

After you click OK, all those separate elements will be grouped together to form your new symbol directly in the schematic.

Using the group function, you can create/edit any symbol in the schematic, easily and quickly.

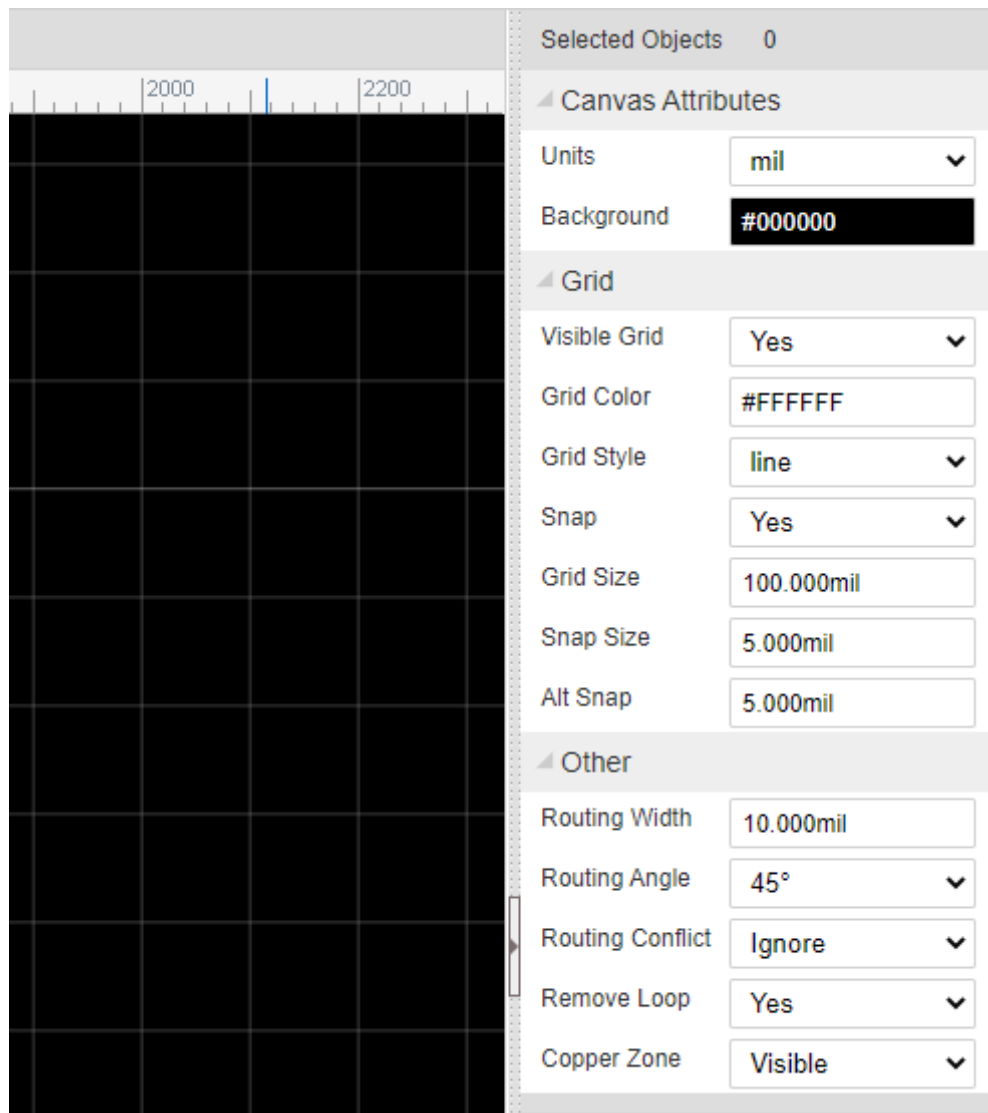
PCB Layout

Canvas Setting

After the initial conversion of a schematic to PCB, it is time to learn how to manage EasyEDA's PCB Design Editor.

Canvas Attributes

Lots of PCB canvas attributes are the same as Schematic canvas attributes. The key is that you can set **units** in PCB canvas attributes.



When you select a object at the canvas, you can modify its attributes at the right panel.

Snap Size: The cursor snapping size.

Alt Snap: When press hotkey ALT the cursor snapping size.

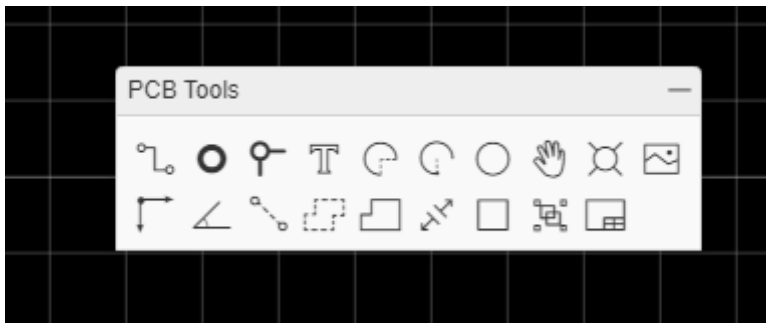
Other

- **Routing Width:** Setting the default routing width.
- **Routing Angle:** Setting the routing angle.
- **Routing Conflict:** When routing the track, what to do when impact the difference net objects.
 - **Ignore:** The track go through the objects.
 - **Block:** The track will stop when meet the difference net objects.
 - **RoundTrack:** The track will go around the difference net objects.
- **Remove Loop:** Remove the track loop.
- **Copper Zone:** Setting the copper zone visible or invisible.

PCB Tools

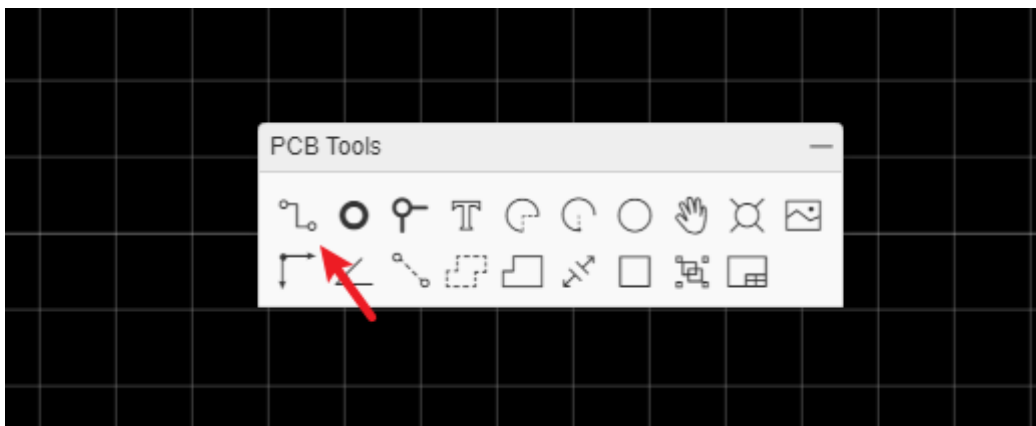
PCB tools provide many function to fulfill your PCB design requirement.

Such as: Track, Pad, Via, Text, Arc, Circle, Move, Hole, Image, Canvas Origin, Connect Pad to Pad, Copper Area, Solid Region, Measure/Dimension, Rect, Group/Ungroup. etc.

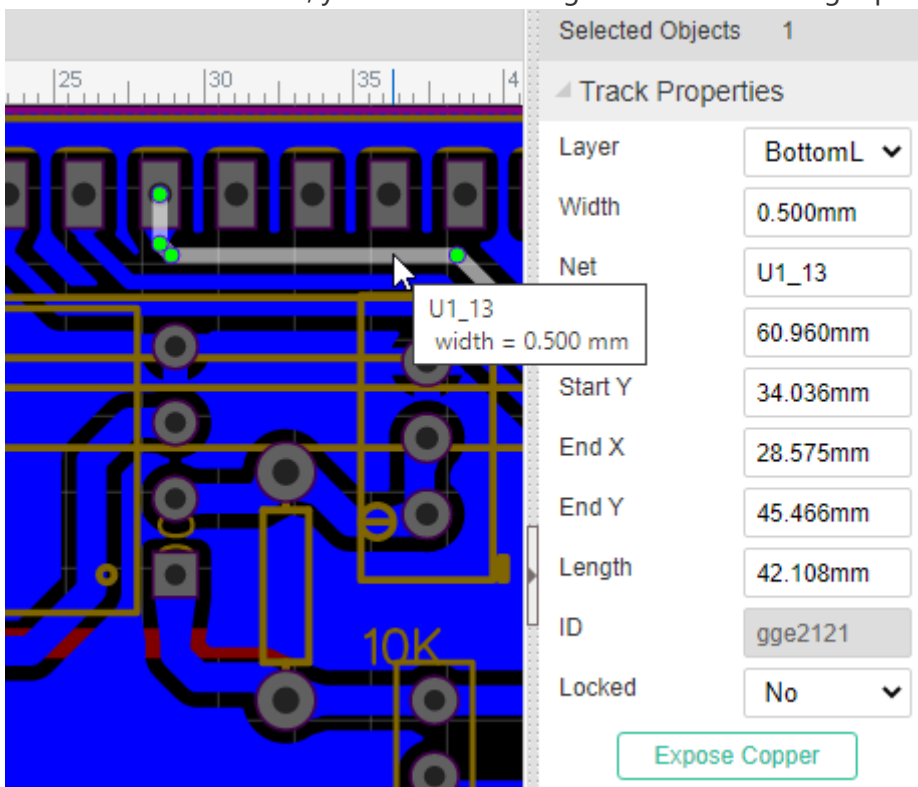


Track

In the schematic editor, we use Wire or the **W** Hotkey to connect Pins, in a similar way in the PCB editor, we use Track to connect Pads. Track allows you to draw PCB tracks and can be found on the PCB Tools palette or using the **W** Hotkey (not T: see above!).



When a track is selected, you can find its Length attribute in the right panel.



If you want to create solder mask for the track, you can click the "Expose Copper" button at the right-hand property panel.

When click the track, you will see the nodes, you can drag it or right-click delete it.

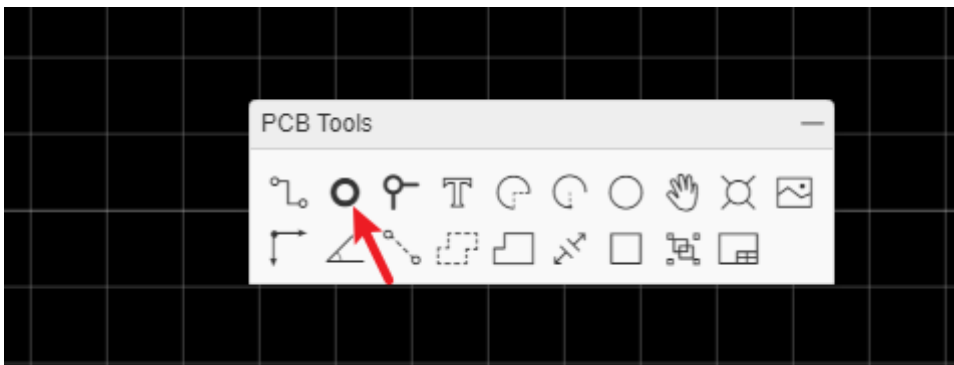


when select the point to point separated tracks, you can convert them as Solid Region or continus track at right-click menu.

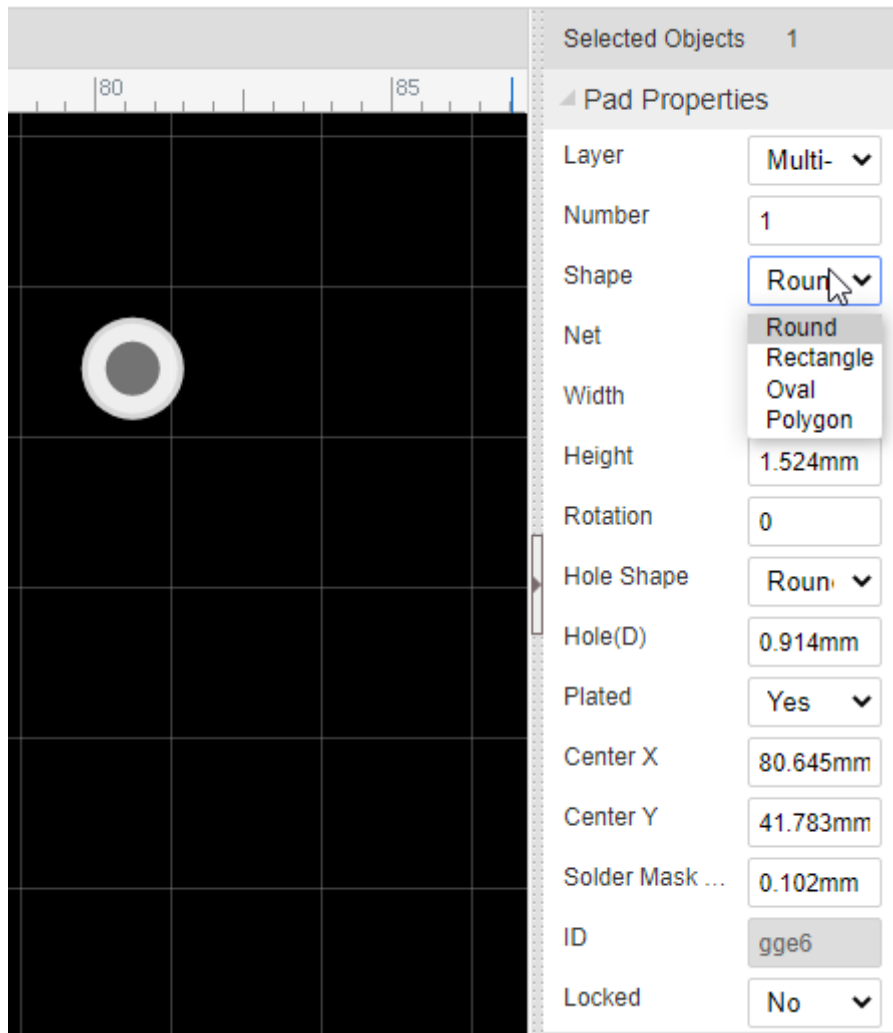
The more information of routing, please refer at [PCB: Route Tracks](#)

Pad

You can add pads using the Pads button from the Footprint Tools palette or using the **P** hotkey.



After selecting one of the pads, you can view and adjust its attributes in the right hand Properties panel.



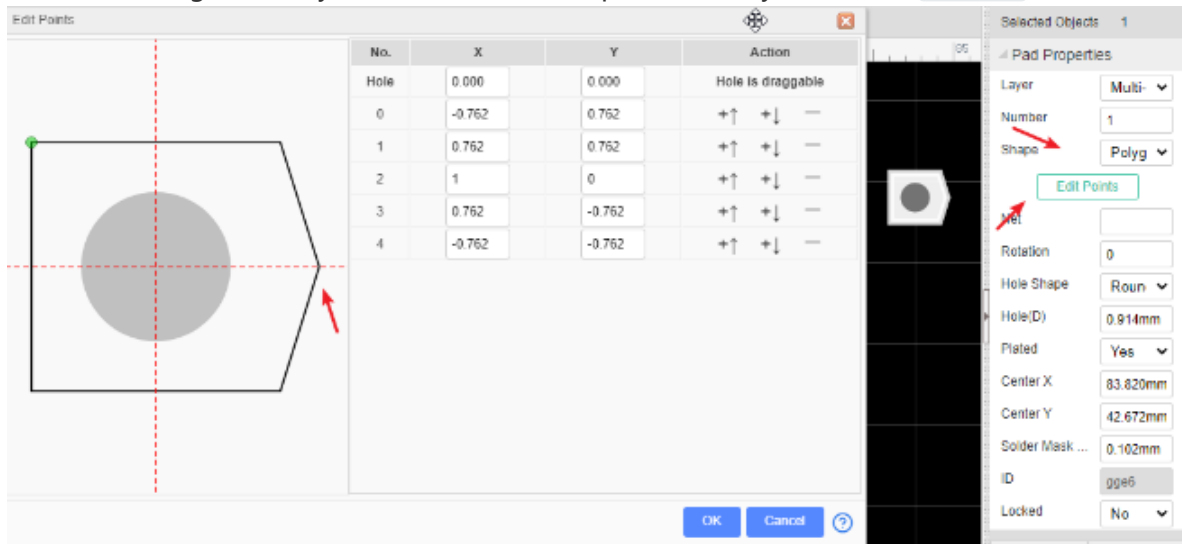
Number: Remembering the pin numbers you set in the schematic symbol in your Schematic Lib: to connect those schematic symbol pins to the pads in your PCB footprint, the pad numbers you set here in the Footprint footprint must be the same.

Shape: Round , Rectangular , Oval and Polygon.

EasyEDA supports four shapes: Round , Rectangular , OVAL and POLYGON.

- OVAL PAD will give your more space.
- POLYGON PAD will let you to create some strange pad.

Like in the image below, you can edit the PADs points when you select a POLYGON PAD



Layer: If the pads are part of a **SMD** footprint, you can set it to **Top layer** or **Bottom layer**. For through hole components you should set it to **Multi-Layer**. If it setting as multi-layer, it will connect with all copper layers.

Net: You don't need to enter anything here because at present this footprint is not connected to anything in a circuit.

Width and Height: When the shape is set to Round, Width will equal Height.

Rotation: Here you can set the Pad's rotation as you want.

Hole(D): This is the drill hole **diameter** for a through hole pad. For a SMD Pad, set its layer to **TopLayer** or **BottomLayer**.

Hole Shape: Round and Slot. When it is set as a slot, the Gerber is generated through the stitching of multiple drill holes in the corresponding position. If your hole is round, please do not set it as a slot, so as to avoid the overlapping error of holes during the production of DFM detection.

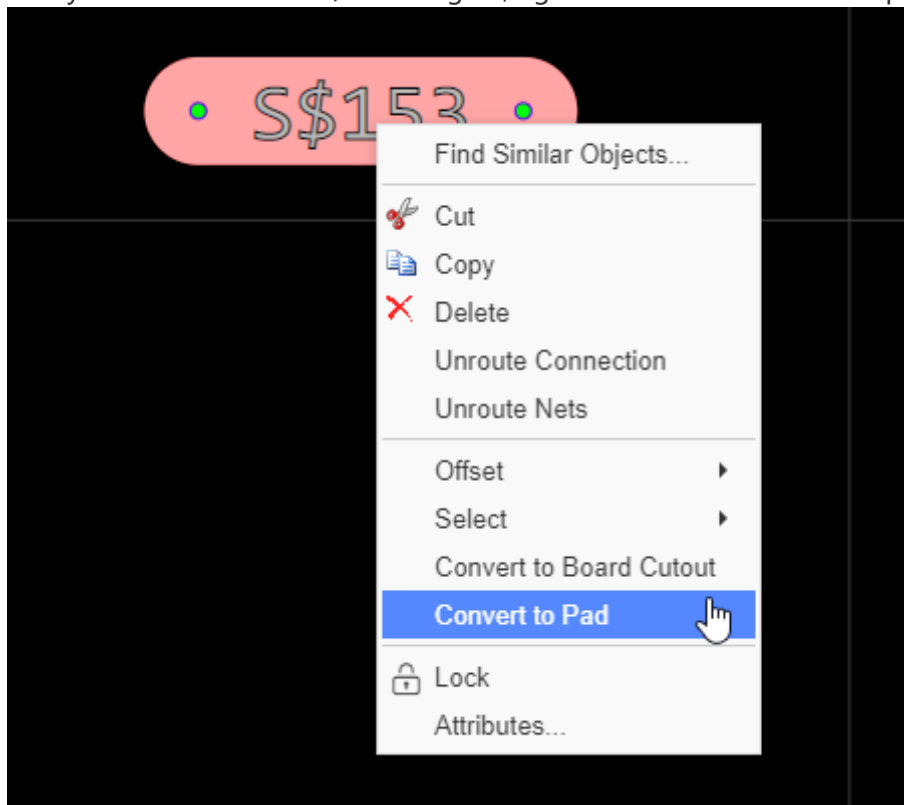
Center-X and Center-Y: using these two attributes, you can set the pad's position with more precision, compared to using the mouse.

Plated: Yes or No. When you set it as No, this pad Inner wall do not metallization.

Paste Mask Expansion: For single layer pad. This property affects the size of the tin area on the plate of the steel mesh. If you want to set a pad that is not open in the steel mesh, you can set the value to be negative, which is usually larger than the diagonal of the pad.

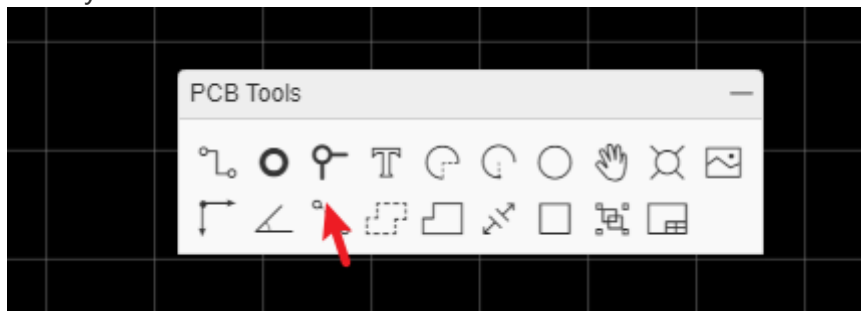
Solder Mask Expansion: This property affects the size of the green oil area cover on the pad. If you want to set a pad not open covered with green oil, you can set the value to be negative, the value is usually set larger than the diagonal of the pad.

And you can select a track/Solid Region, right-click it and convert to a pad.



