Right clicking

In EasyEDA, right-clicking opens a context sensitive menu:

- When you are placing a symbol, right-clicking will stop placing and return to select mode. This is the same as the ESC key.
- When you are drawing a shape such as a polyline, after a right click, the polyline will be stopped at the place where you right click but the mouse will remain as a **cross**, so you can draw another shape.
- To get out of the current active context sensitive command such as placement or drawing mode and go back to **select mode**, just double right click or press ESC (sometimes twice).

Right-Click and drag Right-clicking and holding the button anywhere in the Schematic, Waveform or PCB Canvas while dragging the mouse will move the canvas around within the EasyEDA window. Holding the middle button and dragging performs the same operation.

ESC key

Pressing the `ESC` key ends the current drawing action but does not exit the current active context sensitive command mode (i.e. it does not return the cursor to select mode). Pressing ESC again returns to select mode.

Select more shapes

- CTRL+Left Clicking on items adds those items to your selection;
- Clicking and holding creates a selection box;
- Creating a selection box, using click and drag to the right, selects everything inside the box;
- Creating a selection box, using click and drag to the left, selects everything inside and intersected by the box;

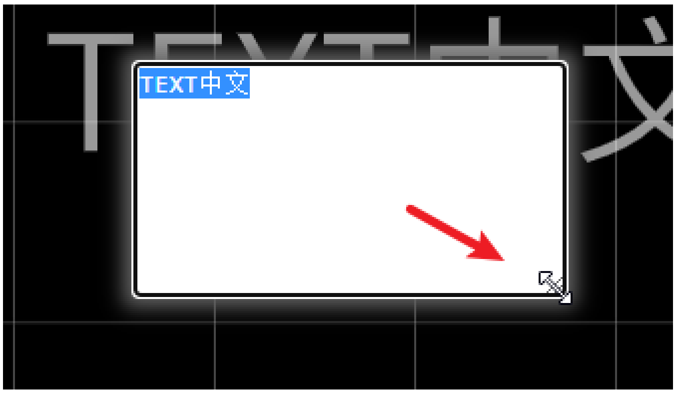
Zoom in and Zoom out

- Using the middle mouse button:
- Roll forward to zoom in;
- Roll back to zoom out;
- Using hotkeys, the default hotkey `A` for zoom in, `Z` for zoom out.

Please note:
Do not scroll your mouse at the same time as pressing the CTRL key when your cursor on the top menu, the browsers will zoom the whole website, if you just want to zoom the canvas in the EasyEDA window, you need to put your cursor onto the canvas. If the "zoom the whole website" happens, just press `Ctrl+0` to reset the browser view zoom.

Double clicks

Double clicking any text area opens a resizable text box that allows you to edit the text.



Press the enter key to save your changes. Click outside the box or press ESC to discard your changes.

Pan/Move Canvas

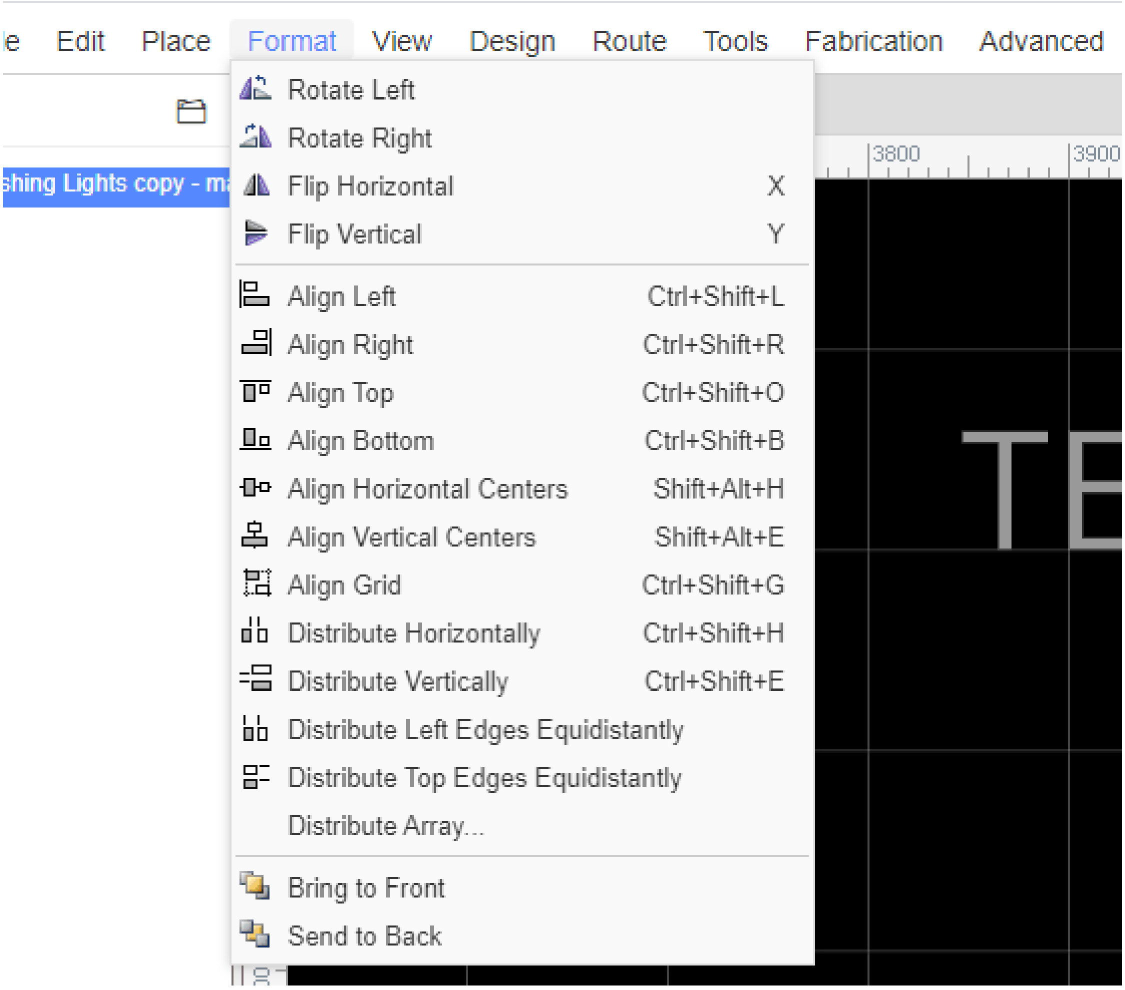
- Right click anywhere in the Schematic, WaveForm or PCB Canvas and Hold down right button to drags the canvas around within the EasyEDA window.
- If your canvas is bigger than the EasyEDA window and is showing scroll bars, you can use either the scroll bars or the Arrow keys to scroll the canvas to pan.
- When drawing a wire, a graphic line or shape that you wish to extend beyond the edge of the EasyEDA window holding down the left mouse button after starting the line will pan the canvas to keep the drawn item inside the window.

Tip:
If you use Chrome, and cursor is in the canvas while pressing CTRL or ALT key and rolling your mouse, the canvas will move vertically, and when pressing SHIFT and rolling your mouse, the canvas will move horizontally.

Rotate

After selecting one or more items, you can rotate the selected items using:

Top Menu > **Format** > **Rotate** or by pressing the default rotate hotkey: ‘Space’ .

