Schematic Constructor 1.3

Freepcb2 compatible schematic editor


© 2021 FREEPCB.DEV

Quick Start Guide
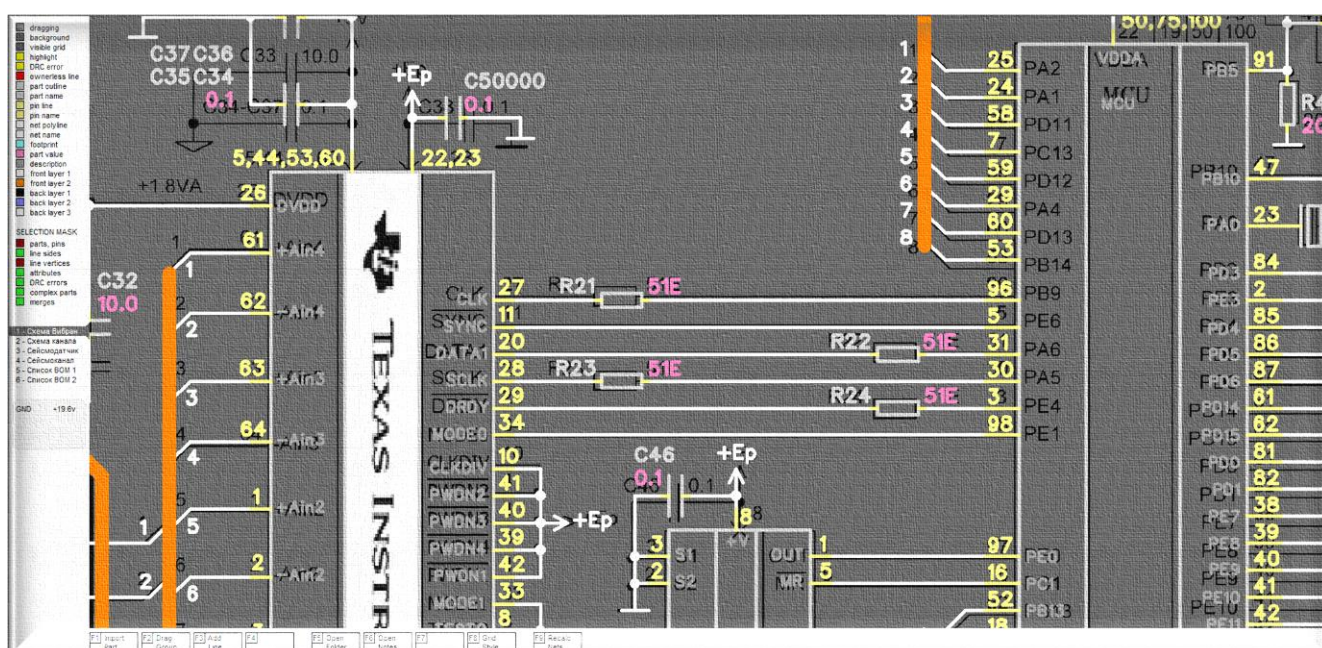
The multi-page editor is designed to quickly create small electronic circuits without any libraries. Using a simple tool, polylines, you can draw a schematic diagram and create a netlist. (taken from https://freepcb.dev/ECDS.html page)

Table of contents:

Abbreviations in the file: UGO - conventional graphic designation

Beginning of work

Place the directory Freepcb-2 to the hard disk of the computer, preferably not deep from the root of the disk. You cannot place the folder in C: \\ ProgramFiles since this folder is write protected and will prevent the program from creating temporary files necessary for correct operation. Run the executable file FreeCds.exe located in the BIN folder. At the very beginning, after starting, you need to select a directory in which all your projects of circuits and printed circuit boards will be stored. Create this directory using Windows tools, and then specify the path to the directory in the schematic editor via the main menu "File >> Open from project folder >> Select Folder". The program will save the path to this directory in the configuration file default_cds.cfg and you don't have to re-specify it ever again.
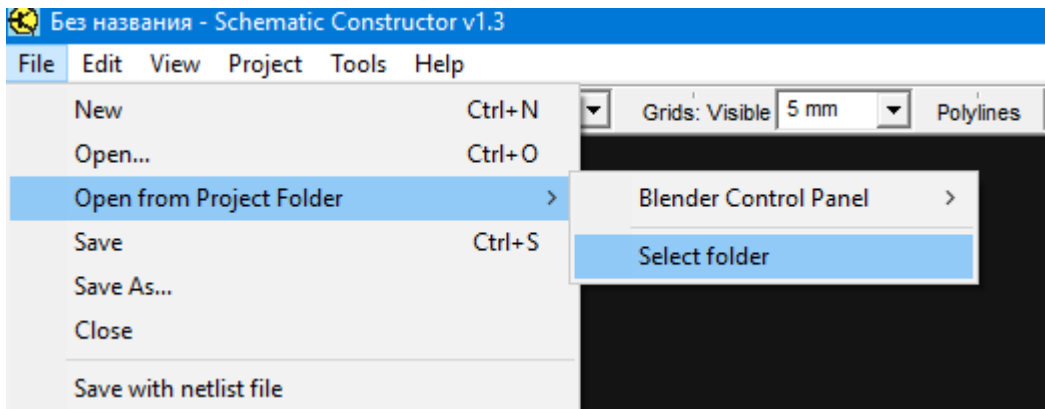
Now you need to create a new project, for this we press the button "File >> New".
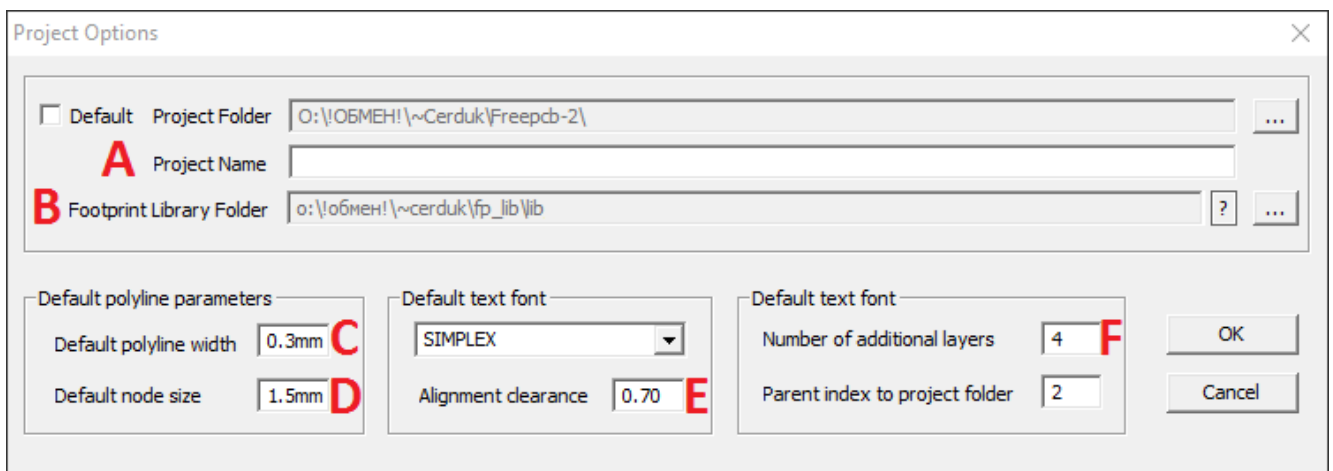
A project options dialog will pop up, in which you must enter

A - the name of the project,

B - specify the path to the Freepcb-2 footprint library. During netlist generation, the program will check the selected library for the presence of all footprints used in the circuit. (In this project we will use the library fp_lib \ lib included with Freepcb-2)

C - specify the default line thickness. The best schematic drawing style is to choose one dominant line thickness throughout the project.

D - specify the size of the knot at the junction point of two or more nets. These nodes are created automatically when a button is clicked F9 (Recalc.Nets)... A node is created only in those places where the vertex of the

polyline abuts against the side of the polyline, as shown in the figure, but it is not created when two polylines are connected by their ends.
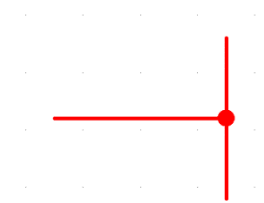

E - select clearance coefficient at alignmenttext attributes of the part. Any part generally has three attributes: RefDes, Footprint and Value. These attributes can bealignedin order in a column, taking into account the alignment factor, so that the circuit looks neat. More on this later.

F - select additional layers for drawing polylines if necessary.

Now click OK and the program will create a folder in the project directory. The folder will have the name of the project, inside this folder will appear a folder named "version-01", and inside it there will already be a schematic and PCB file. What is the "version-01" folder for? Read on ...

Create a project branch

When we have completed the work on the project and have already made the printed circuit board, we can no longer make changes to the current project branch, but must create a new one in order to make all the changes and new ideas into it. To do this, Schematic Constructor1.3 provides an option to create a project branch. This function is located in the main menu "Project >> Create branch"
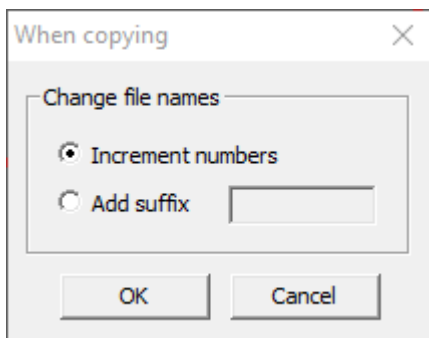
By default, the number increment option is set, which means that by clicking OK next to the "version-01"Folder appears"version-02", And the schematic and PCB file names will also be incremented by one. But you can also add a suffix to the file name instead of incrementing the number and create a branch for example "version-01-mod" (if you want to create a temporary branch, for example). What are the advantages of creating a branch through the program menu in this way? Why can't you just copy the "version-01" folder and rename it manually? This can of course be done, but it will probably take much longer. In addition, PCB files can contain text with the file name in a silkscreen layer for version control, and when a branch is created via the menu, these texts are automatically adjusted according to the new file name. Also, the schematic file can include text blocks in which the names of the PCB files appear.

Page template

So, now we have a project in which there are no objects. The program comes with a file Template.cds, which contains the default A4 page border. To insert this object into our project, select the menu "Edit >> Paste page from file". In the dialog box, select the Template.cds file, and then in the select page dialog box, select Page1. A frame like this will appear:
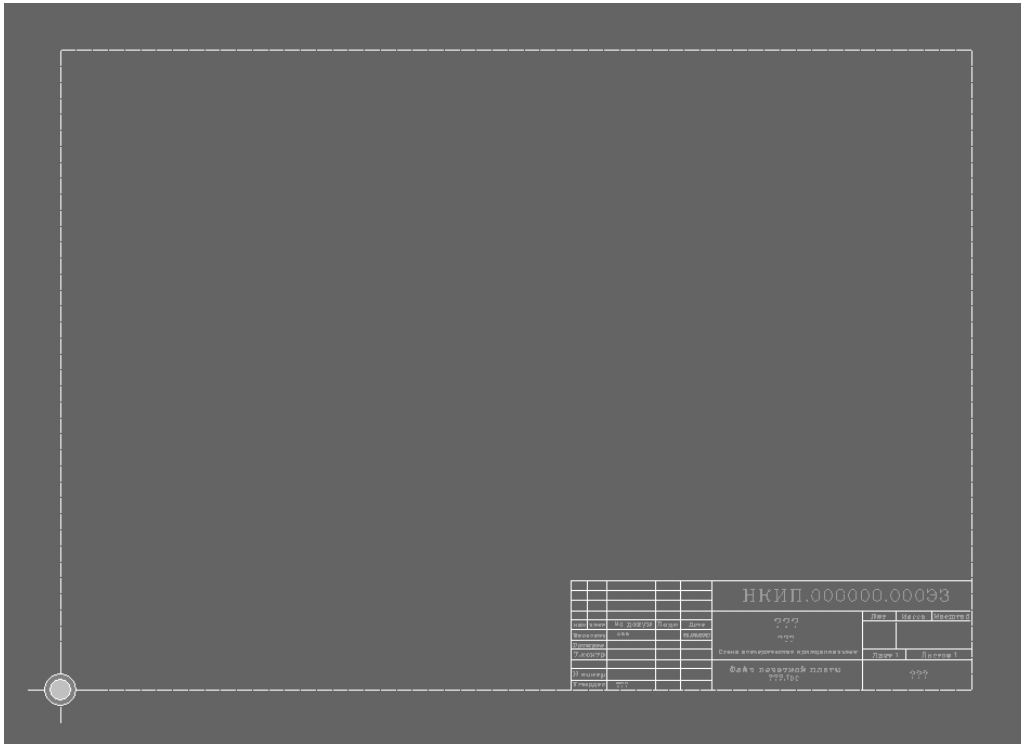

This frame is made in the project in the form of a solid part, and therefore, when trying to select the side of the polylines, the entire frame is selected at once. But you can select the corners of the frame polylines (the vertices of the polylines) to move these nodes (move using the keyboard arrows, or by pressing the button F4 (Move) when the vertex is highlighted).
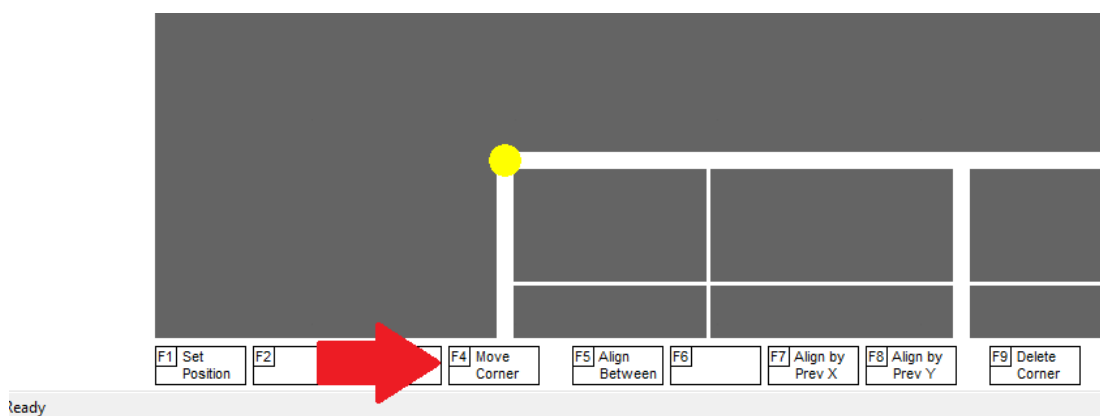
The program will enter the drag-and-drop mode. To complete dragging, click with the mouse to the desired place, and to cancel dragging, right-click. If you want to move the side of any polyline in the part (and not the

whole part), then you need to activate the blocking of the part in the selection mask. On the left of the main window there is a selection mask that allows you to prohibit / allow the selection of different types of objects
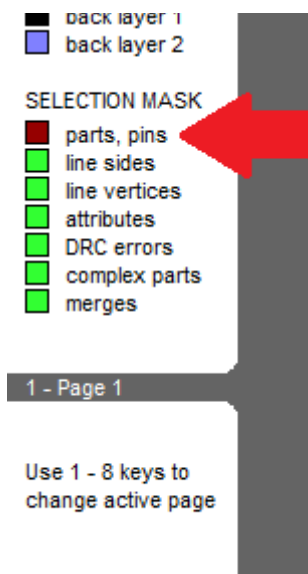

Click on the "Parts, Pins" line to disable the selection of whole parts. Now you can select any side of the polylines included in any part and change its position with arrows or by entering drag mode using the button<mark>F4 (Move)</mark>... You can also select a group of polylines using the mouse frame by dragging the mouse while holding the left button from the upper left corner to the lower right corner. When two or more objects are already selected, then you can add other objects to the selection simply by selecting them with the mouse frame (without pressing additional CTRL keys, etc.). To deselect the selection locally, drag the mouse in the opposite direction, i.e. from the bottom right to the top left corner. Thus, you can selectively deselect, for example, the vertices of polylines.

Draw a polyline

Let's draw a polyline and see what function buttons appear on the selected side of the polyline, and what can be done with this polyline in general.

In the mode when nothing is highlighted, press <mark>F3 (Add line)</mark>, the polyline properties dialog box will appear:
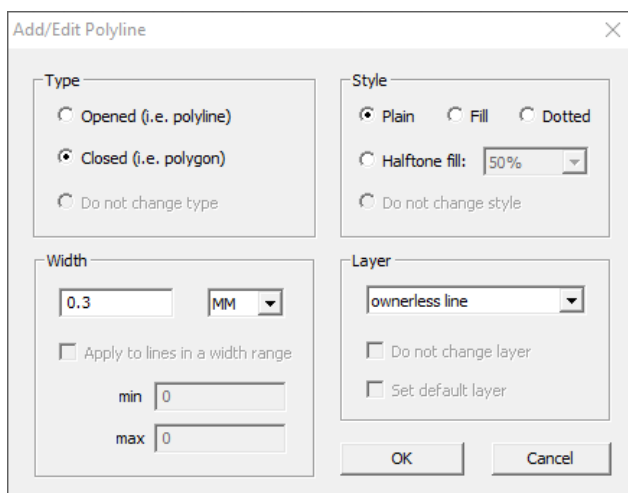

1) <mark>Type</mark>GroupBox. Here you must select the closedness of the polyline. This property, when the beginning and end of the polyline is automatically connected to each other, is used in polygons when you need

to fill the polygon with a solid color, also for example when drawing a contour line of a microcontroller, but is not used when drawing net connections or when drawing pins of parts. Net pins and connections must be open polylines.

2) Style GroupBox. Plain is a simple polyline. Fill - in the case of a closed polyline, this is a full filling of the polygon with the color of the polyline layer. Dotted - dashed polyline, the size of the dotted line depends on the thickness of the polyline. Halftone fill - filling a closed polyline with a diluted tone of the color of the polyline itself.

3) Width GroupBox.

4) Layer GroupBox. The "Ownerless Line" layer is selected by default. When you draw an electrical circuit (part outline, pins or circuit connections), then you do not need to select any other polyline layer in this window, leave the polyline layer "Ownerless Line"So that the program automatically moves this polyline to the layer corresponding to the polyline's attributes. For example, if a polyline has an attribute "Pin" (that is, a pin name is specified), then the program automatically moves this polyline to the "Pin line" layer. Or if, for example, a polyline connects two pins, then the program automatically moves this polyline to the "Net polyline" (connection) layer ... If you change the polyline layer and select one of the additional layers, then there will be no automatic layer detection and movement, and the polyline will not be assigned pin or circuit status. Thus, additional layers can be used only for graphic objects that are not related to either part pins or net connections.

Click OK and the polyline drawing mode will turn on. Click one point, then another to draw a linear section, then finish drawing by clicking the right mouse button. Select the side of the drawn polyline and the following bottom softkey menu appears:
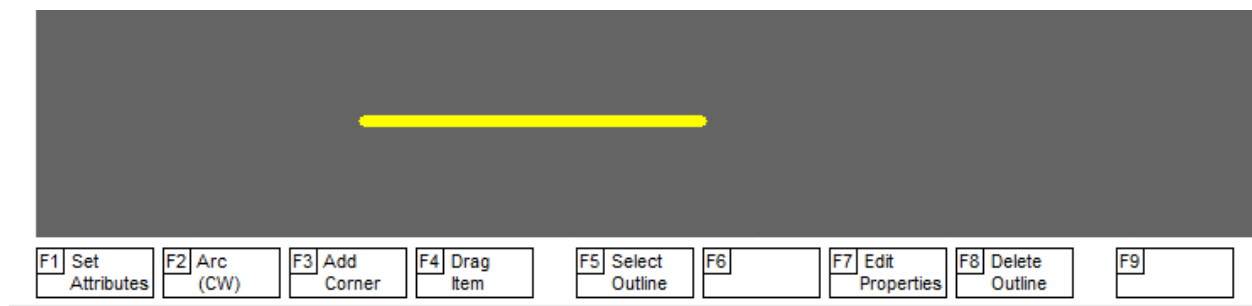
"F8- Delete outline "button. With this button you can completely remove the polyline.

"F7- Edit Properties "button. With this button you can bring up the "Add / Edit Polyline" dialog, which was shown in the previous screenshot. This window is also available when selecting a group of polylines, i.e. when one or more polylines are selected, or a part consisting of several polylines is selected, then you can change the properties for this group. By default, when this dialog box is called, the "Do not change ..." checkbox will be checked, to change any of the polyline properties, you need to uncheck this checkbox. In the GroupBox with the choice of the line width, uncheck the "Apply to lines in a width range" checkbox, and then the new width will be applied to all selected polylines. But you can also apply the new width not to all selected polylines, but only to those that fall within the range of widths indicated below in the MIN and MAX boxes.

We have specified the values "0" and "0.2", which means that the new width of 0.3mm will be applied only to lines that are currently no more than 0.2mm wide.

In the GroupBox of the layer selection there is a "Default layer" checkbox, which resets the polyline layer in accordance with what attributes the polyline currently has. For example, for a polyline with the "Net Name" attribute it will be the "Net Polyline" layer, for a polyline with the "Pin Name" attribute it will be the "Pin Line" layer, for a polyline with the "RefDes" attribute it will be the "Part outline" layer.

"F5- Select outline "button. This button selects the entire outline of the entire polyline. The function is also available when selecting several polyline segments (for a group).

"F4- Drag Item "button. This button activates the mode of dragging the polyline segment. The function is also available when selecting several polyline segments (for a group).

"F3- Add Corner "button. This option allows you to insert an additional vertex on the selected side of the polyline. For example, let's insert a new vertex so that the side of the polyline makes an angle of 45 degrees to the horizontal
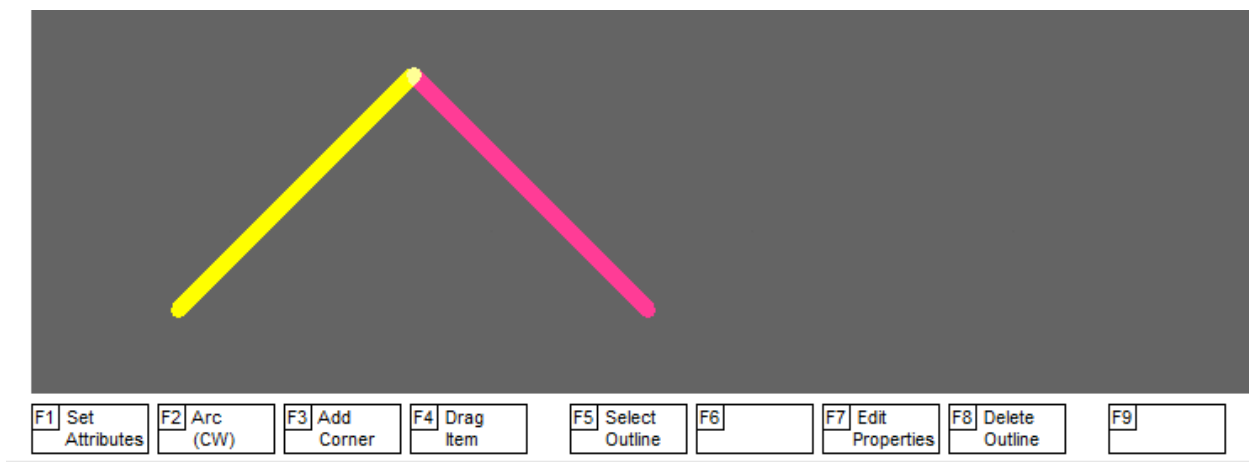
"F2- Arc (CW / CCW) "button. This option allows you to turn a slanted line segment into an ellipse element. Press several times to get the desired twist direction.

"F1- Set Attributes "button. Click on F1 to bring up the "Set Attributes" dialog box. This dialog box stores all the text attributes of any polyline. In total, the polyline in Schematic Constructor 1.3 has 6 text attributes:

1) Reference Designator
2) Value text
3) Footprint name
4) Pin name
5) Net name
6) Description text

Depending on which attribute is active, the polyline can take the status of a net polyline of a circuit, a polyline of a pin of a part, or a contour line of a part.
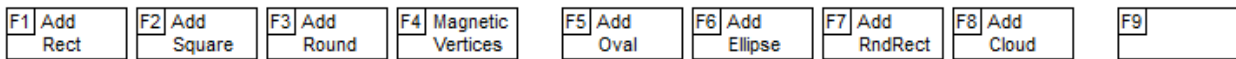

On the right, for each of the text attributes, you can select an individual letter size and font line width. These values are saved when this dialog is closed, and restored the next time it is called, and saved to the project file.