Via

When you want to lay a multilayer PCB, you need to add Vias for nets getting through layer and layer.



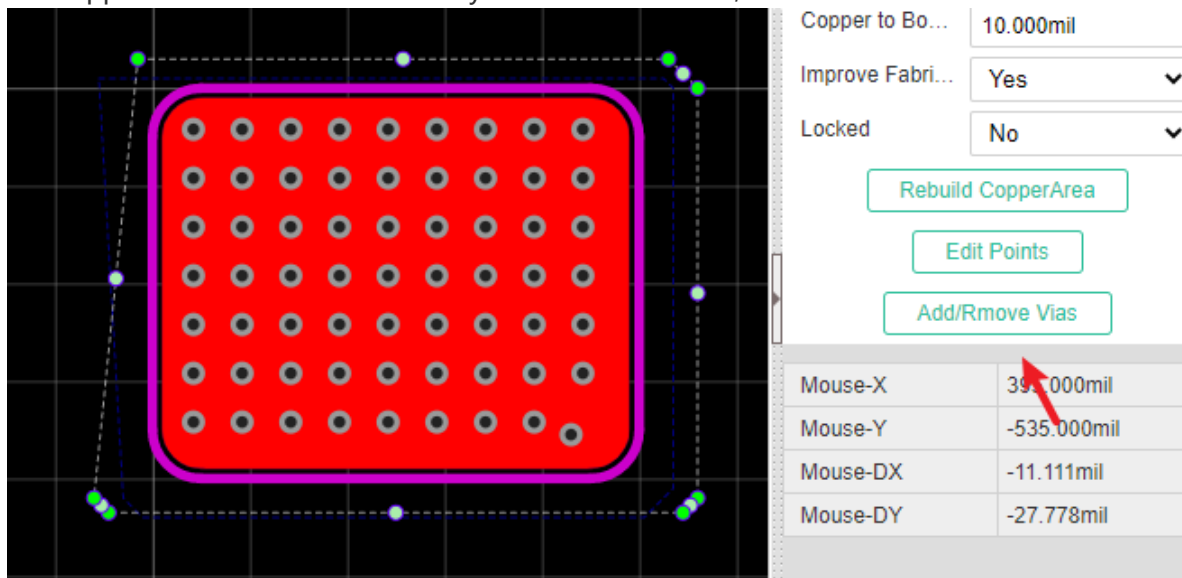
Place a Via on a Track

When placing a **via** on a track, the track will be cut to two segments, and the via net will follow track's net. Placing two vias on a tracks, you will get three segments, then you can change one segment to other layer id, or remove one of them.



Place Multiple Vias

Click the copper area outline, click the "Add/Remove Vias" button. this feature needs the same net copper areas on two and more layers in the same time, the cross area will add the vias.

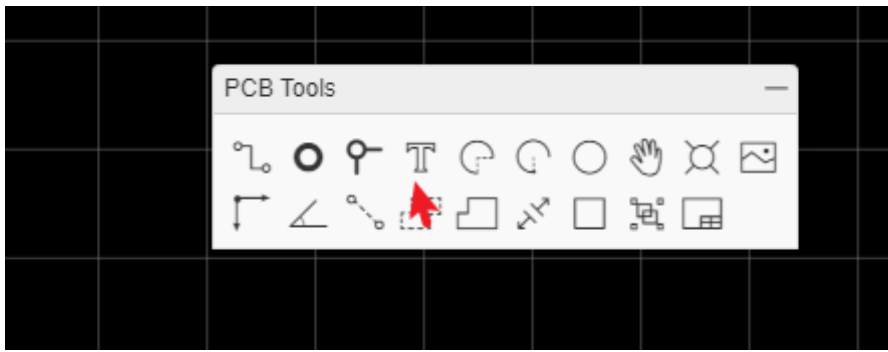


Notice:

- EasyEDA only support the through via for all layers, doesn't support the buried/blind via.

Text

You can add more fonts from your computer or download [free fonts:](http://www.fontspace.com/)



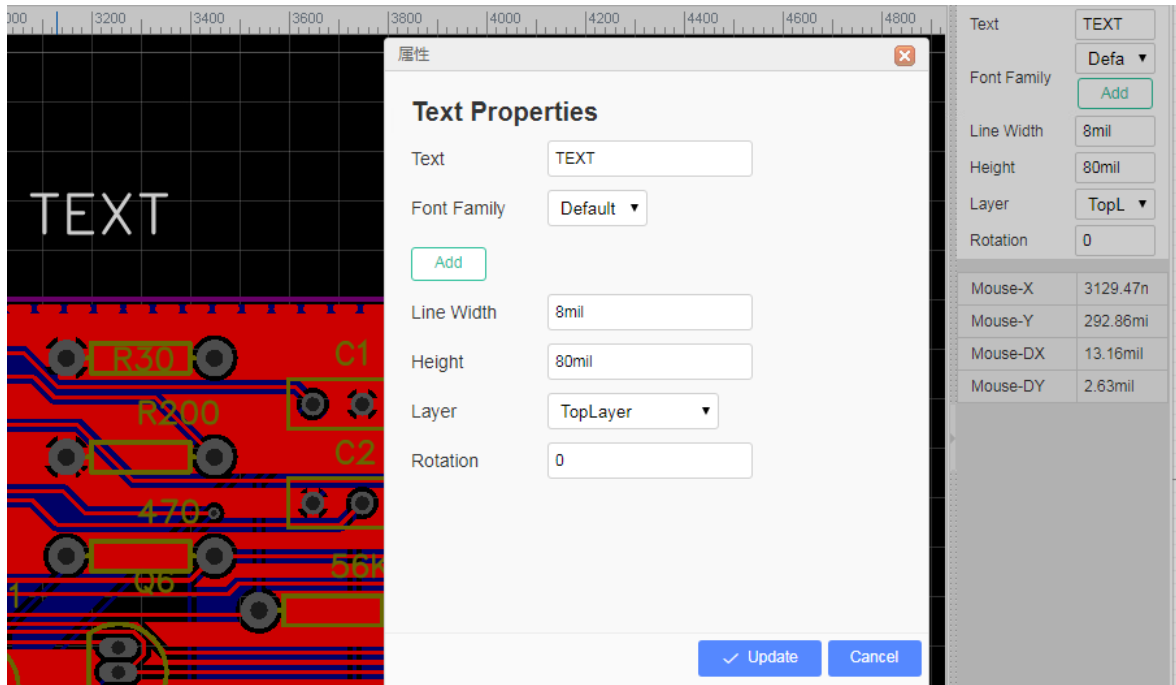
if you need Japanese or Korean you can use [Google Noto fonts](#)

The editor including fonts are:

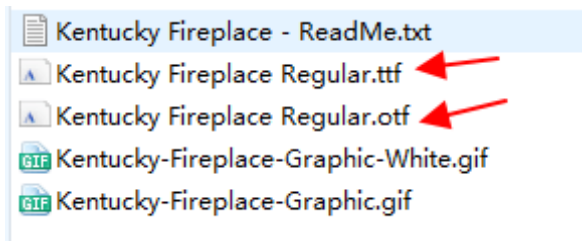
(NotoSansCJKsc-DemiLight)[<https://github.com/googlefonts/noto-cjk/blob/main/Sans/OTF/SimplifiedChinese/NotoSansCJKsc-DemiLight.otf>]

(NotoSerifCJKsc-Medium)[<https://github.com/googlefonts/noto-cjk/blob/main/Serif/OTF/SimplifiedChinese/NotoSerifCJKsc-Medium.otf>]

Select the text, then you can find a Font-family attribute on the right panel like in the image below.



Click the add button, then choose the font, the font file must be `ttf` or `otf`.

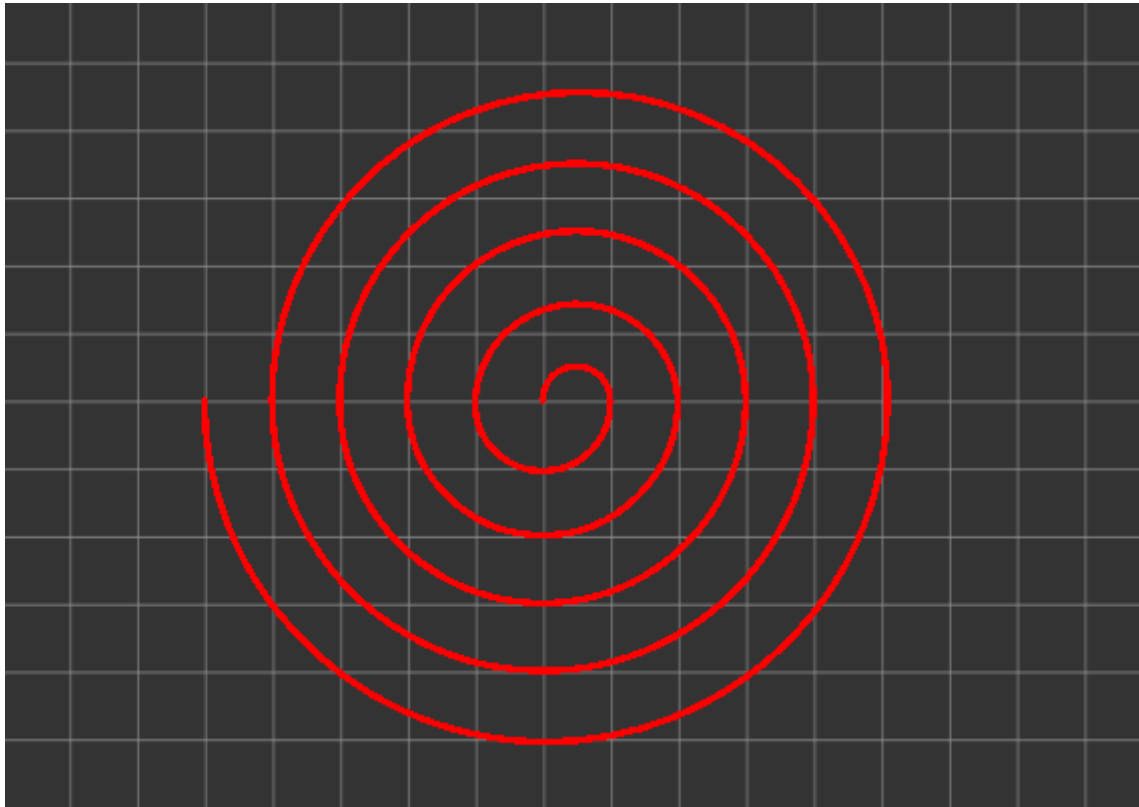


So you can add any fonts by yourself. EasyEDA doesn't cache the font on our server, so if you close the editor, you need to add the font again by yourself.

Note: If you use the other font, the `Linewidth` attribute is useless, because it will be automatically set by changing the `Height`.

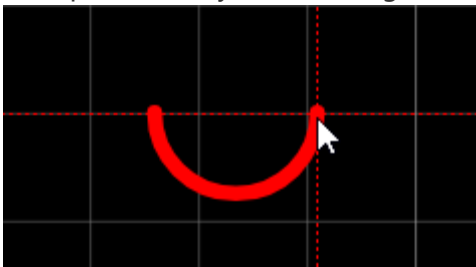
Arc

You can draw many Arcs with different sizes, it's easy to create a pretty cool PCB as you like.

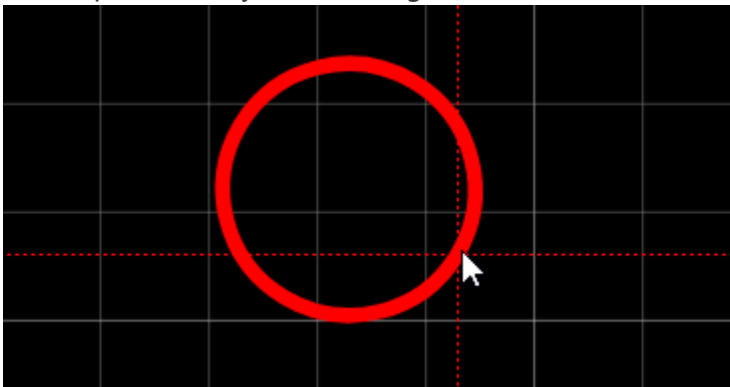


EasyEDA provides two Arc tools:

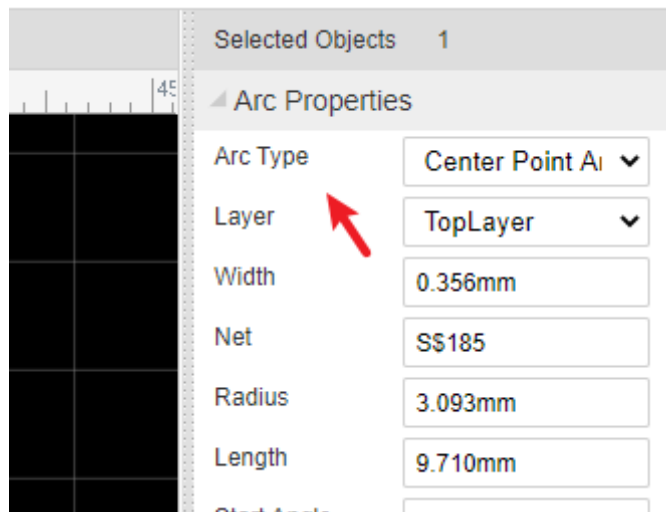
- Start point fixed, you can change the end point position and radius.



- Center point fixed, you can change the radius.



Select the arc, you can change the arc type at property panel, different arc type has different drag behavior.



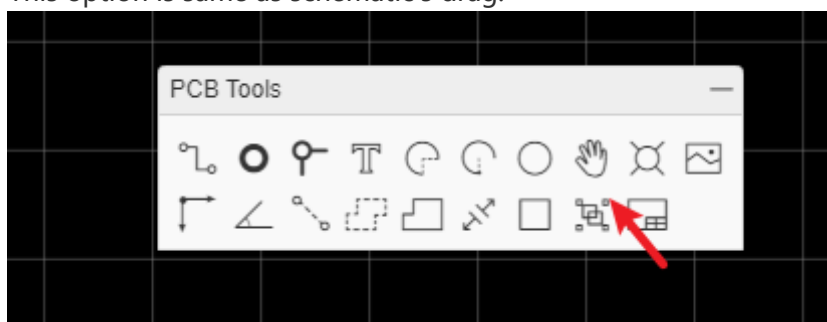
Circle

You can draw a circle in PCB. If you want to draw a circle at TopLayer or BottomLayer, please use Arc.



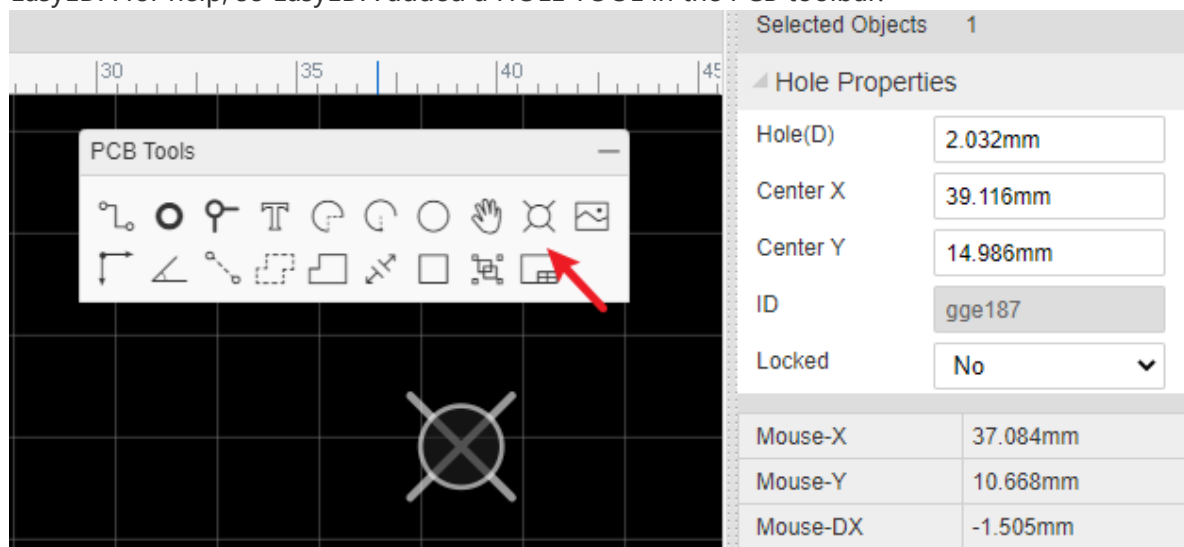
Move

This option is same as schematic's drag.



Hole

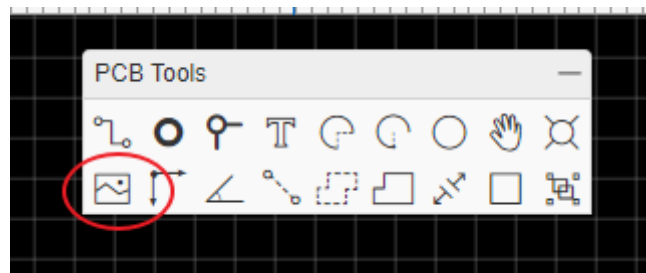
There were lots of users that didn't know how to use PAD or VIA as a HOLE, they asked EasyEDA for help, so EasyEDA added a HOLE TOOL in the PCB toolbar.



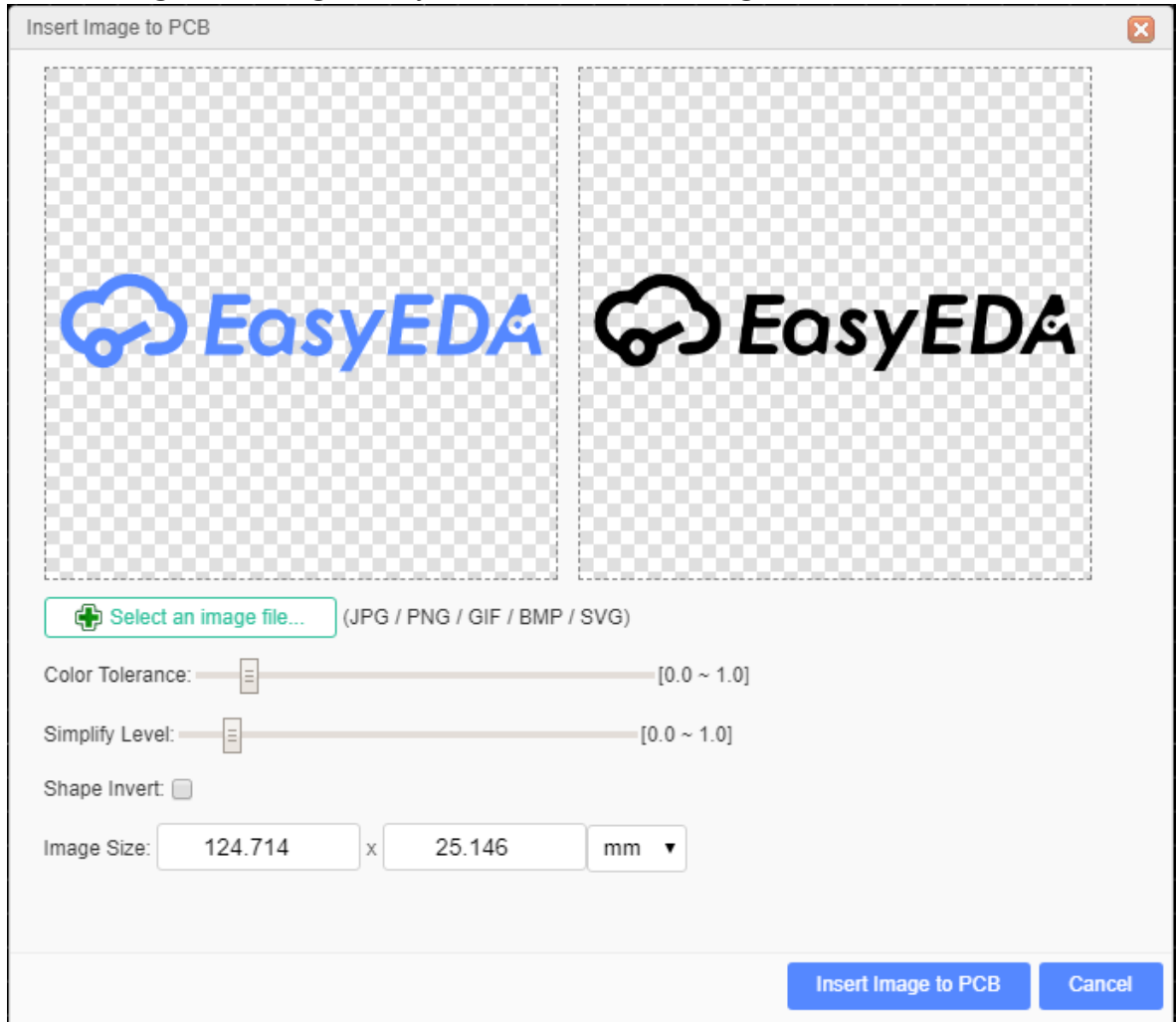
And if you want to create the slot hole, you can use solid region(Type: NPTH), or route a track, and then right-click the "Convert to NPTH" menu.

Image

On PCB and Footprint editor, there is a nice feature on the PCB Tools bar.



After clicking on the image icon, you will see the Insert Image window as below.