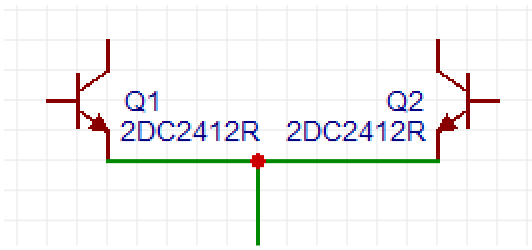


When in PCB view mode you can click the footprint and change its rotation in the right property panel.

Please note:
Rotating a multiple selection rotates each item about its own symbol origin. It does not rotate the items about the centroid of the group of items.

Flip

To place a Q2 as shown in the schematic below you need to Flip the item. Via: Top menu - Format - Flip.



You can Flip one or more selected items using:

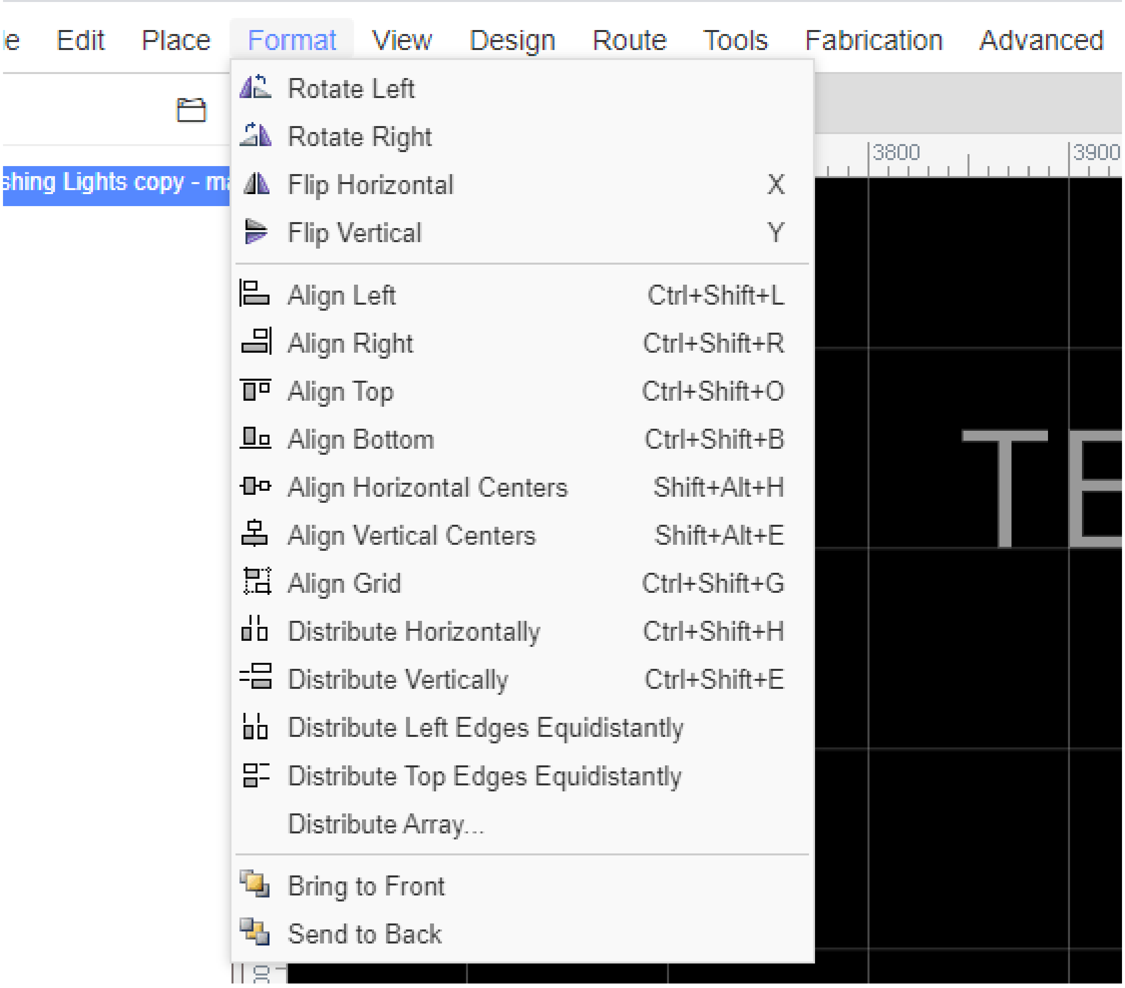
Rotate and Flip > Flip Horizontal or Flip Vertical from the toolbar,

or by pressing the default flip hotkeys: `X` to Flip Horizontal, `Y` to Flip Vertical.

Notice: Footprint does not support the flip command.

Align

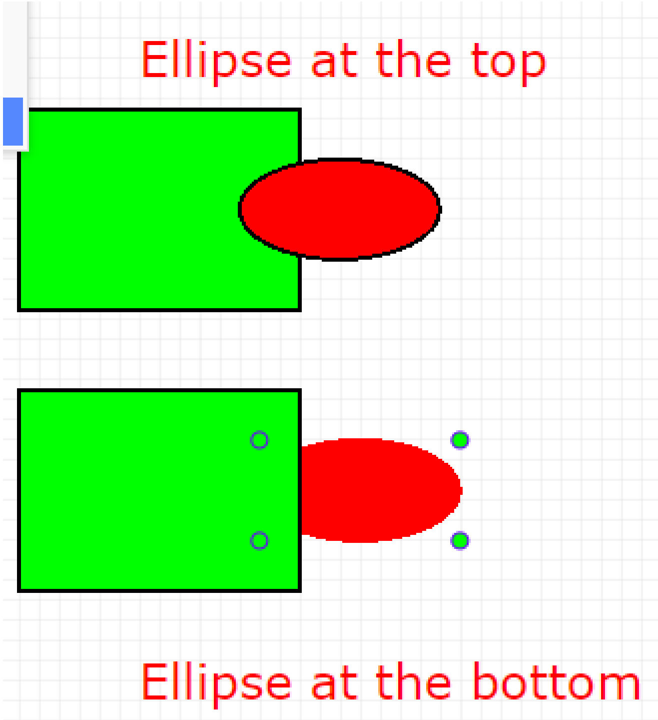
EasyEDA provides many align option features, you can align your symbols or footprints very easily using: Top menu - Format - Align. There are also icons on the toolbar for this.



Bring to Front and Send to Back

In the image below, both the rectangle and the ellipse are filled. Use: Top menu - Format - Bring/Send to Front/Back.

If you draw an ellipse before drawing a rectangle, the rectangle will overlap and therefore hide the ellipse. To reveal the ellipse, select the rectangle and then use the Bring and Send function, as shown:

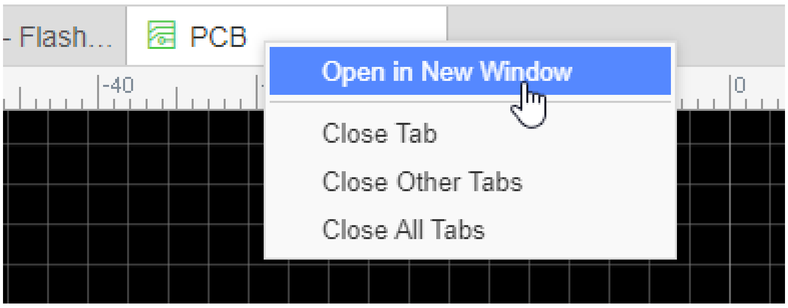


Multiple Windows

Since v6.4.0, EasyEDA supports multiple windows design.