Draw simple shapes

Click on a segment of any polyline, and then reset the selection by clicking on an empty space or pressing the button<mark>ESC</mark>... Now in the mode when nothing is selected the button "<mark>F4 - Repeat Polyline</mark>". By pressing this button, the program will immediately switch to the polyline drawing mode by applying the properties of the last selected polyline on which you clicked with the mouse. When you first entered this drawing mode, the bottom menu with simple objects became available. Click on the button corresponding to the object you want to draw

| F1 Add Rect | F2 Add Square | F3 Add Round | F4 Magnetic Vertices | | F5 Add Oval | F6 Add Ellipse | F7 Add RndRect | F8 Add Cloud | | F9 |

<mark>F1</mark> - Add rectangle

<mark>F2</mark> - Add square

<mark>F3</mark> - Add circle

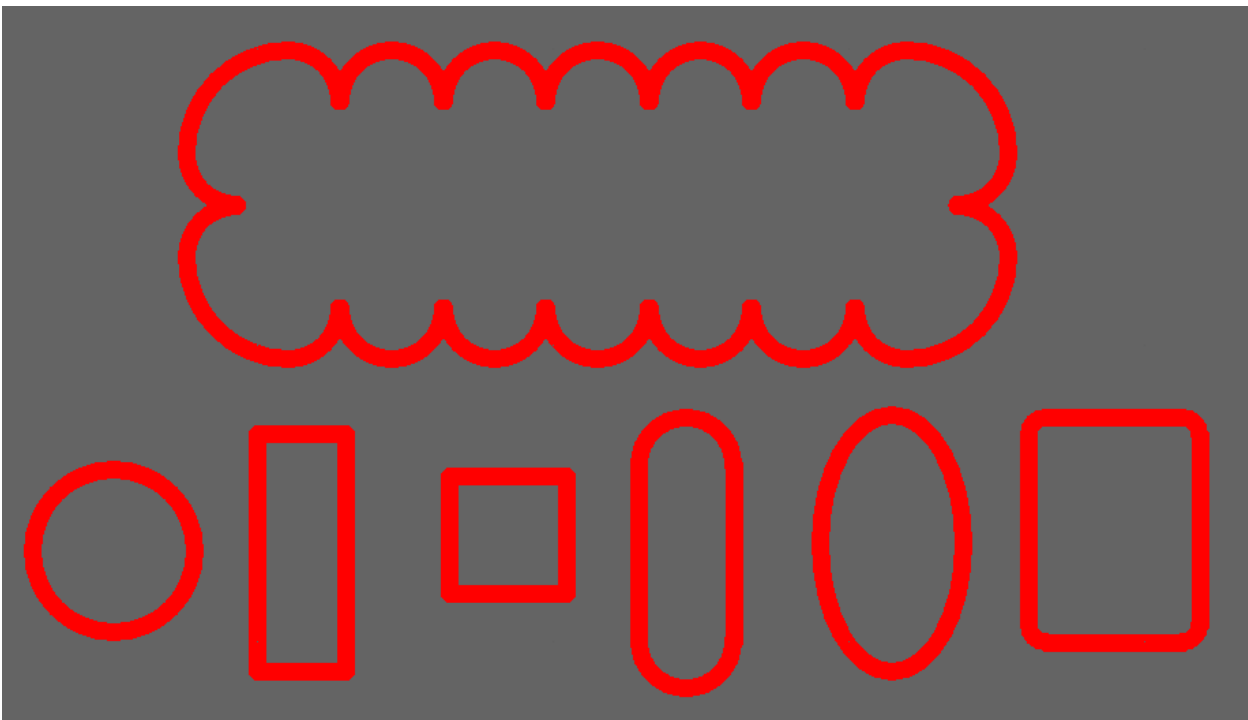<mark>F5</mark> - Add an oval

<mark>F6</mark> - Add ellipse

<mark>F7</mark> - Add rounded rectangle

<mark>F8</mark> - Add cloud

After selecting the object, drag the mouse while holding the left button from the upper left to the lower right corner. When the button is released, the program will enter the object that was selected into this rectangle.

You should be aware that the rectangle and square object at the end of drawing are automatically snapped to the current grid, and can be distorted in the case of a large grid, when the size of the drawn object is commensurate with the grid step. Select an adequate grid spacing once and do not change it whenever
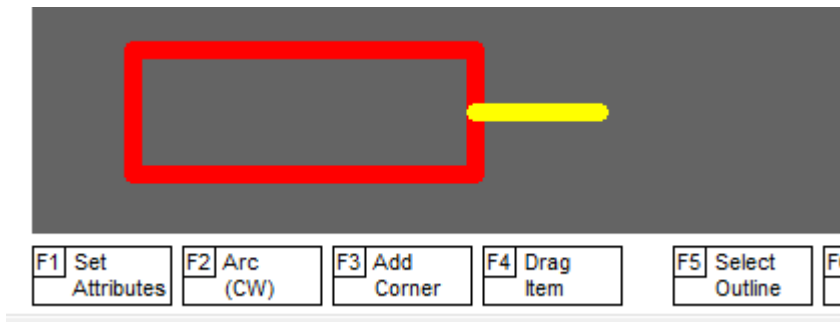
possible, so that, for example, when importing a part from another project, the part is imported without distortion. Typically, the spacing of the grid of polylines is in the range from 0.1 to 1.0 mm.

| Grids: Visible | 5 mm ▼ | Polylines | 0.5 mm ▼ | Texts | 0.1 mm ▼ | Angle | 45 ▼ |

As already mentioned, once the rectangle is finished drawing, its vertices are automatically snapped to the grid. This binding can be canceled by pressing the standard CTRL + Z key combination.

Draw a PCB detail

Draw a rectangle, and a polyline to the right of it, consisting of one segment.

| F1 Set Attributes | F2 Arc (CW) | F3 Add Corner | F4 Drag Item | F5 Select Outline | F6 |

Select the line on the right and set the pin name via the "F1 - Set attributes"

Click OK and the program will add the text "1" next to the polyline. Please note that after adding a new attribute, the text will have a selection so that the developer can move it to the desired location. Therefore, without deselecting the selection, press F4 (Move) to move it closer to the pin line. When you select any text,

the polyline to which this text is anchored is also highlighted. You cannot add text to the Schematic Constructor without binding to any polyline.
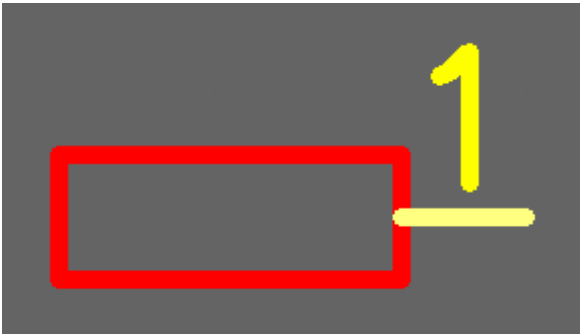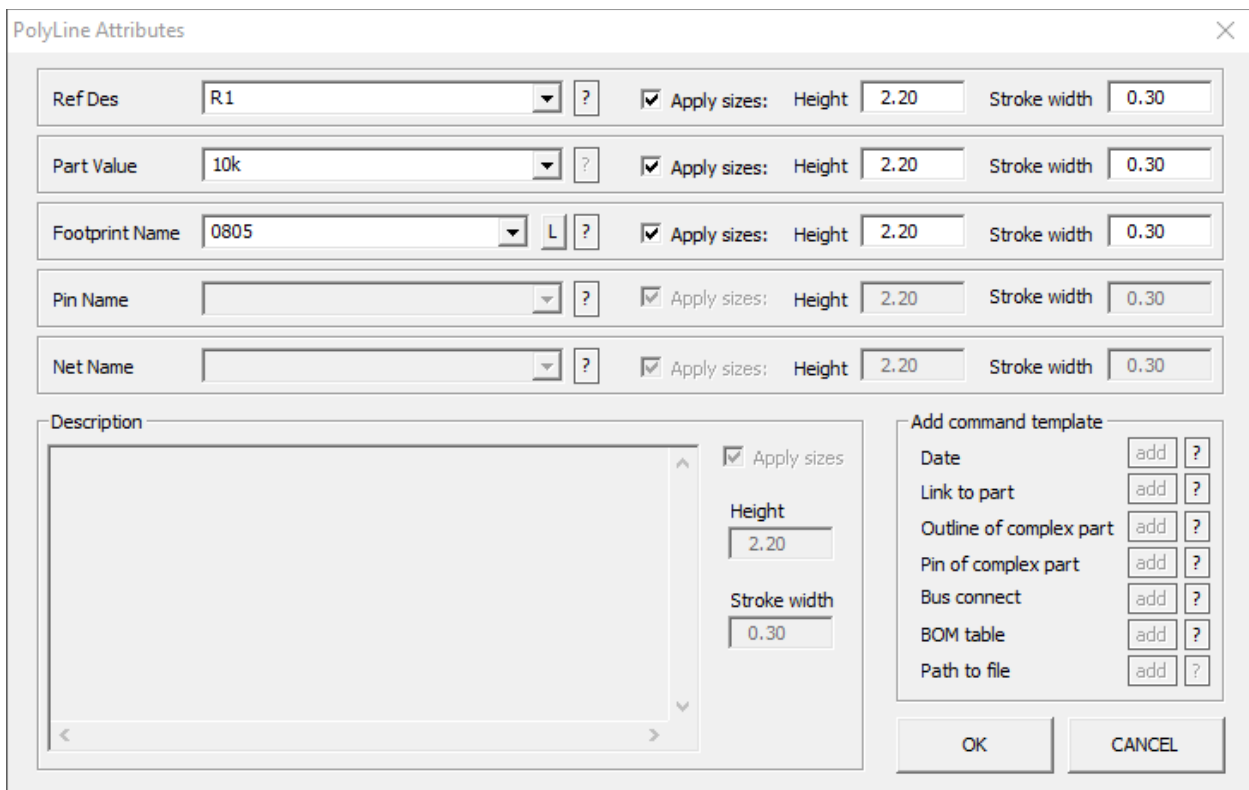

Since the resistor does not need to display the pin name, we must hide this text. To do this, select the text "1" and press "F1 - Edit Text". In the dialog box, enter text size = 0 and text stroke width = 0.

Now select with the mouse with the frame all the objects that you want to combine into a part, and press "F1 - Set Attributes".
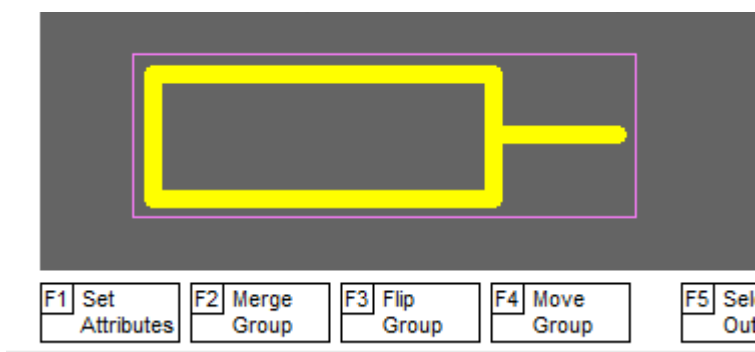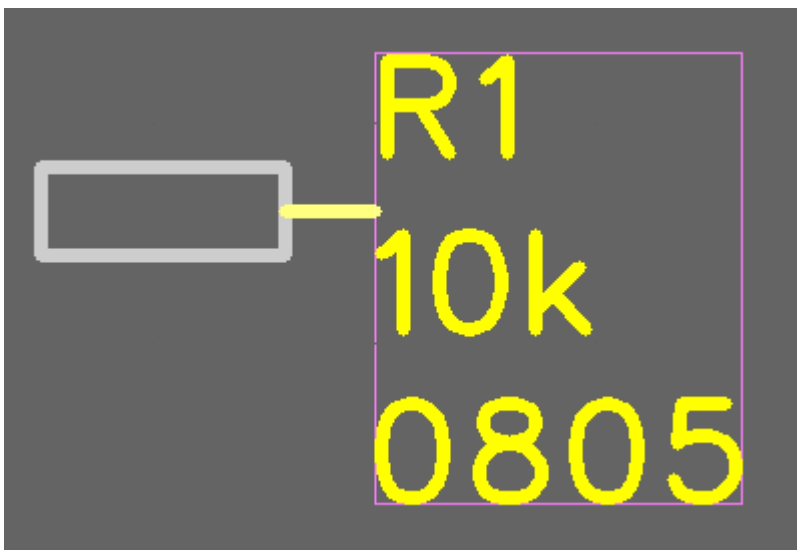
Set the Positional Designator (for example R1), value (for example 10k), and the name of the footprint (for example 0805)

Three text objects will appear, which will also have a selection for easy movement of these objects immediately after they appear. Without deselecting, press F4 (Move) to move them to a suitable location.


Attribute alignment:

Now let's return to the point of aligning these attributes on the working area. There are two ways to align these text attributes:

1) On the highlighted reference designator, right-click and select any of the 4 alignment types. The alignment order "Ref Value Footprint" is fine for us. The program will arrange these texts and also

apply clearance coefficient, which is specified in the project settings (see page 2 Paragraph "Beginning of work")

2) The second way - in the mode when nothing is selected, press "F2 - Drag group"To enter continuous drag-and-drop mode. In this mode, click on the RefDes text and the program will capture three attributes of the part at once. Then click on the point where you want to place these attributes, and the program will automatically apply the alignment taking into account the clearance coefficient. This option is useful when you need to move or align the attributes of several parts in a row.

Do not forget to enable the part selection mask so that when you click on the contour polyline of the part, the entire part is selected. In the allowed mask mode (green square), you can copy pins from any part, and the program will automatically increment the names of new pins. In the mode of the forbidden part mask (red color of the square), any copied part will be destroyed into its constituent polylines
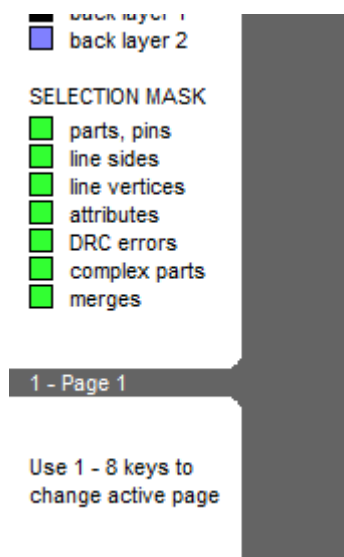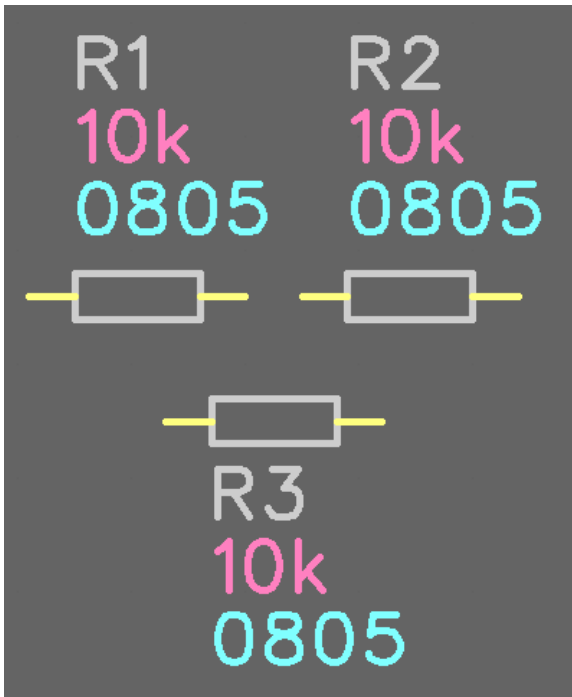

So, in the resolved part mask mode, select the part pin on the right and copy it using the standard shortcut keys CTRL + C and CTRL + V. Insert it to the left of the part outline polyline. We got a resistor with two pins named "1" and "2".
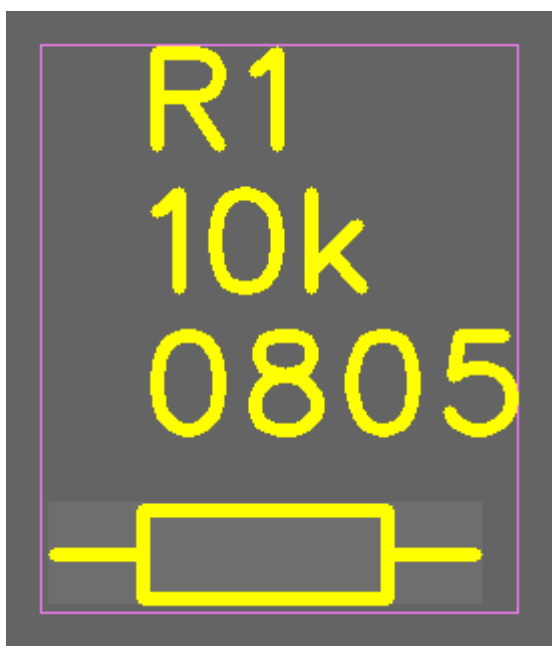
Copying a part

Now select the resistor and copy it twice using the standard keyboard shortcut CTRL + C and CTRL + V.

Now we are going to draw the pin connections. To do this, click "F4 - Repeat Polyline"To enter the polyline drawing mode. In drawing mode there is a button "F4 - Magnetic Vertices", Which enables / disables the attraction of the vertices to each other. When drawing connections between pins, this option will be very useful. You can turn this magnet on and off, depending on the need for it at the moment, by repeatedly pressing the buttonF4...

| F1 Add Rect | F2 Add Square | F3 Add Round | F4 Magnetic Vertices | | F5 Add Oval | F6 Add Ellipse | F7 Add RndRect | F8 Add Cloud | | F9 |
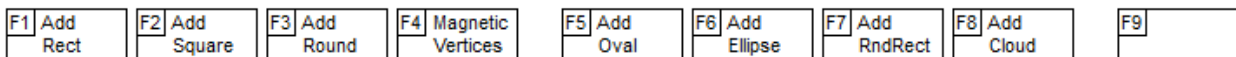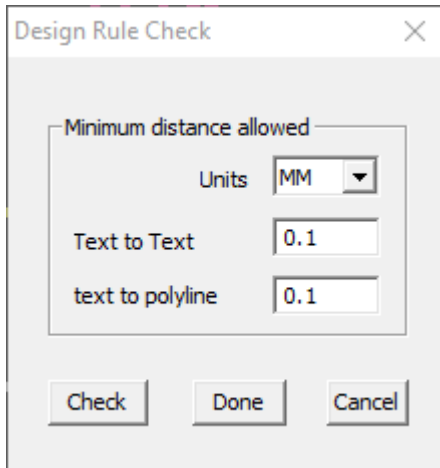
Connect the pins as shown in the picture.
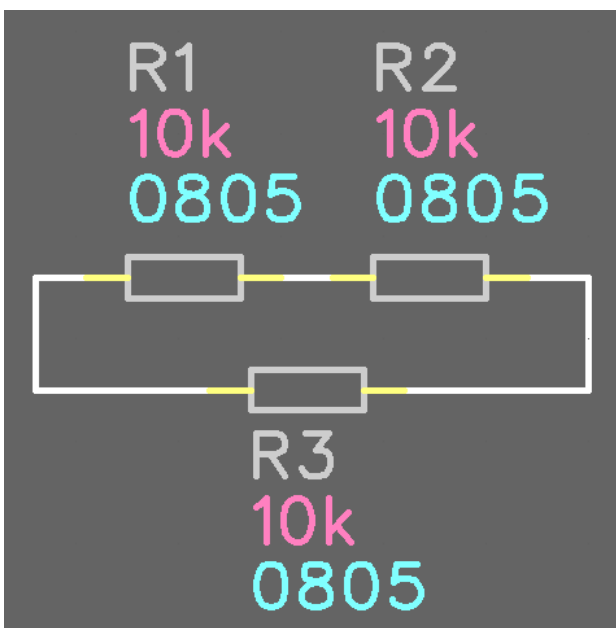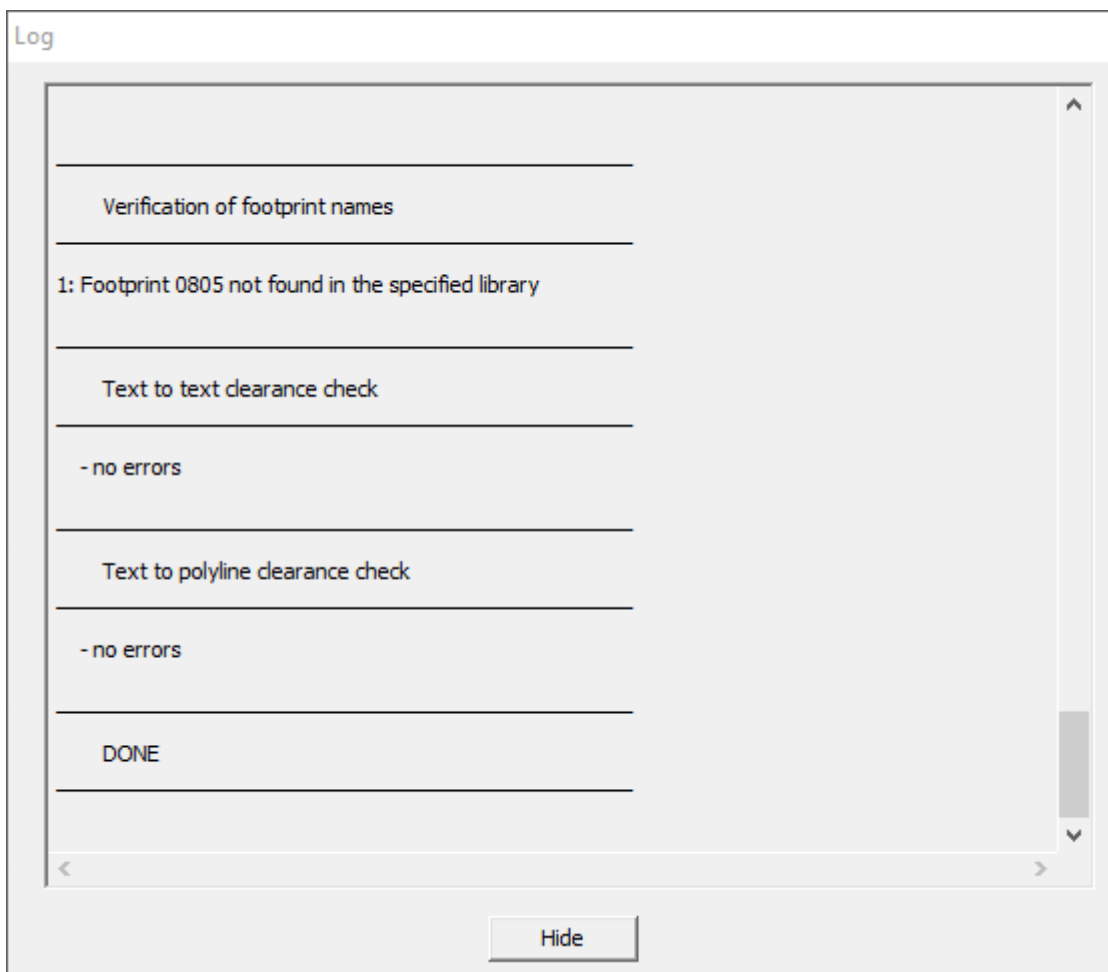
DRC circuit design check

Schematic Constructor1.3 is equipped with a simple tool for checking the clearance of text attributes on a polyline. The program checks the clearance of the text object rectangle to any polyline. Enter the required values and click the Done button.

Design Rule Check

Minimum distance allowed

Units	MM ▼
Text to Text	0.1
text to polyline	0.1

Check   Done   Cancel

Then press the shortcut DRC: CTRL + D. As a result of the check, we got one varning: "footprint 0805 is not present in the specified library".