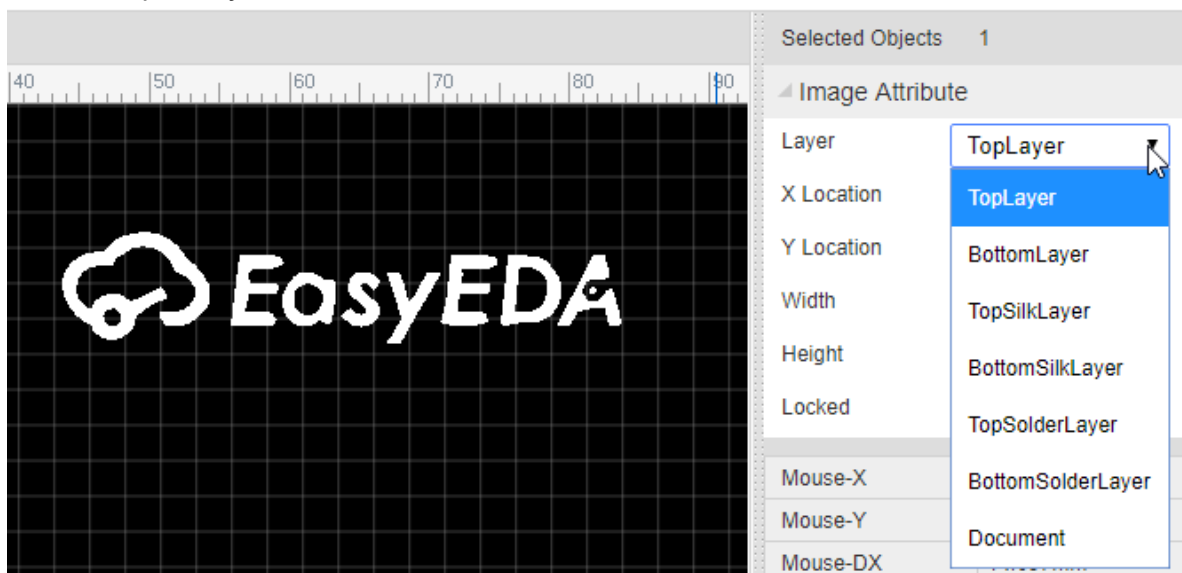


In this dialog, you can choose your favorite image, EasyEDA support `JPG`, `BMP`, `PNG`, `GIF`, and `SVG`. Unlike some other EDA tools which only support a Monochrome Bitmap image, EasyEDA supports full color, but Monochrome Bitmap is welcome.

You can adjust the color tolerance, simplify level and reset the image size there.

And you can select shape invert.

The image will be inserted to the active layer, if it is not right, you can change the attribute. Such as TopSilkLayer.



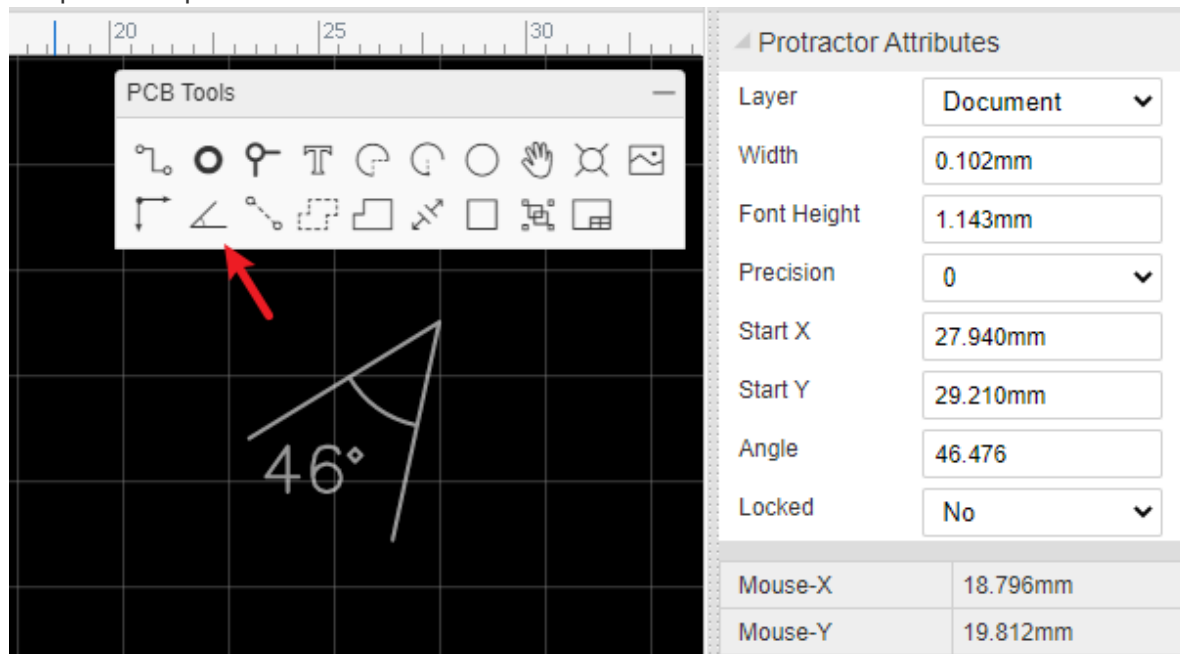
Canvas Origin

This option is the same as schematic's Canvas Origin.



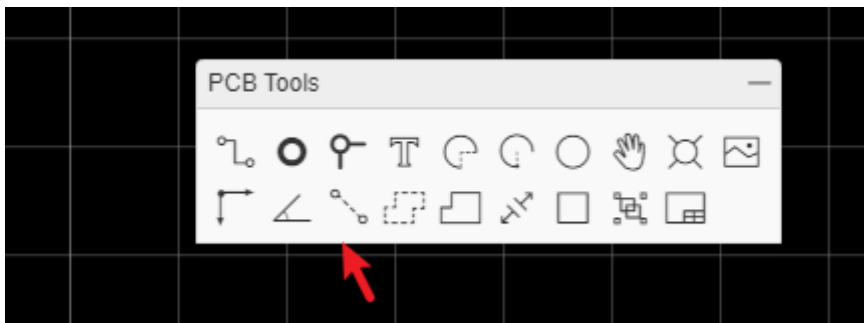
Protractor

We provide a protractor for PCB tools.



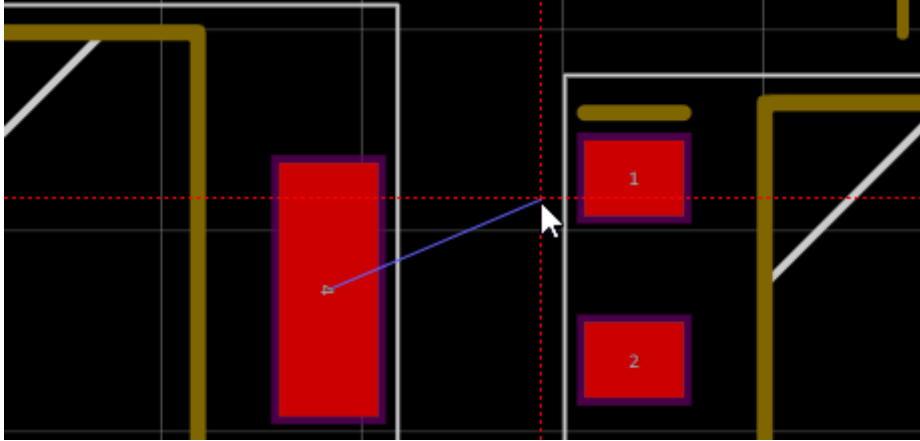
Connect Pad to Pad

When creating a PCB without a Schematic, none of the pads on the Footprints have nets connecting them so there will be no ratlines.



Rather than try to track the pads from scratch, it is a good idea to connect them up by hand first using [Connect Pad to Pad](#) from the PCB Tools palette. This will help you to remember to track the pads correctly with fewer mistakes.

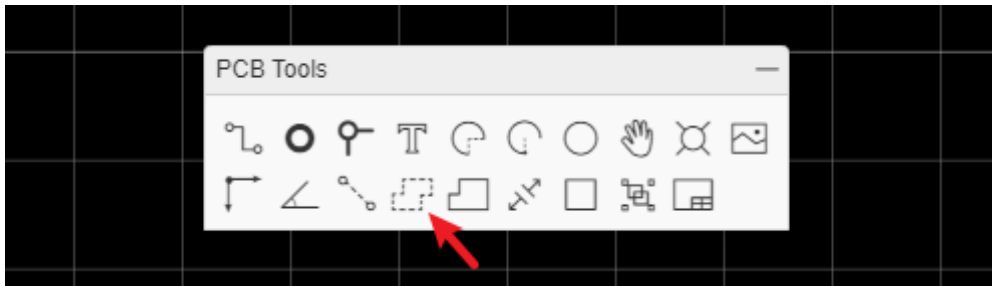
You could also do this by setting net names for all the pads: if the two pads are given the same net name then EasyEDA will understand that they are connected together and will automatically create a ratline between them.



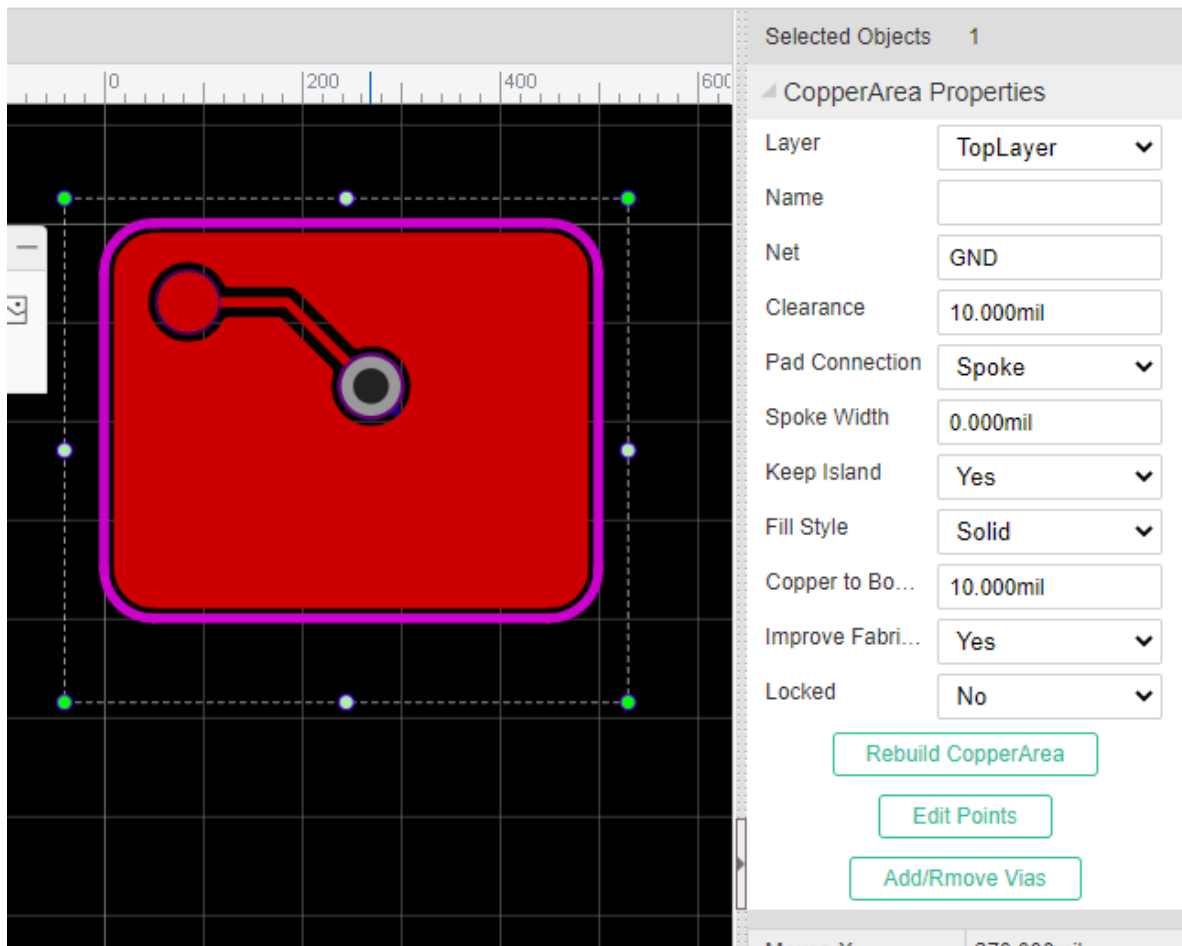
Or you can set these two pads with the same net name at the right panel Pad Properties after you click the pad.

Copper Area

Sometimes you will want to fill in or flood an area with copper. Usually this copper area will be connected to a net such as **GND** or a supply rail. You can draw the outline of a flood using the **Copper Area** button from the PCB Tools palette.



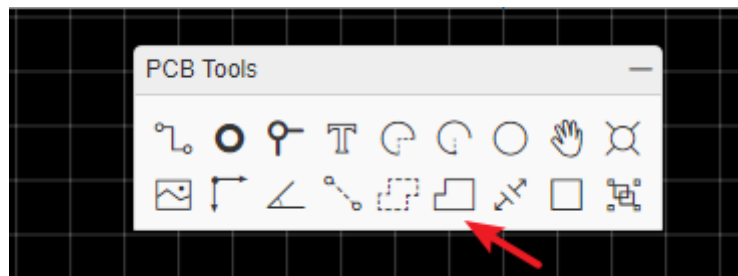
When selecting a copper area, you can find its attributes from the right hand **Properties** panels.



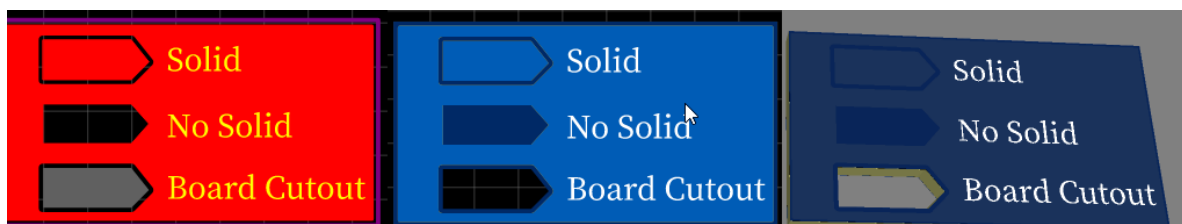
The more information please refer at [PCB: Copper Pour](#)

Solid Region

EasyEDA has added a new tool Solid Region for PCB design



This is a very useful, quick way to connect Pads. You can draw a Solid Region to include all of these pads with same net name, then set the region to the same net name as the pads. It is like Copper Area but easier to use for small areas. To use Solid Region like this, set the Type attribute (in the right hand Properties panel) to Solid.



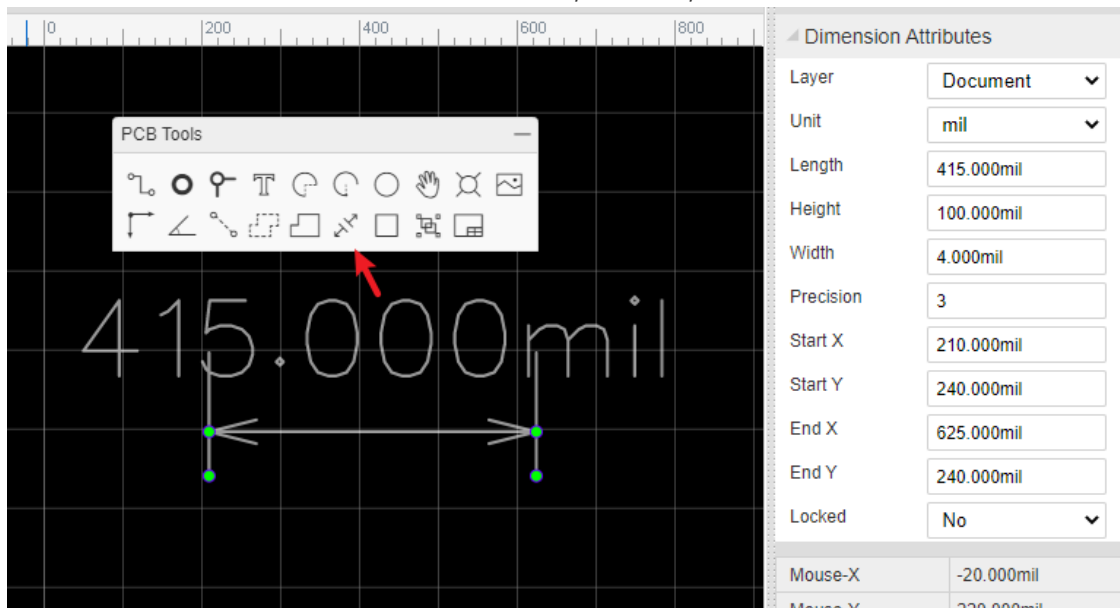
The more information please refer at [PCB: Solid Region](#)

Measure/Dimension

Making and adding measurements is useful in PCB design. EasyEDA provides two methods to do this.

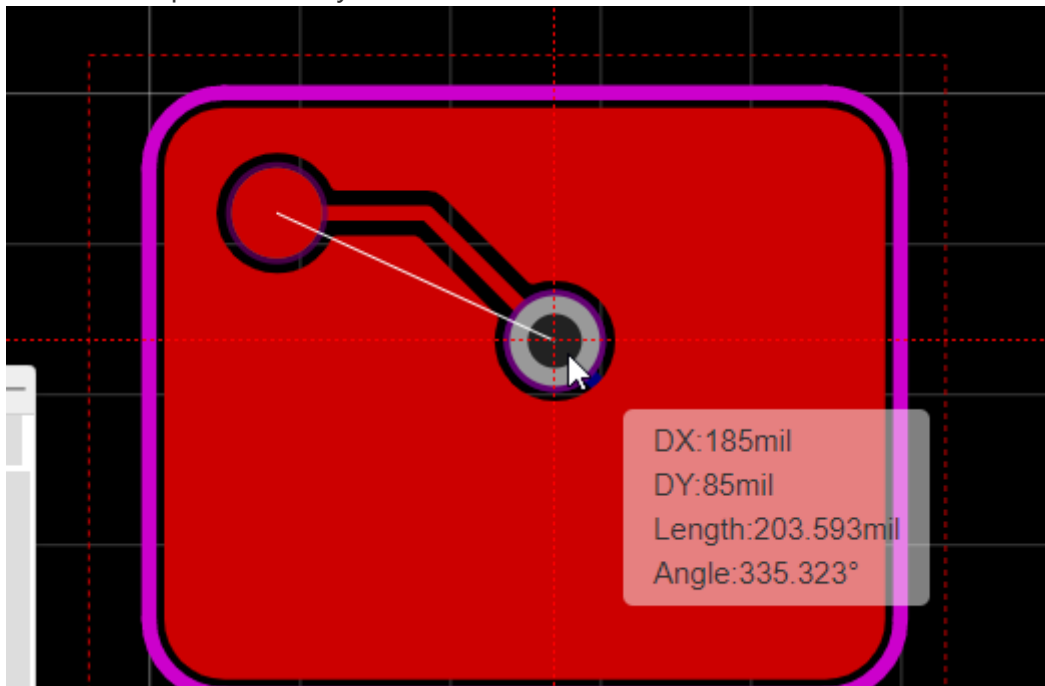
1. Dimension tool in the PCB Tools palette:

This tool can show three units on the canvas, millimeter, inch and millimeter.



When you click one side of the dimension on the PCB, you can drag it for any directions or change its length.

2. Measure a distance using Hotkey **M**, Or Via: **Top Menu > Edit > Measure Distance**, then click the two points which you would like to measure.

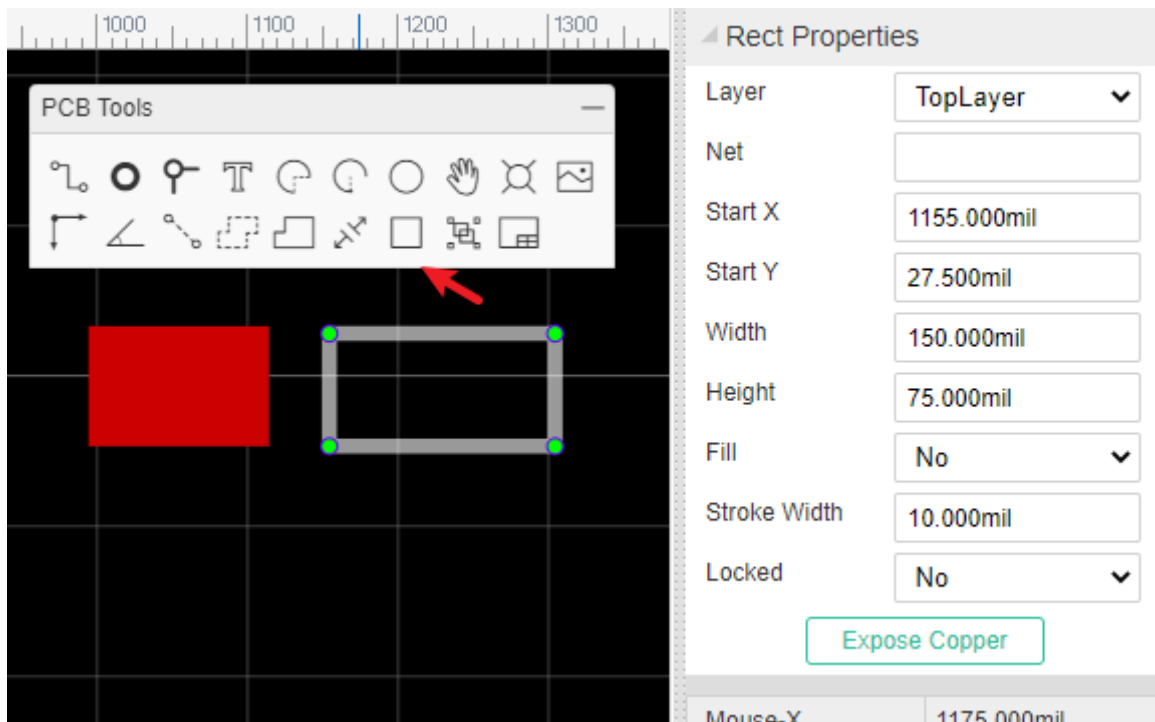


Tips:

- *It's unit follows canvas's units.*
- *You can disable the snap option to measure at the canvas property panel.*

Rect

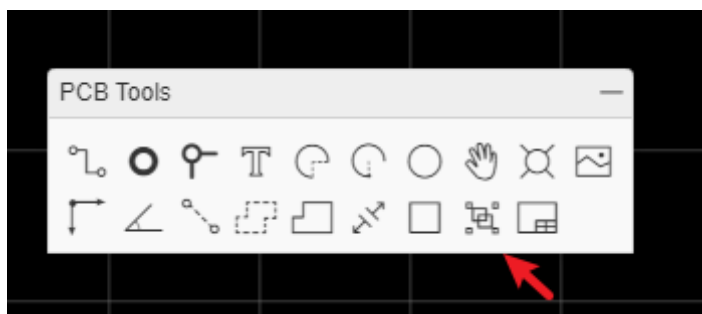
It looks like a Solid Region, but it can't be set Nets and you can't set the Layer as NTPH.