How do it works?

1. Open schematic and PCB
2. Right-click the schematic or PCB tab, click "Open in New Window"

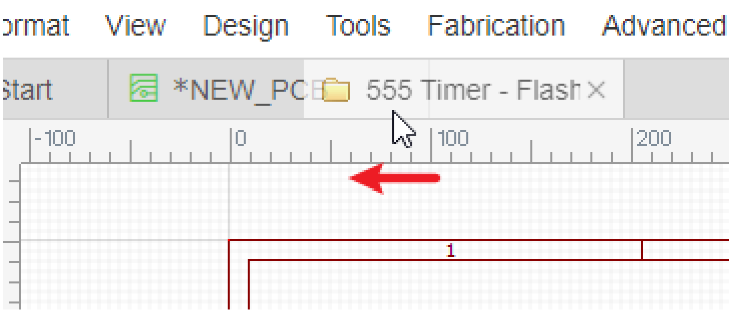


3. This will open the document in a new window, then you can do the cross probe: Click the component, pads, click the Design Manager list, the "Cross Probe and Place" also works.

Documents Tab Switch

It is easy to modify the tab positions of your documents.

Simply drag the tab location, or use the hotkeys SHIFT+1 and SHIFT+2

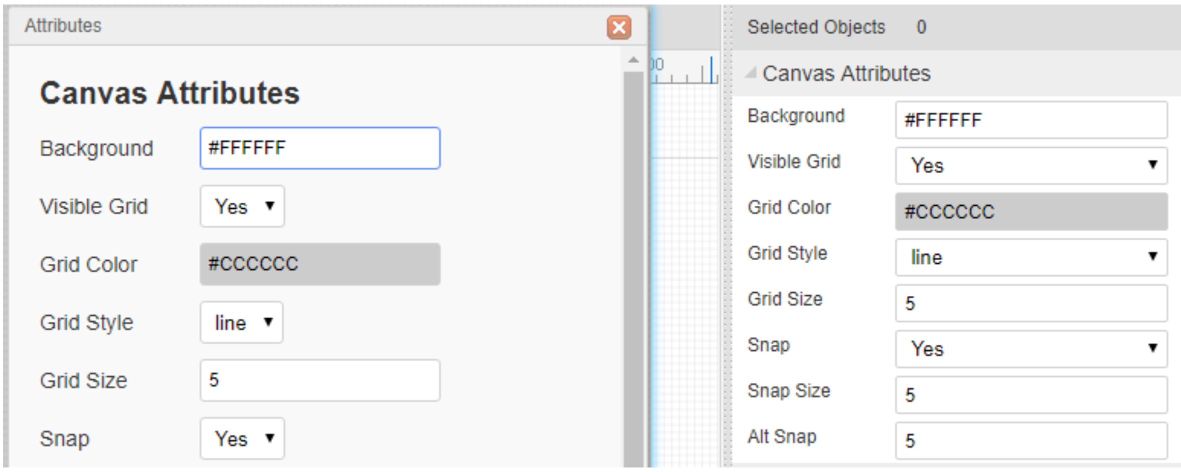


Schematic Capture

Canvas Setting

During this tutorial we will create a simple Schematic design to guide you in using EasyEDA Schematic capture.

You can find the canvas Properties setting by clicking on any the blank space in the canvas.

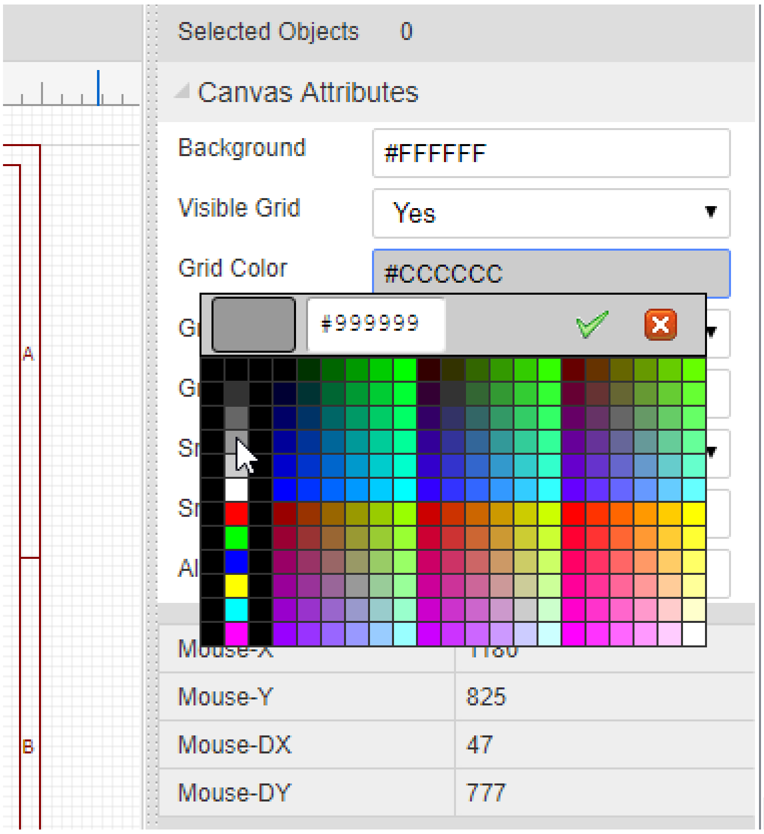


As described earlier, background and grid colours and the style, size, visibility and snap **attributes** of the grid can all be configured.

The canvas area can be set directly by the Width and Height or by using the available preset frame sizes.

Grid:

- **Visible Grid** : Yes or No
- **Grid Color**: Any valid colour
- **Grid Style**: Line or Dot
- **Grid Size**: To ensure proper alignment of all EasyEDA parts, it is advisable to set in 10, 20, 100. the unit is pixel.
- **Grid** (and background) colour can be set directly by entering the hexadecimal value of the colour you want or by clicking on a colour in the palette that opens when you click on the colour value box:



Snap:

- **Snap**: Yes or No. Pressing this key toggles switching snap to grid on and off.
- **Snap Size**: To ensure proper alignment of all EasyEDA parts, it is advisable to set in 10, 20, 100 but any valid number can work, such as 1, 5, 10.

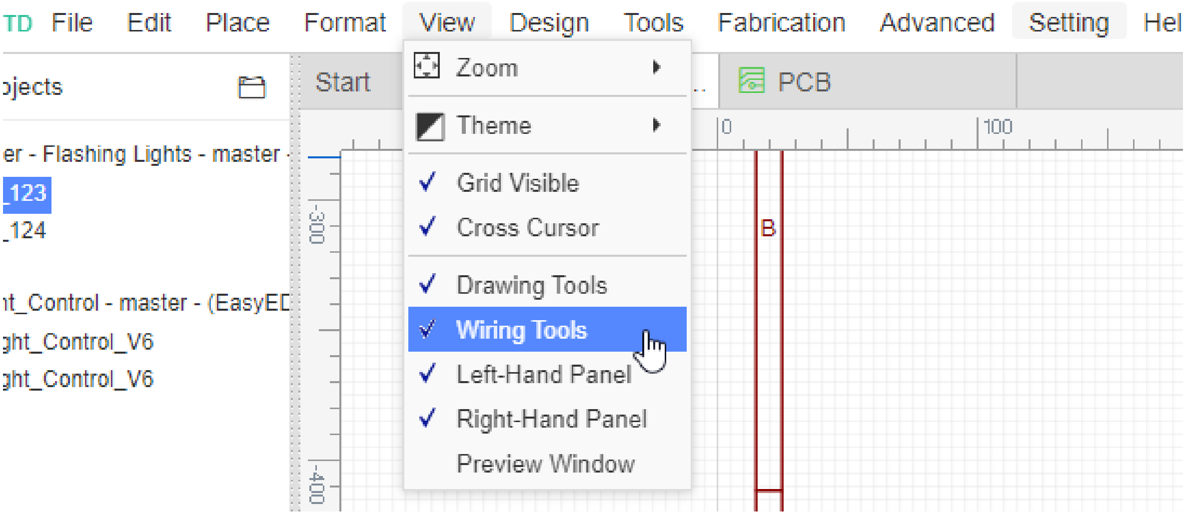
It is strongly recommended that you keep **Snap = Yes** all the time. Once items are placed off-grid it can be very difficult to reset them back onto the grid. Off-grid placement can result in wires looking as though they are joined when in fact they are not and so causing netlisting errors that can be hard to track down.