Log

Verification of footprint names

1: Footprint 0805 not found in the specified library

Text to text clearance check

- no errors

Text to polyline clearance check

- no errors

DONE

Hide

Let's try to select all the text objects 0805 using the menu option of the right button - finding similar texts. On the selected text 0805, call the context menu with the right mouse button and select "Select Similar Texts"
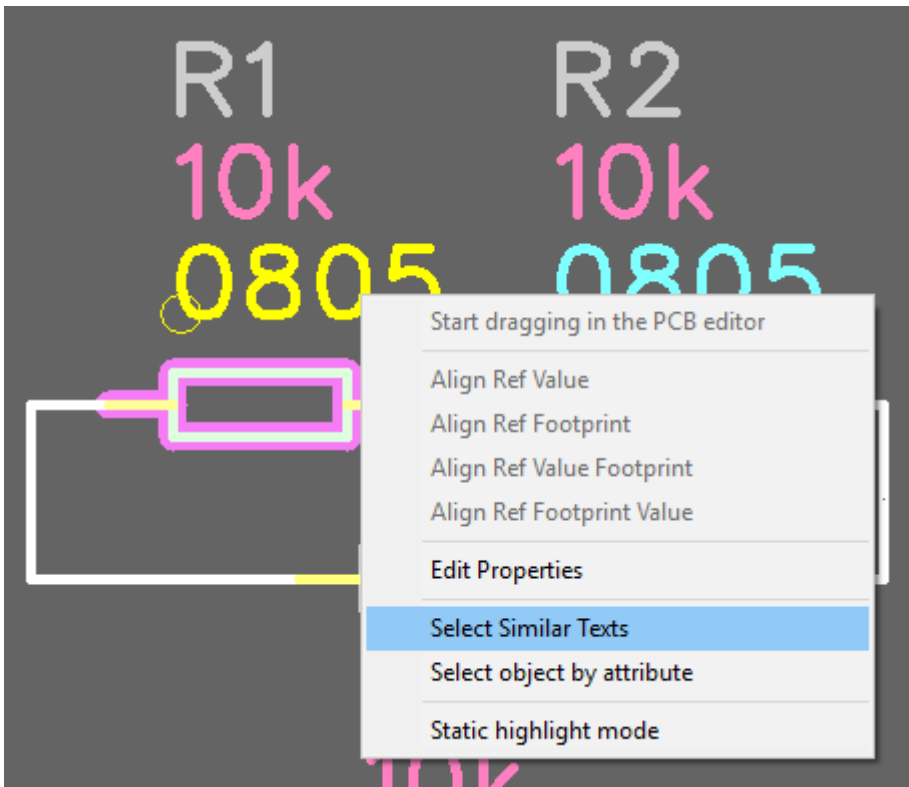
Now, when the text is selected, call the context menu again and select "Select Parts by Attribute"
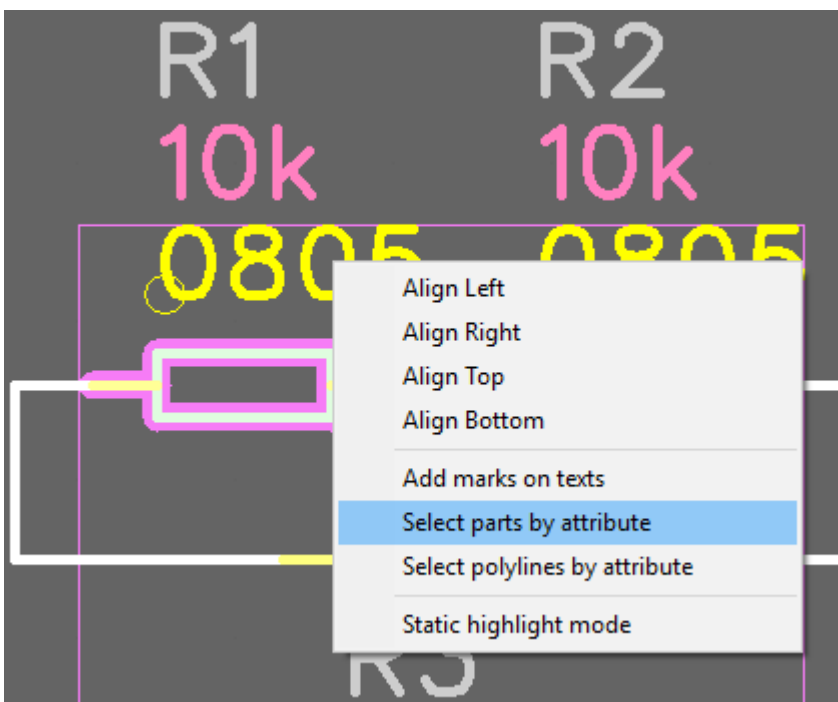
There are two ways to change the footprint name for a group of selected parts:

1) Click on F1 (Set Attributes) and choose a different footprint name for these three resistors. The advantage of the way to select a footprint through the "Set Attributes" dialog is that we see a complete list of footprints in front of us in a drop-down combo box. The disadvantage of this method is that the Value attribute for all selected parts must be the same.
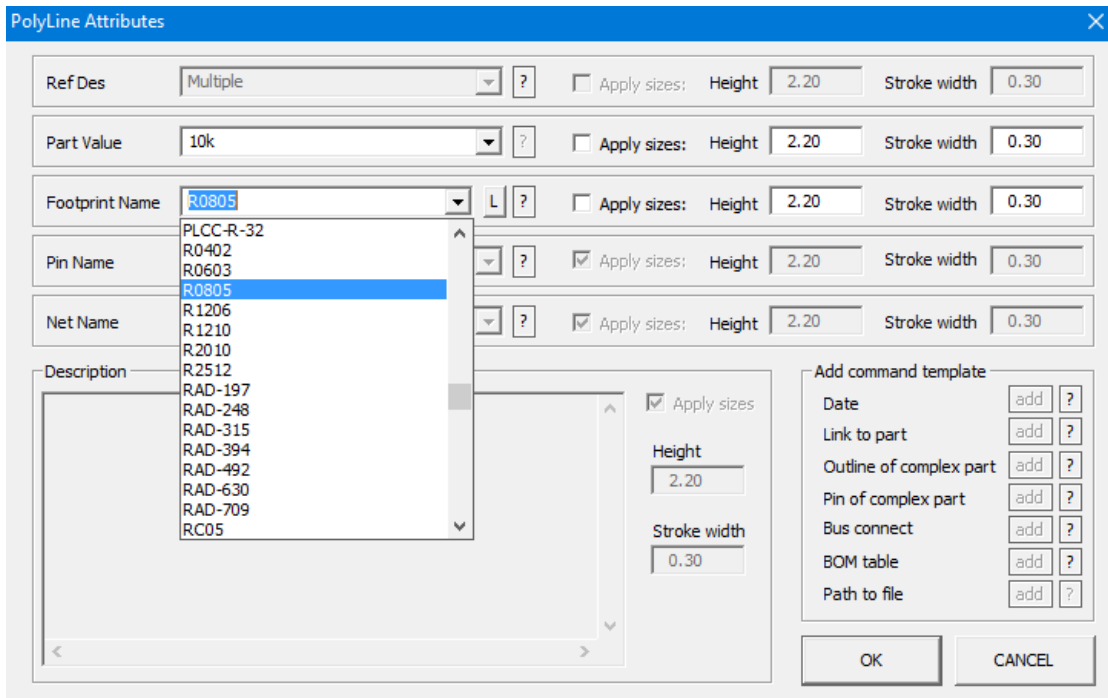
Convenient search: if you do not know the exact name of the footprint, you can enter only part of the name of the footprint and then open the drop-down list. In this case, the program will sort the list by making the first those footprints that have a part of the entered string in their name.

2) For the next method to change the name of the footprint, select the "Tools >> Change Part Attributes >> Footprint". Using this option, you can change the name of the footprint regardless of the Value parameter. That is, the Value parameter can be different for the selected parts. The disadvantage of this method is that there is no drop-down full list of footprints.
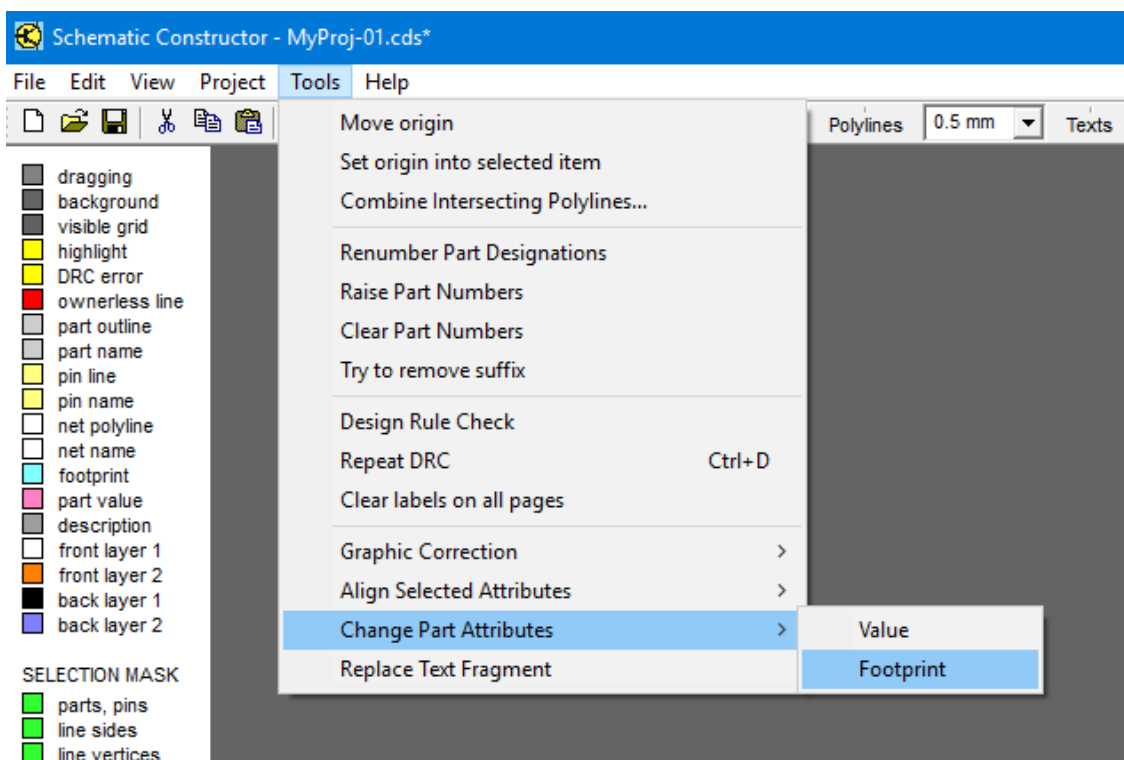
PADS-PCB netlist generation

Call the netlist settings dialog via the menu "File >> Netlist Settings".
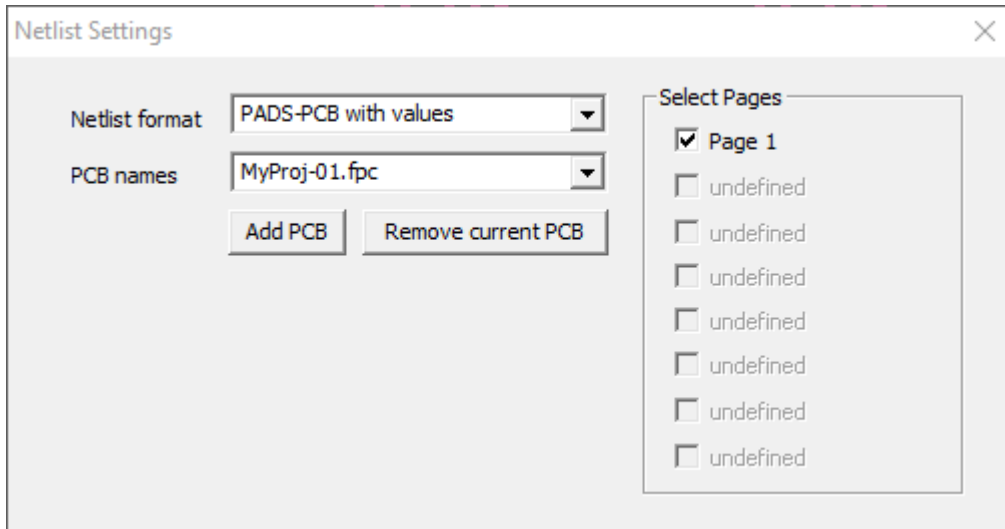
Select the PADS-PCB netlist format that is supported by the compatible PCB editor FreePcb-2.3. The next dialog box below contains a list of connected PCB files, where you can add PCB file names and link them to project pages. Since we have a one-page design, we cannot do this (since we do not have free pages in the project that we could link to other PCB files). When creating a project, the program automatically creates a PCB file name that matches the schematic file name. You can change the name by deleting the current name and adding a new one. We see that we have a PCB file called MyProj-01.fpc. In fact, this file is not yet in the directory next to the schematic file, but it will appear, as soon as we press the view switch button on the PCB-editor. (Select the menu item "View >> Switch To PCB Editor", And the program will create (if not already created) and switch to the PCB file by opening it in Freepcb-2 editor. But now this is not at all necessary)
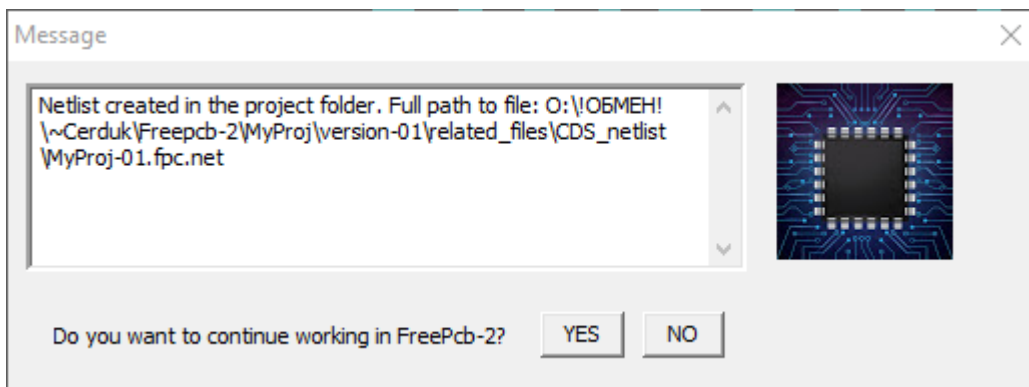
Now let's create a netlist of our schematic consisting of three resistors and load it into the PCB editor. The menu button is called "Save with Netlist file", And it is located in the main menu"File". The program will do an additional design check and check the contacts of the circuits, and then create a netlist and display information about the file path in a dialog box

To automatically load the path to the created netlist into Freepcb-2, press the YES button in the dialog box. The PCB editor will start and the external netlist import window will pop up, in which the import options will be set by default as in the screenshot below.

Import Options

You are importing a netlist file into a project that already contains parts and/or nets

For parts in project that are not in netlist file
- ⊙ Remove parts
- ○ Keep parts but remove connections to nets
- ○ Keep parts and connections to nets

For parts in project whose package or footprint doesn't match netlist file
- ⊙ Change footprints
- ○ Keep footprints

For nets in project before import
- ☑ Remove all nets that are not in netlist file
- ☑ Total control of nets present in netlist file on parts present in netlist file
- ☐ Rename nets of the NETxxxxx format when names match

For traces and copper areas connected to pins whose nets have changed
- ☑ Retain traces
- ☑ Retain stubs
- ☑ Retain areas if possible

| Import | Save project with these settings then import | Cancel |

It remains to press the button Import and the file will be loaded

Log

```
Reading netlist file "O:\!ОБМЕН!\~Cerduk\Freepcb-2\MyProj\version-01\related_files\CDS_netlist\MyPrc
Indexing library folder "o:\!обмен!\~cerduk\fp_lib\lib"
Indexing library: "Ivex, Allan Wright.fpl"
Indexing library: "Ivex.fpl"
Indexing library: "unknown.fpl"


Importing parts into project:

Importing nets into project:
    Adding pin R2.1 to net "NET00001"
    Adding pin R3.1 to net "NET00001"
    Adding pin R1.1 to net "NET00002"
    Adding pin R2.2 to net "NET00002"
    Adding pin R1.2 to net "NET00003"
    Adding pin R3.2 to net "NET00003"

Moving traces and copper areas whose nets have changed:


Checking tees and branches:
  0 tee_IDs in array:

************** DONE ****************
```
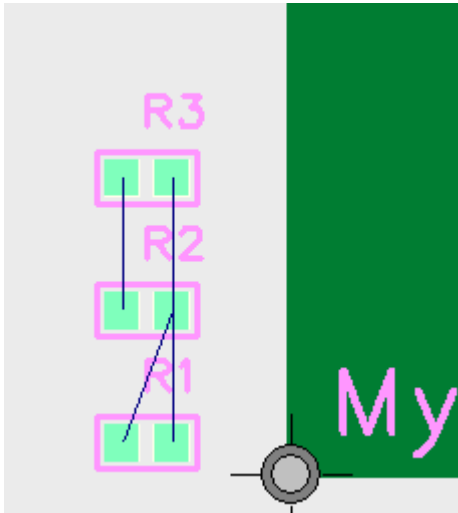
| Hide |

Here, on the left, three resistors have appeared, connected according to the diagram.

Remember to save the PCB file so that the schematic editor does not display an error when trying to modify a schematic with an unsaved PCB file.

Working in the PCB editor

Let's take a quick look at the Freepcb-2 PCB editor. Schematic Constructor1.3 created a PCB file in which there is a sample of the PCB outline with a size of 50 x 100mm, as well as texts with the name of the board in the layers of the upper and lower silkscreen. The board outline is a closed solid polyline. You can move the vertices of this polyline by clicking on the vertex and pressing the function buttonF4 (Move)... There is also another way to change the configuration and dimensions of the contour line. Since we have only three parts, we need to make the outline much smaller - let's make it 10 x 15mm. Let's start by cutting the board into 2 pieces, one of which is then simply removed. Select the side of the contour polyline and press the "F7 - Add Cutout"
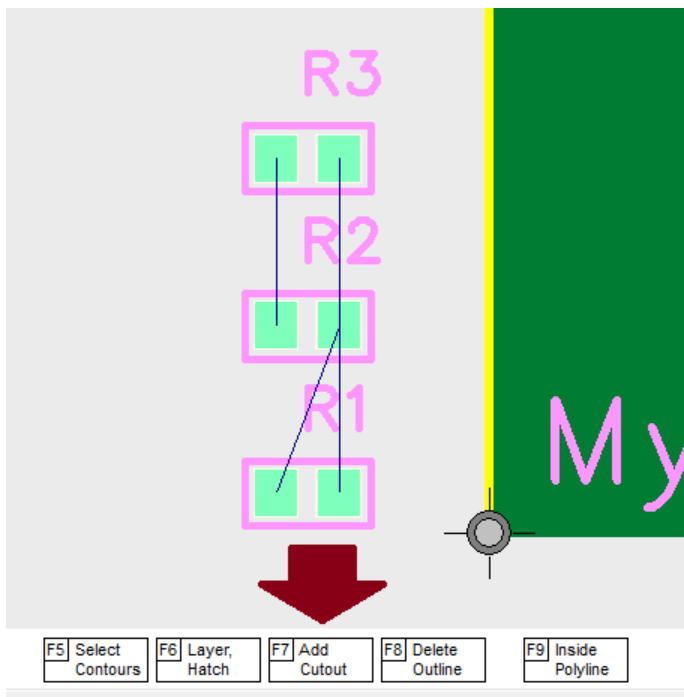
The program will switch to the mode of drawing polylines. Add a cutout in the form of a polygon as shown in the following screenshot by left-clicking on the points of the working area in drawing mode.
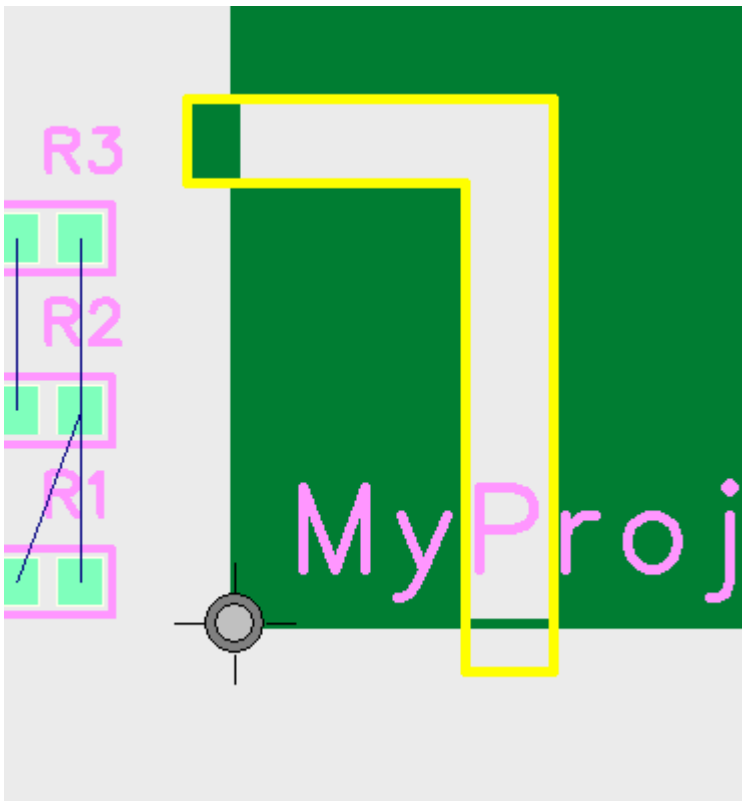
Now you need to combine the contour and cut using the option found in the main menu "Project >> Combine Intersecting Polylines". Thus, you can connect not only a contour with a cut, but also two intersecting contours.
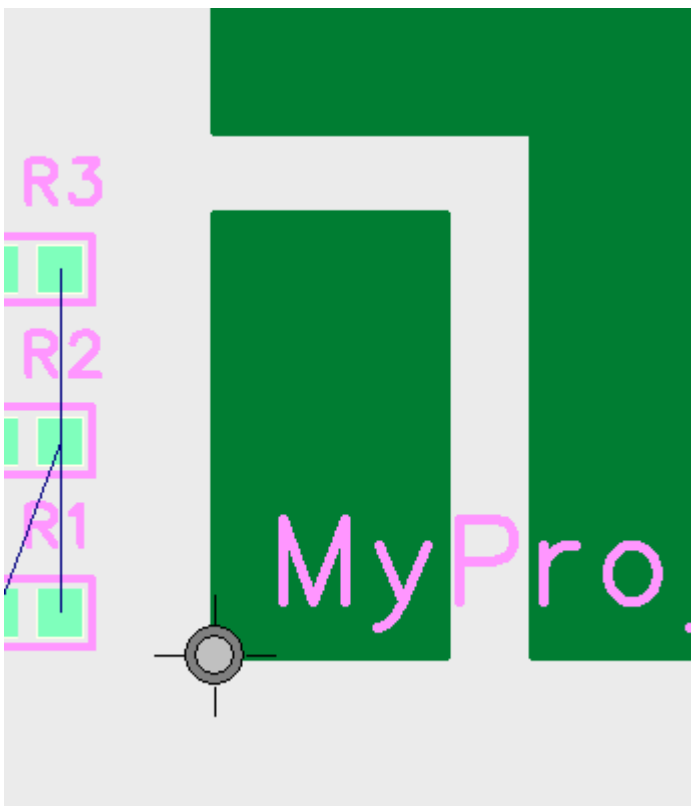
Delete the large piece of the path by selecting its side and pressing the "Delete" button. Now we have to, as we wanted to make the size of the outline 10 x 15mm. Since the origin of coordinates coincides with one of

the vertices of the contour (lower left vertex), we will set the coordinates of the opposite vertex, located diagonally from zero. Select this vertex with the mouse frame as shown in the picture.
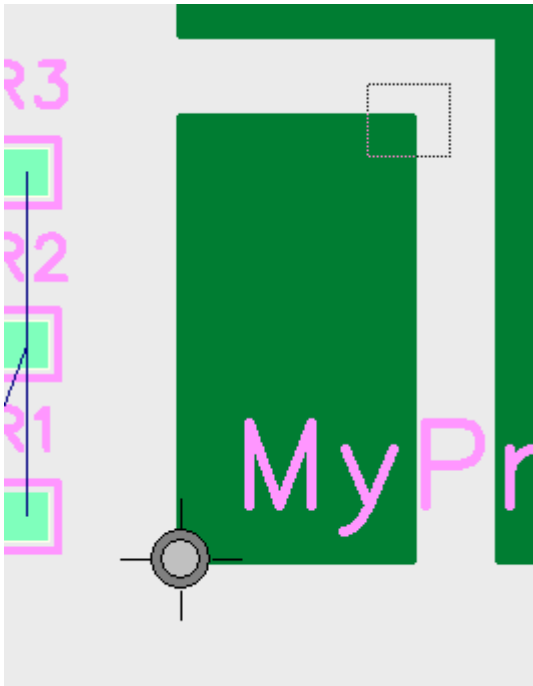
Why with a frame, but simply because it is more convenient and faster than aiming at the corner of the polyline, trying to hit it with the mouse pointer. In the mode when the vertex is already selected, press the function button "F1 - Set Position", And in the dialog box that appears, enter the coordinates X = 10, Y = 15.
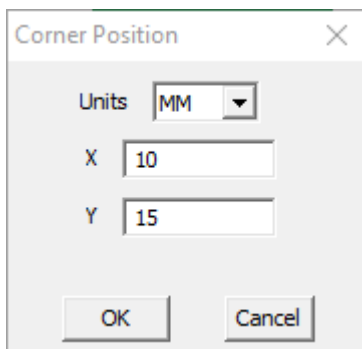
Since we want to move only the selected vertex, and not the entire path, then click the NO button as shown below
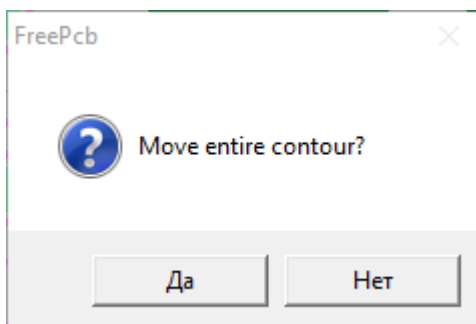
The result is this form
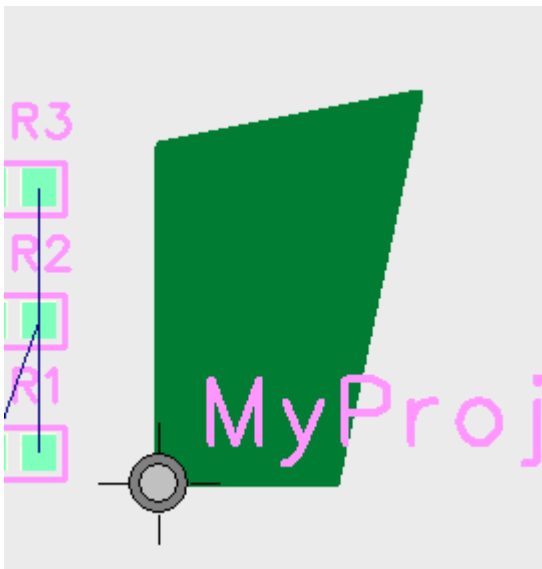
Now select the lower right vertex with a frame and press the function button "F3 - Align"To align the vertex with a multiple of the alignment angle. The alignment angle must be set in the top toolbar (the rightmost combobox in the screenshot)

The angle is now 45, which is the most commonly used value. This angle is also used when routing conductors on a printed circuit board, which aligns the tracks in multiples of a 45 degree angle.