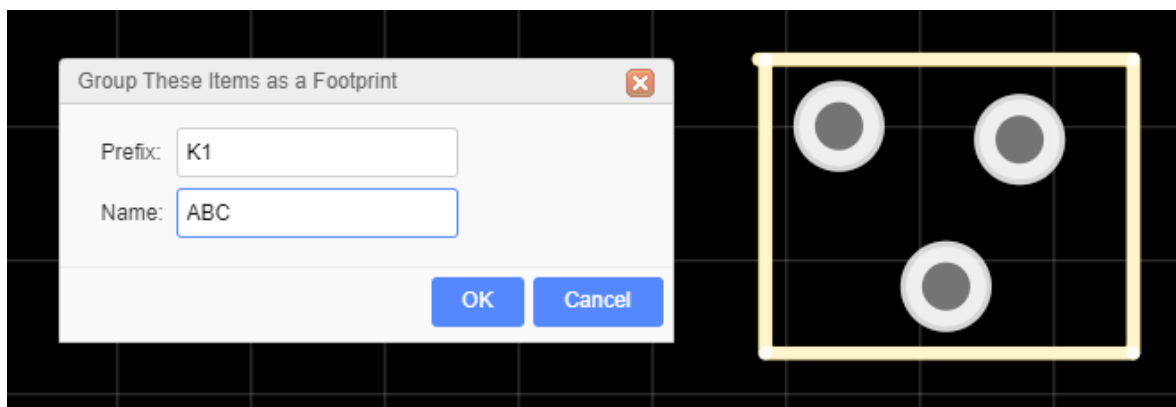
The rect doesn't rotate, you can change its width and height.

Group/Ungroup

Just like Group/Ungroup in the Schematic Editor can be used to create a schematic lib symbol, you can use Group/Ungroup from the PCB Tools palette to create a Footprint footprint in the PCB editor.

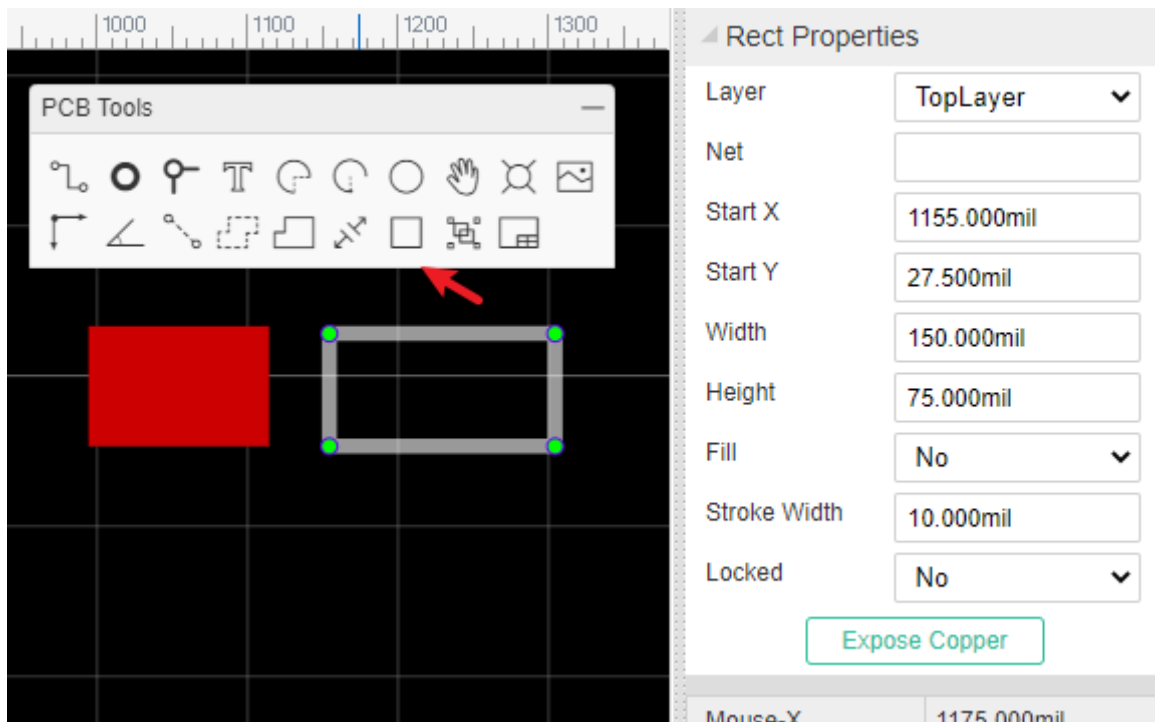


For example, place Tracks and Pads on the canvas, then select all of them and click **Group/Ungroup** to group them like as a footprint in the image below:



Notice:

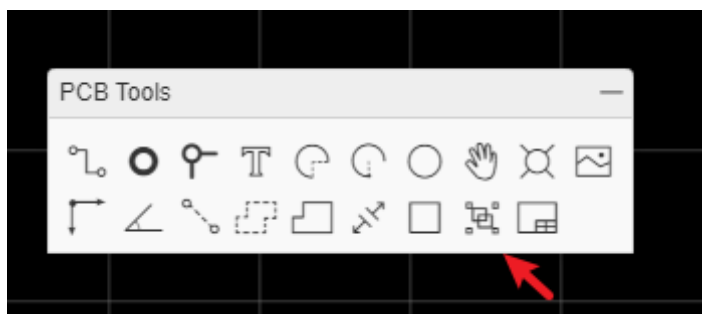
- Before ungroup the footprint, please change it's layer to top layer first, because of the footprint after grouping will at top layer.



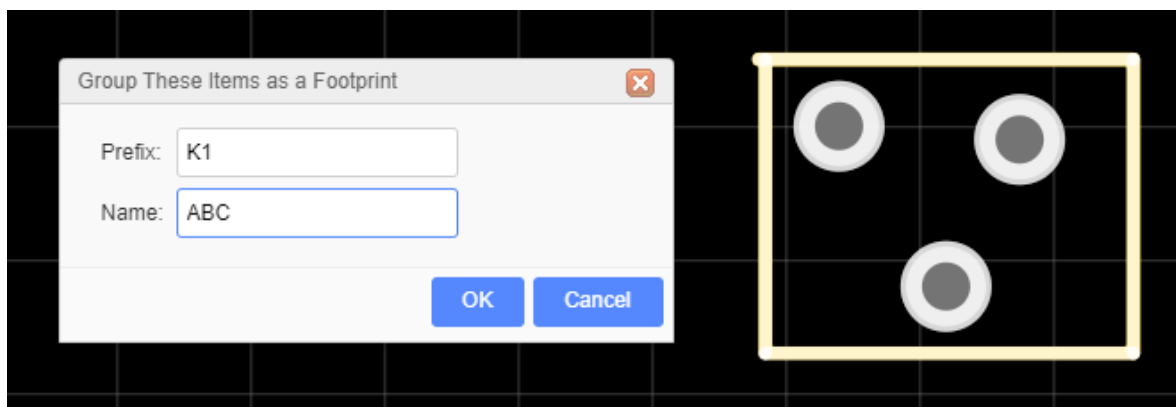
The rect doesn't rotate, you can change its width and height.

Group/Ungroup

Just like Group/Ungroup in the Schematic Editor can be used to create a schematic lib symbol, you can use Group/Ungroup from the PCB Tools palette to create a Footprint footprint in the PCB editor.



For example, place Tracks and Pads on the canvas, then select all of them and click **Group/Ungroup** to group them like as a footprint in the image below:



Notice:

- Before ungroup the footprint, please change it's layer to top layer first, because of the footprint after grouping will at top layer.

- The grouped footprint doesn't support Import Changes, it will be removed if you Import Changes.

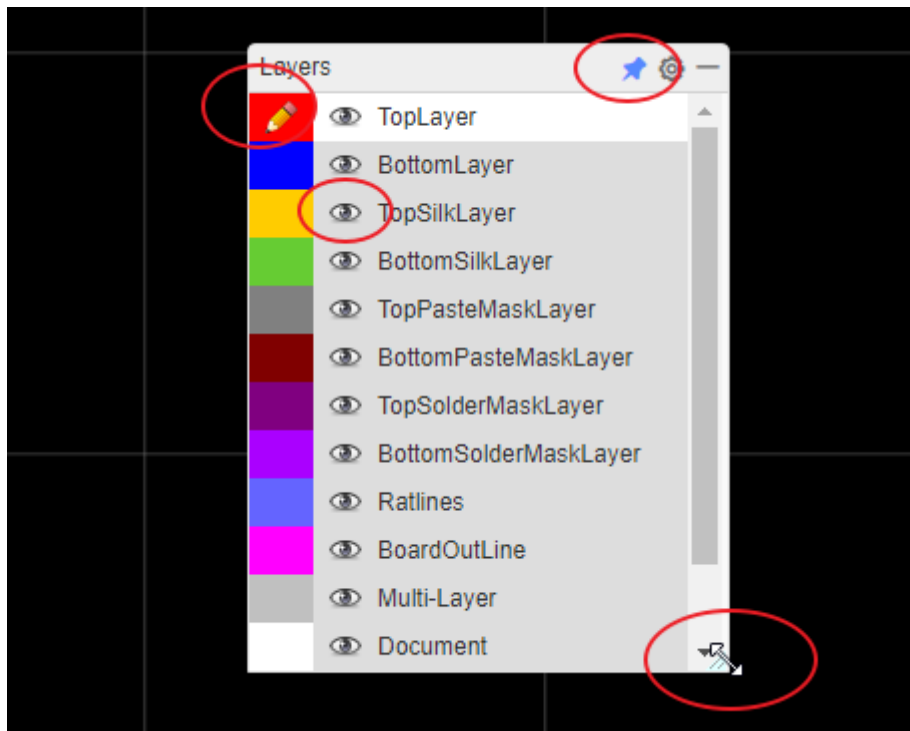
Layers Tool and Objects

Layers Tool

Active Layer: The colours of the layers in the **Layers Tool** are defined in the Layer Options Settings. To work on a layer then you must make it the Active layer.

To do this,

- Click on the eye icons to show/hide layers.
- The pencil icon in the coloured rectangle indicates that this is the active layer.
- Click the pin icon to fix the layering tool without automatically closing it.
- The height and width of the layer tool can be adjusted when dragging the lower right corner of the Layers Tool.



HotKeys for layer activation:

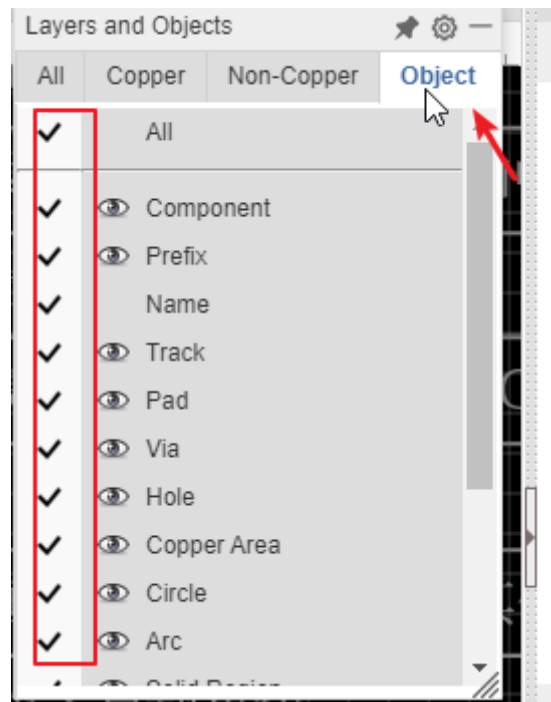
- **T**: Top Layer is active
- **B**: Bottom Layer
- **1**: Inner1 Layer
- **2**: Inner2 Layer
- **3**: Inner3 Layer
- **4**: Inner4 Layer

The more information for the PCB layers please refer at [PCB Layout - Layer Manager](#)

note: the hidden PCB layer is only visually hidden. The corresponding layer will still be exported during photo preview, 3D preview and Gerber export.

Objects Filter Tool

Click "Object" to switch to object filtering.



Select: When the tick in front of the object is checked, the corresponding object in the canvas can be manipulated with the mouse. Uncheck will not allow mouse operation. Including click selection, box selection, drag and other operations.

Eye: Click eyes to modify the display and hiding of corresponding objects in batches.

- Component: Displays or hides the entire components, excluding the component's name and prefix
- Prefix: Displays or hides the entire components' prefix
- Name: Displays or hides the entire components' name
- Track: Displays or hides the entire tracks, for all layers
- Pad: Displays or hides the entire free pads, excluding the pads in the component
- Copper Area: Displays or hides the entire copper areas' fill area, excluding copper outline
- Text: Displays or hides the entire normal texts, excluding the text of the component

Note:

- The layer and object invisible and visible will not go into Undo and Redo.

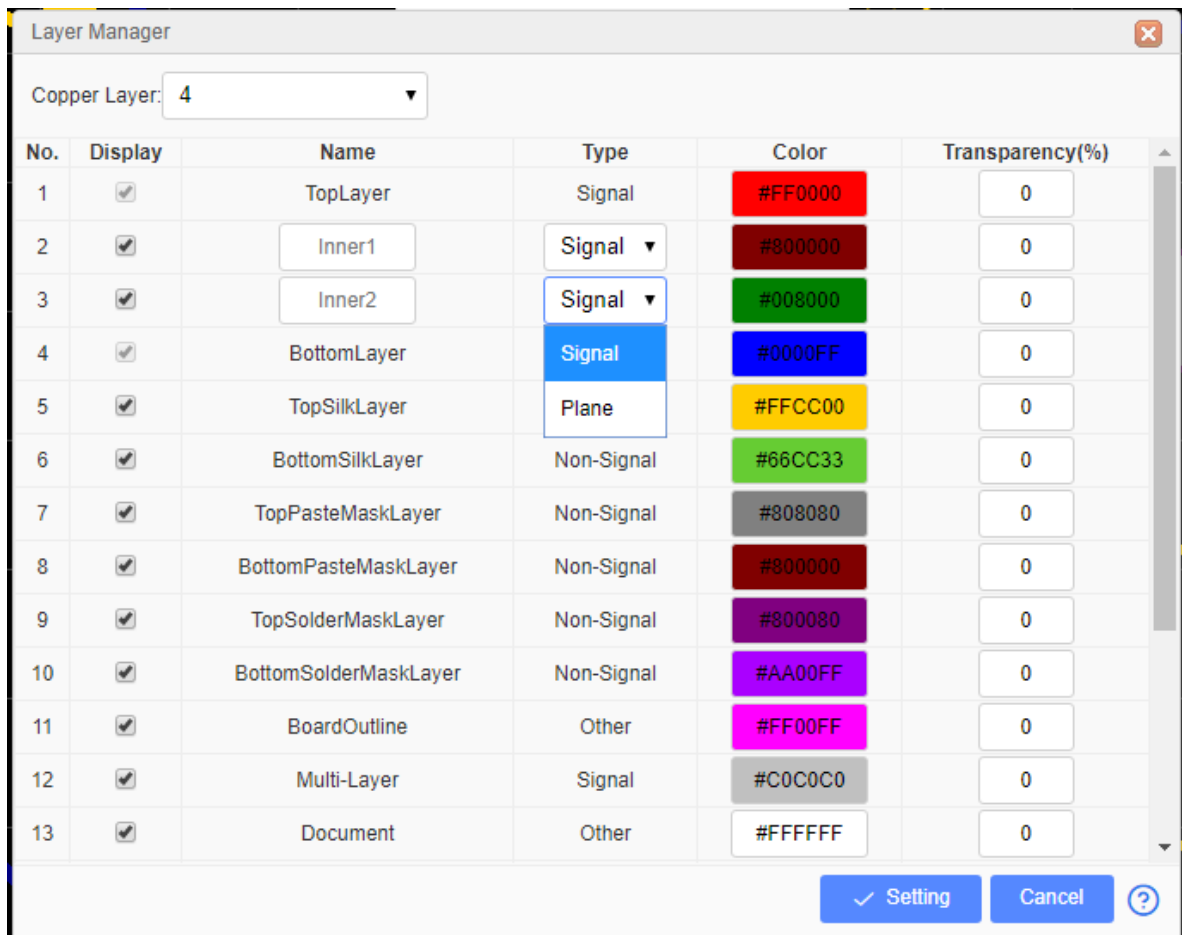
Layer Manager

Layer Manager

You can set the PCB layer's parameters at the Layer Manager.

Via **Top Menu > Tools > Layer Manager...**, Or Click **Layers Tool** gear icon. Or right-click the canvas - Layer Manager menu.

The Layer Manager dialog:



The Layer Manager setting only works for the current editing PCB.

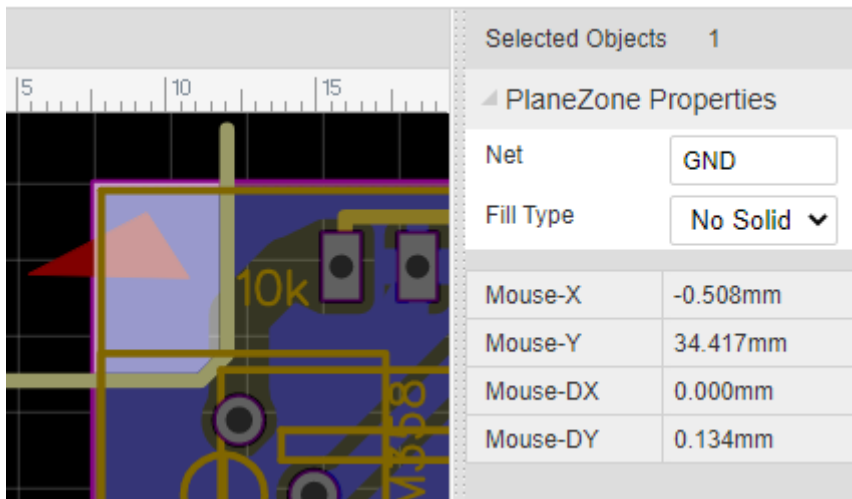
Copper Layer: The copper layer of your PCB. EasyEDA support 34 copper layers. The more copper layers the PCB will be more expensive. The TopLayer and BottomLayer is default layer, can not be disable. If you want change the copper layers from 4 to 2, you must delete the inner layers objects first.

Display: If you don't want a layer doesn't display at "Layers Tool", you can disable the checkbox. Notice: This option only hide the layer name on the "Layers Tool", the objects of the hidden layer still exist, when you generating the Gerber, they will appear.

Name: Layers name. For the inner layer, you can define the name.

Type:

- **Signal:** Which is working for the signal. Such as Top and bottom layer.
- **Plane:** When the inner layer type is "Plane", this layer will be copper poured, if you want to separate the copper area you can draw the Track or Arc. You can treat this layer is a only has the copper area, but its easy than draw the copper area. The track you routed will generate the clearance when generating the Gerber. The "Plane" usually is using for the Power or Ground copper pour on the inner layer. You can set the net for the plane zone.



Notice:

When draw the track to separate the plane zone, the track start point and end point must over the middle line of the board outline track. Otherwise, the plane zone will not be separated; When using the plane layer, the PCB can not exist two closed board outline, only one closed board outline will generate the plane zone.

- **Non-Signal**: Such as silk screen, mechanical layer, document layer etc.

Color: You can define the color for each layer.

Transparency: You can change the layer transparency.