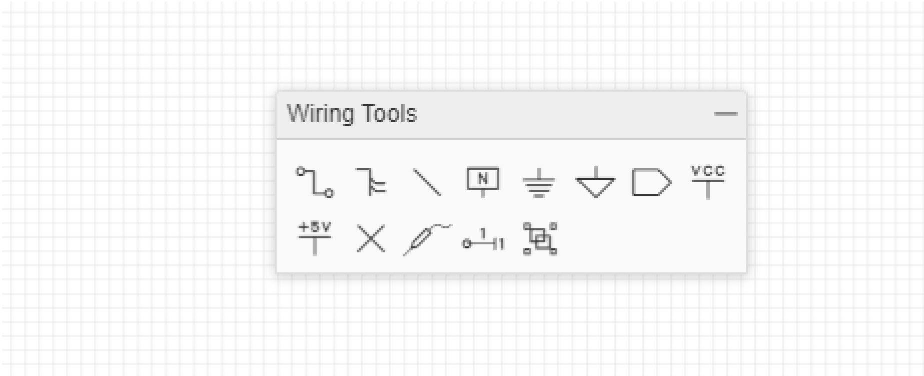
If you need to draw detailed parts of new symbols or footprints that need to go between grid points, try to reduce the grid spacing to draw these elements and then reset the grid back to your chosen default value as soon as you have completed that part of the drawing. Setting Snap=No should only really be used as a last resort.

- **ALT Sanp**: Snap size when pressing the `ALT` key.

Wiring Tools

If you have hidden your tools , you can open them from here:
Top toolbar **Top Meun** > **View** > **Wiring Tools...**

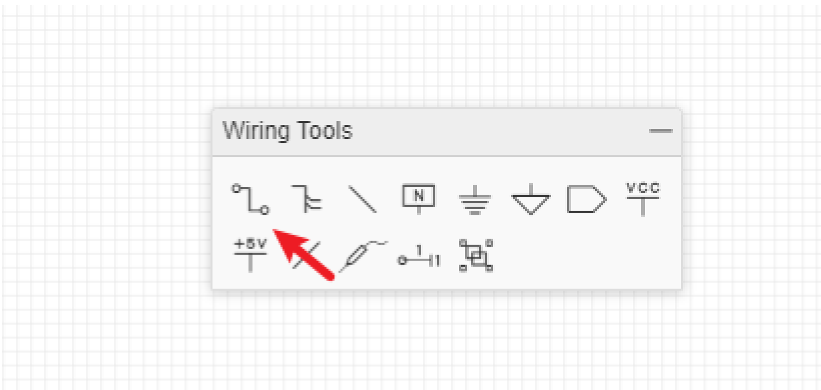




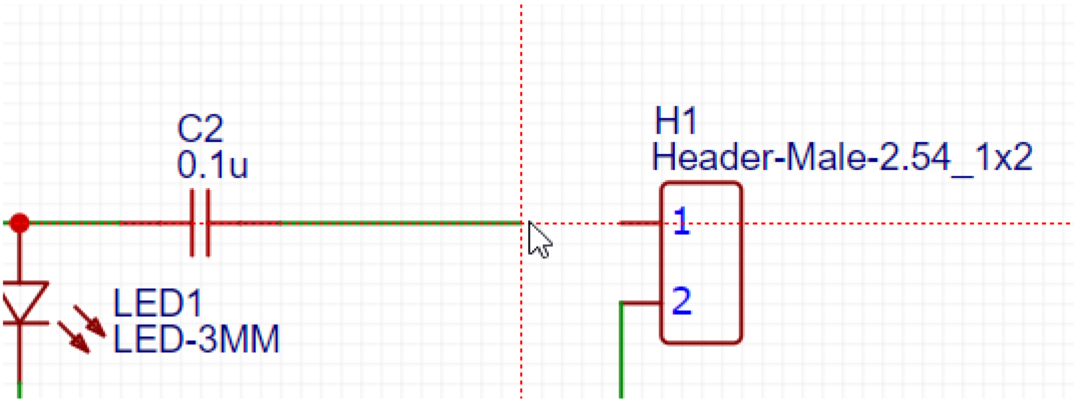
Note: *All of the commands in Wiring Tools are electronics related. Don't use a wire when you just need to draw a line, shape or an arrow: use Drawing Tools instead.*

Wire

There are three ways to enter the wire mode in EasyEDA.

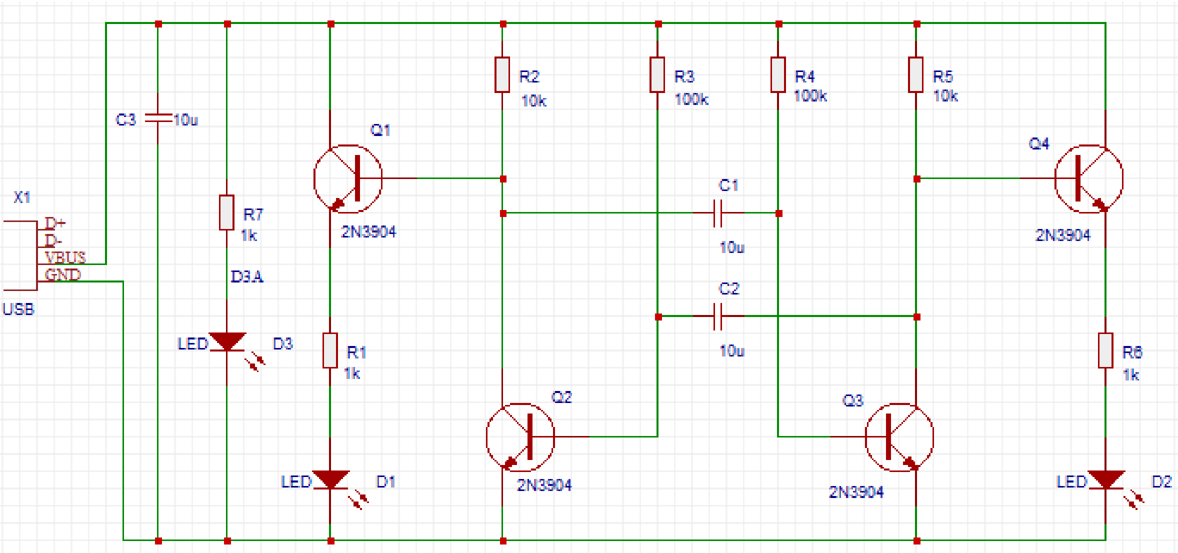


1. Click the **Wire** button from the **Wiring Tools** palette.
2. Press the `W` hotkey.
3. Click on the end of a component pin (where the grey pin dot appears if you select the component):



EasyEDA automatically enters **Wire** mode.