Align the top left vertex in the same way.

As a result, we have a contour of 10 x 15mm exposed by coordinates.

A very useful option when working with polylines is to move the origin to any vertex of the polyline. To move the origin to another vertex, first select that vertex and then choose the "Tools >> Set Origin Into Selected Item", Or press the hotkey"O". You will find a list of all hotkeys in the "Help >> Keyboard Shortcuts".
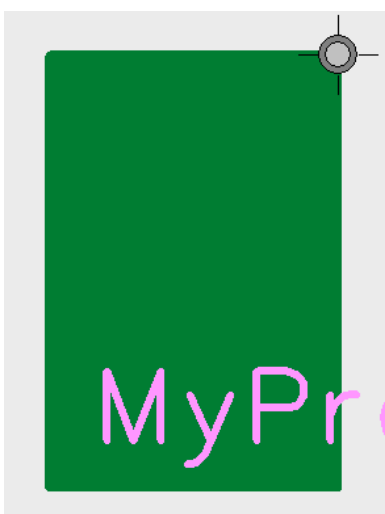
Now we need to move the parts onto the PCB by placing them inside the contour line. Although in our example there are only three parts and it is quite easy to find them on the board, imagine as if we have a huge project and it is impossible to find the desired part according to the diagram on the board. First, let's select a

full screen view of the printed circuit board, with center alignment. Select the menu item "<mark>View >> Show Board Outline</mark>".

Now switch to the schematic editor window and select the text R1 (RefDes). Call the context menu and select the "Start dragging in the PCB editor" item.
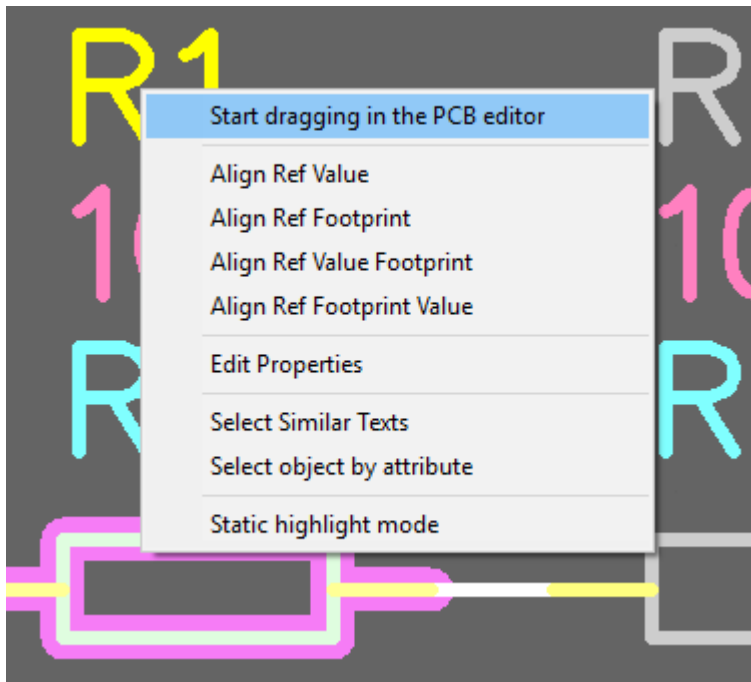
The program will switch to the PCB editor and move the selected part to the center of the screen, wherever it may have been. Click the left mouse button to complete the move. Move the remaining parts R2 and R3 in the same way. It turned out the following location:
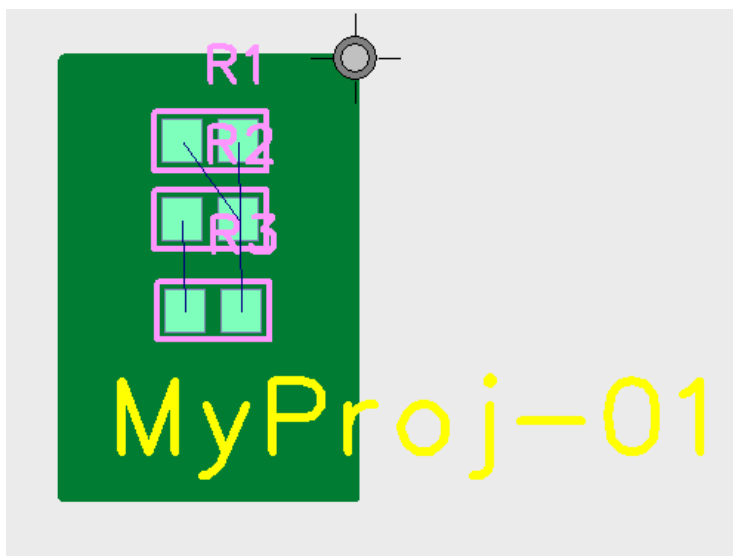
By the way, it is desirable to reduce the text of MyProj-01. Highlight the text and click "<mark>F1 - Edit Text</mark>". In the Edit Text Parameters dialog box, set the new text height and font line width
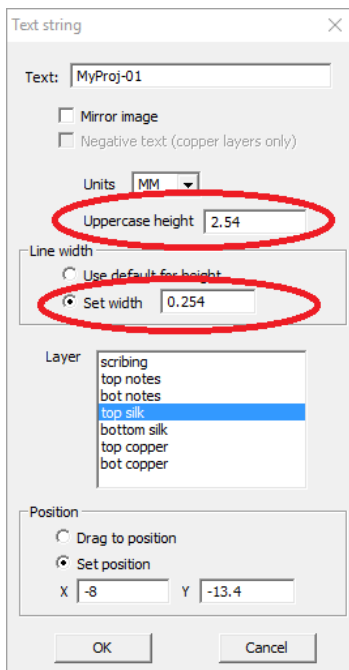
Enter a value of 1.0 x 0.15mm so that the text fits inside the contour line. Move the text to the desired location by highlighting it and pressing the function button F4 (Move)...

There is another way to select parts on a printed circuit board from the schematic editor, which allows you to immediately find a group of parts and select them on the board. This will not move the parts to the center of the screen; on the contrary, the focus of the screen will move to the parts. Switch to the schematic editor and select all three resistors with the mouse frame at once. The button "F9 - Switch To PCB". Click it, the program will switch to the PCB editor window and highlight the selected parts on the board. Without deselecting the selection, you can press F4 (Move) to move the group in the PCB Editor. It happens that in this group the parts are far from each other, so it is advisable to collect them in a heap. To do this, first move this group (with the details spaced out on the sides, as it is) away from all objects somewhere to the side in order to then group them automatically (bunch together) using the function button "F5 - Part Autopos.. ". Press it several times to change the positioning algorithm, and achieve the desired location. It is possible that the program will not arrange the parts the way you want, so manual override is fine.

What function buttons are available in the drag-and-drop mode of a part. Select any part by clicking on it with the mouse and press the function button F4 (Move)... The program will enter drag-and-drop mode, in which the following function keys will become available:

F1 - change the side of the part (Top \ Bottom)

F2 - rotate the part clockwise 90 degrees

F3 - rotate the part counterclockwise by 90 degrees

F4 - recalculate (optimize) rat lines

F5 - if copper traces are connected to the part, then demarter them to the via, in other words, delete the routing before changing the layer of the trace.

F6 - if copper paths are connected to the part, then de-route them from start to finish

F7 - block the connections connected to the pins of this part.

F8 - unblock the connections connected to the pins of this part

F9 - break connections with all tracks.

Silkscreen display

You can resize the text of the reference designators in the silkscreen layer, and you can turn off the visibility of the text, for example, for the VALUE parameter of parts, or turn off the visibility of the lines of the silkscreen layer for parts.

Hold down the CTRL key and click on the Parts line in the selection mask, which is located on the left side of the main window. The mask will be inverted and will look like this
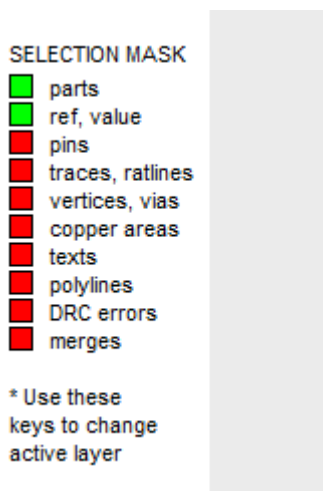
Thus, we have just enabled the selection of parts and silk-screen printing, and disabled the selection of other types of objects. In this mode, when a group of parts is selected, its own menu opens, in which there is a button for editing silk-screen printing. Select all three resistors with the mouse frame and click the "**F9 - Edit Silk**". The program will display the second page of the menu, in which all buttons will be somehow related to editing silkscreen parameters.
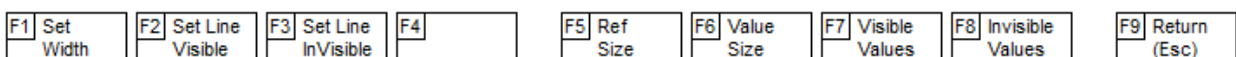
F1- sets the width of all silk-screen printing lines of the selected parts equal to the size specified in the global settings for gerber files. Call this dialog using the CTRL + G combination, or through the menu item "**File >> Generate CAM files**". Set the width of the lines in this dialog box (circled in an ellipse).

F2 - enable display of silk-screen printing lines on the screen and in gerber files.

F3 - disable the display of silkscreen lines on the screen and in gerber files

F5 - set the size of the tag text for the selected parts

F6 - set the text size VALUE for the selected parts

F7 - make VALUE-text visible.

F8 - make VALUE-text invisible.

F9 - return to the previous menu.

Auto-alignment RefDes

In the same selection mask mode, which allows only parts to be selected using the mouse frame, select a group consisting of several parts. It is not worth highlighting a huge amount of detail at a time, because the processing will become suboptimal and the function will slow down. The best option is to select 10 to 15 details. With the details highlighted press the button F7 and the program will place reference designators around the parts. With a dense arrangement of parts, there is a possibility that the algorithm will not be able to find a location for the text. In this case, these parts (for which the program could not find a location for RefDes) will remain selected on the board, so that you can, for example, reduce the size of the Ref-text for these parts with the button F5, and the selection is automatically deselected from the rest of the details. After placement, the board takes the form:
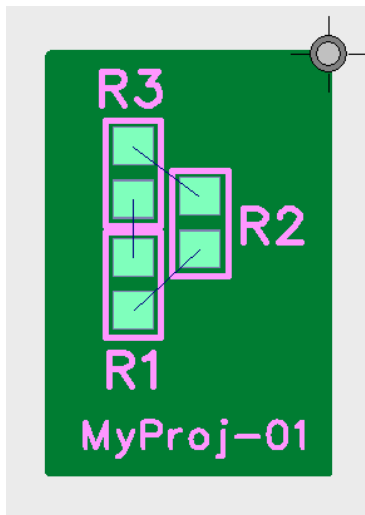
Copper tracing

Return the selection mask to its original state by clicking on the "Parts" line while holding down the CTRL key.

Then highlight the rat line segment and press the "F4 - Route Seg"
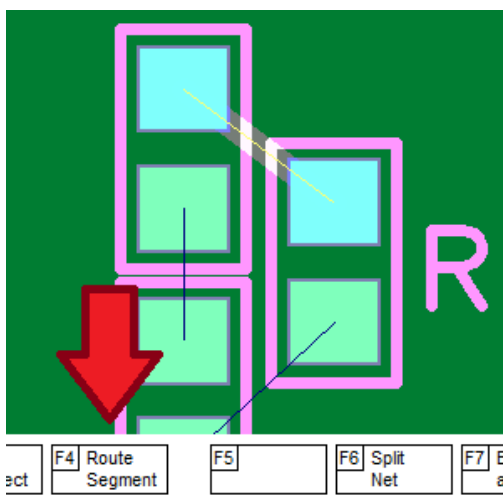
Tracing will start from the pin that was closer to the mouse cursor at the time the key was pressed. Therefore, move the mouse cursor in advance to the pin from which you want to start tracing, and then click F4... The program will switch to trace mode. In this mode, you can change the width of the tracks using the buttons F1 and F2 (increase decrease). In this case, the program iterates over the track widths according to the list specified in the project settings. (The project options window is located in the "Project >> Options".) Using the button "F3 - As Pad" You can make the track width equal to the pad width of the footprint. Click to indicate the location of the first vertex of the copper trace. Then press the button "F4 - Complete" To end tracing.
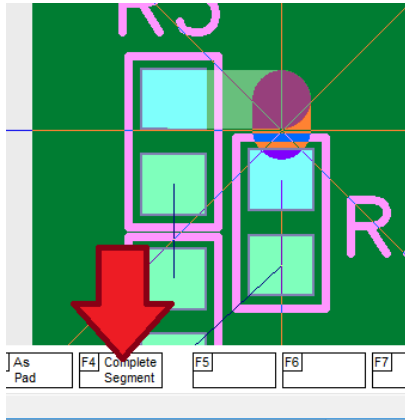
After that, the last vertex of the alignment will be selected, which you can align (relative to adjacent vertices) by pressing the "F3 - Align".
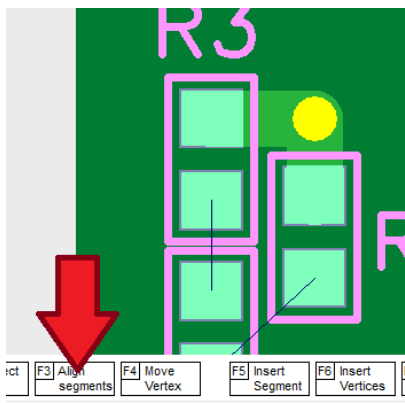
Then, without deselecting the vertex, you can insert an additional route segment in place of this vertex by pressing the button "F5 - Insert Seg".
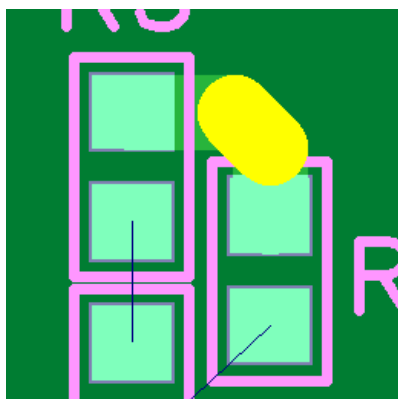
Trace the remaining undistributed rat lines in the same way. You can find more detailed information on tracing on the website. https://freepcb.dev/How_to.html

PCB design check

Call the DRC check window via the menu "DRC >> Design Rule Check". Enter the acceptable values of ground clearance for different types of objects and click the "Check" button. (To quickly access the design check, press the keyboard shortcut CTRL + D.)
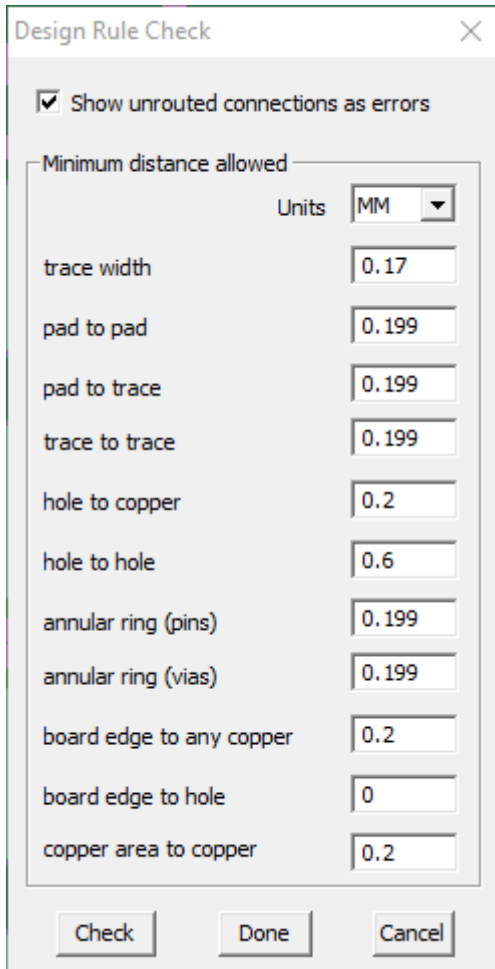
Since we have a simple design, this is where we will end the test. For a more complex design, you need to do a full check by selecting each DRC menu item:

- Check parts and nets
- Check traces
- Check connectivity
- Check copper areas
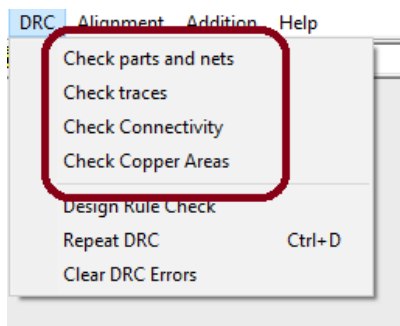
Generating gerber files

Call the CAM dialog using the keyboard shortcut CTRL + G, or use the menu "File >> Generate CAM Files".
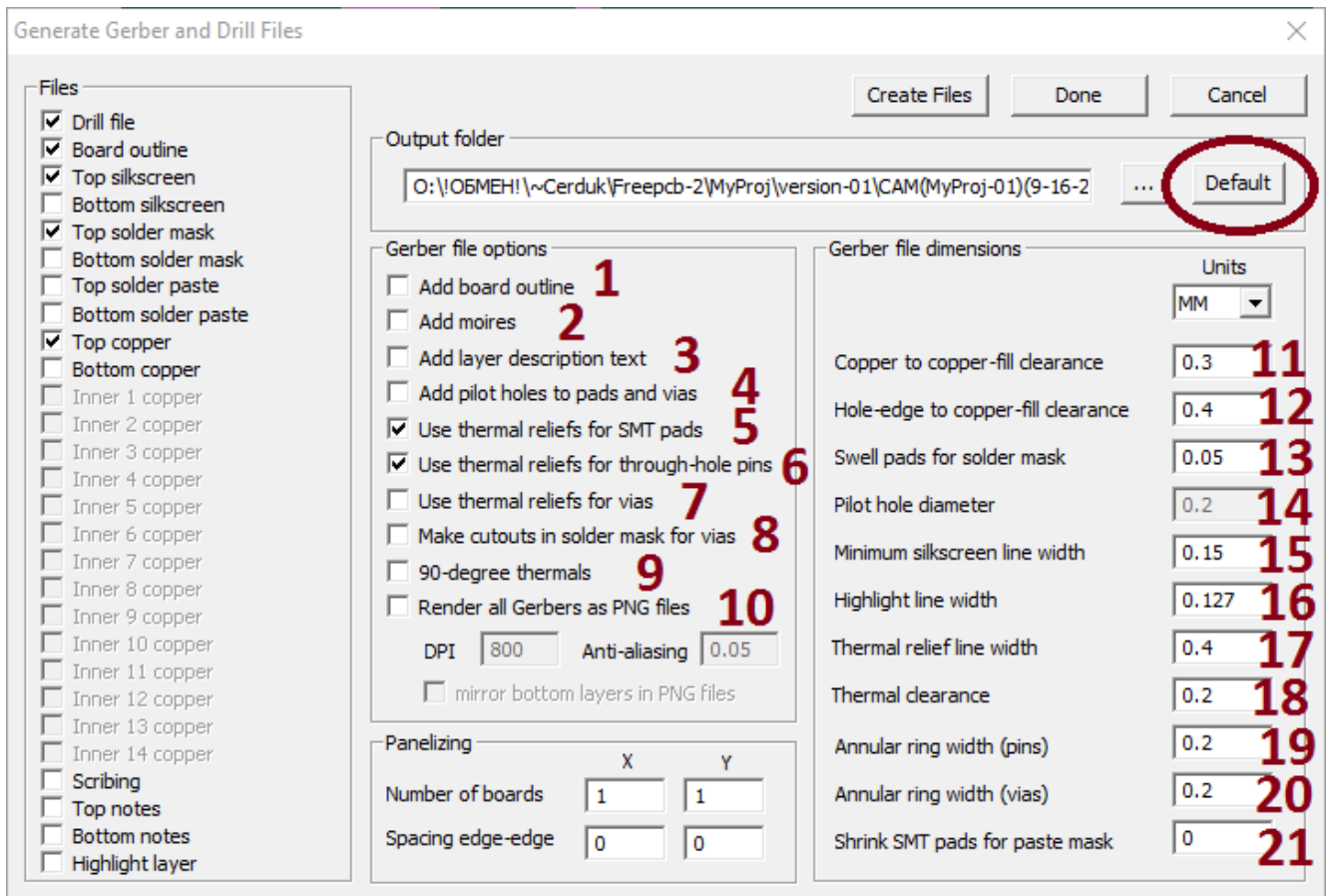
Click the DEFAULT button to generate a default path in the head directory of the project. Let's briefly describe the configurable options in this dialog box (see the numbers in the screenshot).

1) Add PCB outline polyline to all layer gerbera files. (you can select layer files in the left block of the dialog box)
2) Add moire to align layers

3) Adding text with the layer name
4) Add a pilot cut for manual hole drilling
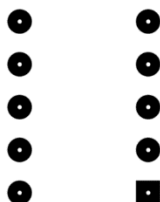
5) Adding a thermal barrier for SMT pads connected to the copper area G36 (G36 is a copper polygon for which gaps to other objects and thermal barriers are generated at the stage of creating gerbera files,

and in the program window this type of polygon is displayed simply as a contour polyline). When pouring copper areas with "Ghost", this checkbox also affects the fill parameters, instructing the program to add thermal barriers to the copper filled area by default (for SMT pads).

6) Adding thermal barrier for through pads connected to the copper area of the G36. When pouring copper areas with "Ghost", this checkbox also affects the fill parameters, instructing the program to add thermal barriers to the copper filled area by default (for SMT pads).

7) Adding Thermal Vias

8) Opening the protective mask on vias

9) Thermal barrier at an angle of 90 degrees to the horizontal

10) Create PNG files for all generated layers

11) Clearance for copper objects for polygons type G36

12) Clearance for holes for polygons type G36

13) Increase of protective mask on pads

14) Pilot cut diameter (see point 4)

15) Minimum silkscreen line width

16) The minimum line width of the highlight layer (the very last layer in the list of layers on the left)

17) Thermal barrier line width

18) Thermal barrier clearance on pad

19) A copper ring around a through pin, which is automatically generated on inner layers if the pin has no drop in the inner layer, but has a connection to a copper trace or polygon in that layer.

20) A copper ring around the via, which is automatically generated on inner layers if the via connects three or more copper traces on different layers

21) Reducing the paste mask on pads

Click the "Create Files" button to generate the selected gerber files. Then click the Done button to hide the dialog. In the mode when nothing is highlighted, press the function button "F5 - Open Folder"To open the current project folder and view the gerbera folder.