Layer Defination:

- **TopLayer/BottomLayer**: The top side and bottom side of the PCB board, copper layer.
- **InnerLayer**: Copper layer, routing track and copper pour.
- **TopSilkLayer/BottomSilkLayer**: Board silkscreen.
- **TopPasteMaskLayer/BottomPasteMaskLayer**: This layer is the layer used to make the stencil for the SMT pads, helping to solder. This layer has no effect on production if the board is not required to make the stencil.
- **TopSolderMaskLayer/BottomSolderMaskLayer**: The top and bottom cover layers of the board are typically green oil, which acts to prevent unwanted welding. This layer belongs to the negative film drawing mode. When you have wires or areas that do not need to cover green oil, draw them at the corresponding positions. After the PCB is generated, these areas will not be covered with green oil, which is convenient for operations such as tinning.
- **BoardOutline**: The board shape definition layer. To define the actual size of the board, the board factory will produce the board according to this shape.
- **TopAssemblyLayer/BottomAssemblyLayer**: Simplified outline of components for product assembly and repair. Used to export document printing, without affecting PCB production.
- **MechanicalLayer**: Record the information on the mechanical layer in the PCB design, and only use it for information recording. By default, the shape of the layer is not manufactured at the time of production. Some board manufacturers use the mechanical layer to make the frame when using Altium file to production. When using Gerber file, it is only used for text identification in JLCPCB. For example: process parameters; V cut path etc. In EasyEDA, this layer does not affect the shape of the border of the board. If the mechanical layer has closed wires, JLCPCB will give priority to using the mechanical layer as the shape of the board when producing the board. If there is no outer frame of the mechanical layer, GKO will be used as the frame (historical influence of Altium file). It is necessary to pay attention to the use of the mechanical layer in the design.

- **DocumentLayer:** Similar to the mechanical layer. But this layer is only visible in the editor and will not be generated in the Gerber file.
- **RatlineLayer:** PCB network ratline display, this layer is not in the physical sense, in order to facilitate the use and set color, it is placed in the layer manager for configuration.
- **HoleLayer:** Similar to the RatlineLayer. For Hole(Non-Plated Hole) display.
- **Multi-Layer:** Similar to the RatlineLayer. For multi-layer hole(Plated hole) display. If the PAD setting layer property as multi-layer, it will connect with all copper layers.
- **DRCErrorLayer** Similar to the RatlineLayer. For DRC(Design Rule Error) marking display.

Layout Single Layer PCB

The PCB copper layers of EasyEDA are double, EasyEDA doesn't support layout a single layer directly. if you want to layout a single layer PCB(such as only layout on the bottom layer),

There are two methods:

Method 1:

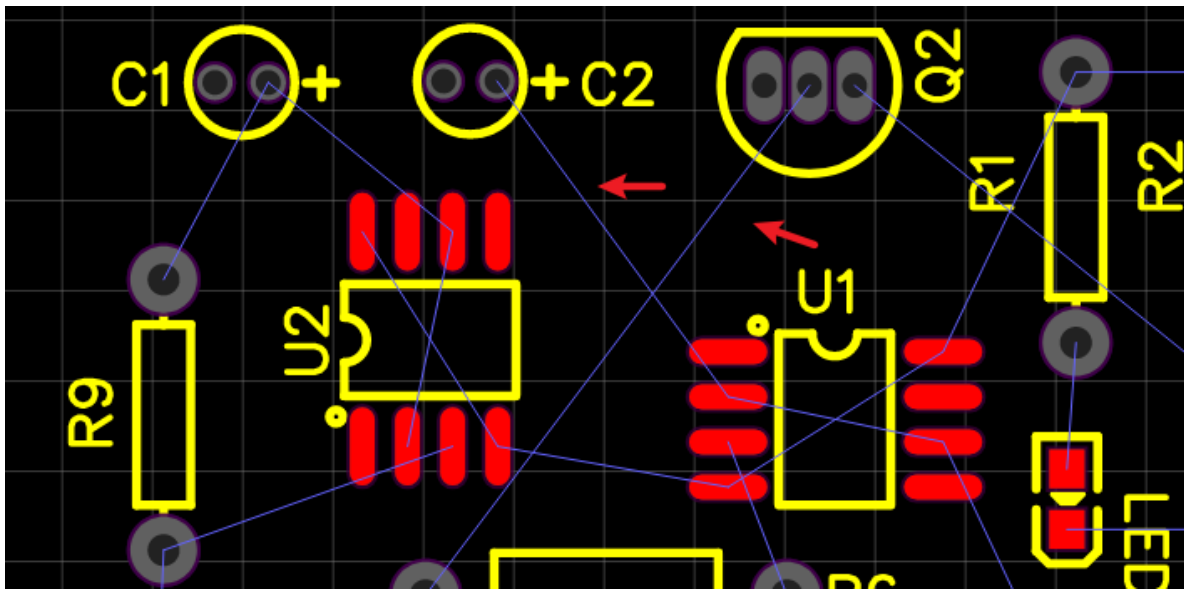
- Route the track and copper on the bottom layer, and without placing via.
- If you are using the footprints which have the multi-layer pads, that will appear on the top and bottom layer, then you need to change all multi-layer pads "Plated" as "No".
- Generate the Gerber, decompress the Gerber zip file, delete the layers which you don't need(such as Gerber_TopLayer.GTL, Gerber_TopSilkLayer.GTO, Gerber_TopSolderMaskLayer.GTS, Gerber_TopPasteMaskLayer.GTP).
- And re-compress the Gerber to a zip file, and order it.

Method 2:

- Design your PCB at one side, if other side has pads etc, you don't need to deal with them.
- Generate the Gerber.
- Add the comment for mention that you need to order the single layer PCB when order the PCB.

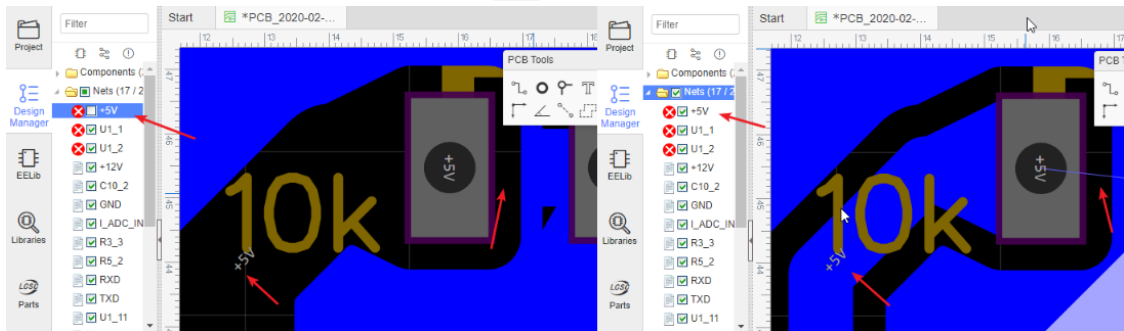
Ratline

When you layout the track in the PCB, Between Pad and Pad as they have the same net name, a Ratline will be automatically shown among them to reveal that they can be connected with a track.



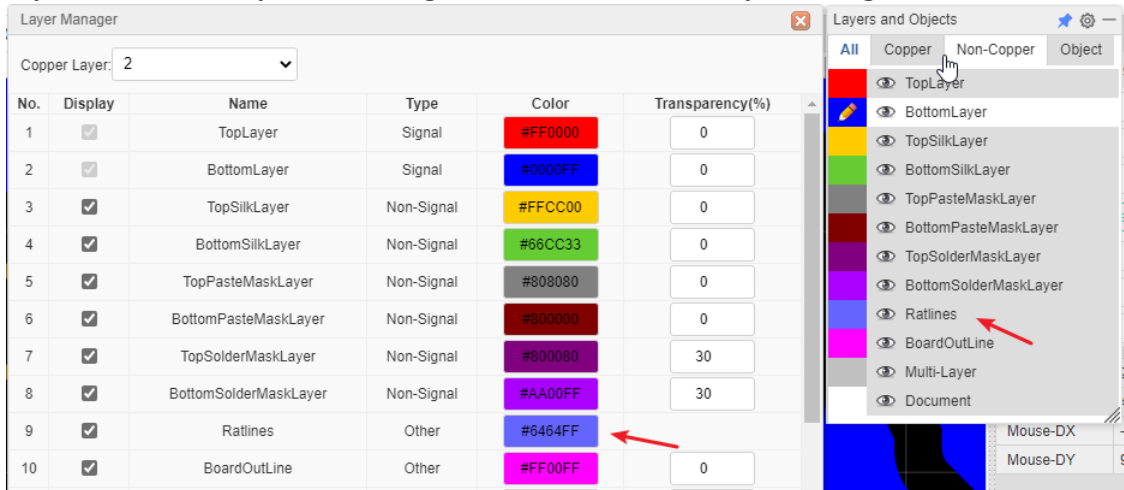
1. If you want one ratline do not show on the PCB editor, you can deselect the net in the design manager, as below deselect +5V:

If you still draw a track in +5V after deselecting, canvas will not display this track and ratline, but it will show a net text with +5V as below.



Based on this skill, you don't need to lay GND net before copper area in the PCB.

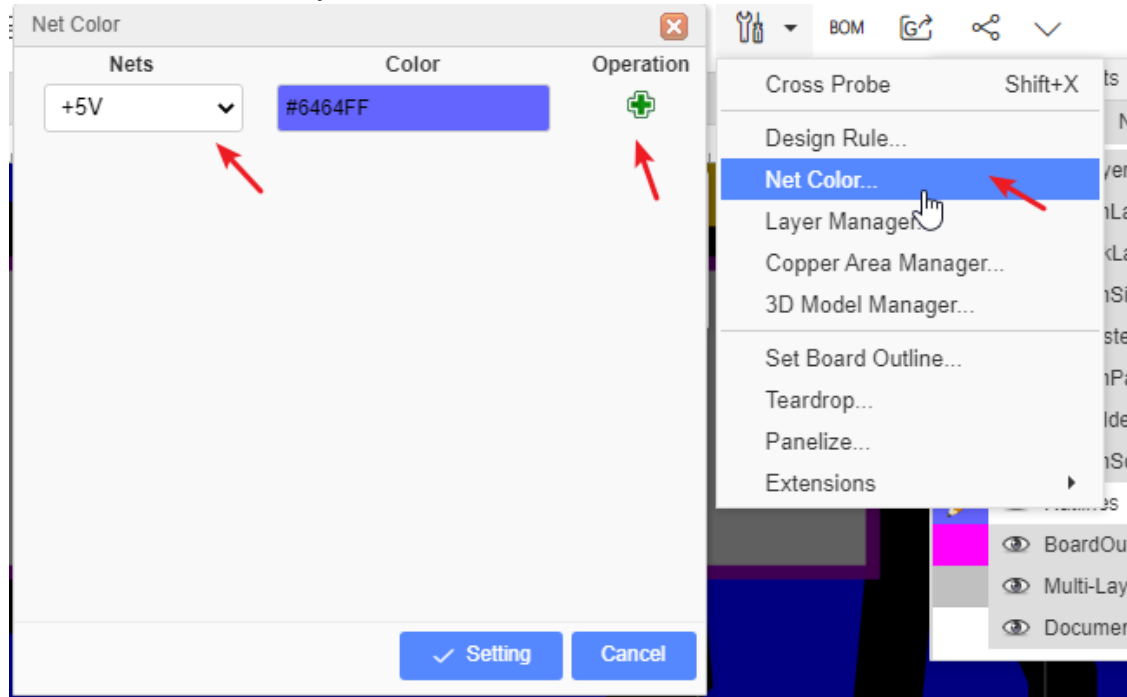
2. If you want to check the ratlines with highlight, you can click the pencil on the Ratlines Layer as below, and you can change the ratline's color at Layer Manager.



3. If you want to highlight one ratline all the time, you can click a pad, press hotkey H, press it again unhighlight.

4. If you want to change one ratline's color, you can set it at: - Tools - Net Color. After setting the color, you need to click the plus icon on the right. The color is not affected by

the color of the ratline layer.



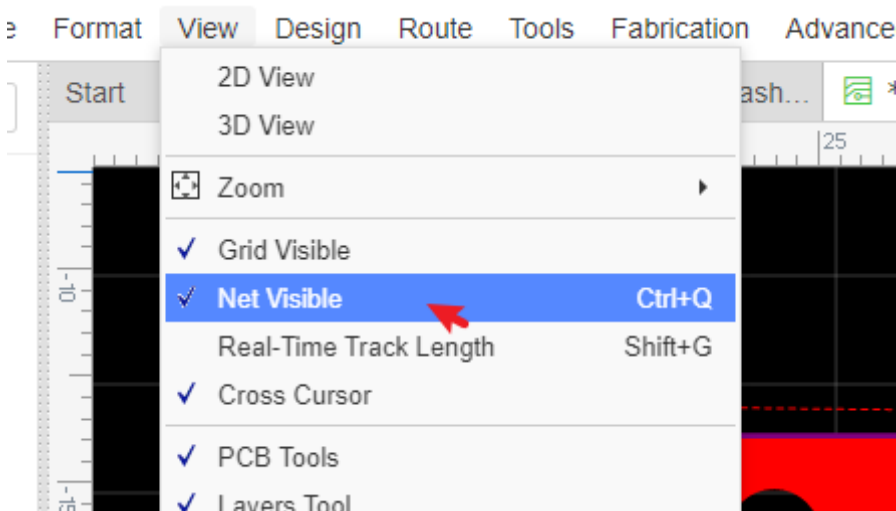
5. If you want to remove one ratline, you just need to remove objects' net. Select it and empty the net.

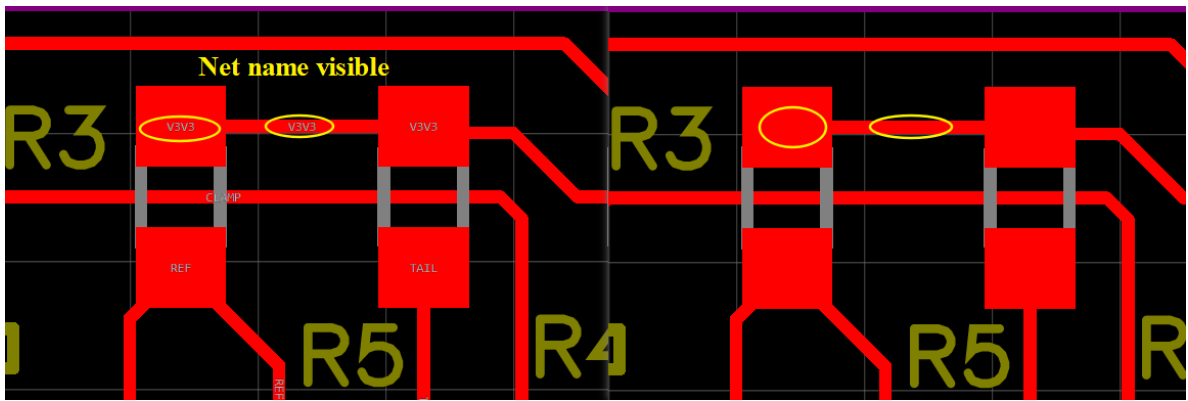
PCB Net

Net Name Visible

PCB editor can display net name in the track or Pads, if you don't need this feature, just need to turn it off via:

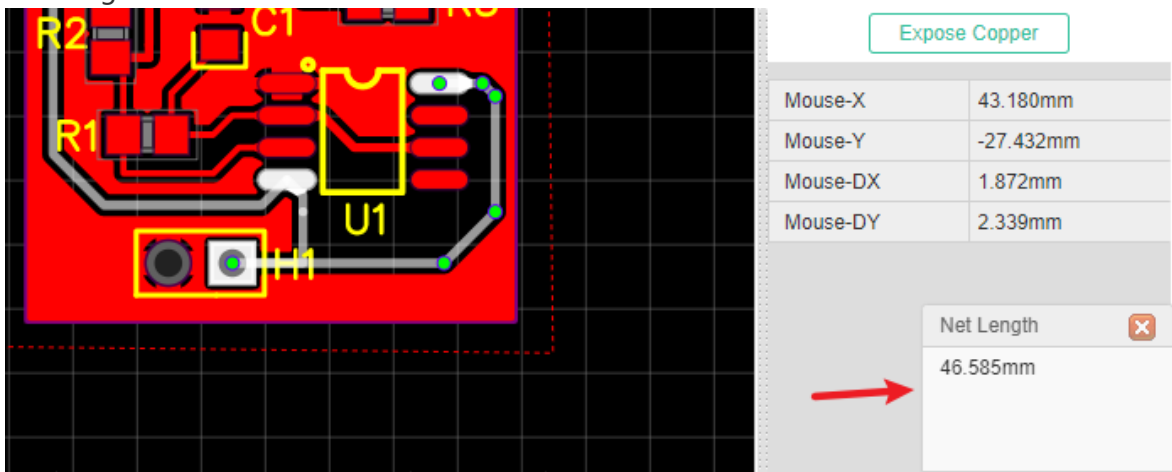
Top Menu > View > PCB Net Visible, or press hotkey **CTR+Q** .





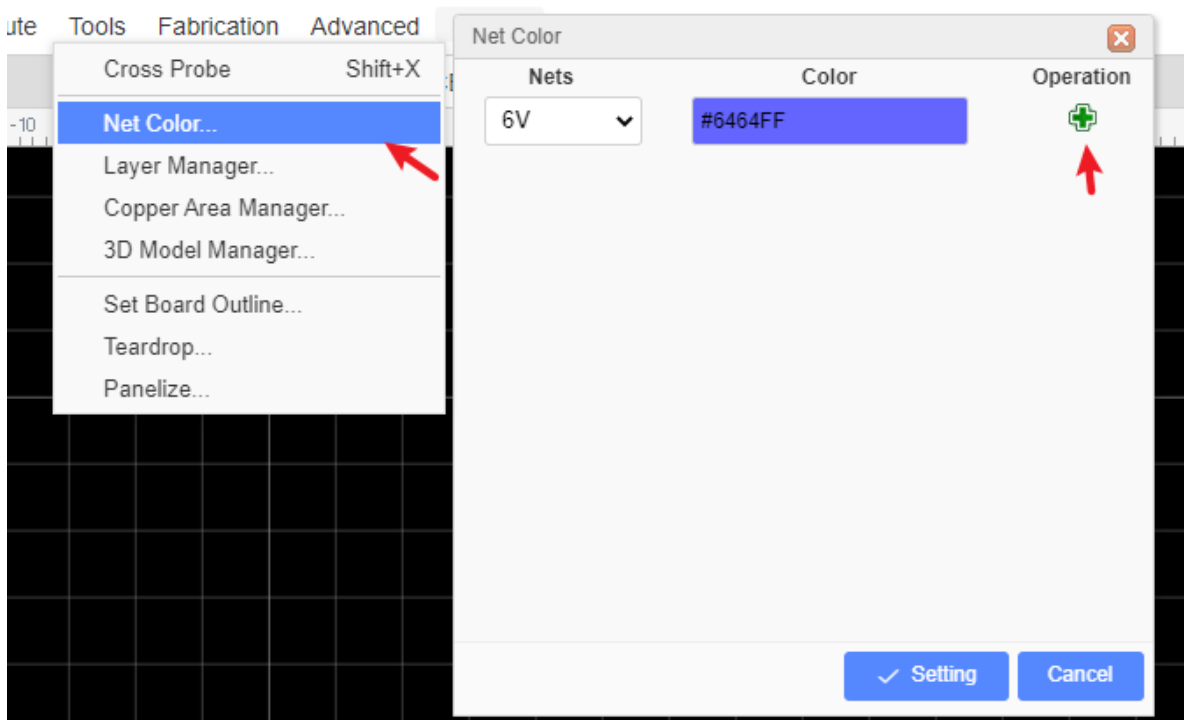
Net Length

After selecting a track, and then pressing **H** key or click its net at Design Manager, EasyEDA will highlight the whole net and pop a message box to tell you the whole net's length. like in the image below



Net Color

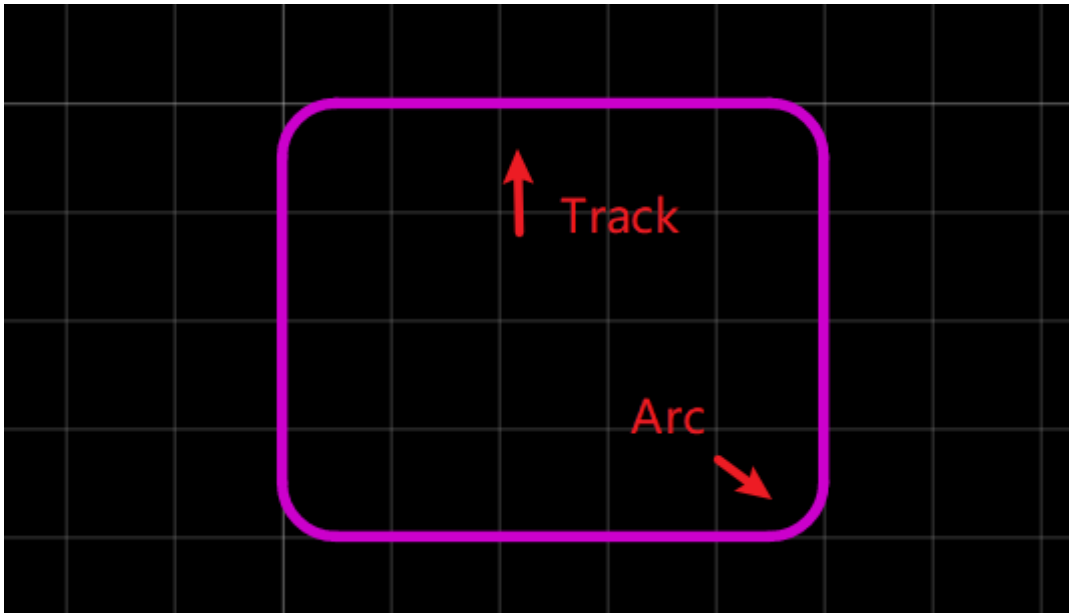
If you want to change one Ratline's or Net's color, you can set it at: **Top Menu- Tools - Net Color**. After setting the color, you need to click the plus icon on the right. The color is not affected by the color of the ratline layer.



When you set a color for a net, you need to click the + button to make it work.