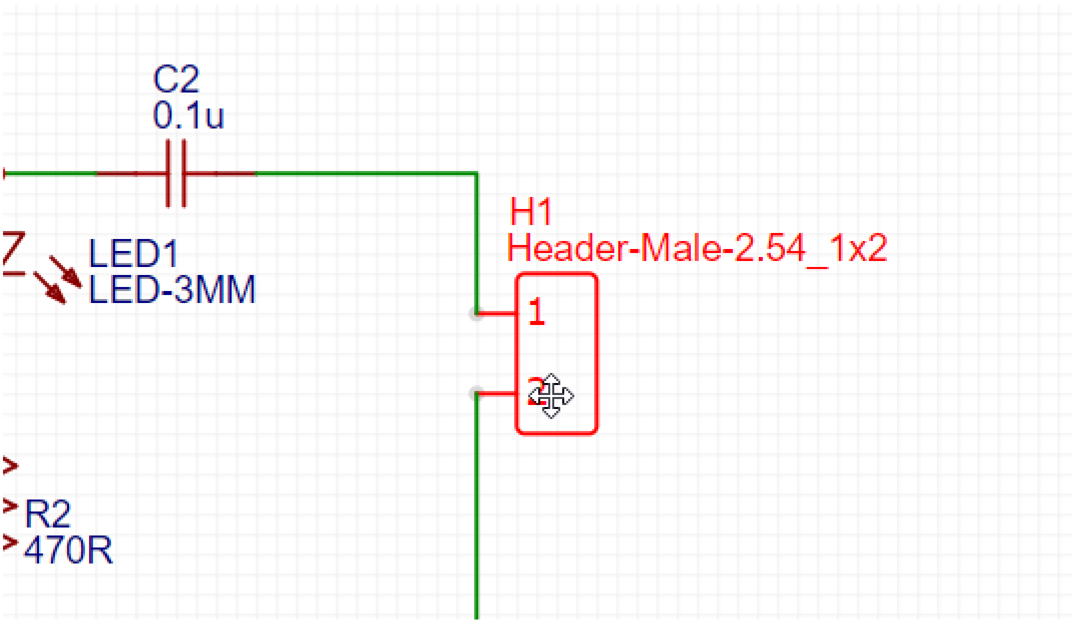
Here is a screenshot of the **Astable Multivibrator LED project schematic** after wiring:



Moving Components and Wires:

If you place a component, such as a resistor, on top of a wire then the wire breaks and reconnects to the ends of the component.

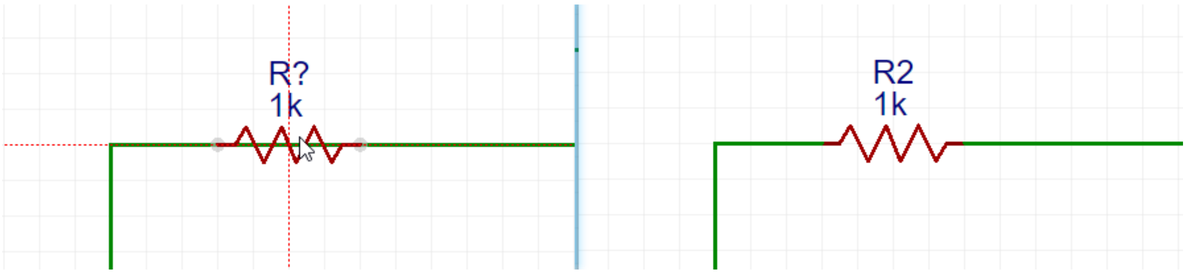
When moving selected components using the mouse, they will drag attached wires with them (“rubber band”) to some extent but please be aware that the rubber banding feature has some limitations. When moving selected components most wire will move vertically and horizontally. Using the arrow keys will not rubber band. Selected wires do not rubber band.



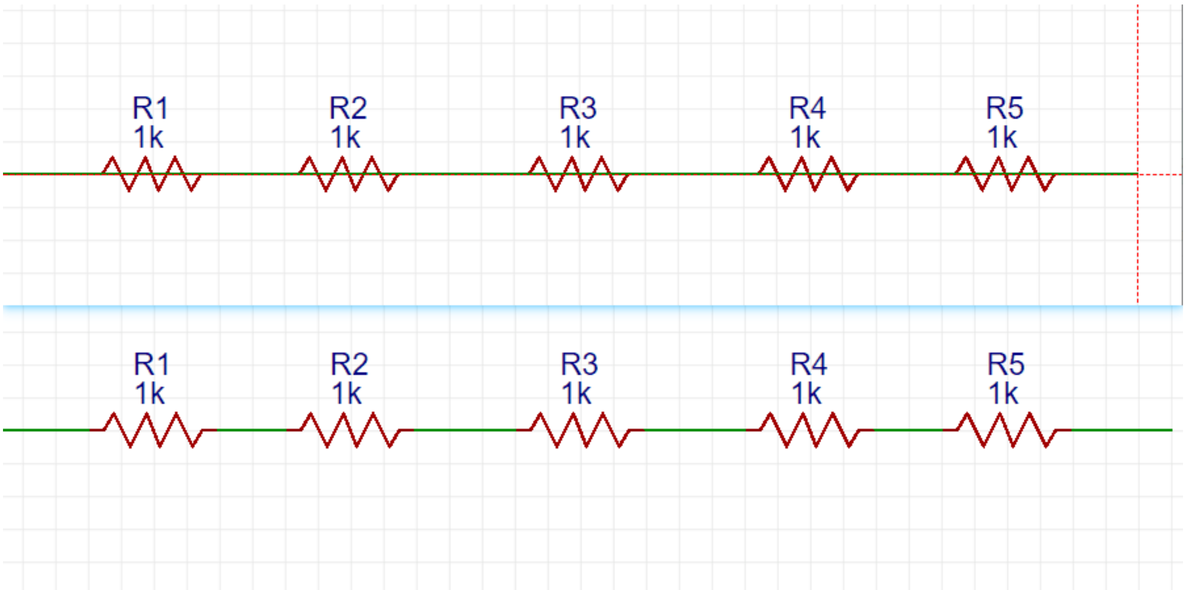
A selected wire can be moved directly by clicking on it using the mouse or by the arrow keys. If a wire is selected by clicking on it using the mouse then green grab handles will appear at the ends and vertices.