Create Net Labels

Let's go back to the schematic editor and create a shortcut to the net named GND. Draw a polyline consisting of three segments as shown in the screenshot
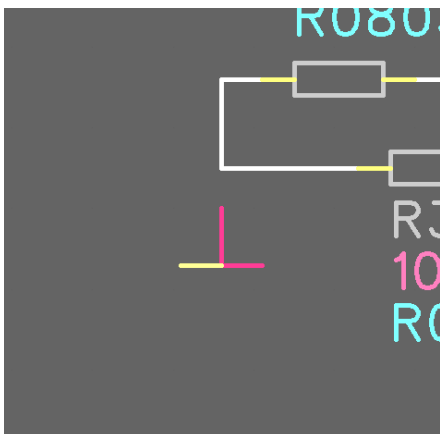
Select any segment of this polyline and set the attribute "Net Name" for it in the attributes dialog box, which is invoked by the function key F1... Enter the text "GND" in the "Net Name" field

The result is such a chain mark.

The net label can be connected only by one of the ends on the part pin or any polyline in the net, otherwise the program will generate an error when checking the DRC. Now we need to move it so that its end falls on another line of the chain. Click on the label segment and click "F5 - Select outline"To select the entire polyline. Click onF4to start dragging. The result is such a scheme
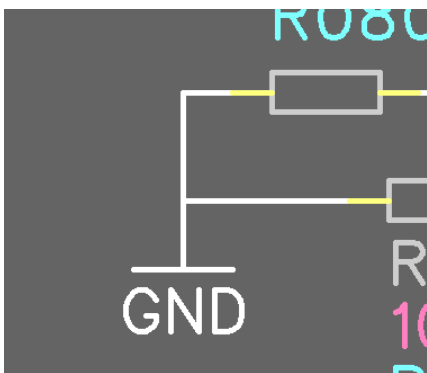
Click "F9 - Recalc.Nets"(In the mode when nothing is selected) to generate nets and create nodes at the junctions of nets.
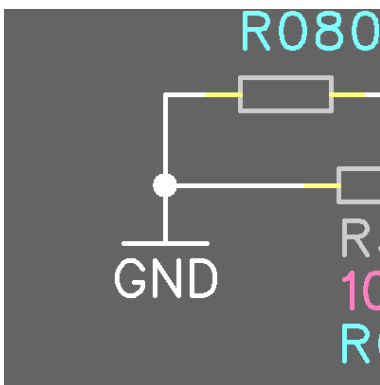
Add this mark to the selected net markers for subsequent quick copying of this mark to the center of the main program window: select the side of the mark polyline and select "Add to Favorite Nets". Now on the left, under the selection mask, a window has appeared with a list of favorite net labels.
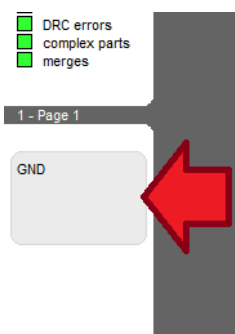
By clicking on the text GND in this window, you will get a copy of this mark in drag-and-drop mode in the center of the screen on the working area.

Now let's update the netlist and transfer it to the PCB editor. Press the main menu button "File >> Save with Netlist File"And in the dialog box that appears, click YES (switch to the Freepcb-2 PCB editor window). In the external netlist import dialog box, press the buttonImport, after which the netlist will be loaded into the PCB editor. Remember to save the PCB file (so that the schematic editor does not display an error when trying to modify the schematic with an unsaved PCB file).

Importing parts from related projects

Let's create a new project and import into it the resistor that is in the current project (10k @ R0805). Select the menu item "File >> New"And enter a name for example" MyProj2 ". Now we have an empty project into which we will import a part from the "MyProj" project. To do this, we need to open the import dialog. In the mode when nothing is highlighted press "F1 - Import Part". The program will index the project folder and display a list of ever used parts in VALUE @ FOOTPRINT format in the import dialog.

Import part by attribute   ✕

Attribute string
10k@R0805   ①   ▼

Text line options   Search among
☐ text line start   ☑ part reference   10k@0805
☐ text line end   ☑ part value   10k@R0805
☐ text line present   ☑ part footprint   Template@???
☑ whole word   ☑ pin name
☐ case sensitivity   ☑ net name   ②
☐ it's part prefix   ☑ description

Pin count
Ignore min -1 max -1   Start

☑ search all pages   Parts used only in the current project
☑ search outside this project...   All parts used in the current project
☑ Choose from a list of ever used parts   Parts used once in any project   ?
☑ Apply default polyline width and text sizes

O:\!ОБМЕН!\~Cerduk\Freepcb-2\bin\Autosave\02.cds
O:\!ОБМЕН!\~Cerduk\Freepcb-2\bin\Autosave\03.cds
O:\!ОБМЕН!\~Cerduk\Freepcb-2\bin\Autosave\04.cds
O:\!ОБМЕН!\~Cerduk\Freepcb-2\bin\Autosave\05.cds   ③
O:\!ОБМЕН!\~Cerduk\Freepcb-2\MyProj\version-01\MyProj-01.cds

Let's describe some of the fields of this dialog box:

1) A search string to search for a part by attribute, as well as to filter the list of parts.
2) List of parts ever used in projects.
3) List of files in which this part was fixed during indexing.

Pay attention to an important point - in our case, the BIN folder with the executable program is inside the project folder, and therefore the contents of the BIN \ AUTOSAVE folder are included in the list of files. If you do not want this folder to be indexed, then move the BIN folder with the program from the project folder to some other location.

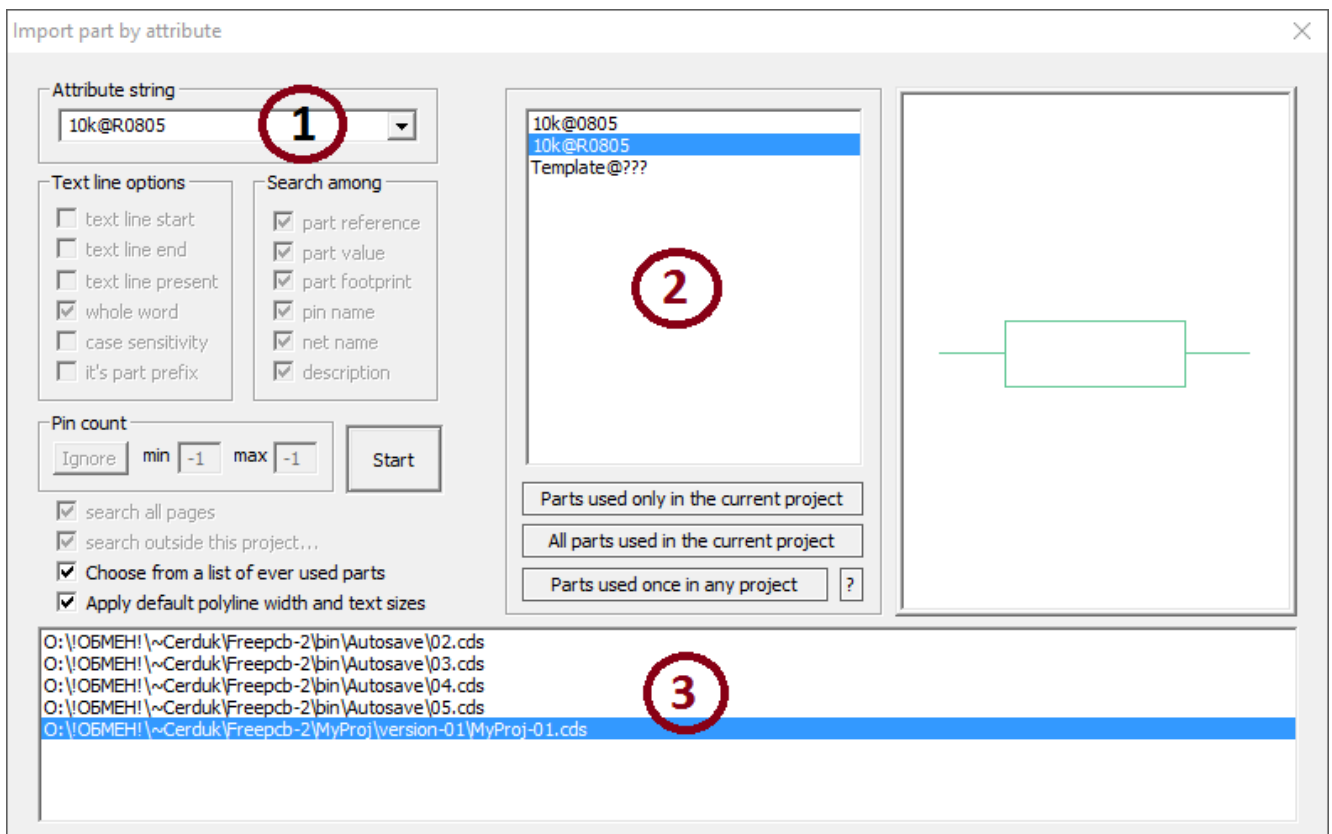By default, the program imports the part, applying to it the width of the polylines specified in the options dialog box of the current project, which is called from the menu "Project >> Options". The size of the text attributes specified in the attributes dialog box, which is invoked by the function button "F1 - Set Attributes"On the selected polyline. If you do not want to apply these parameters, but want to import the part in its original form, then uncheck the "Apply default polyline width and text sizes". So, click on 10k @ R0805 in list # 2 (see the screenshot above), then click on the file from which you want to copy a part in list # 3, then click "Start". The program will find the part and copy it to the current project. If you do not select a file in the list # 3, then the program will take a part from the first file in this list.

Let's draw a circuit of a multivibrator on LEDs in this project. To do this, in addition to the already existing UGO resistors, we will need transistors, LEDs and capacitors.

Composite transistor

Schematic Constructor1.3 allows you to create parts consisting of several graphic symbols located in different places on the page. We will now create two independent BC856B transistor graphics that are integrated into one TI / DBV6 package. The TI / DBV6 footprint is in another library: fp_lib \ lib_extra, which comes with the program, so we will use this library in this project. Change the footprint library via the menu "Project >> Options".

Draw the UGO of the transistor, as shown in the screenshot below, and at the end of the reference designation add ".1" or "-1", which will mean for the program that this part is composite.
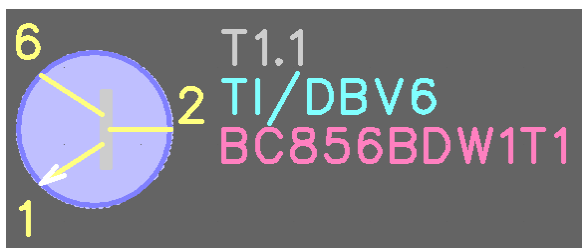
Now copy this transistor with CTRL + C and CTRL + V. The program will paste a copy named T2.1. We need to change this reference designator to T1.2. To do this, select the copied element completely by clicking, for example, on the circle (but not on the pin, because in this case only the pin will be selected, and not the whole

part). Now open the attributes dialog with the button "F1 - Set Attributes". Change the value from T2.1 to T1.2 and click OK.

Having selected the entire transistor, make a mirror image of it with the function key "F3 - Flip Group".

Then change the pin numbers to 3-collector, 4-emitter, 5-base according to the transistor documentation. To do this, you can click on the pin text and press "F1 - Edit Text". The second option is to click on the pin polyline and press "F1 - Set Attributes".

As a result, we got the second element of the same component.
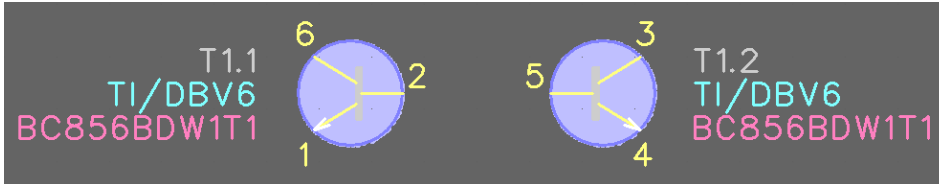
Draw the remaining elements of the multivibrator circuit.
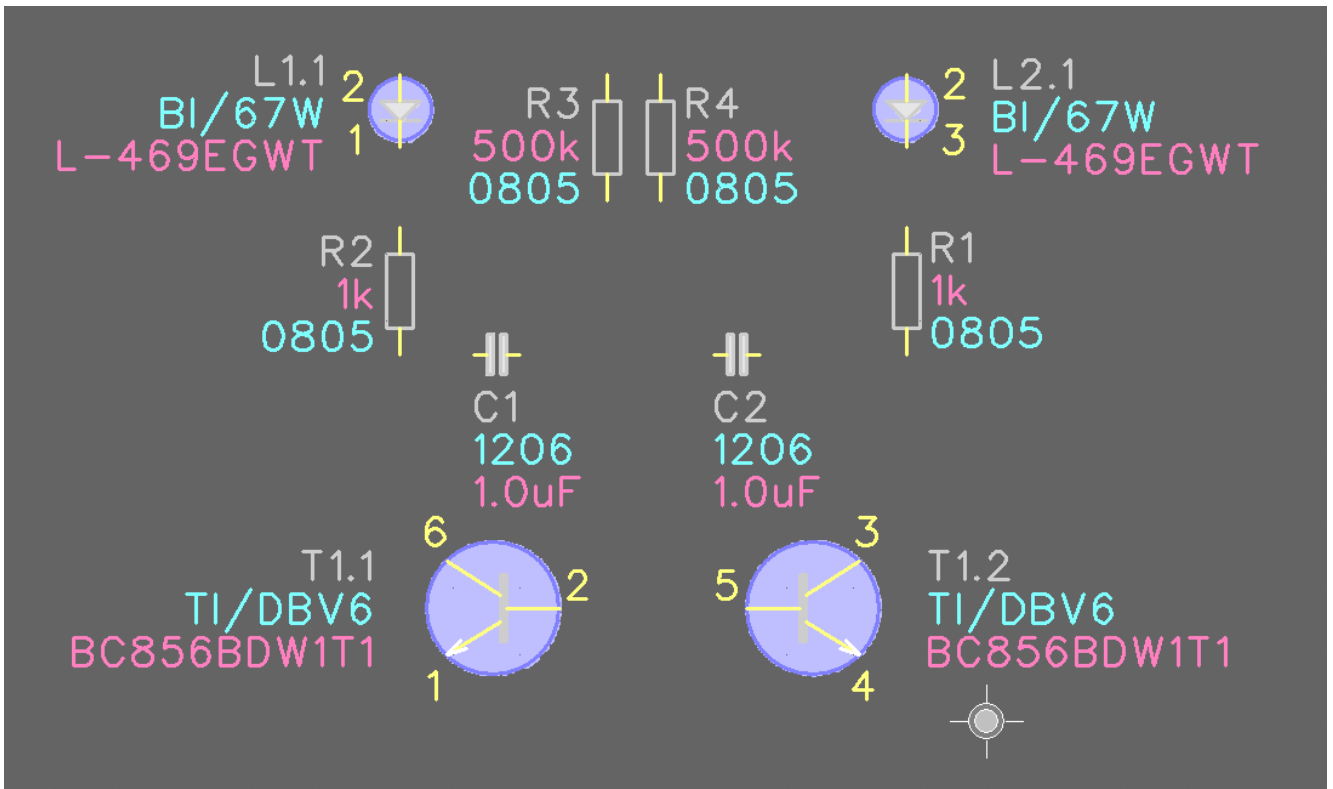
When you enter the VALUE and FOOTPRINT attributes in the dialog box, you can run the external Freepcb-2 library manager utility to help select a footprint from the library, and select a footprint from the list. The library manager displays the appearance of a footprint and the distance between adjacent pins (which is also very useful), has a filter window to search for the desired footprint by parameters. For example, you can enter in the N_PINS column the number of pins that the desired footprint should have. You can also enter the same value in the N_HOLES column to strip off all SMT components and leave only through. The footprint library manager is launched from the polyline attributes dialog box by pressing the button "L".

PolyLine Attributes

Ref Des	R2	☐ Apply sizes:	Height 2.20	Stroke width 0.30
Part Value	1k	☐ Apply sizes:	Height 2.20	Stroke width 0.30
Footprint Name	0805	Apply sizes:	Height 2.20	Stroke width 0.30
Pin Name		☑ Apply sizes:	Height 2.20	Stroke width 0.30
Net Name		☑ Apply sizes:	Height 2.20	Stroke width 0.30

Description

☑ Apply sizes

Height
2.20

Stroke width
0.30

Add command template

Date	add	?
Link to part	add	?
Outline of complex part	add	?
Pin of complex part	add	?
Bus connect	add	?
BOM table	add	?
Path to file	add	?

OK   CANCEL

Freepcb-2 footprint library manager screenshot

FreePcb footprints library

Filter

Name: 0805
Package:
nPins:
nHoles:

☐ Pad style (top, bot, inn)
◉ none   ○ mdrect
○ round ○ oval
○ square ○ octagon
○ rect

Lib: o:\!обмен!\~cerduk\fp_lib\lib_extra   ...

Total footprints = 799

"Ivex" 0805
"Ivex" 0805
"Ivex" 0805

Edit | Open PDF | Open Lib | Delete

Combine footprints
Add String to References
Add String to Footprint name for panelization
Replace selected Footprint in current File
Replace all footprints in current File

1,9mm

Selected

FootPrint: 0805
Package:
Author: Ivex
Description: none

Mod

Sorting style
○ Alphabetically
○ The number of pins
○ The number of holes
◉ Author
○ Description   R

☐ Remove duplicates

Rewrite all lib

Load file

www.freepcb.dev

Complete the circuit by connecting all the pins as shown in the screenshot
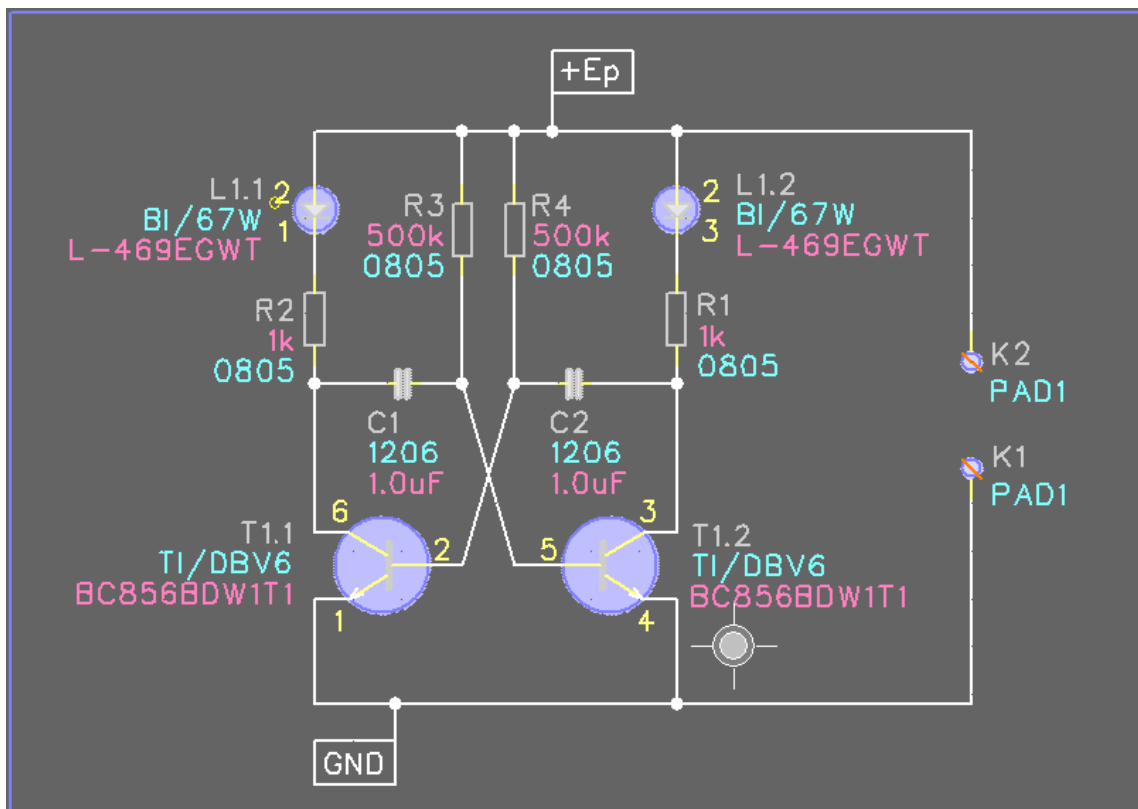
We got a circuit containing two components - a microcircuit with 2 transistors and a two-color LED.

Ref-Lists

You can create a parts list that includes the parts you choose. This list will be stored inside the schematic file, and not as a separate file as in the PCB editor Freepcb-2. Then, when you want to see the list, you must select it from the menu item "Project >> Ref lists >> [ref_list_name]"And the program will highlight the parts on the working area that are present in the list. To create a list, highlight the parts on the work area, and then press the menu "Project >> Ref lists >> Create by selected parts". You can use reflists in BOM tables to include / exclude any details.

Auto update date

The "Description" attribute of a polyline can contain text commands that are converted by the program into the execution of any logical operation. One such command is the auto-update date, which is updated when changes are made and the project file is saved. Select any free polyline that is not part of any part or part of a chain, then open the attributes dialog with the function button "F1 - Set Attributes". Here, on the right side of the window, there is a block of command text templates. A command template is a text block with parameters that the user can change as needed. Click the ADD button next to Date, circled in an ellipse in the following screenshot. As you can see, text appeared in the Description field, which will be converted by the program to the date format. Let's break down a text string to understand how it works.

First, there are three two-digit numbers 00.00.00 (this is a sample date), followed by a vertical bar, which is a key and divides this command line into two parts - the one on the left will be displayed on the screen as a result of the command, and the one on the right is the actual command text and is not displayed on the working area or when printing on paper. The command is the part of the line after the pipe (| DD.MM.YYYY DATE) that you can make changes to:

 - you can replace dots with slashes

 - you can swap places like MM and DD

Press OK and the text of the current date will appear on the screen in the format you have chosen.


Detail reference

The next command in the polyline attributes dialog box is a part reference. Click the ADD button next to "Link To Part" to automatically insert the template into the "Description" field. The command template looks like this:
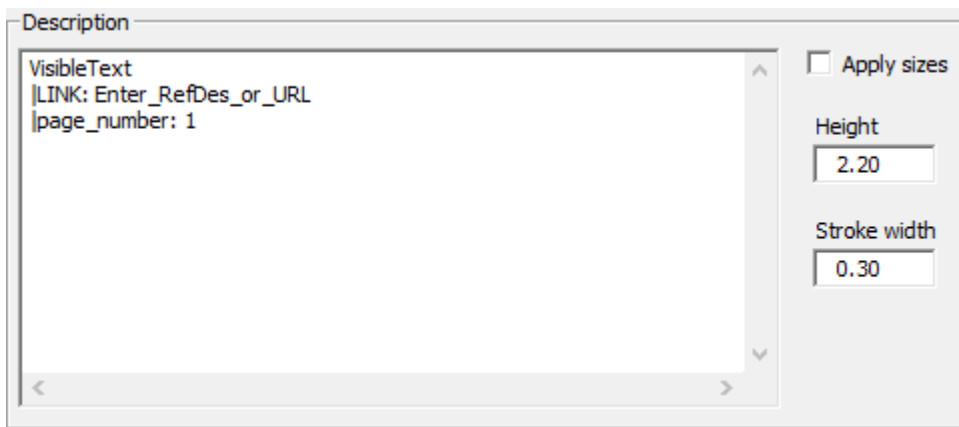
Replace the text "VisibleText"To the one you want to display on the screen. The second and third lines contain the command text. Instead of text, enter "Enter_RefDes_or_URL"Reference designation of the part. On the next line, enter the page number on which this part is located. Click OK to save your changes and close the dialog box. When you hover the mouse cursor over the link text, the text should be highlighted. This tells us that the text object is a link. To follow the link, click with the mouse while holding down the keyCTRL...

Complex part (hierarchical symbol)

In Schematic Constructor1.3 you can create a complex part, inside which there can be a part of a schematic consisting of simple parts. In this case, the main scheme will contain the UGO of a complex part (a rectangle with pins) in any quantity. This engineering approach is useful when creating multi-channel circuits containing identical blocks. The role of the channel will be played by a complex detail. The schematic inside a complex part should be drawn on a separate page. Therefore, the entire project with complex symbols will be located on at least two pages. Let's try to create such a project using the example of a circuit of the same multivibrator. We will have to abandon the composite transistor and the two-color LED, because we will split the multivibrator circuit into two halves vertically, and these will be two independent blocks. So let's get started. First, we need to create a second page on which we will place the diagram of half of the multivibrator in the form of a complex part. Select the menu "View >> Add New Page"And enter the name of the second page. Go to this page and draw a rectangle about 60 x 90mm as shown in the image below.
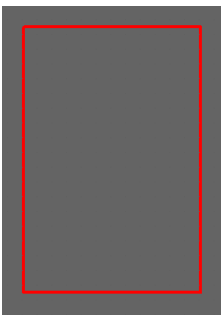
We need to output a complex part as pins:

1) LED anode,
2) transistor emitter,
3) the base of the transistor,
4) and the junction point of the capacitor C1 and the resistor R3.