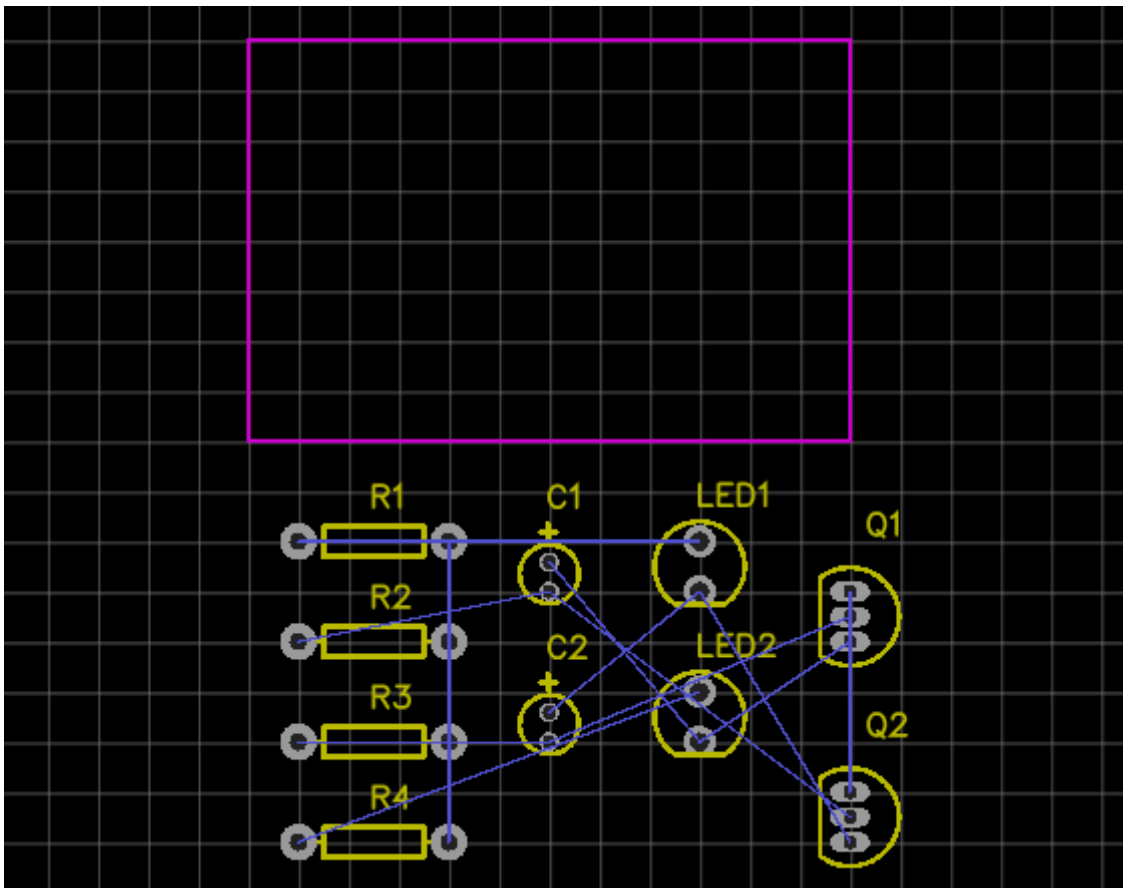
Board Outline

Before placing footprints we need to create a board outline. The board outline must be drawn on the **Board Outline** layer. So first, set **Board Outline** as the active layer, then draw the board outline using **Track** and **Arc** from the PCB Tools palette.



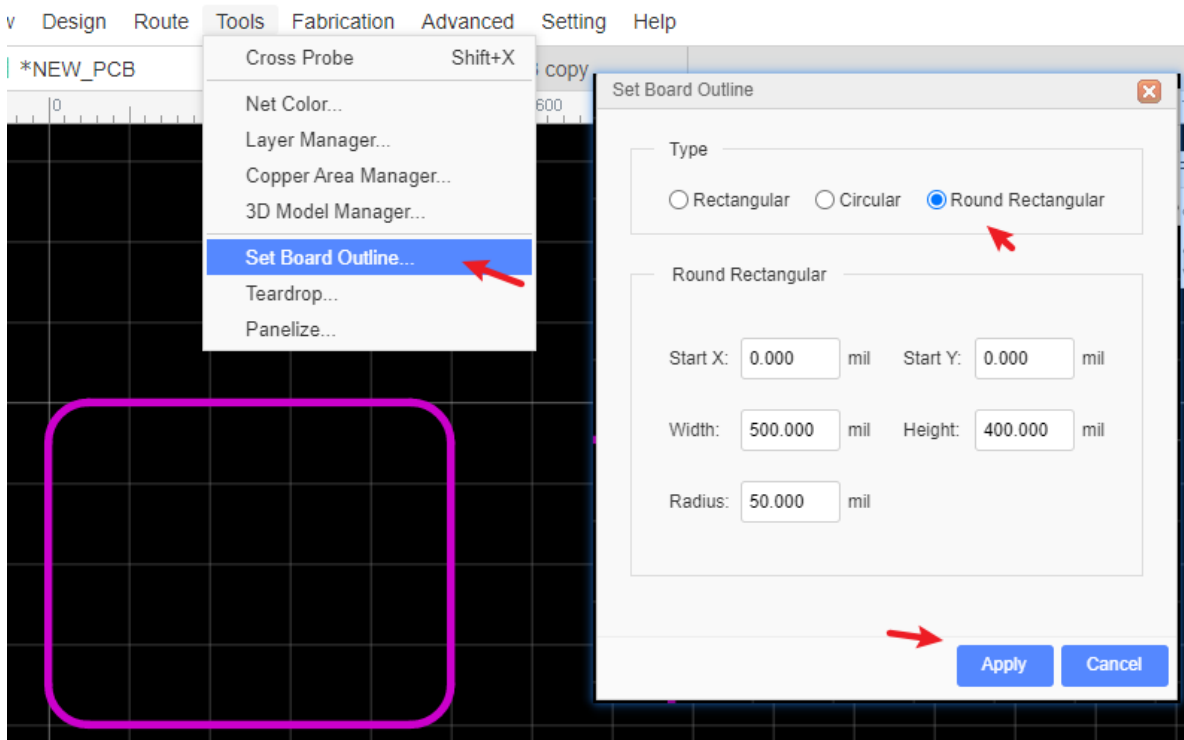
When converting a Schematic to PCB, EasyEDA will try to create a board outline for you.

The area of the default board outline area is 1.5 times the sum of the area of all of your footprints, so you can place all of your footprints into this board outline with some allowance for tracking. If you do not like the board outline, you can remove the elements it is made up from and draw your own.



To create a simple rectangular board outline, this arc can be removed and the line X and Y end points edited - either directly in the Properties panel or by dragging the line ends - to close the rectangle.

And EasyEDA provides a **Board outline wizard**, so it is very easy to create a board outline. Via: **Top Menu > Tools > Set Board Outline**, Or find it on the toolbar.



In this dialog, there's a choice of 3 types of board outlines, Rectangular , Circular, Round Rect. If you need a different more complex board outline, you need to import a DXF file.

Notice:

- When generate the Gerber, EasyEDA will show error if the board outline doesn't closed or the board outline tracks overlap.
- You can cutout the hole by using the board outline, or use `Hole`, or `Solid Region(Type: Board Cutout)` to create the hole instead of using the board outline.
- You can right-click track or circle to convert to board cutout.
- If the board outline doesn't closed, the copper pour will not show up.

Route Tracks

Route Tracks

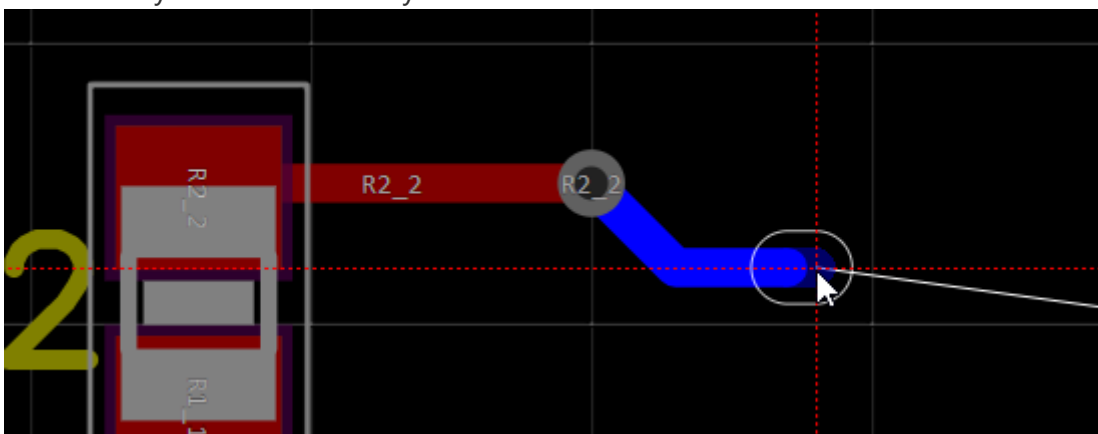
In the schematic editor, we use Wire or the `W` Hotkey to connect Pins, in a similar way in the PCB editor, we use Track to connect Pads. Track allows you to draw PCB tracks and can be found on the PCB Tools palette or using the `W` Hotkey (not T).

Some Tips about Track

- Single click to start drawing a track. Single click again to pin the track to the canvas and continue on from that point. Right click to end a track. Double right-click to exit track mode.

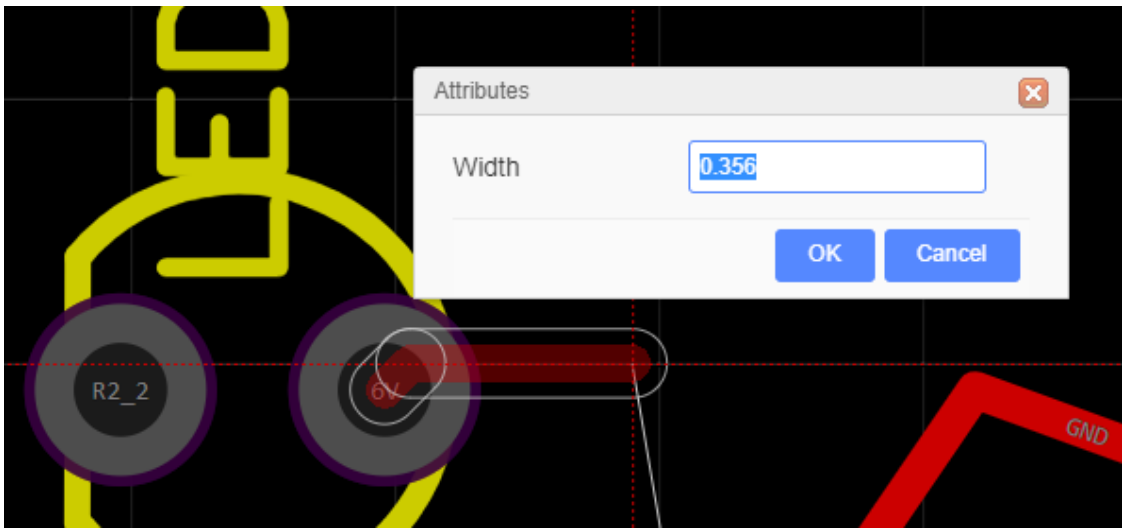


- Drawing a track at the same time as using a hotkey(for example hotkey `B`) for changing the active layer will automatically insert a Via:



If you start drawing a track on the top layer, you will see it drawn in red, then press the B key to change to bottom layer and you will see EasyEDA insert a grey via and then the track will continue being drawn but now on the bottom layer in blue.

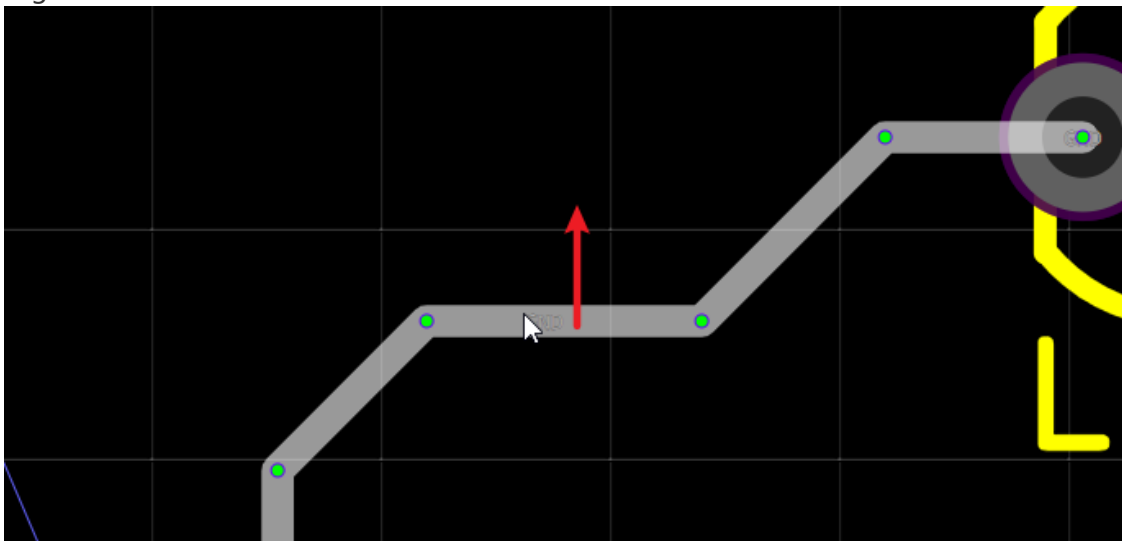
- Pressing the `+` or `-` Hotkeys when drawing the track will change the width of the track on the fly. Use the hotkey `TAB` to change the track width.



- Double clicking on a drawn section of the track will add a new vertex at that point. You can drag the vertex to form a new corner. And you can right-click the point and delete it.

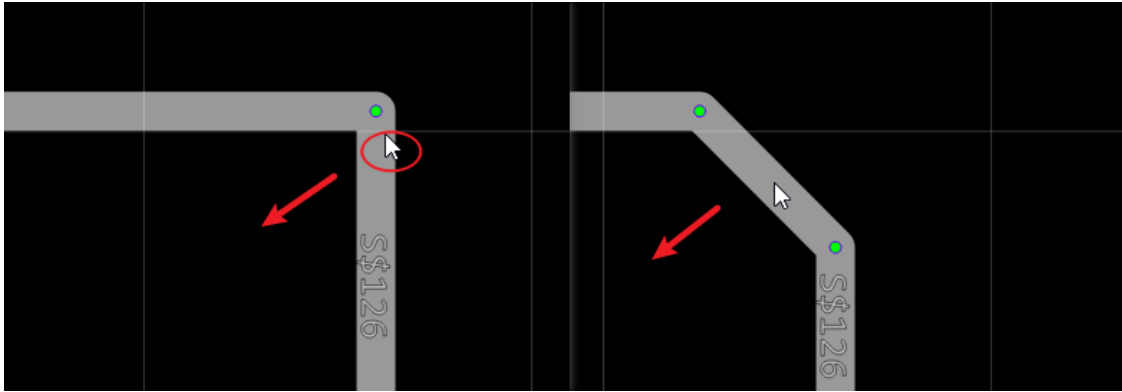


- Click to select the track and then Click and Drag on a segment of the track to adjust the segment between vertices.

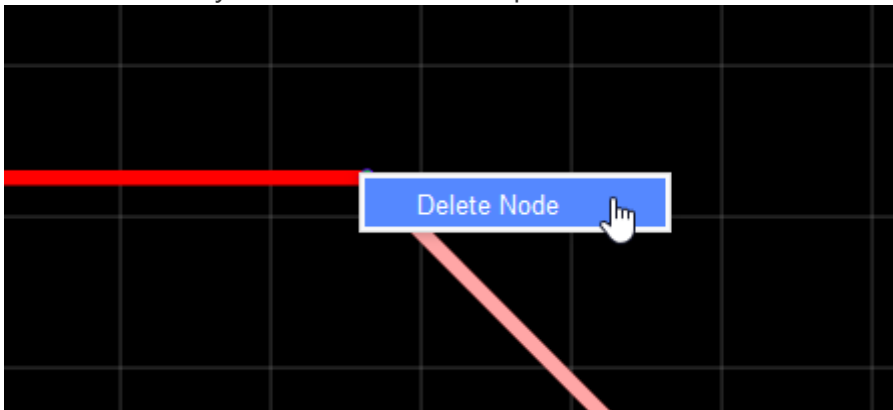


- When the track's corner is a right Angle and the routing corner is 45, drag the track next to the right Angle node to make a bevel. Instead of dragging a node directly, dragging a

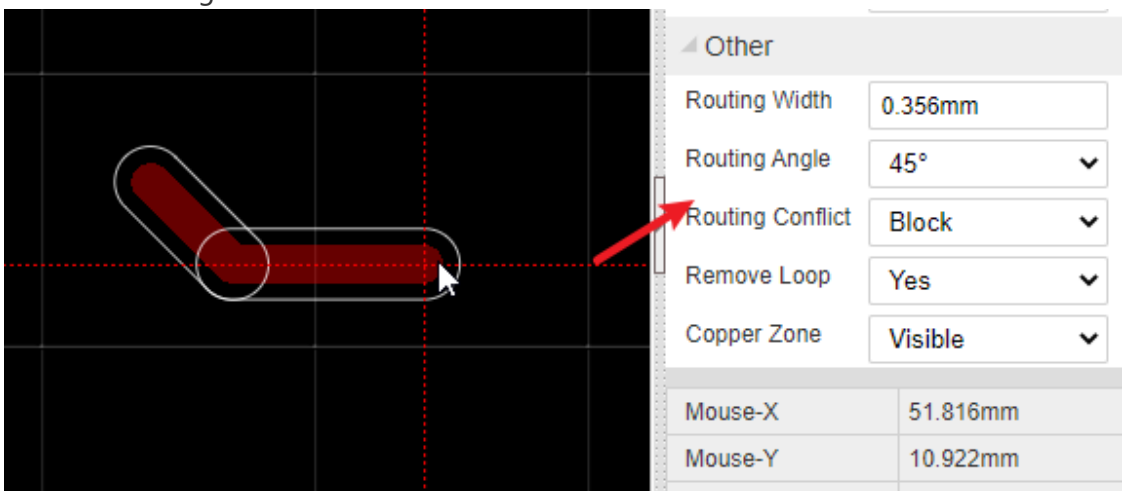
node directly will drag the entire track.



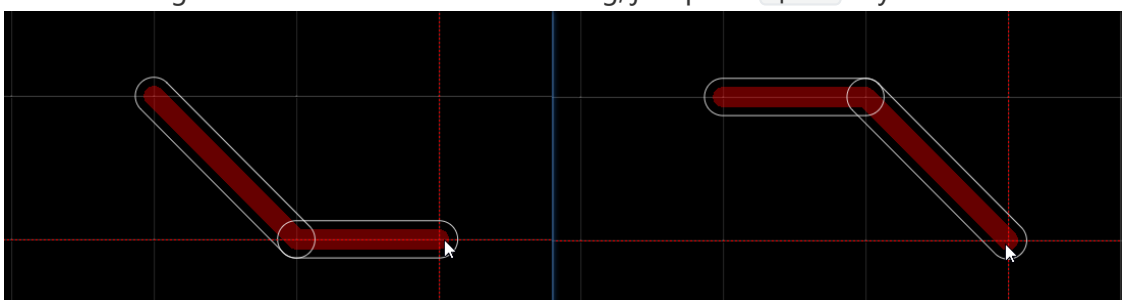
- Selected a track, you can delete its node point.



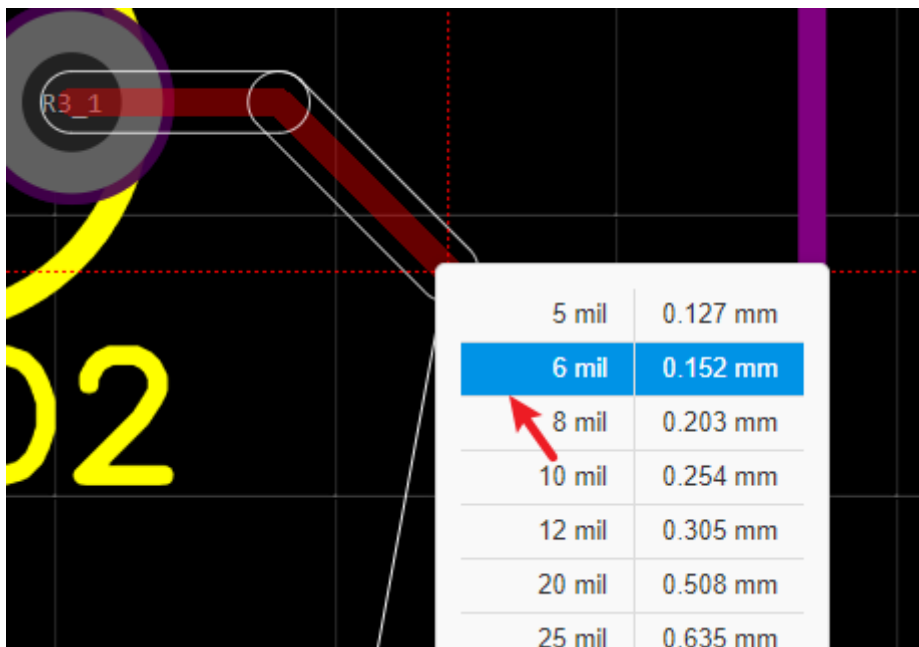
- Pressing the **L** Hotkey when drawing the track will change the track's Route Angle on the fly. And you can change Route Angle on the Canvas Attributes of the right panel before the next drawing.