Auto adjust connection

If you put a resistor or capacitor on a wire, the wire will auto connect the pins as below:

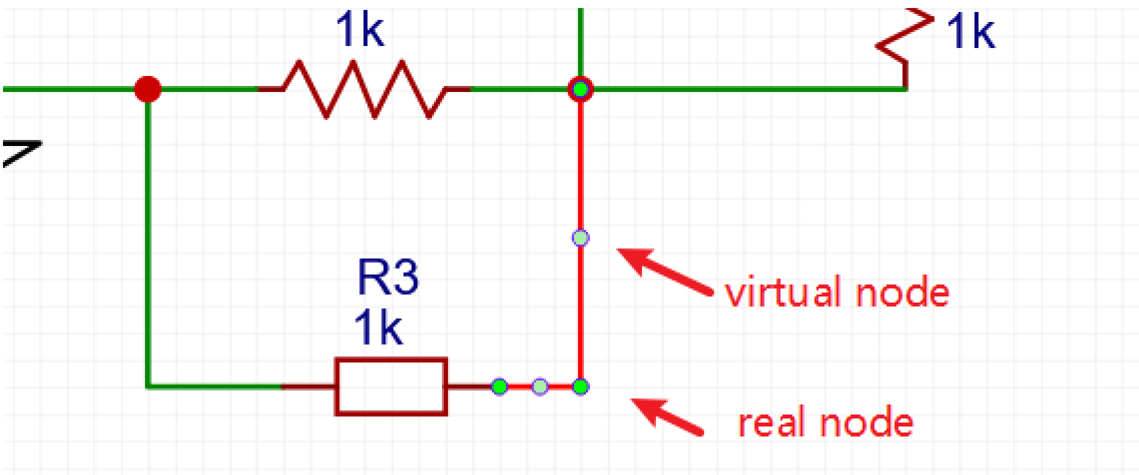


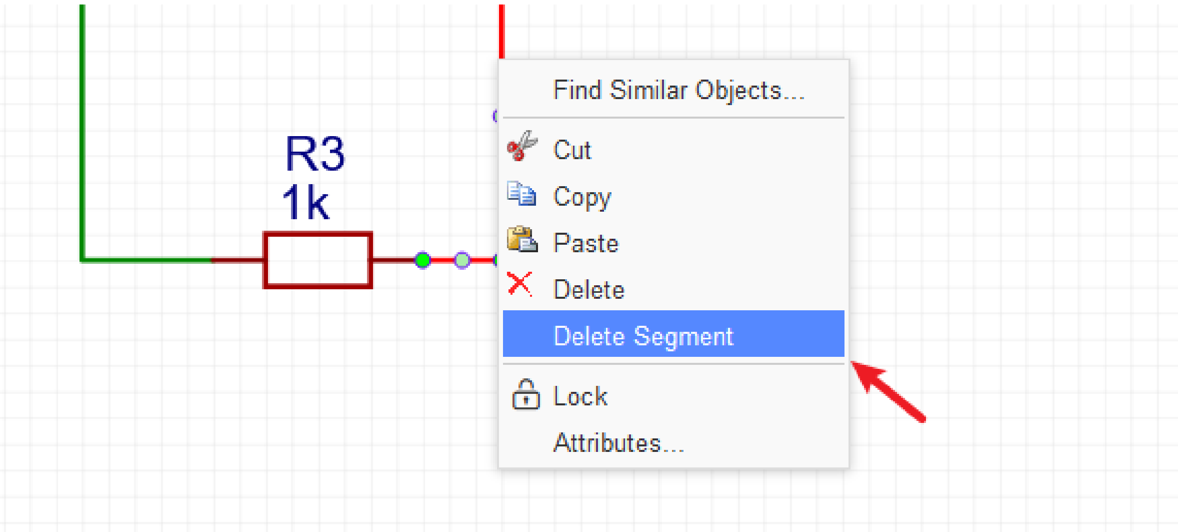
When you want to wiring a series of resistors which are in a row, you can just wire through them, and then you will find they all be connected.



Wire Node

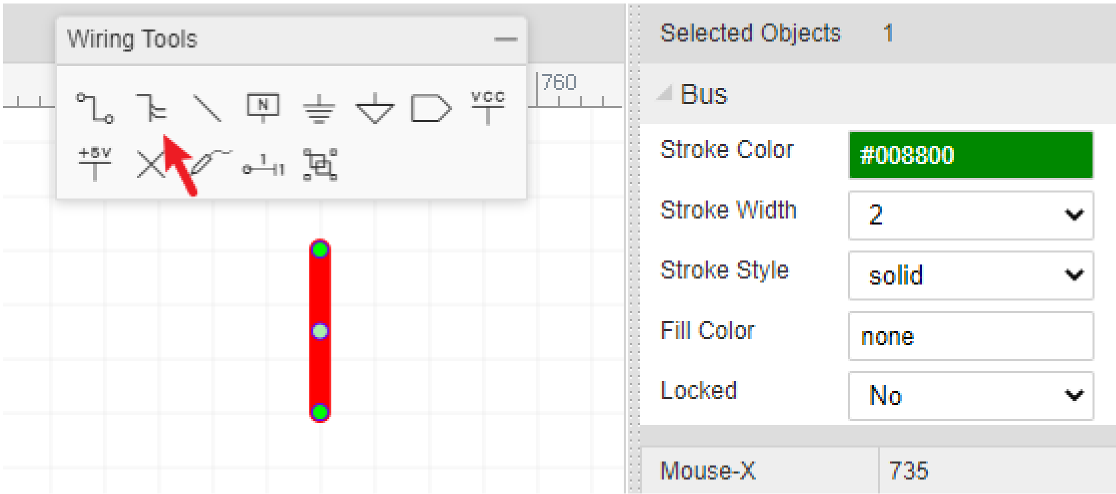
When you click on the wire, you can see the nodes on the wire, where the white is the virtual node, the red is the real node, drag the virtual node to generate the real node, and right-delete the line segment is to delete the line segment between the real nodes.