In total, 4 pins are obtained, which our complex part should have. Let's draw the first pin at the top and add the attribute "Pin Name"As if we were creating a regular part.
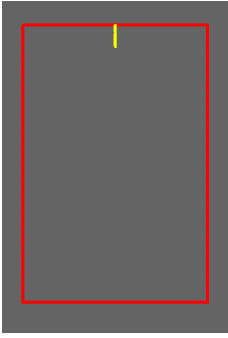
Select the pin line and call the attributes window by pressing F1... Next to "Pin Name" enter "1" and click OK. Then select all objects (rectangle and pin) with the mouse frame and call the attributes window again to enter the name of the complex part in the RefDes line. Enter for example "Part" (or "Channel"), any name, no number. Click OK.

We've got a part consisting of a contour polyline and pin "1".

Select the first pin and duplicate it three times (using the shortcut CTRL + C, CTRL + V) in order to position the pins as shown in the figure to complete the shell of the complex part.
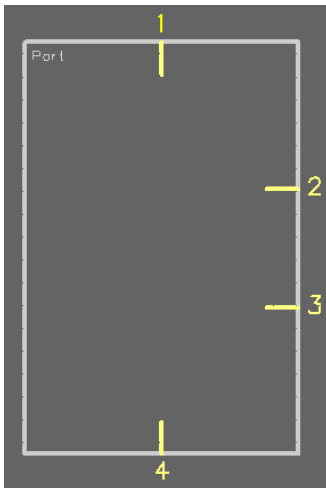
This name of the part will serve as a shortcut located on the main scheme of the project, more on that later. Now let's add a half of the multivibrator circuit inside the contour of this part.
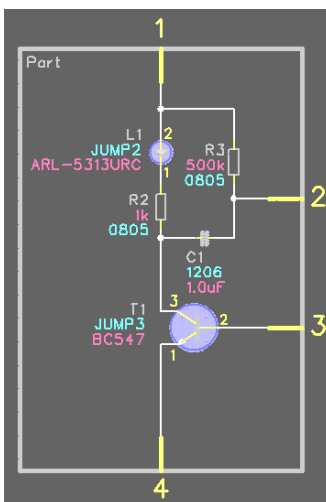
We have finished drawing a complex part. Thus, the current page contains only an outline of this part and nothing else. Now let's return to the first page of our project to create the UGO of a complex part and the main circuit of the multivibrator. You can switch the page by pressing the corresponding number on the

keyboard: press "1" to select the first page. Delete all parts except PAD1, PAD2 and net markers + EP and GND. Draw a simple rectangle next to it, this will be the UGO contour polyline of the first complex part.
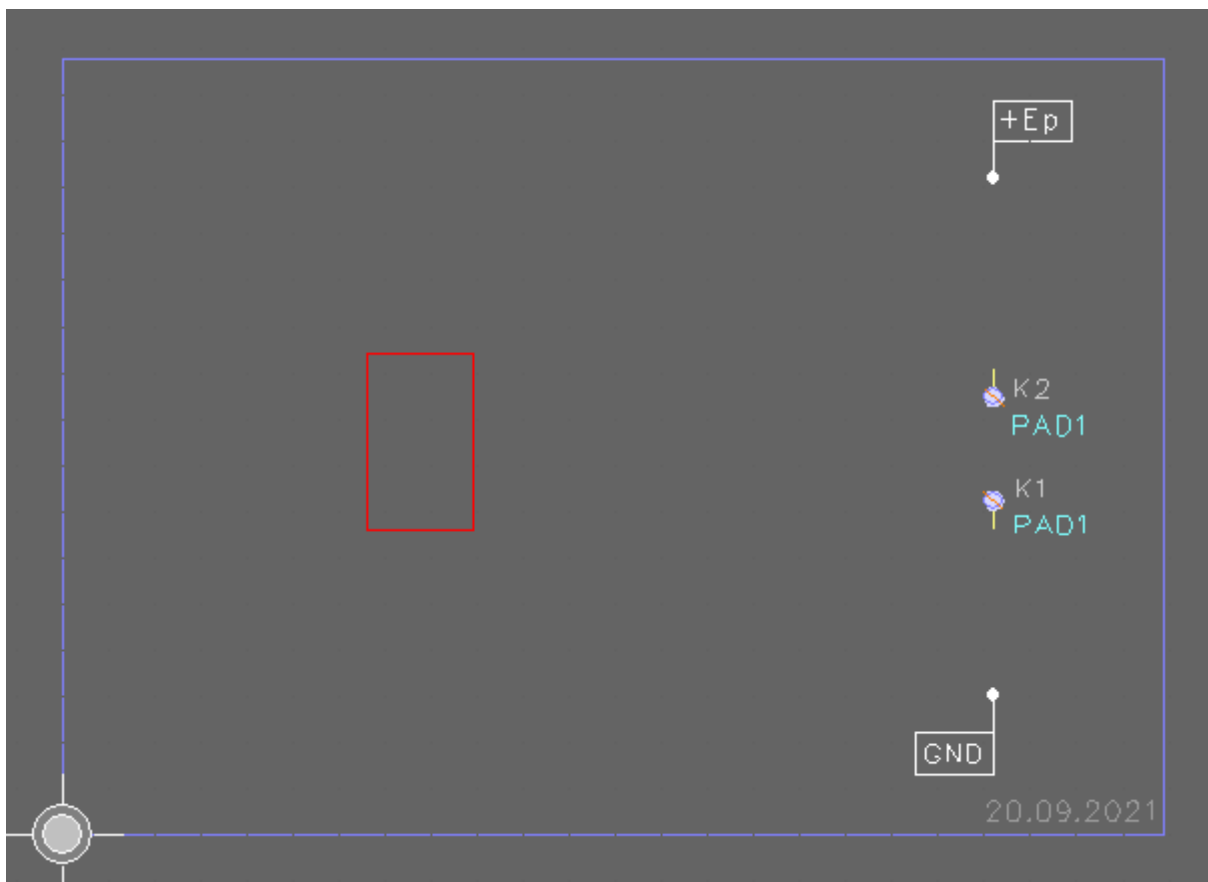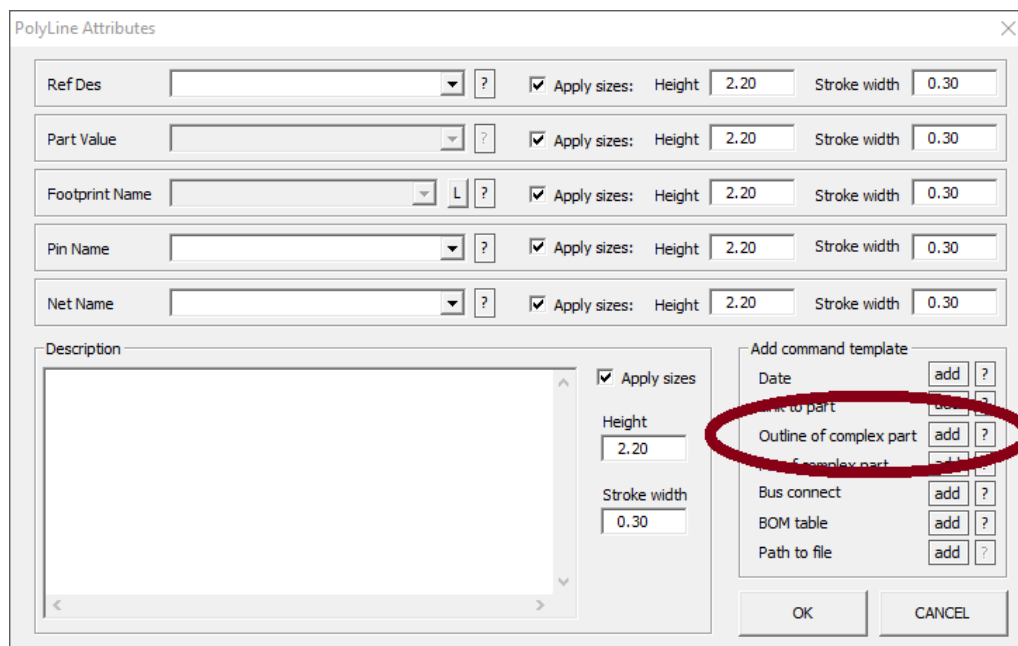

Select the side of this polyline and press the key F1 to bring up the attributes window. In the attributes dialog box, click the ADD (Outline of complex part) button.

In the "Description" field, the program will automatically insert a command template indicating that this polyline is a contour polyline of an UGO complex part.

```
Description
VisibleText
|suffix: xx
|command: BLK
```

Apply sizes ☑

Height
2.20

Stroke width
0.30

This is an empty template containing only one variable parameter, "| suffix: xx", which you must specify. Instead of "xx", enter a complex part suffix, for example "A" and click OK.
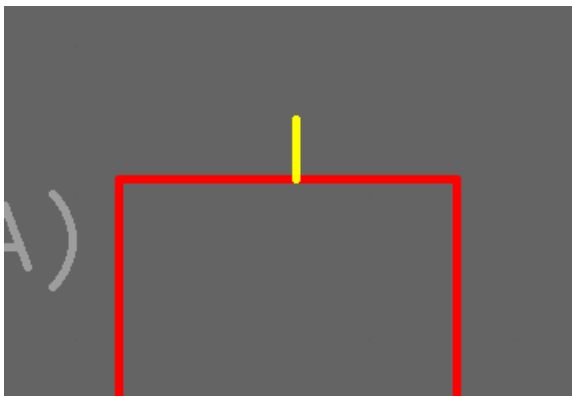


```
Description
VisibleText
|suffix: A
|command: BLK
```

Apply sizes ☑

Height
2.20

Stroke width
0.30

If you click now on this contour polyline, the program will display a warning with the text: "Error! Complex part has no pins", So the next step is to add four pins to this UGO of a complex part, corresponding to the pin names that we set for the part"Part"(On the second page). Draw a polyline consisting of one segment, as for a regular pin of any part.

Highlight a side and press the key F1to bring up the polyline attributes window. In the Attributes dialog box, click the ADD (Pin of complex part) button.

In the "Description" field, the program will automatically insert a command template indicating that this polyline is a pin on the UGO of a complex part.

This is an empty template containing four modifiable parameters:

1) "| Suffix: xx", (suffix - enter the same as for the contour line)
2) "| LINK: xx", (the name of the part with the contour on the complex part diagram, we have it "Part")
3) "| Pin_name: xx", (pin name of the complex part)
4) "| Page_number: xx", (the page where the complex part schematic is located)

enter "xx" instead of the values as shown in the picture below and click OK.

```
Description
VisibleText
|suffix: A
|LINK: Part
|pin_name: 1
|page_number: 2
|command: BLK_PTR
```

☑ Apply sizes

Height
2.20

Stroke width
0.30

If everything is done correctly, the text "1" will appear next to the polyline.
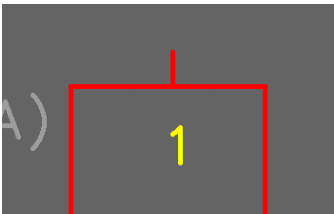
Move it to the top immediately by pressing F4 (Move)... Check if the program will give any warning when you select a complex part UGO. To do this, click on the UGO contour polyline. In this case, both objects should become selected: both the contour polyline and the pin.

With the pin polyline selected, duplicate it using the CTRL + C, CTRL + V combo. The program will enter drag-and-drop mode for the copied pin. In this mode, pressF3to rotate the pin 90 degrees.
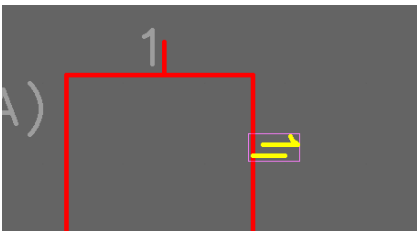
Select the pin polyline again and in the attributes dialog change the pin name from "1" to "2" as shown in the screenshot


```
Description
1
|suffix: A
|LINK: Part
|pin_name: 2
|page_number: 2
|command: BLK_PTR
```

☐ Apply sizes

Height
2.20

Stroke width
0.30

Click OK to save your changes and close the attributes dialog box. Highlight the text "2" and enable drag-and-drop mode by pressing the function keyF4... In drag mode, pressF3to rotate the text 90 degrees.
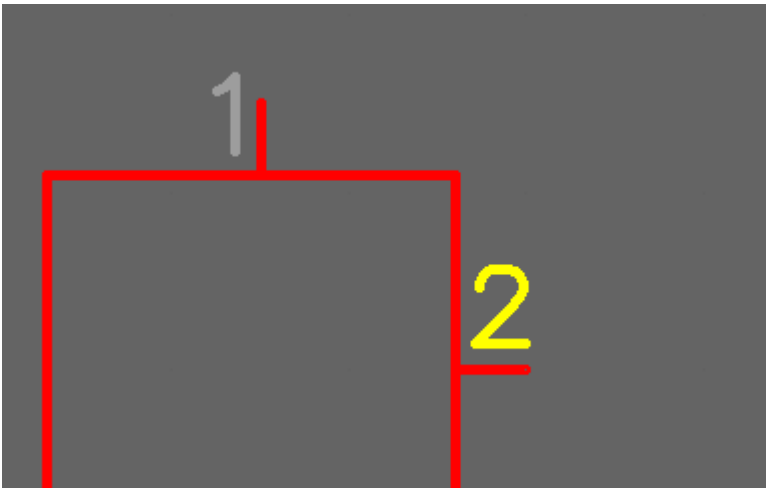
In the same way create two more pins named "3" and "4" for the complex part. As a result, we get a HUO of a complex part.
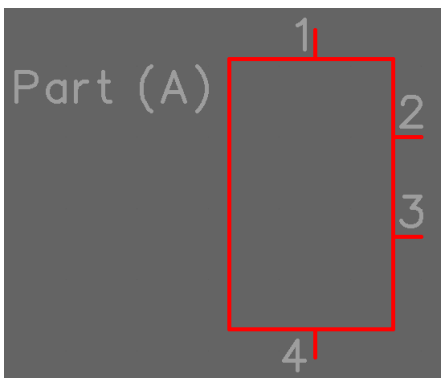
Click on the UGO contour polyline. In this case, all 5 objects should become selected: the contour polyline and 4 pins. The program should not issue any warning when the UGO highlights a complex part.

Now we need to create a copy of the same UGO, placing it on the right, and make it a mirror image. Select the entire UGO of the complex part by clicking on the contour line, then duplicate it by pressing CTRL + C and CTRL + V. Mirror the selected group with the function buttonF3...
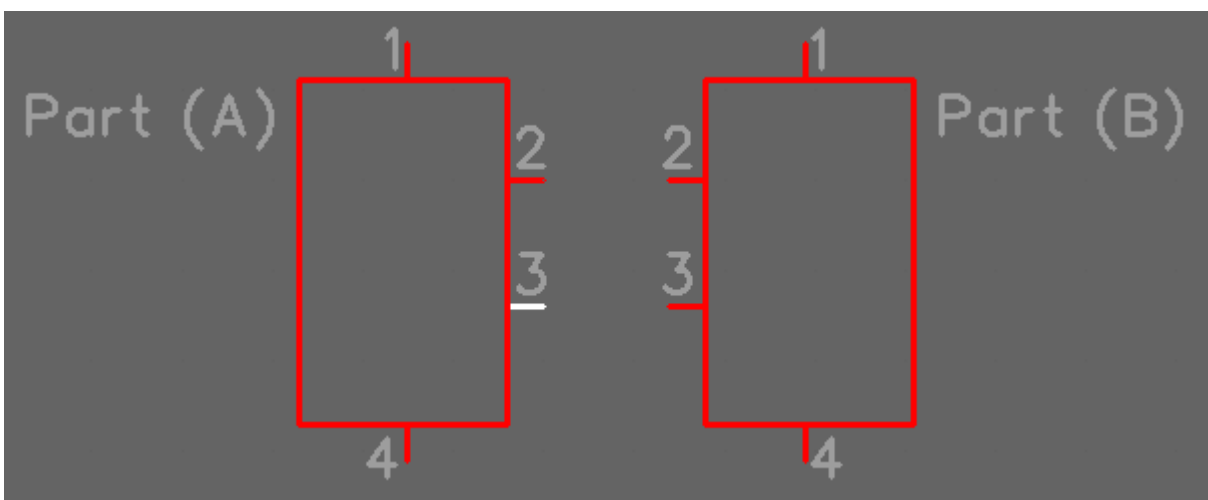
At the end, you need to connect the pins of the complex parts according to the multivibrator diagram.
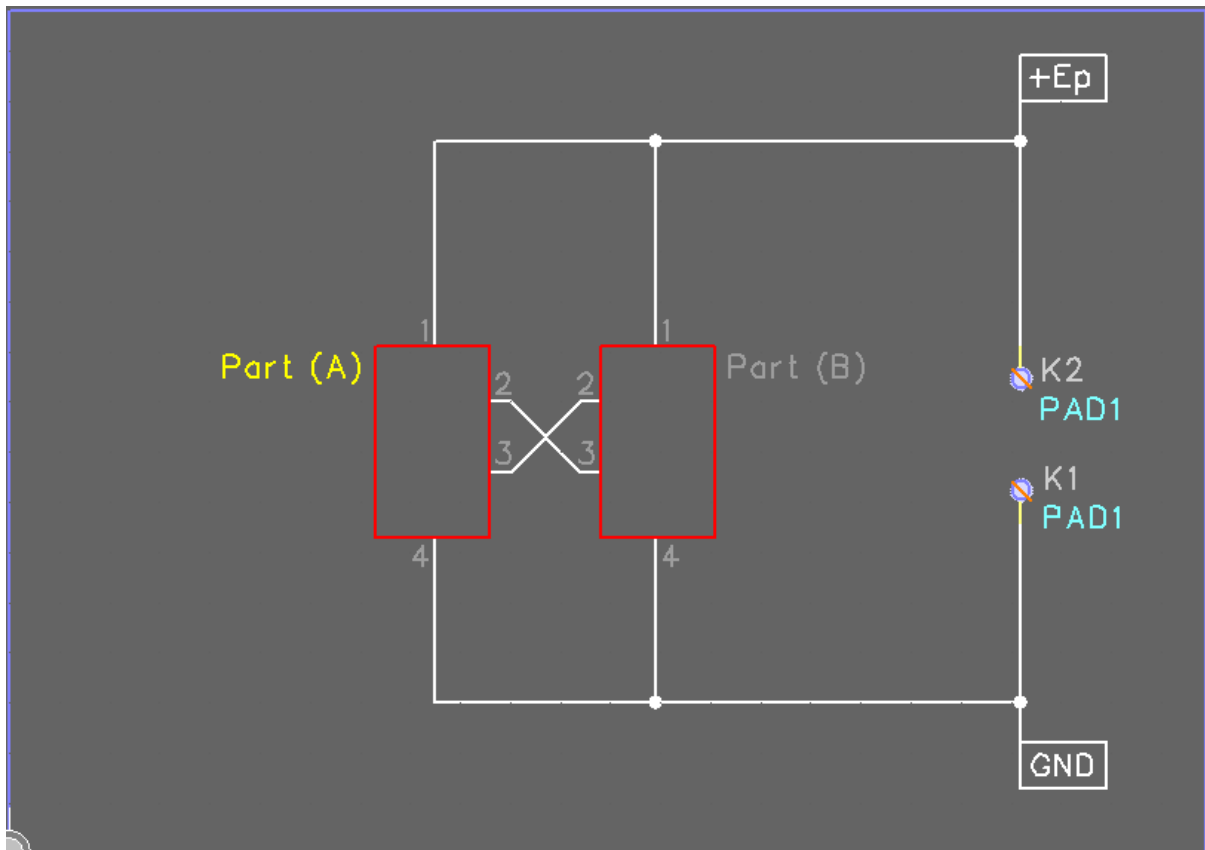
The circuit is ready. Generate a netlist by choosing the menu item "File >> Save with netlist file".

Electronic bus

Let's make the connection of pins "2" and "3" using the electronic bus. Select the objects on the left, which are circled in an ellipse in the screenshot, and move them to the left to make space between the left and right parts of the electrical circuit. Move objects with the arrows on your keyboard.
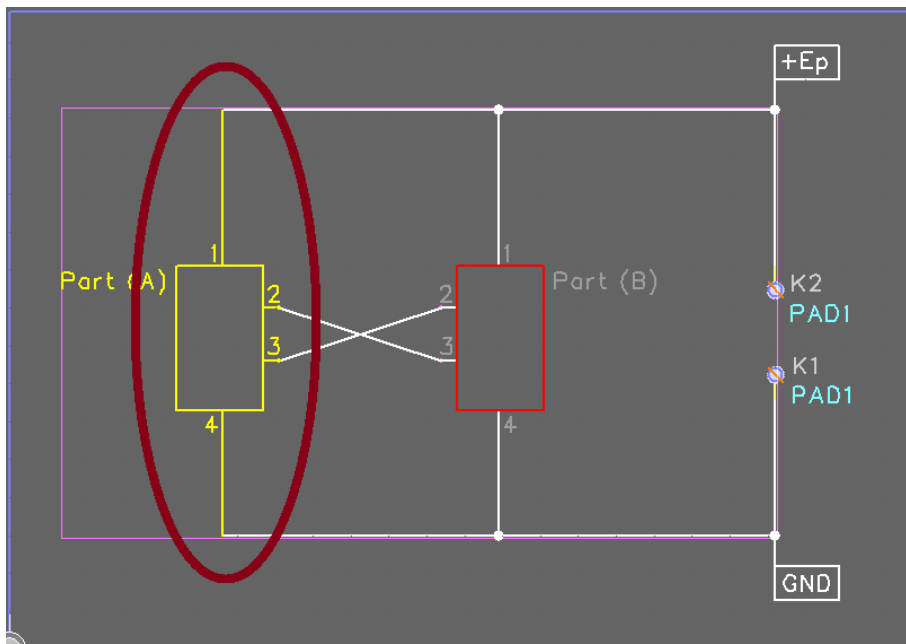
Remove the links connecting the pins "2" and "3", And draw a vertical e-bus polyline. Since the e-bus must be thick, highlight it and press the softkey "F7 - Edit Properties"To change its width. Set 1.5mm, uncheck "Apply to lines with a range width", Select the polyline layer for example" Front Layer1 "and click OK to save the changes.
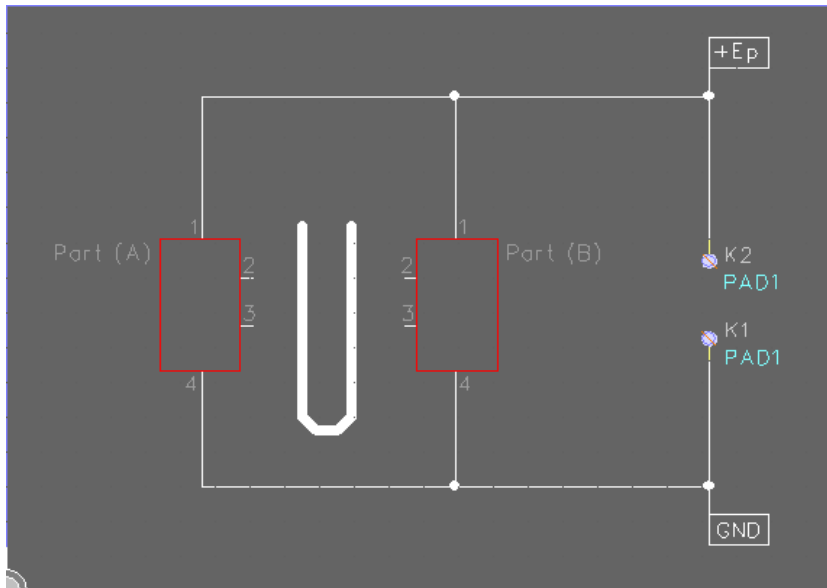
Select this polyline and call the attributes dialog using the button F1... In the "Description" field, enter the name of the e-bus (for exampleBUS-1), and click OK.

The electronic bus is almost ready, all that remains is to supply the electrical circuits in order to connect to it. Draw four polylines at a 45 degree angle like in the screenshot.
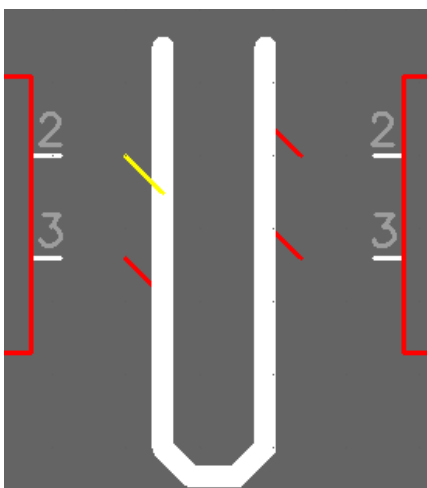
Select one of them and in the attributes dialog, click ADD next to the Bus Connect command.