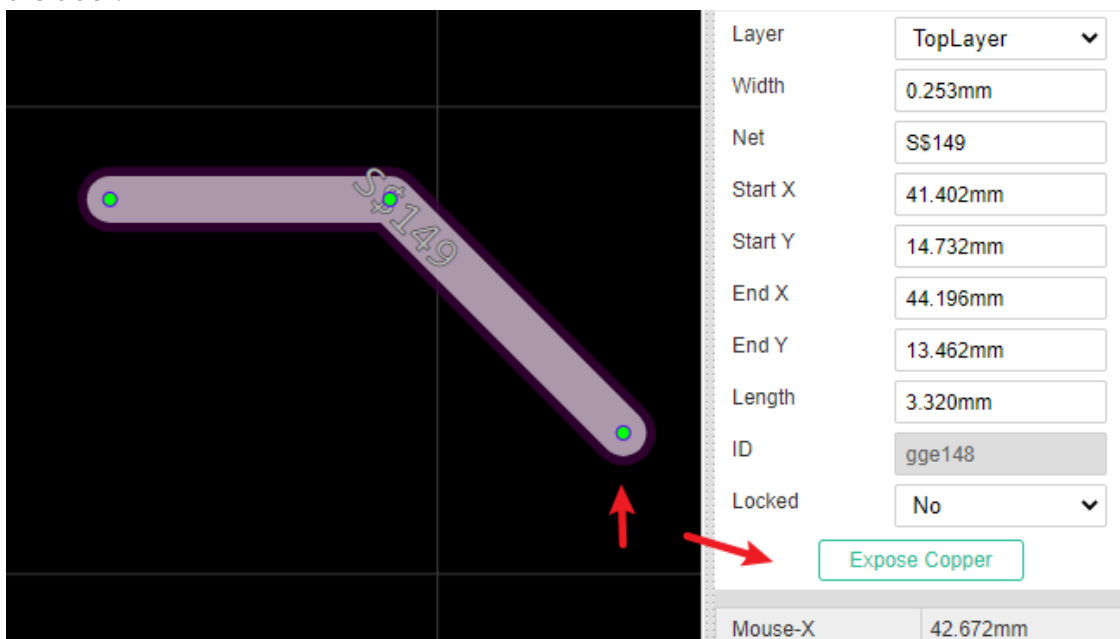
- You can change inflection direction when routing, just press **Space** key.



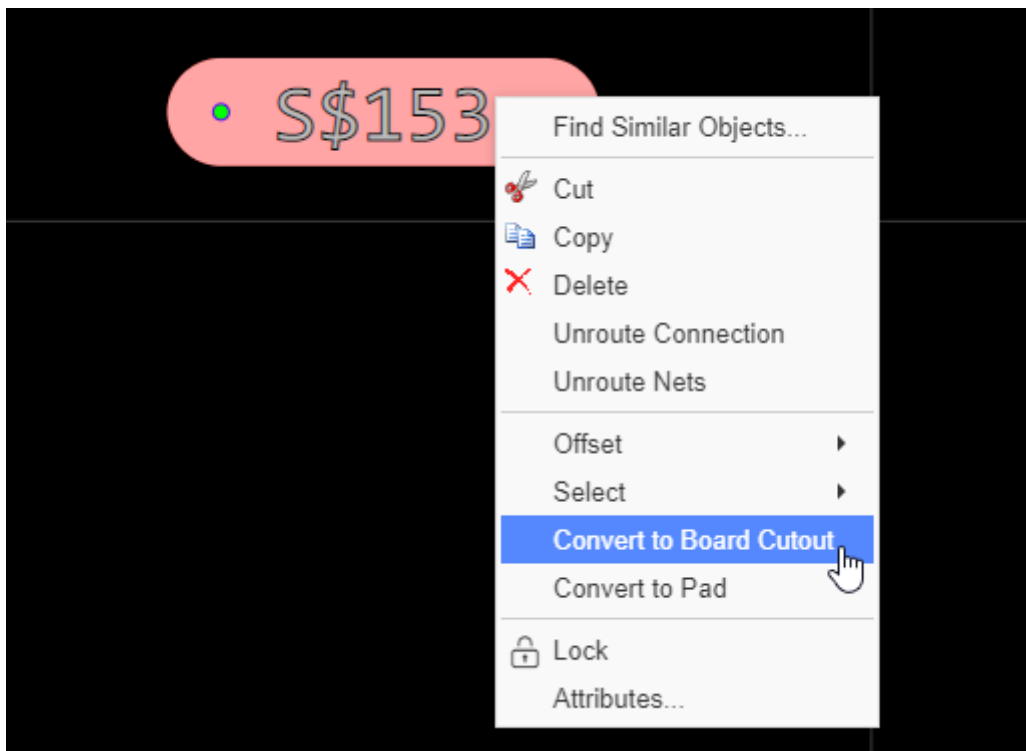
- If you want to route a track and use "L", and the then press "+", you will get two different size track segment. or press "SHIFT+W".



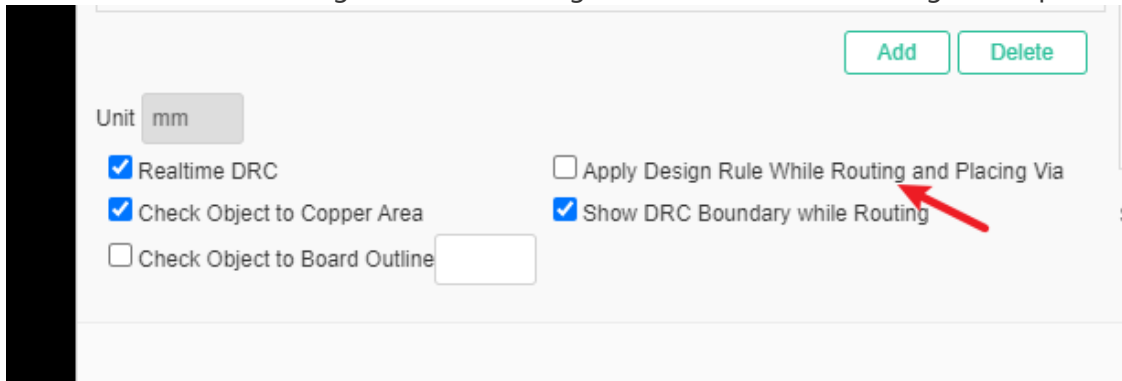
- If you want to create the solder mask aperture for the track, you can use "Expose Copper" when you select the track on the right-hand panel. The solder mask will be bigger than the track.



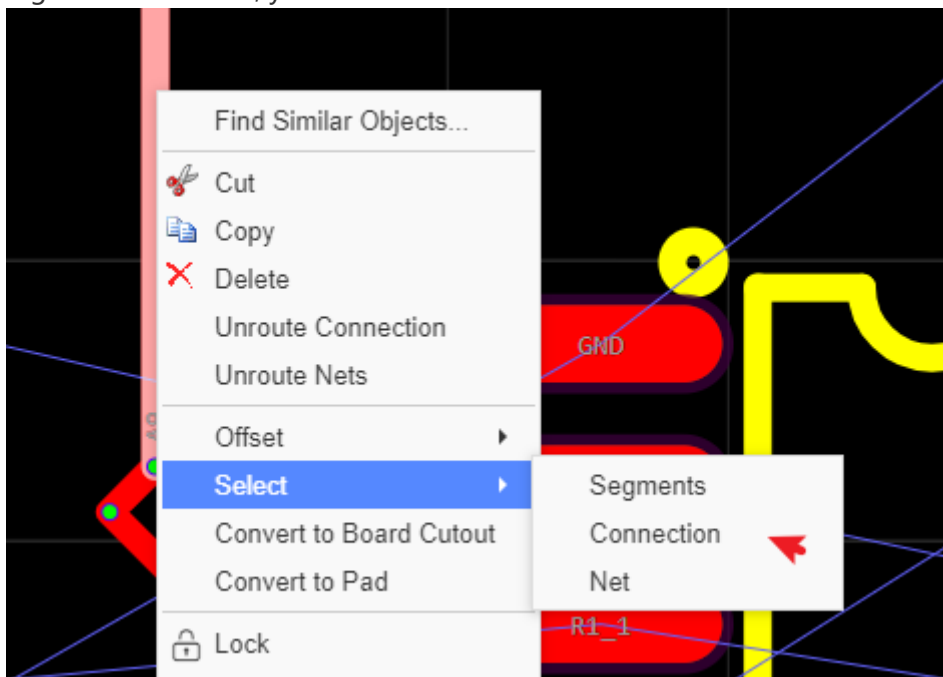
- And if you want to create the slot hole, you can route a track, and then right-click the "Convert to Board Cutout" menu.



- You can make track routing width follow design rule, after enable the design rule option.

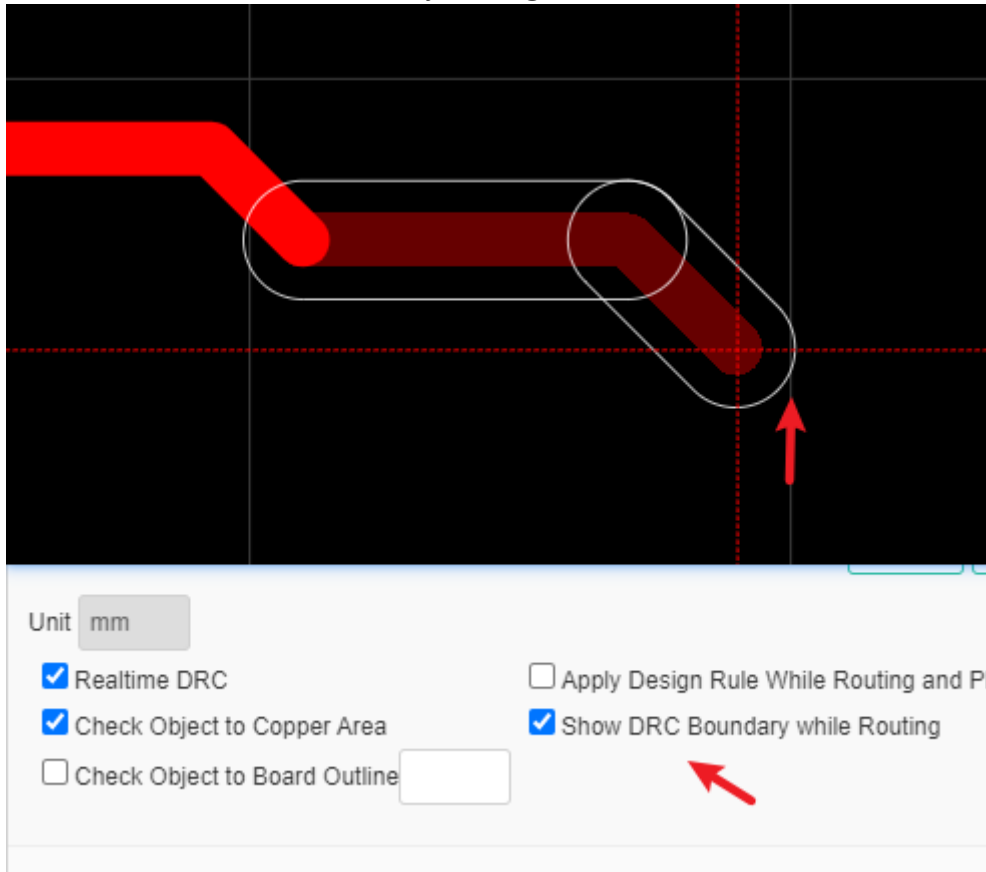


- Right-click the track, you can select the track connection or a whole same net tracks.

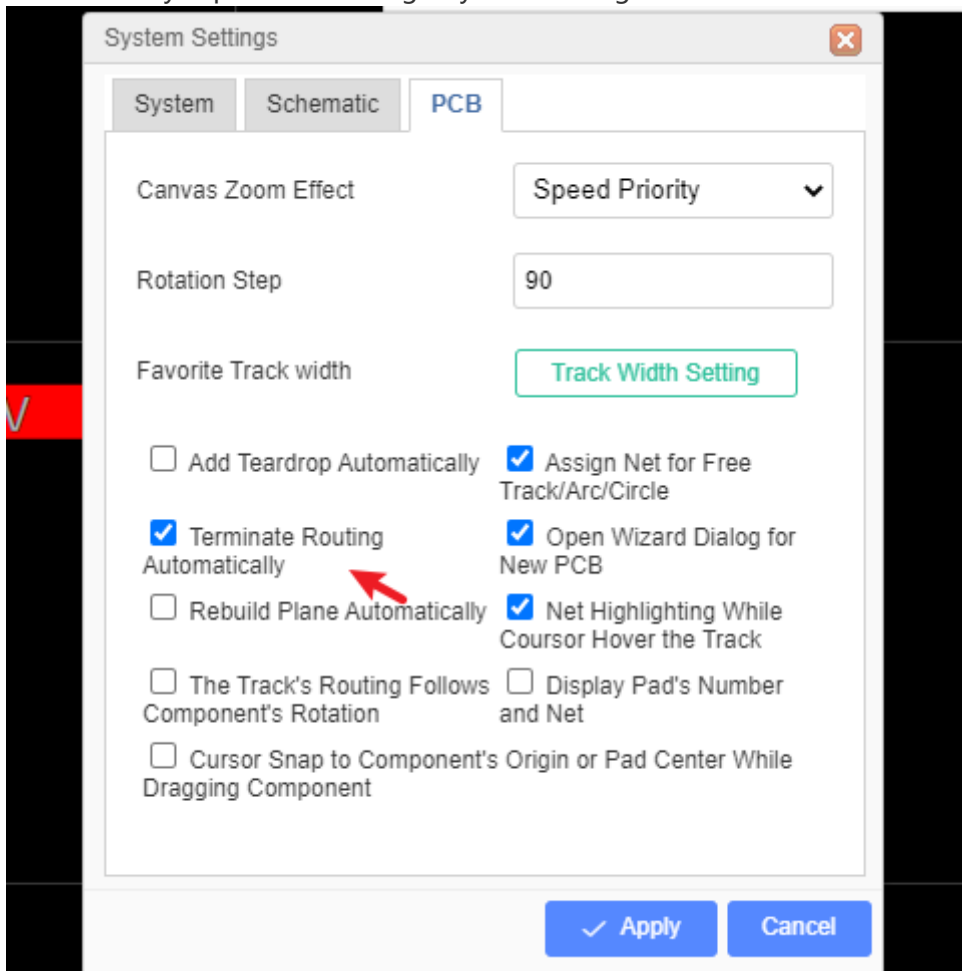


- If you want to the whole track, you can press **SHIFT** and move it.

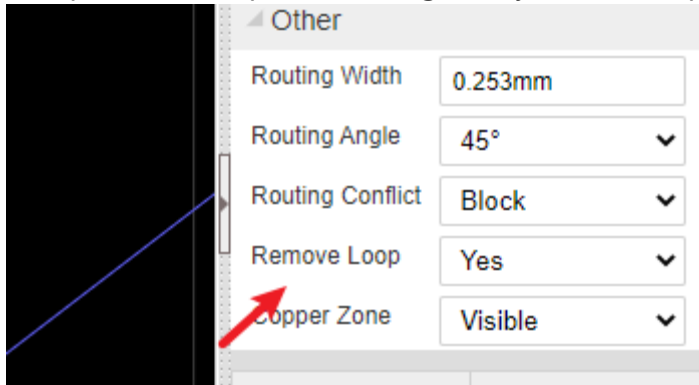
- You can disable the DRC boundary at Design Rule. The size follow the rule.



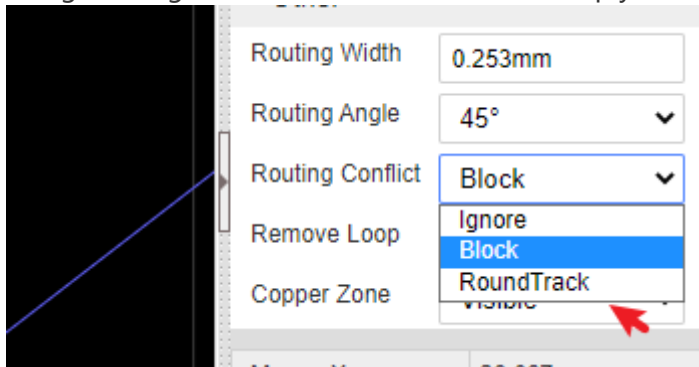
- If you want to continue routing for a net, you can disable the "Terminate Routing Automatically" option at "Setting - System Setting - PCB".



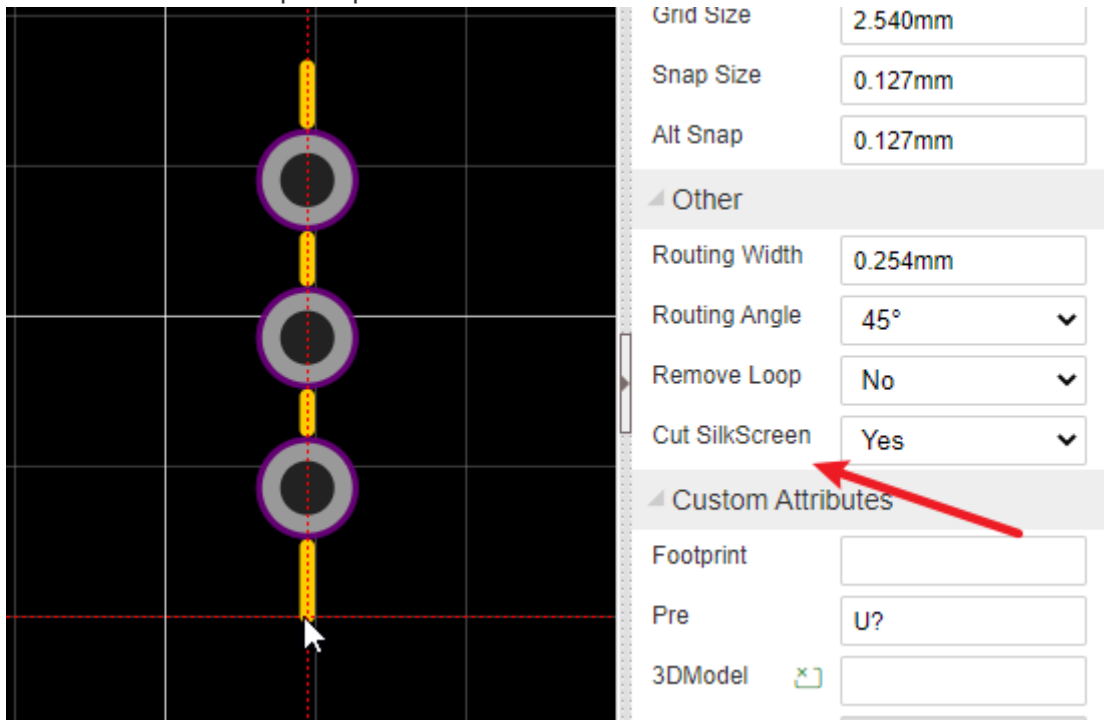
- Set up Remove Loop while routing, it only works on copper layer.



- Using Routing Conflict as "RoundTrack" will help you finish routing quickly.

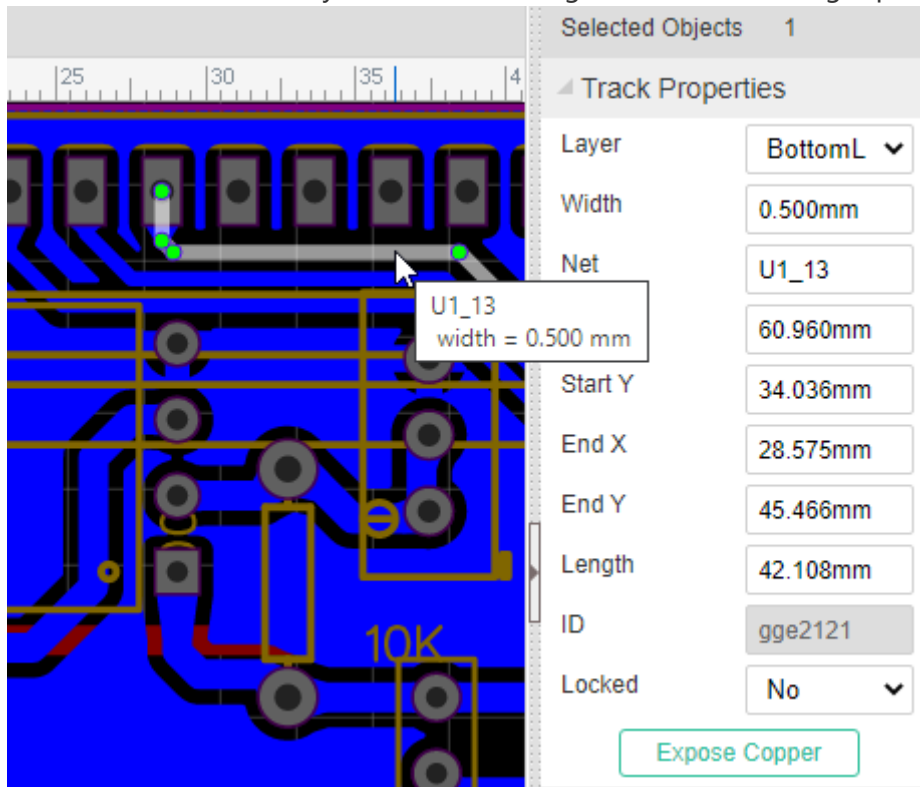


- When edit the footprint document, you can set up the "Cut Silkscreen" to avoid the silkscreen track overlap the pad.



Track Length

- When a track is selected, you can find its Length attribute in the right panel.