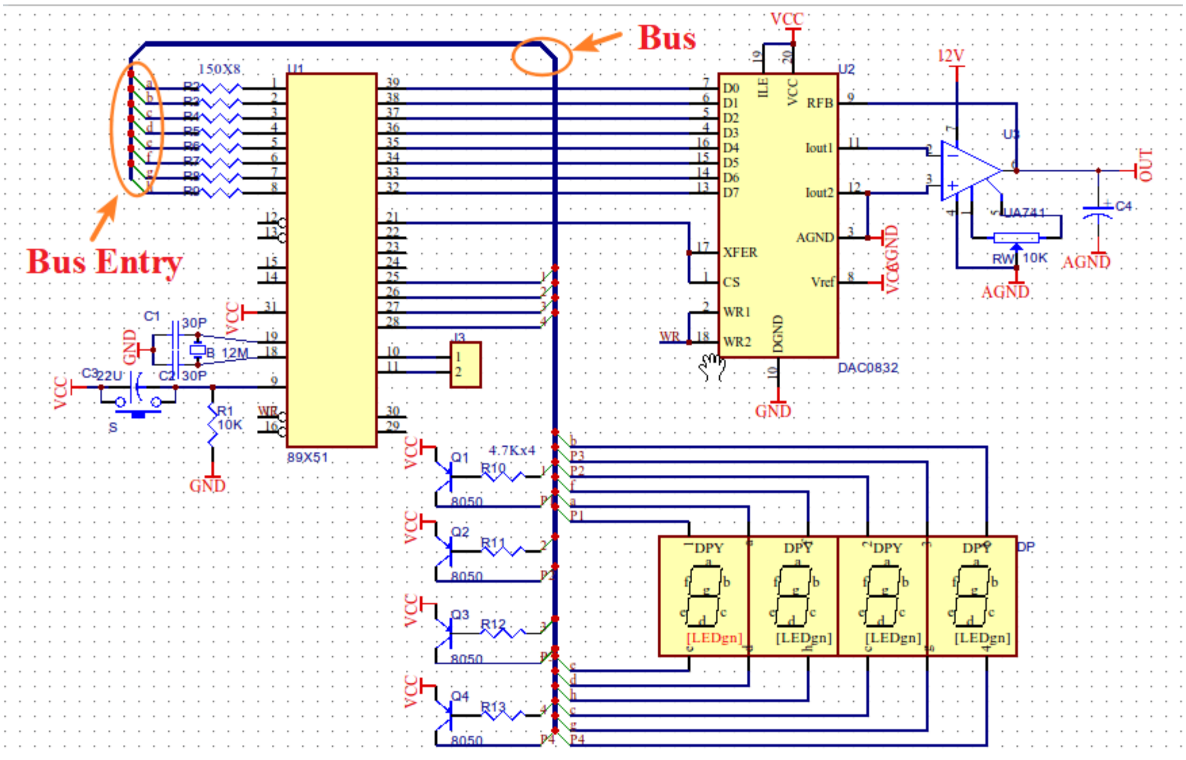




Bus

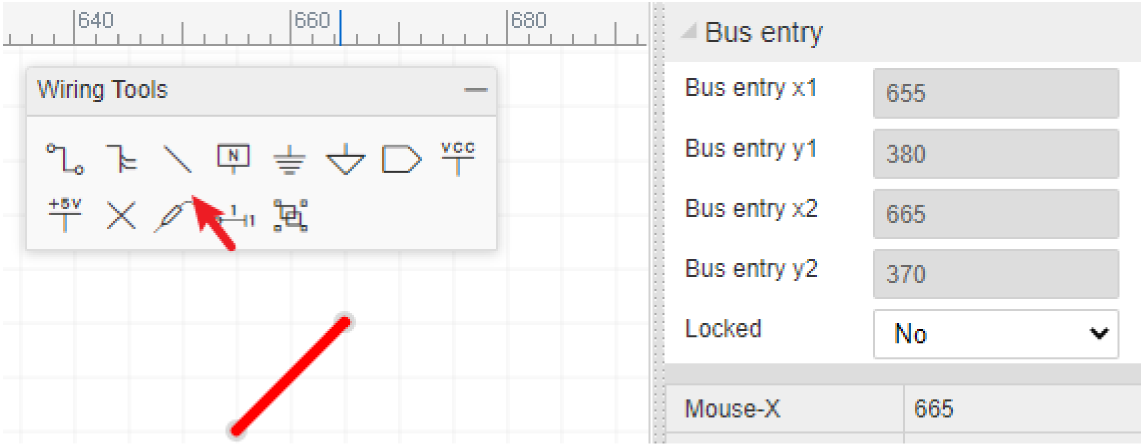
When you design a professional schematic, perhaps it will use a lot of wires. If you wiring one by one, much time would be wasted, and then you need to use `Bus`.





Bus Entry

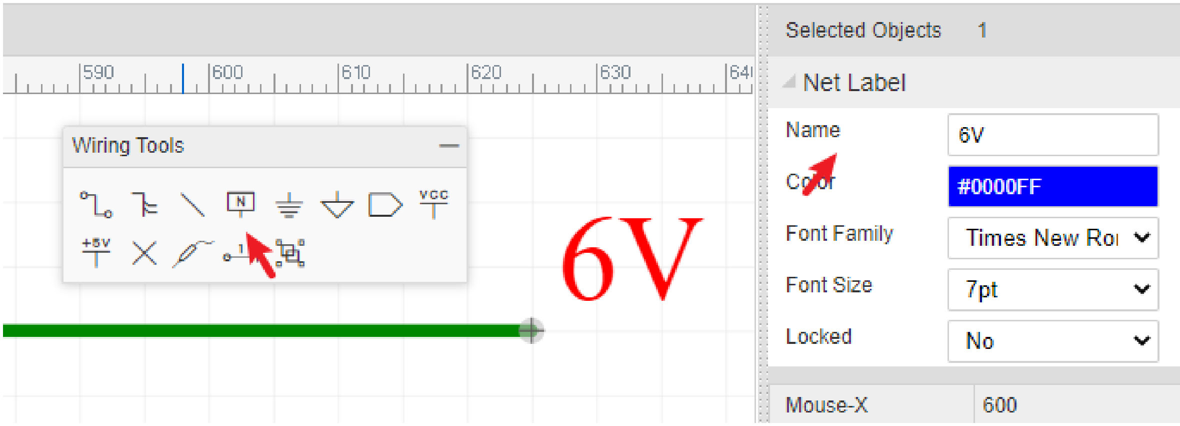
If you decide to wire with `Bus` , the `Bus Entry` must connect to Bus and other nets with wires. such as in the above image.



The "Bus" and "Bus Entry" just for the indication, because when you place Bus and Bus Entry, you have to place the netlabel on the Bus Entry dot point.

Net Label

NetLabel can be used to give your wires names to help you find them and identify any misconnections. You can find the **NetLabel** from the Wiring Tools palette or by using the `N` hotkey. When selecting the netlabel, you will find its attributes in the right hand Properties panel:

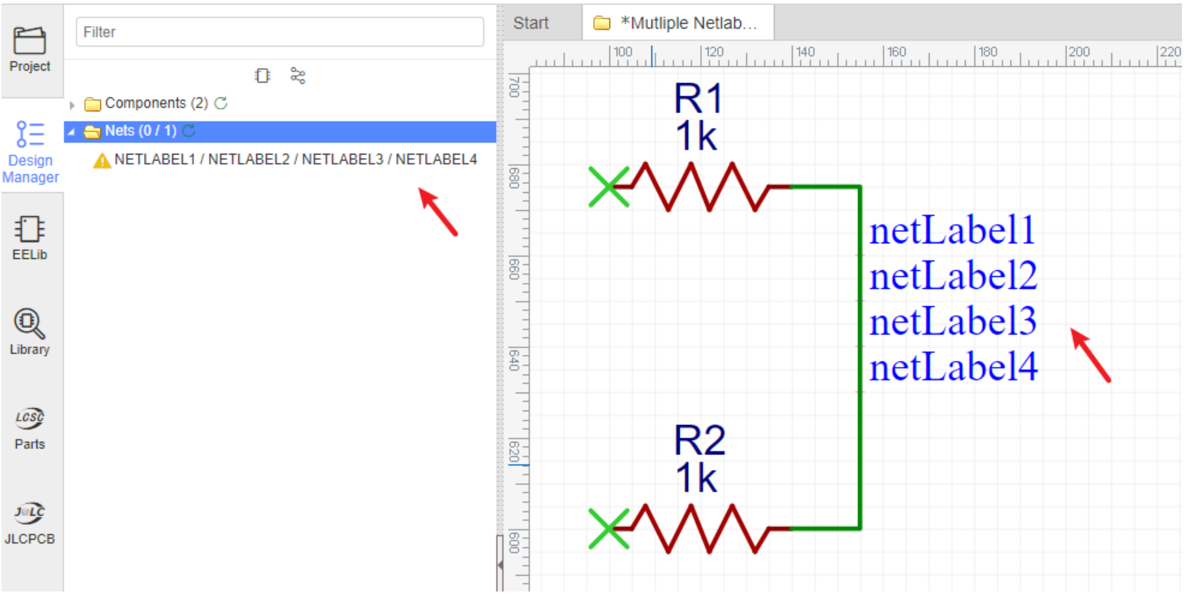


You can change its name and colour. If you only want to change its name, it may be easier to just double click the netlabel.