In the "Net Name" field, the program will automatically insert a command template indicating that this polyline is an electronic bus label.

This is a blank e-bus label template. Enter the text "1" to the left of the vertical bar (instead of "EnterNetName"), And to the right of the vertical bar (instead of the text"EnterBusName") Enter the name of the bus we have already named"BUS-1".

Click OK to complete the label creation. Create the other three marks for the electronic bus in the same way, and then connect according to the diagram.
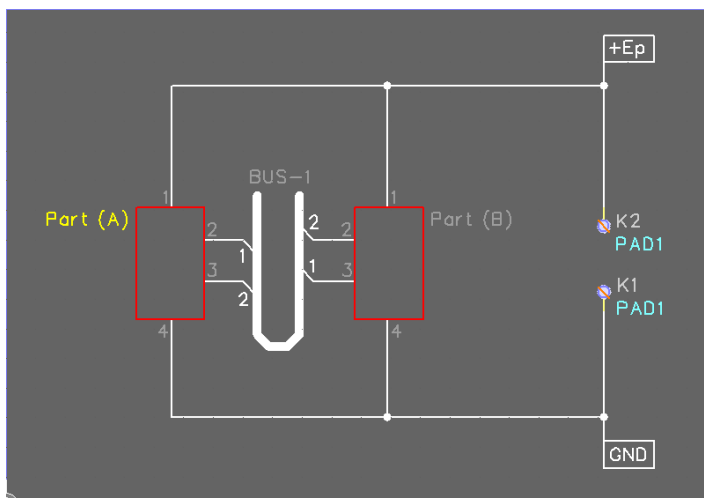
Auto updatable BOM table

In Schematic Constructor1.3, you can insert a BOM table into any page of the project. The table can be inserted inside a rectangular closed polyline using a special text command. In this case, the table is inserted automatically into the rectangle occupied by the area of the polyline, no matter what shape and how many corners the polyline has. All this is done using the text command "BOM Table"In the" Description "field of the polyline attributes. Add any rectangle or square to the artboard. To do this, click the function key "F3 - Add Polyline", The polyline properties dialog box will open, click OK to continue without changing any options. The program will switch to drawing mode and a menu for selecting standard shapes will appear. Please selectF1 or F2...

F1 Add Rect	F2 Add Square	F3 Add Round	F4 Magnetic Vertices	F5 Add Oval	F6 Add Ellipse	F7 Add RndRect	F8 Add Cloud	F9
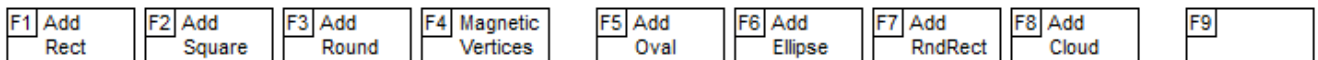
Select the side of the drawn rectangle and call the polyline attributes window by pressing "F1 - Set Attributes". Click the ADD button next to the "BOM table" command template header.
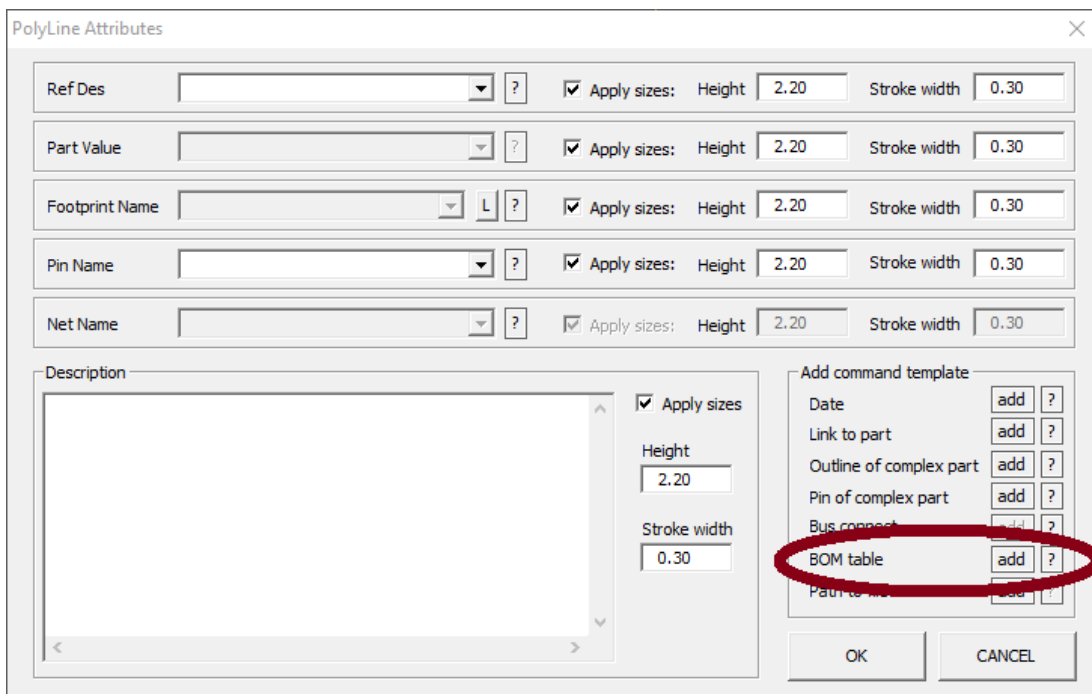
A text template with customizable BOM table parameters will automatically appear in the "Description" field.
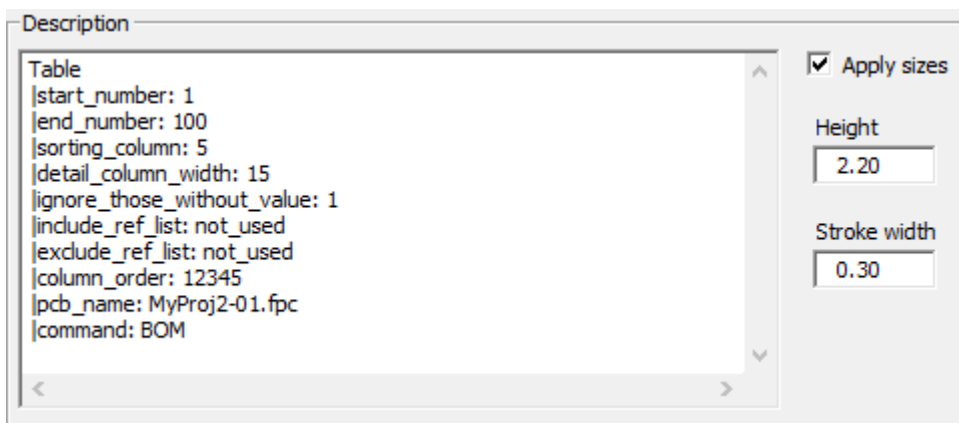
In this command template, in contrast to the commands for creating a complex part, almost all the parameters are already set initially, so you can immediately click OK to complete the creation of the BOM table. Before

clicking OK, it is worth checking the correctness of the PCB file name, this is the penultimate line (| pcb_name: … ") so that the program does not generate an error.

```
PCB: MyProj2-01.fpc
ITEM    VALUE         FOOTPRINT    CNT    DETAILS

1       1.0uF         1206         2      C1A, C1B

2       ARL-5313URC JUMP2          2      L1A, L1B

3       500k          0805         2      R3A, R3B
4       1k            0805         2      R2A, R2B

5       BC547         JUMP3        2      T1A, T1B
```

The table contains five columns:

1) Item
2) Value
3) Footprint
4) Count
5) Details

Description of command items:

1) | start_number: 1= the value of the starting number in the table. If you want to insert a fragment of the table, and not the entire BOM-table, then simply change the value of the start number, and the table will start from the number (according to the complete list) that you specify.

2) | end_number: 100= the value of the end number in the table. If your BOM table contains many rows, then you can split the table and create two different tables, for example, the first table contains rows from 1 to 25, and the second contains rows from 26 to the end of the full list. For the first table:
| start_number: 1
| end_number: 25
For the second table, specify:
| start_number: 26
| end_number: 999

3) | sorting column: 5= sorting items in the table, the fifth column is selected by default - this is sorting by RefDes. You can change this value by entering a number between 2 and 5.

4) | detail_col_width: 15= width of the "Details" column. You can increase or decrease this number

5) | ignore_thouse_without_value: 1= ignore parts that do not have a "Value" attribute. You can enter either "0" or "1"

6) | include_ref_list: not_used= creating a BOM-table from the list of a reflist. In this case, the BOM table will contain only those parts that are present in the stored list (see paragraphReflists). Enter the name of the reflist, or enter the text "not_used" to not use this option

7) | exclude_ref_list: not_used= Creation of a BOM table excluding the parts that are present in the stored list. If you want to exclude any parts from the BOM table, first go through the menu "Project >> Reflists"Create a Ref-list with these details, and then at this point, instead of the text not_used, enter the name of your reflist.

8) | column_order: 12345= the order of the columns in the table. You can swap these numbers

9) | pcb_name: <pcb_file_name.fpc>= the name of the circuit board for which you are creating the BOM table. The list of printed circuit boards is in the menu "File >> Netlist Settings".

Cross-page attribute sync

If your schema fit into one page, then you can use the option "Inter-page synchronization of attributes". This option is used when you want to create two or more single page layouts in one file. Each next scheme is located on a separate page and is a modification of the main scheme located on the main page. The part numbers must match across all diagrams, and the program will then keep the VALUE and FOOTPRINT attributes synchronized on all pages. Each time a change is made to the VALUE and FOOTPRINT attributes of a detail on the master page, the program will copy these attributes to other pages. So, first, make a duplicate of the main schema by placing it on another page. To duplicate the main diagram, you need to select everything on the main page with the diagram by pressing CTRL + A, and copy the content of the master page using CTRL + C. Then you need to switch to the next page and paste the copied diagram onto it using CTRL + V. Now you can make some changes to it, if necessary, and then turn on the synchronization of attributes. Select from the main menu the item "Project >> Interpage Attribute Synchronization"
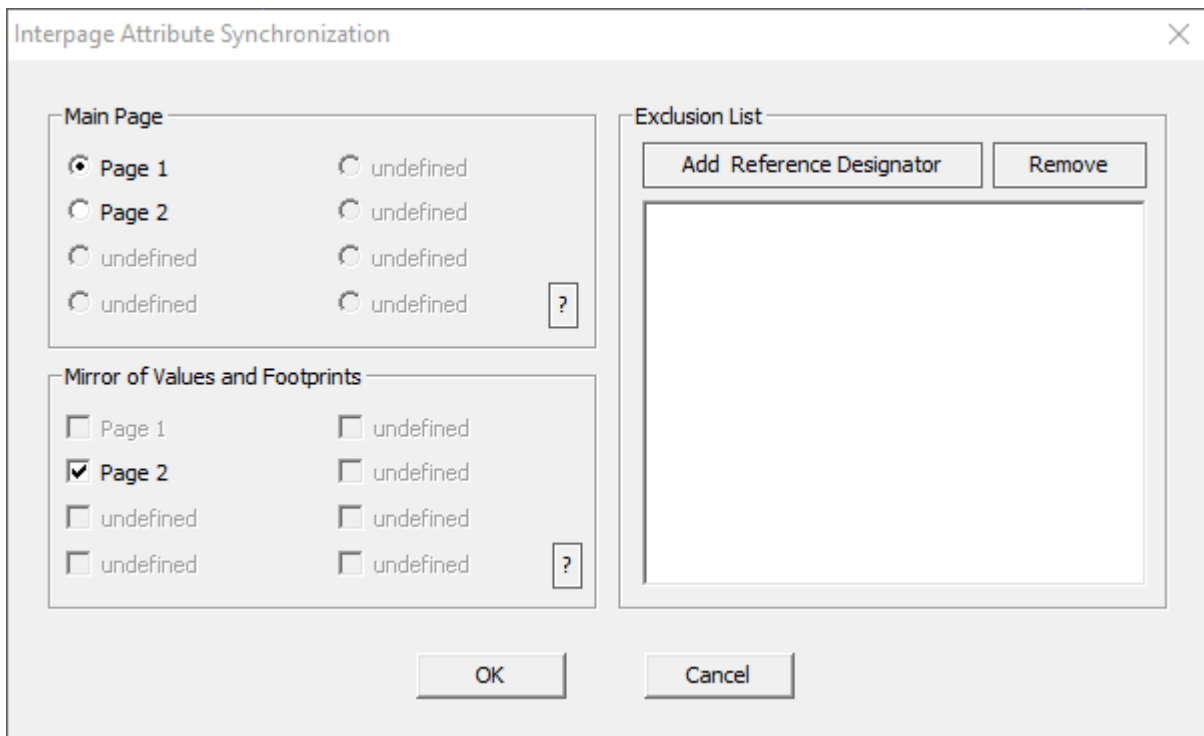

Select the main page and the pages to which the program will copy the attributes.

On the right side of the window, there is a list of exclusions, containing those details to which synchronization should not be applied at the request of the developer. You can add a detail by clicking the "Add Reference Designator" button and choosing RefDes from the list.

Compare pages

When your project includes several modifications of one-page schematics that are similar to each other (as described in the previous paragraph), it is sometimes necessary to find differences in schematics and part attributes. This can be difficult without a page comparison tool. Call the page comparison window from the menu "<mark>Project >> Compare >> Pages</mark>". This window runs in the background, so it can remain open even when editing the circuit. Select the pages you want to compare. The upper box will be conventionally considered the main page, and the lower one - the secondary page.
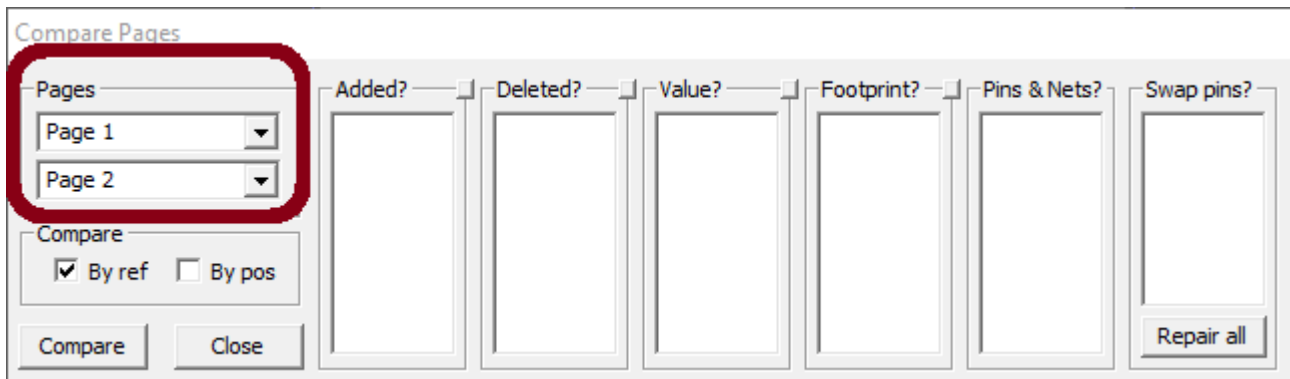
Then, under the page selection combobox, select the comparison mode:

- by reference designators

- by part positions

Comparison by RefDes of the part is selected by default. In this mode, the program scans the parts on the first page and looks for a part with the same RefDes on the second page, in order to then check the differences in the attributes and pin connections. But if you do a global renaming of parts on one of the pages, then page comparison becomes impossible. The only option in this case is to select the second comparison mode (by part positions)
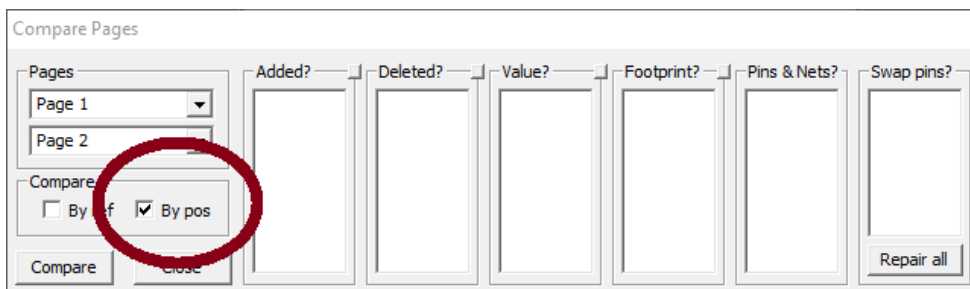
But this option will work only if the coordinates of the corresponding parts coincide on both schemes with an accuracy of 1 mm. The challenge is to make the origin in the same place for both pages. If you are using the page frame, then select the corner of the frame, and press the hotkey "O" (Origin) or use the menu "<mark>Tools >> Set Origin into selected item</mark>"To set the origin to the selected vertex. Do the same for the other page.
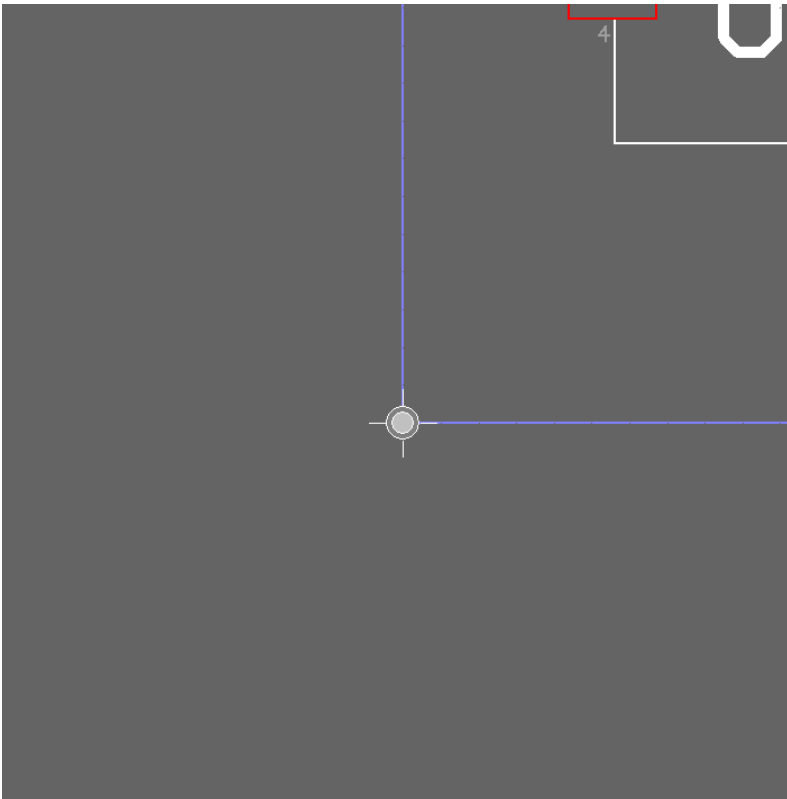
Now click the "Compare" button to compare the pages. The dialog box has 6 list boxes that will be filled in if any differences are found.

Let's create two test circuits and compare them using this option. We will place the diagrams on pages 3 and 4.

SCHEME1 (on page Page3):
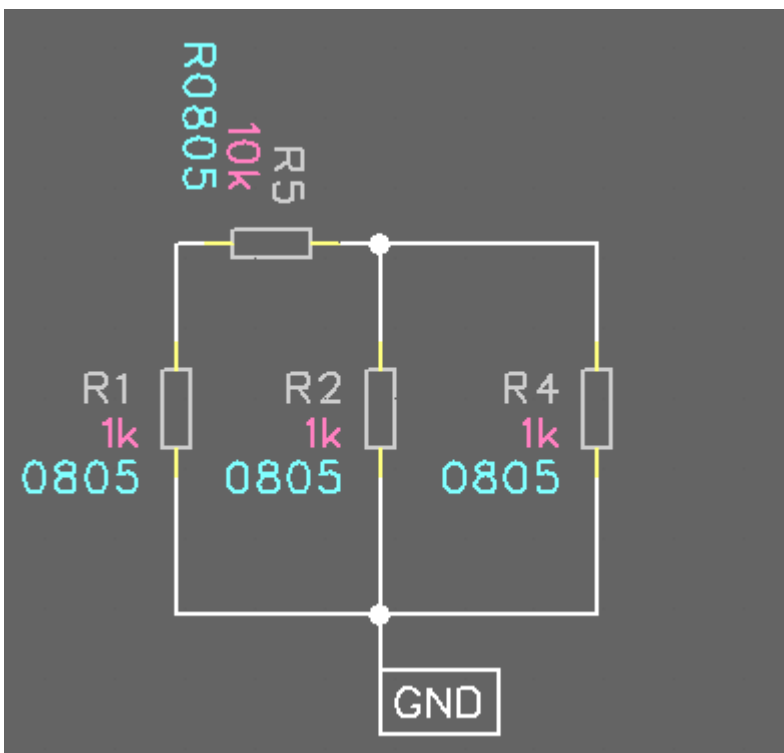
SCHEME2 (on page Page4):
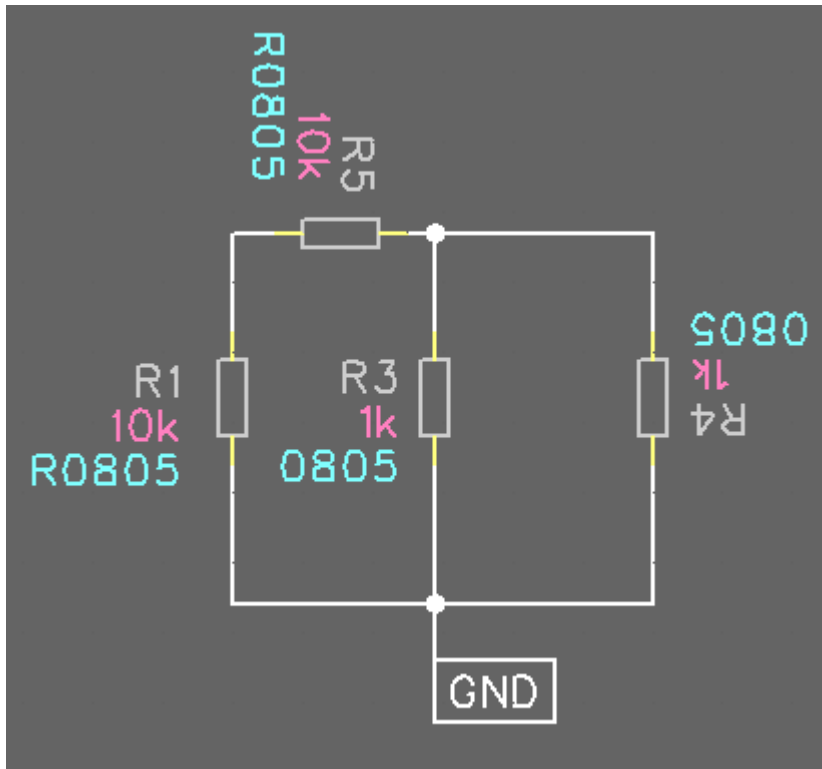
In the second diagram, we changed the VALUE and FOOTPRINT attributes of part R1, then removed R2 and added R3 instead, and rotated the resistor R4 180 degrees. Click the "Compare" button and get the following result:

Let's take a look at the data we received in the dialog box.

Since the top combobox of page selection points to the main page, all received data is interpreted relative to this page (in our case, the main page is "Page 3", the secondary page is "Page 4"). Listbox "Added"Contains details that have been added to the main page relative to the secondary page. Double-clicking on an item in the list moves the focus to the detail.

Listbox "Deleted"Contains details that have been removed from the main page relative to the secondary. Double-clicking on an item in the list moves the focus to the detail.

Listbox "Value"Contains details that differ from each other by the VALUE attribute. Click on the list item and in the title of the dialog box you will see two text values: The first is the VALUE attribute of the detail on the main page, the second is the VALUE attribute of the detail on the secondary page.
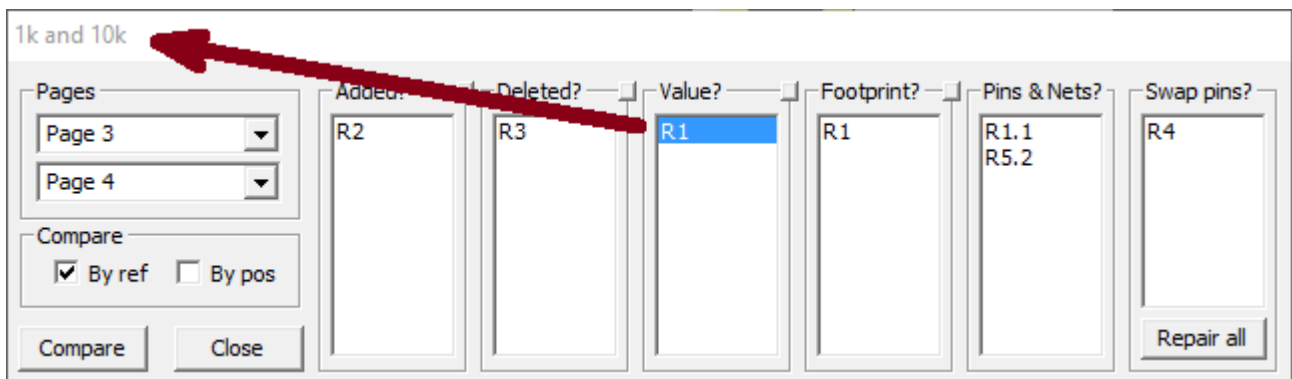
Double-clicking on an item in the list moves the focus to the part located on the current page.

