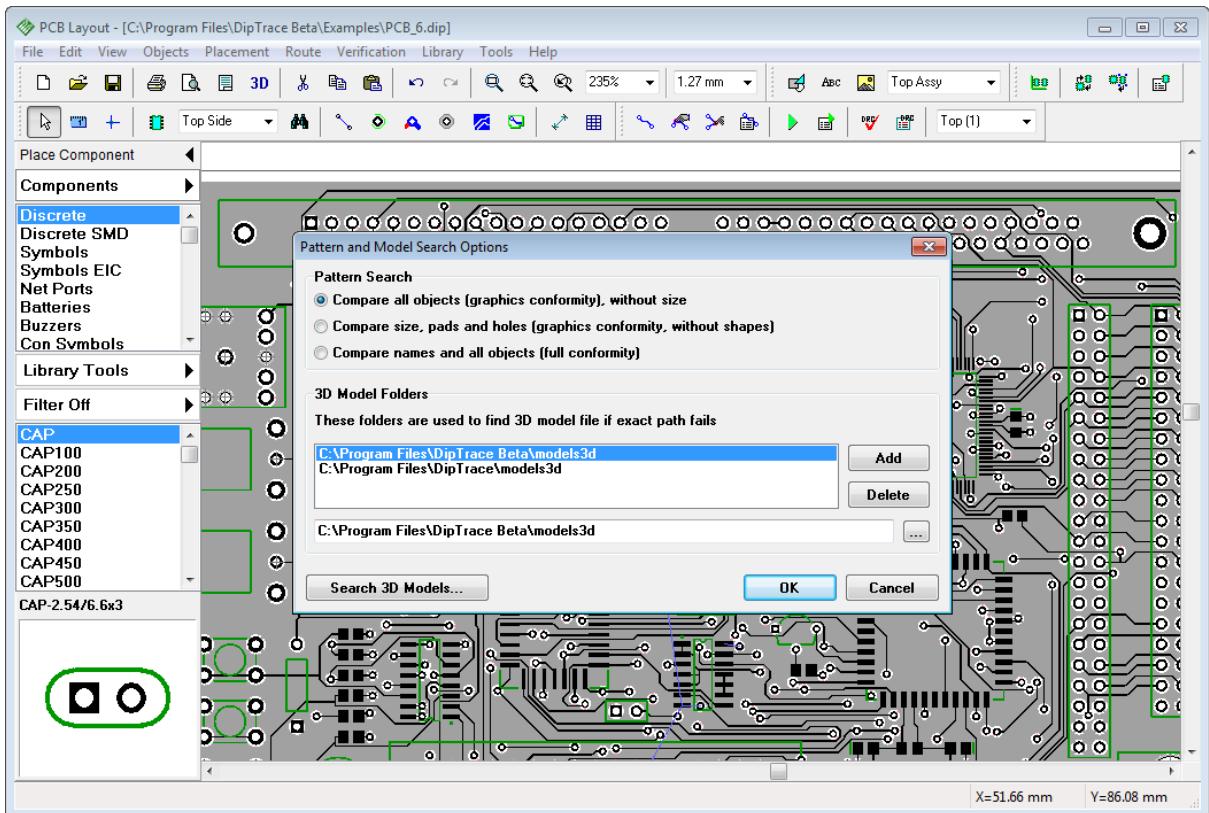


Press "OK" to close 3D model dialog box. Right click on any component on the design area and select "3D Model" from the submenu to open standard 3D Model attachment dialog box.

3D Models Search

Go to "Tools \ 3D Preview \ Patterns and Models Search". In the pop-up dialog box you can change search accuracy and specify HDD search folders.

In Pattern Search section you can select level of conformity that will be used by DipTrace while searching components (more strict requirements means less possible models found). Press "Search 3D Models..." to check results. However, for all new components please attach 3D Model in Pattern Editor while drawing the footprint.



5 DipTrace Links

[DipTrace official web-site](#)

[Technical support \(questions and suggestions\)](#)

[DipTrace Sales](#)

[Download DipTrace latest version](#) (go to "Help \ About" if you don't know your current version)

[Order DipTrace](#)

[Download Libraries](#)

[DipTrace Forum](#) suggest new features, discuss DipTrace and share your experience

[DipTrace Community at Yahoo!](#)

[DipTrace PCB Design Service](#)