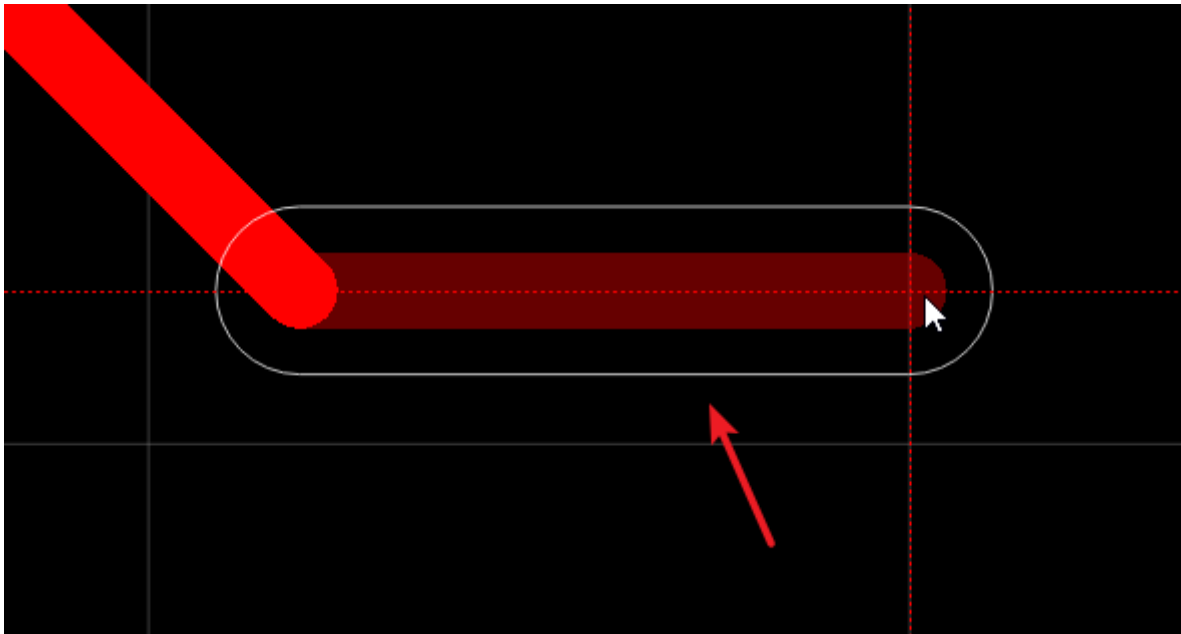
- At left-hand Design Manager, click a net, will pop up a dialog to show you this net track length.
- Click a track, press hotkey H will keep highlight this track and net, and show this net's length.

Delete a Segment from a Track

- While routing, if you want to undo previous track path, you can press key "Delete" or "Backspace".
- Move your mouse to the segment which you want to delete, click it, then hold **SHIFT** and **double click it**. the segment will be removed. Or right-click delete the node.
- Right-click the track node to delete the track
- Click the track, right-click delete it, or press "Delete" key directly.

DRC outline

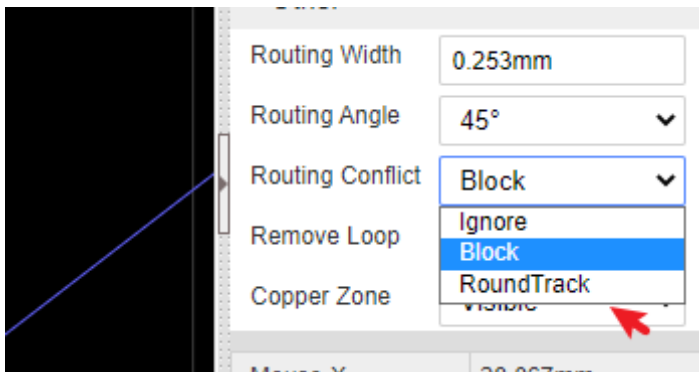
When you routing a track on the signal layer, you will see an outline around the first track, it is the DRC outline, the clearance from outline to the track edge depends on your Design Rule(DRC) clearance setting.



Routing Conflict

When the PCB comes from the schematic converted, the "Routing Conflict - Block" will be opened automatically.

At the right-hand attributes panel - others, you can find a "Routing Conflict" option:



- Ignore: You can route the track overlap the different net name objects.
- Block: If the track net name different with other objects, this track will be blocked when routing.
- RoundTrack: The track while routing will walk around the different net objects.
- Push: Doesn't develop yet.

Differential Pair Routing

EasyEDA provide a easy experience for the differential pair routing.

Via: Top Menu - Route - Differential Pair Routing

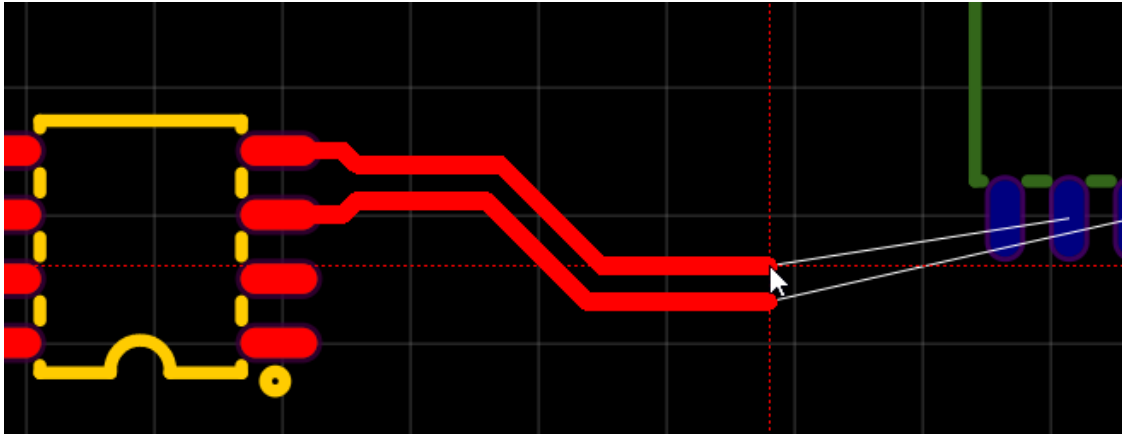
You must make sure the Differential Pair net names must be `XXX_N`, `XXX_P` or `XXX+,XXX-`.

and you need to set Differential Pair net rule at the "Top Menu - Tool - Design Rule" first.

How to route Differential Pair:

1. Set the Differential Pair net name as `XXX_N`, `XXX_P` or `XXX+,XXX-`, and set the rule for the Differential Pair net at the "Design Rule"
2. Click the menu `Top Menu - Route - Differential Pair Routing`
3. Click the one pad of the Differential Pair pads

- 4.Routing

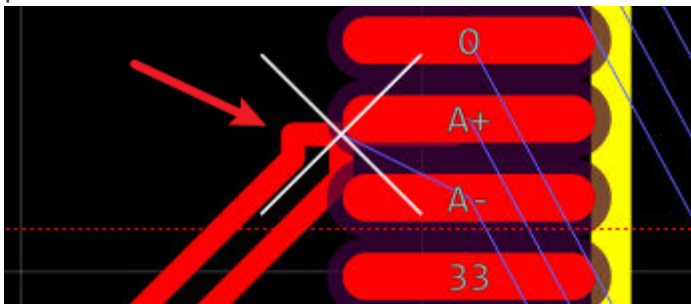


Notice:

- Only for 45 degrees routing, doesn't support hotkey L and Space key.
- Doesn't support the fanout routing.
- Doesn't support the DRC blocking.

Known Issue:

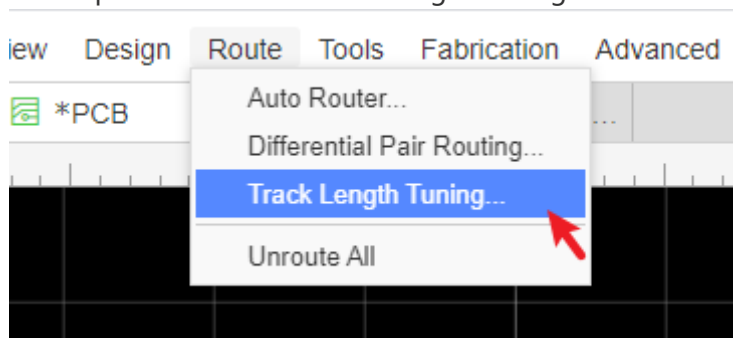
- When finish previous routing location too close with the finish pads, the track will generate the extra segments, please finish the previous location far away from finish pads.

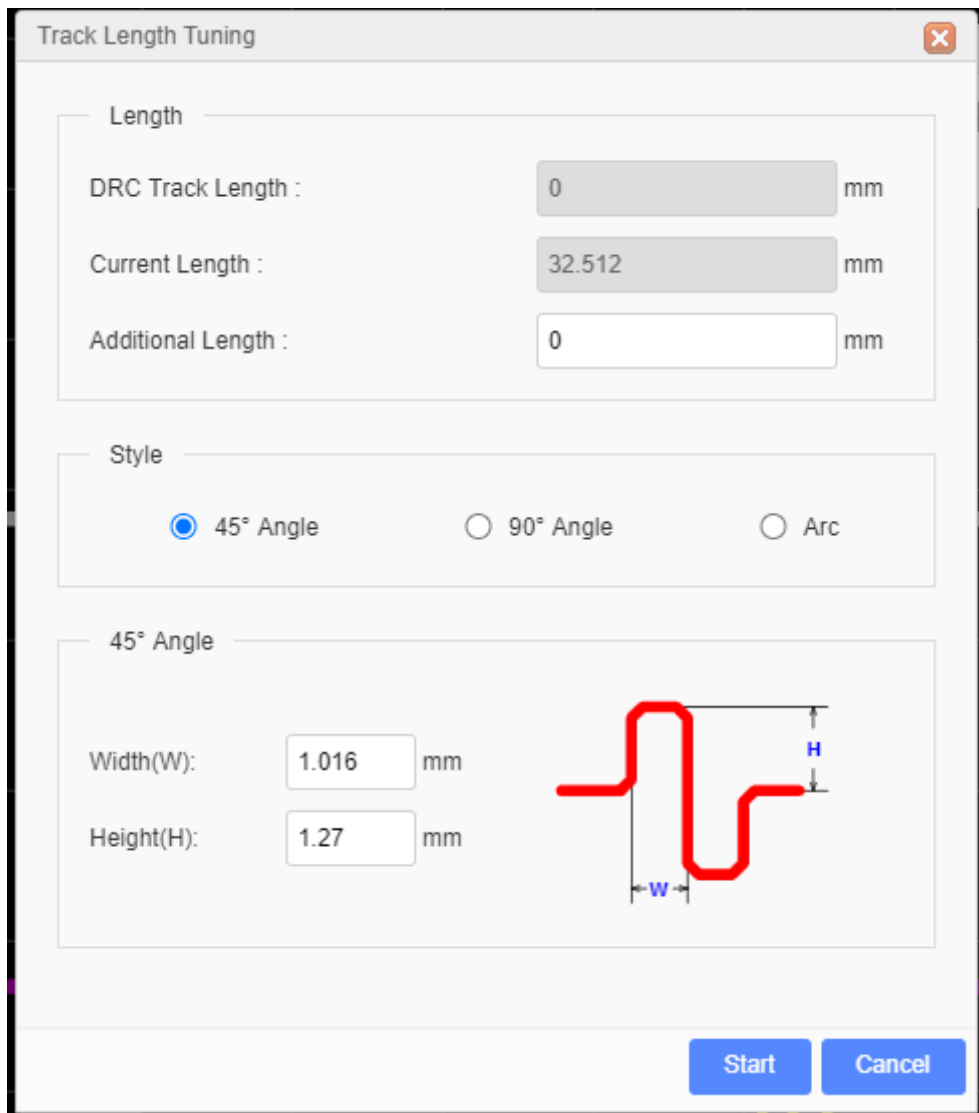


Track length Tuning

You can tuning your track very easy on the editor.

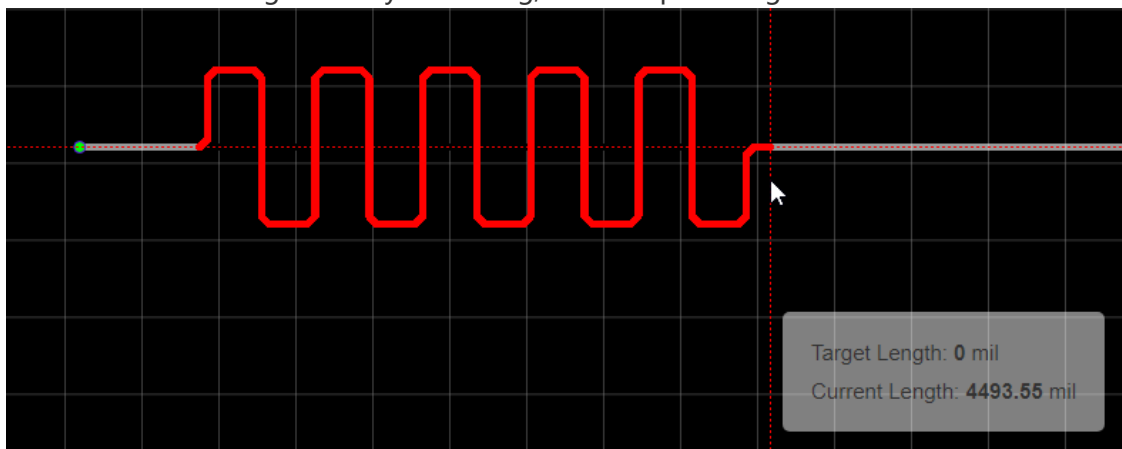
Via: Top Menu - Route - Track length Tuning





How to use:

- 1. Select the track which is you want to tune
- 2. Click the menu: `Top Menu - Route - Track Length Tuning`
- 3. Set the parameter, start
- 4. Left-Click the track where is you want to start, and then move the mouse
- 5. When the track length close your setting, it will stop tuning.



Notice:

- Doesn't support one side tuning for a track yet
- Doesn't support auto push or avoid the nearby tracks yet

Cloud Auto Router

For some simple or prototype PCBs, you may want to use the auto router function to save time. Layout is a time costly and dull job. EasyEDA spends lots of time to provide such a feature and it is loved by our users.

Before using the auto router, you need to set the board outline for the PCB.

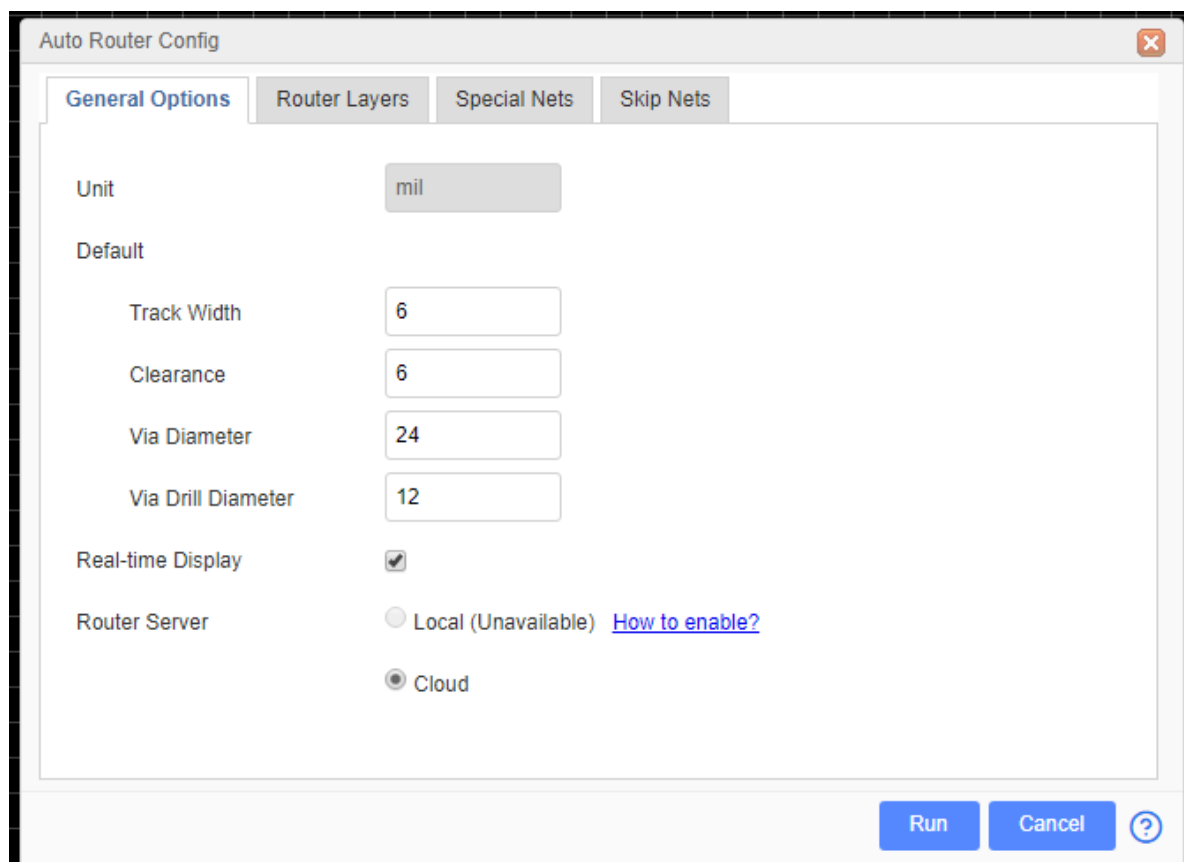
Auto router is not good enough! Suggest routing manually! You can use "RoundTrack(Walk Around)" option to route tracks, via right-hand panel - Routing Conflict.

Steps:

1 Click the the auto router button from the Top Menu"Top Menu> Route > Auto Router"

2 Config the auto router

After you click that button, you will get a config dialog like in the image below.



In the config dialog, you can set some rules to make the auto router result professional. These rule must equalize or more than DRC setting.

General Options

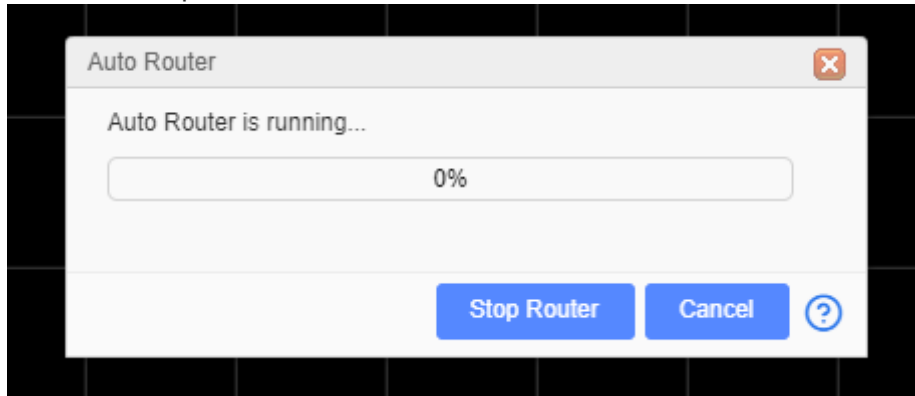
- **Unit:** The unit follows PCB canvas unit.
- **Track width:** The auto-route track width.
- **Clearance:** The clearance of the objects.
- **Via Diameter/Via Drill Diameter:** The via placing by auto-router.
- **Realtime Display:** when you select it , the real time routing status will show on.
- **Router Server:**
 - **Cloud:** Using EasyEDA online server.
 - **Local:** Using the local auto router server, when you click the Auto Router icon, the editor will check the local router server available or not automatically. How to use

please see as below.

- **Router Layers:** If you want to route inner layer, you have to enable the inner layer first.
- **Special Nets:** For the power supply track, you may want it to be bigger, so you can add some special rules.
- **Skip Nets:** If you like to keep the a net with no route, you can skip it. For example, if you want to use copper area to connect `GND` net, you can skip the `GND` net. If you want to reserve the routed track, you need to select the `Skip Routed Nets`.

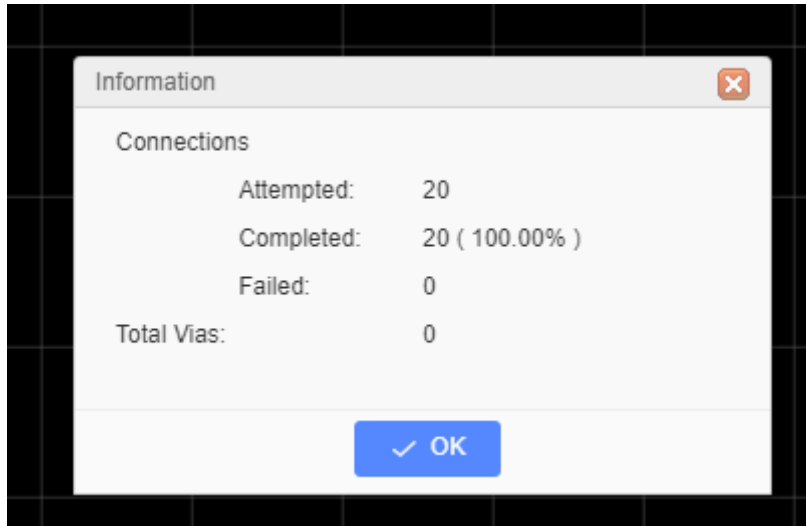
3 Run it

After click the "Run" button , The real time check box will let you see how it is going, but it will make the process a little bit slow.



Waiting for a few minutes, after adding bottom and top copper area, you will get a finished PCB board.

When finish, will pop up a window.



The connection means the track connect times.

Notice:

- The parameter can't less than DRC rule, otherwise will report error.