Listbox "Footprint"Contains details that differ from each other by the FOOTPRINT attribute. Click on the list item and in the title of the dialog box you will see two text values: The first is the FOOTPRINT attribute of the detail on the main page, the second is the FOOTPRINT attribute of the secondary page detail. Double-clicking on an item in the list moves the focus to the part located on the current page.
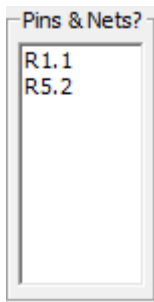
Listbox "Pins & Nets"Contains pins of parts that differ in connection to the circuit. Double-clicking on a list item moves the focus to a pin on the current page.

Listbox "Swap pins"Contains a list of two-pin parts on the main page, presumably rotated 180 degrees relative to the same parts on the secondary page. Pressing the "Repair All" button will automatically correct all the details in the list on the current page.

Comparison of netlists

You can also compare the current open project netlist with any external PADS-PCB netlist. In the Schematic Constructor, we need this option in order to be able to compare two different projects. In particular, you can compare the current project with a previous version of the same project. The program implements this feature by comparing two netlists - the current one and the loaded one. Therefore, you first need to create a current netlist. Click "File >> Save with netlist file"Menu item to do this (close the dialog that appears). Now load the external netlist by calling the "Project >> Compare >> PADS-PCB netlists". In Schematic Constructor netlists are always saved to the current project directory in the folder: ".. \ related_files \ CDS_netlist \ <pcb_file_name.fpc.net>". Select any netlist file. After loading the netlist, the comparison procedure is the same as in the previous paragraph.

Renumber parts globally

The Schematic Constructor supports global renumbering of project details. This option automatically renames the selected parts on the page. If you want to rename all the details on the current page, then select all the components of the page by pressing the standard CTRL + A combination. Then select the menu item "Tools >> Clear part numbers"To erase all part numbers in the selected group. In the dialog box that appears, click OK. After that, without removing the selection, select the menu item "Tools >> Renumber part designations". In the dialog box that appears, you can change the direction of renaming, add a suffix to the parts (for example, R1A, where A is a suffix), and also change the starting number from which the renumbering will start. Usually, there is no need to change the start number, because in any case the program will not allow duplication of RefDes parts for the current project.
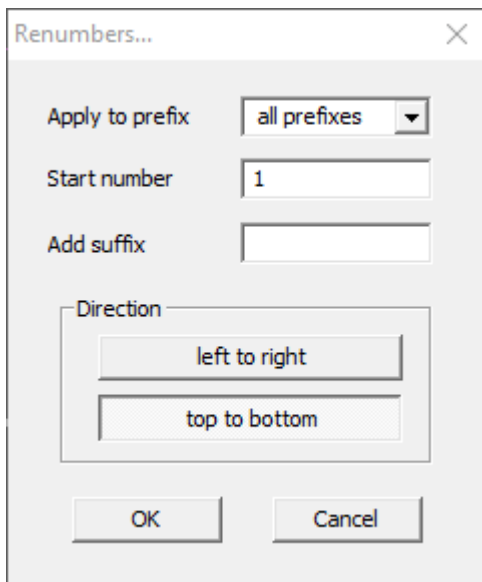
When renumbering parts, the program will create a rename file for the Freepcb-2.3 compatible PCB editor to automatically rename the same parts in the PCB file. Renaming of parts occurs when the netlist is reloaded in the PCB editor. Therefore, after renumbering, immediately generate a netlist and update it in the Freepcb-2.3 editor (In the schematic editor, this menu is "File >> Save with netlist file")

Recommendations:

1) **Renumber only with a synchronized netlist between schematic and PCB. (Netlists are synchronized when the netlist is loaded from the schematic to the PCB. In the schematic editor, select the menu "File >> Save with netlist file", then in the dialog that appears, press the YES button to launch Freepcb-2.3 and update the netlist in the PCB file, and then be sure to save the PCB file by pressing the floppy disk)**
2) **Do not add new parts to the diagram just before renumbering.**
3) **Immediately after renumbering, generate a netlist and update it in the PCB editor.**

Graphical correction of the UGO part

Sometimes we need to correct the pattern of any part - resize, move the vertex of the contour polyline, add the text attribute "Description", etc. If your schematic design contains many identical part patterns (such as resistors or capacitors), then you can add graphic changes to one part and then, selecting a group of similar parts, using the option to replace the old part pattern with a new one.

Example:

1) Change the pattern of one of the resistors (for example R1) by adding a line inside the path.
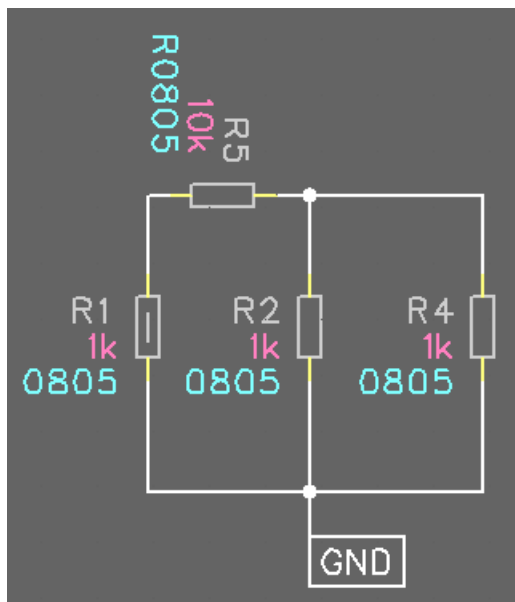
2) Select the text R2 by clicking on it with the left mouse button
3) Call the context menu of the right button.
4) Select the menu item "Select Similar Texts"

Start dragging in the PCB editor

Align Ref Value
Align Ref Footprint
Align Ref Value Footprint
Align Ref Footprint Value

Edit Properties

Select Similar Texts
Select object by attribute

Static highlight mode
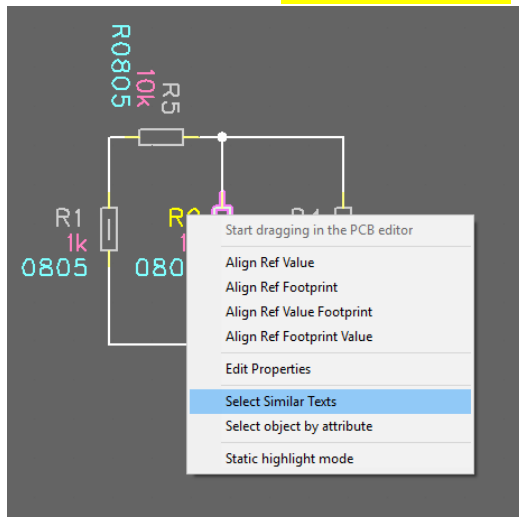
5) Select the menu item "Tools >> Graphic Correction >> Replace Part Pattern"
6) Select in the dialog R1, thus pointing to the reference part, and click OK

Log

***** Part Graphics Correction *****

Attempt number 0, angle 0
Graphics correction of part R2 was successful
Graphics correction of part R4 was successful
An attempt to correct the graphics of part R5 failed. (angels of parts are different)

Attempt number 1, angle 90
An attempt to correct the graphics of part R5 failed. (pin names are different)

Attempt number 2, angle 180
An attempt to correct the graphics of part R5 failed. (angels of parts are different)

Attempt number 3, angle 270
Graphics correction of part R5 was successful
3 parts fixed

Hide

Graphical correction of similar polylines

Sometimes we need to correct the net label by changing the polyline pattern. If your schematic project contains many net labels, then you can make changes to one net label, then select a group of similar polylines and use the option to replace the old polylines with new ones.

Example:

We have three labels, one of which (on the right) has been modified.

Now we will automatically adjust the labels on the left, making them the same as the label on the right. Click on the label on the right (on the sample label) to remember the serial number of the label polyline. The polyline number will be shown in the status bar at the bottom of the window. In our case, this is number 15, you need to remember it.
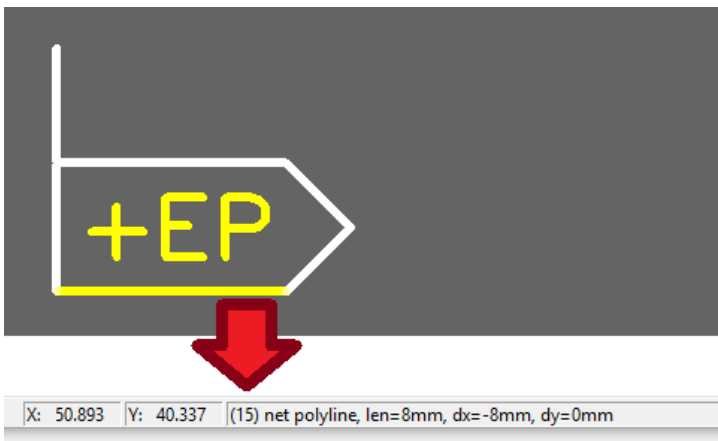
To make the program itself find similar marks, do the following: click on the polyline of any of the marks that you want to fix. Bring up the menu of the right mouse button, and select "Select similar polylines"To automatically select all such polylines on the current page.
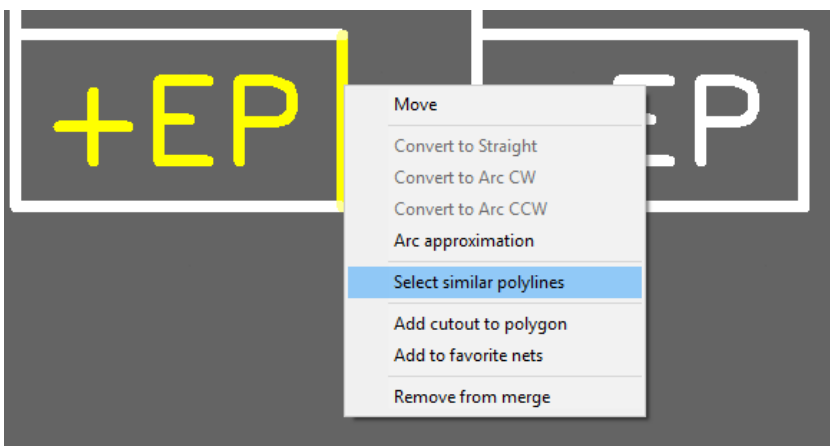
Then call the dialog box for selecting the number of the model polyline to replace the selected ones - "Tools >> Graphic Correction >> Replace Polyline Pattern". Enter the number of the sample polyline (we have this number 15) and click OK. After that, the program will display a dialog in which you need to select the polyline

replacement mode - when you press YES the first vertices of the old and new polylines will be aligned, by pressing NO - the last vertices of the polylines will be aligned.

Schematic Constructor

By default, the coordinates of the first vertices of both polylines will be the same. Align the beginnings of polylines?

Да   Нет
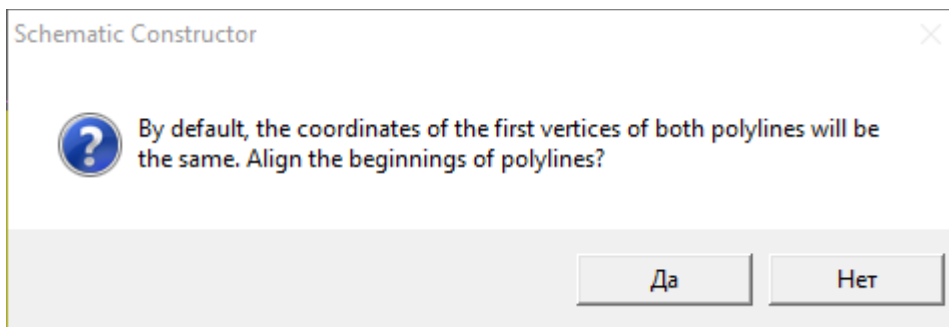
All polylines are now the same

Replacing a text fragment globally

Sometimes we need to replace some text fragment on the whole project page. This can be, for example, replacing a period with a comma. Before calling the dialog box, it is necessary to select the text attributes, among which we want to search and replace the text fragment. For example, if we want to replace the 0.1 text attribute with 0.1 for capacitors, we can do the following:

1) Select any of the 0.1 texts by clicking on it with the mouse
2) Select from the right-click menu "Select similar texts"
3) Call the text replacement dialog box "Tools >> Replace Text Fragment"

Replace Text Fragment

Find   .

Replace by...   ,

OK   Cancel
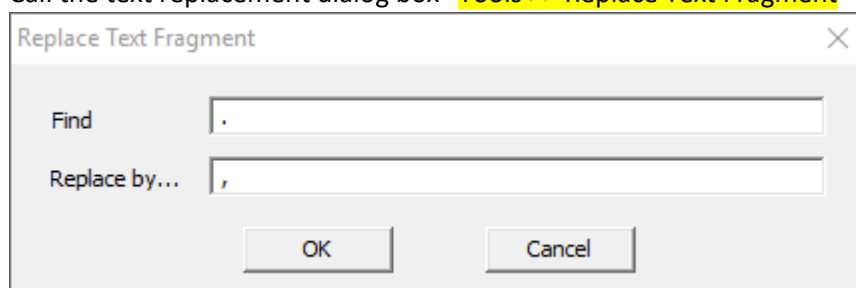
Enter "." In the upper field and "," in the lower field. Click OK, and the program will replace the period with a comma among the selected texts.

You can also use the "Edit >> Select". This menu allows you to select any graphic layer on the current page, be it a polyline layer or a text attribute layer. Select "Edit >> Select >> Attributes" To select all text objects on the current page.

Attaching an image

Schematic Constructor supports import of two types of image files - BMP and JPEG. Any image can be added to the project by attaching it to any polyline. In this case, the image is stretched to fit the rectangle that the polyline occupies on the working area. Therefore, if you want to insert an image into your project, then it is best to do this on a rectangular closed polyline. Draw a rectangle using the function button "F3 - Add Polyline"In the mode when nothing is selected. After that, select the side of the rectangle by clicking on it with the mouse and load the image on it using the menu "File >> Polyline picture >> Attach"

Schematic Constructor can import images in two modes:

1) In normal mode, it can be a color image or a photograph. Displaying the color gamut of the image on the working area exactly matches the content of the image file. These images are displayed when printed to PDF.
2) In the background image mode. For example, it might be a black and white image that contains a circuit diagram for drawing in a circuit editor project. In this mode, any color image is converted to monochrome and displayed with a transparent background. This picture is not printed.

To select the first mode, load an image in JPEG format, to select the second mode, load an image in BMP format (in the dialog box, select the file type - BMP or JPEG and load the file).

When BMP is loaded, the picture is displayed on the screen with the TRANSPARENT property and the background color of the image becomes the same as the background color of the working area. That is why the schematic editor comes with a dark gray background color for the working area. In this mode, the color gamut is optimal - the lines on the background image will be black, and the polylines in the editor are lighter and correspondingly brighter, so it becomes easier to draw a diagram from the picture. In general, this color scheme is easy for most developers to perceive. But you can make changes to the color set of layers through the menu "View >> Colors".
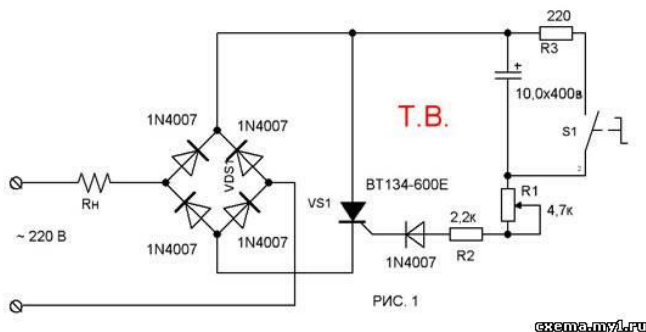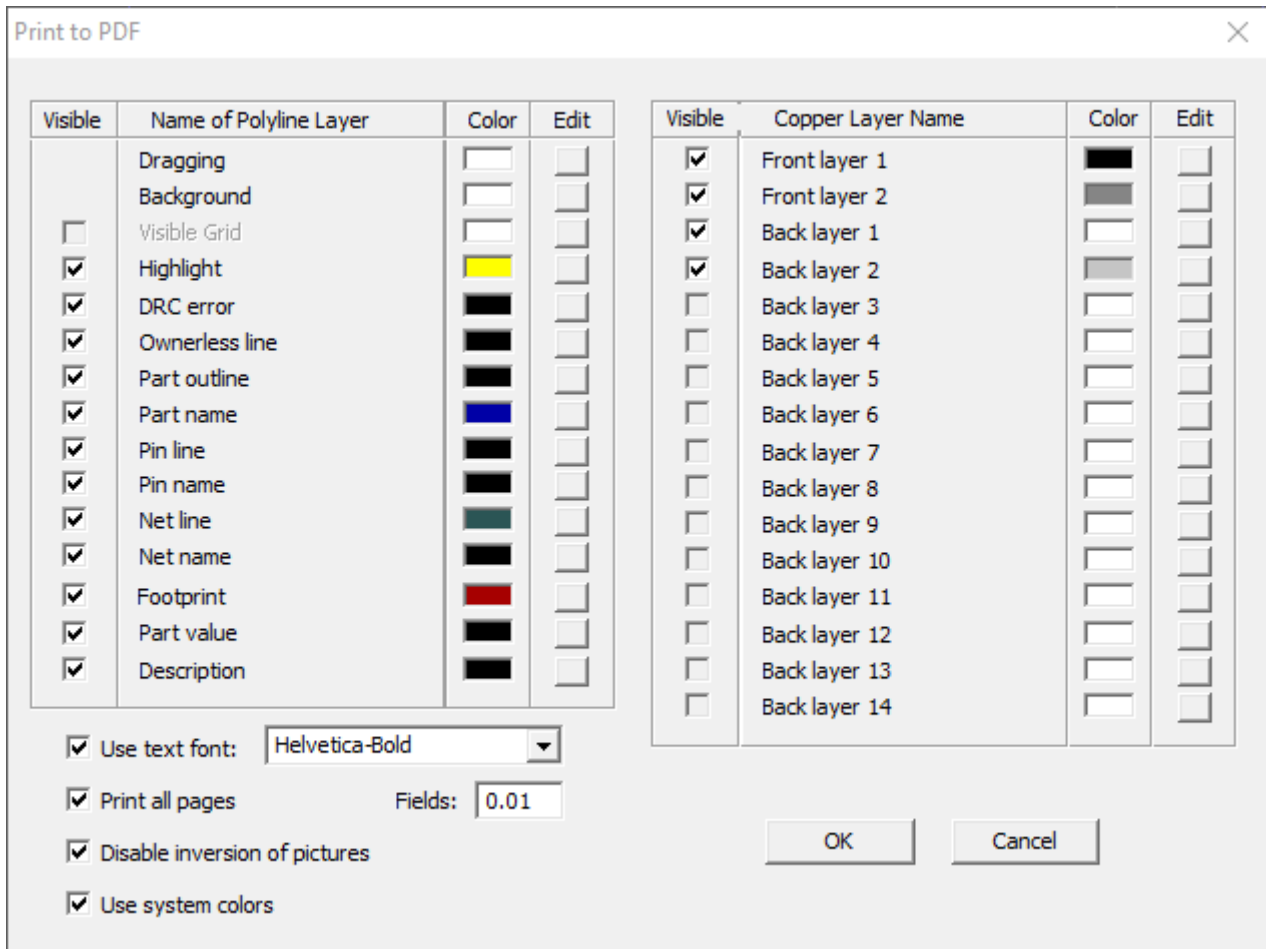

Sample BMP Image
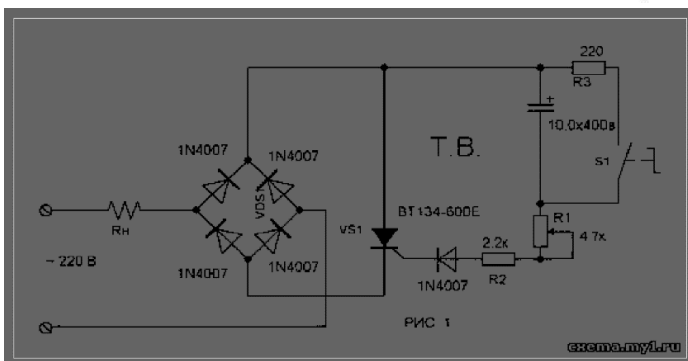
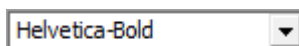*Displaying this file in the program*

Print to PDF

Call the PDF Print dialog box by selecting the main menu item "File >> Print to PDF".

Options:

1) The PDF printer built into the Schematic Constructor has its own local set of fonts that can be used for printing. To use this font, check the "Use text font", And select the name of the font in the combo box to the right of it.

If you do not want to use this set of fonts, then uncheck the "Use text font", In this case the print will use the same vector font as on the working area of the program.

When using PDF fonts, it is important to keep in mind that the size of the letters of this font is regulated by the PDF printer and may differ slightly from the Schematic Constructor font, which is used on the working area in the program window, so the text on paper may have slight displacements. To correct this, any text attribute has a "right-justified" property. You can optionally set the right alignment for any text except the "Pin_name"

attribute, which is automatically aligned. Highlight some text on the work area and press the function button "F1 - Edit Text".
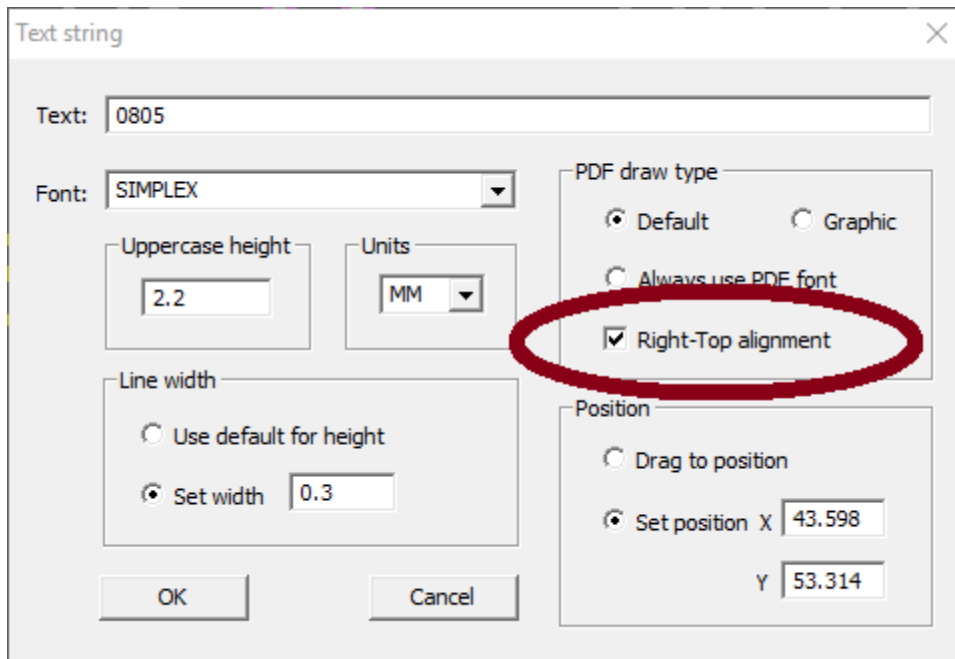
Use this checkbox for right or top alignment of text.

2) Checkbox "Print all pages"Allows you to print the entire project into one PDF file, otherwise only the current page will be printed.
3) Checkbox "Fields"Sets the indentation from the edges in the PDF file.
4) Checkbox "Disable inversion of pictures"Prohibits the inversion of the color of any JPEG image when printing, while on the working area this image may have a color inversion ("File >> Polyline Picture >> Invert").
5) Checkbox "Use system colors"Makes the current layer color setting global, applicable to any project. If you uncheck this box, the PDF color setting, as well as the color set on the working area, will be read from the project file when reading the project file. Thus, in each project, you can customize the individual color scheme.

Print example:

PCB: MyProj2-01.fpc

ITEM	VALUE	FOOTPRINT	CNT	DETAILS
1	1.0uF	1206	2	C1A, C1B
2	ARL-5313URC	JUMP2	2	L1A, L1B
3	500k	0805	2	R3A, R3B
4	1k	0805	2	R2A, R2B
5	BC547	JUMP3	2	T1A, T1B