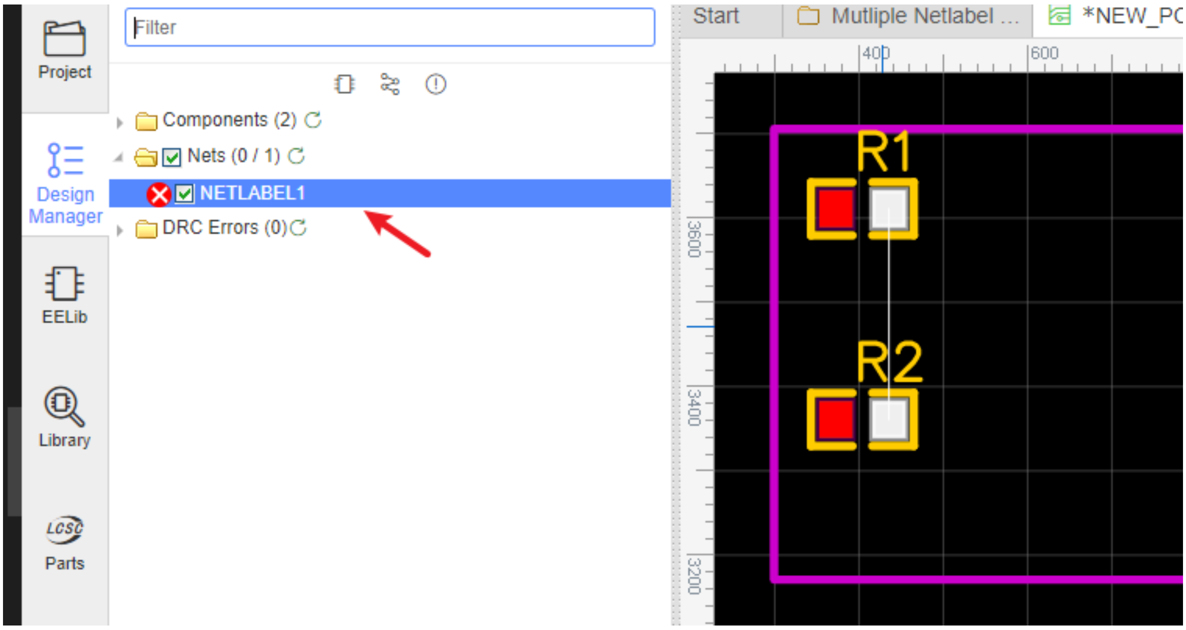
Multi-NetLabels in One Wire

EasyEDA support mutil-netlabel in one wire now.

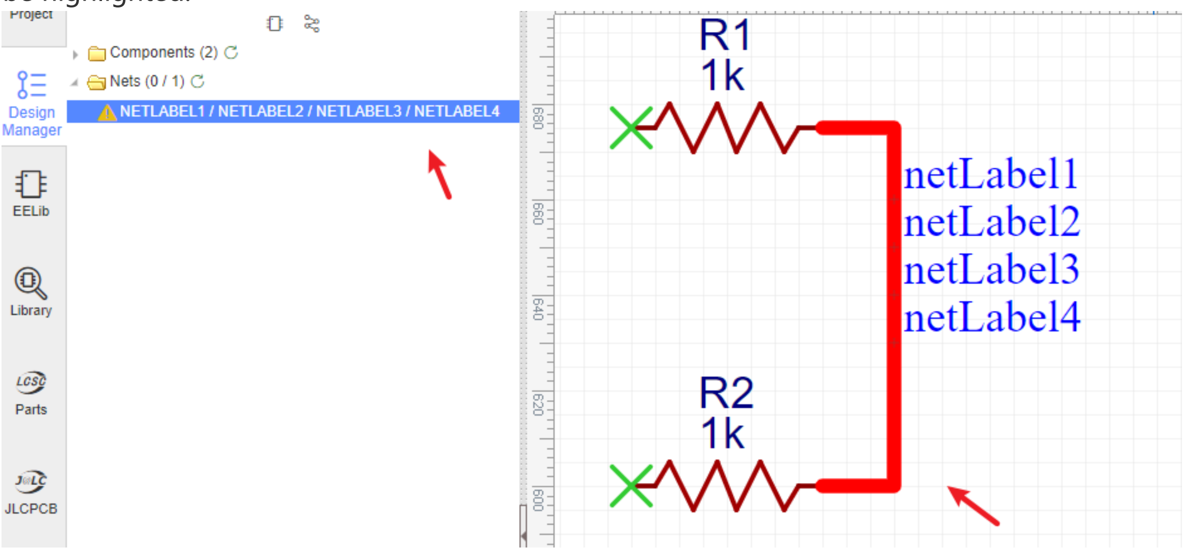
When you convert the schematic to the PCB, the editor will choose the first netlabel you placed as the net name for this wire, as below NETLABEL1.



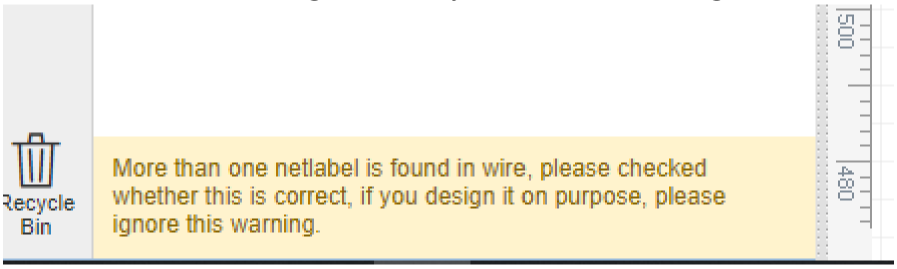
Convert to PCB:



As above image, when you click anyone netlabel's name in the design manager, the wire will be highlighted.



And check the bottom right corner, you will see a warning:

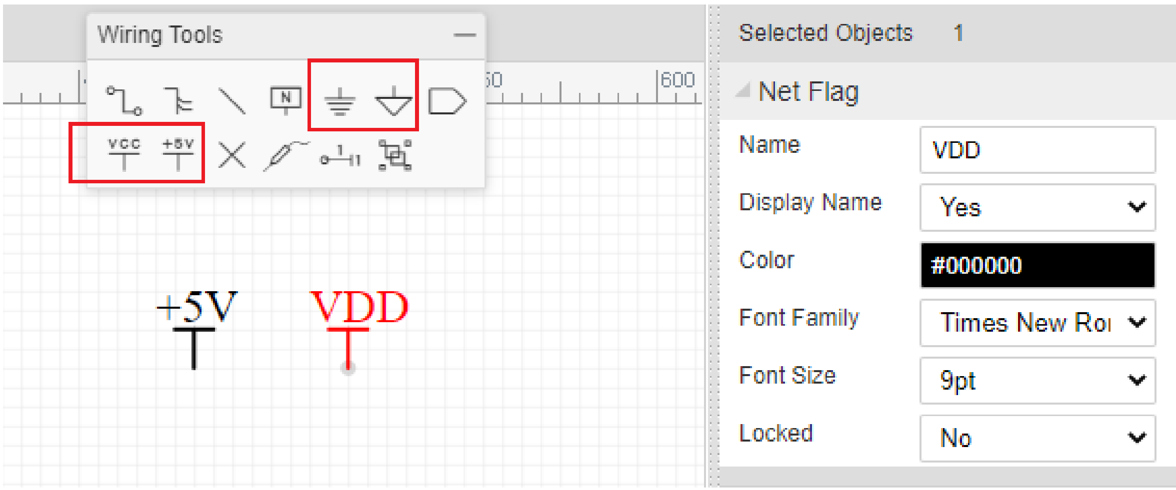


Notice:

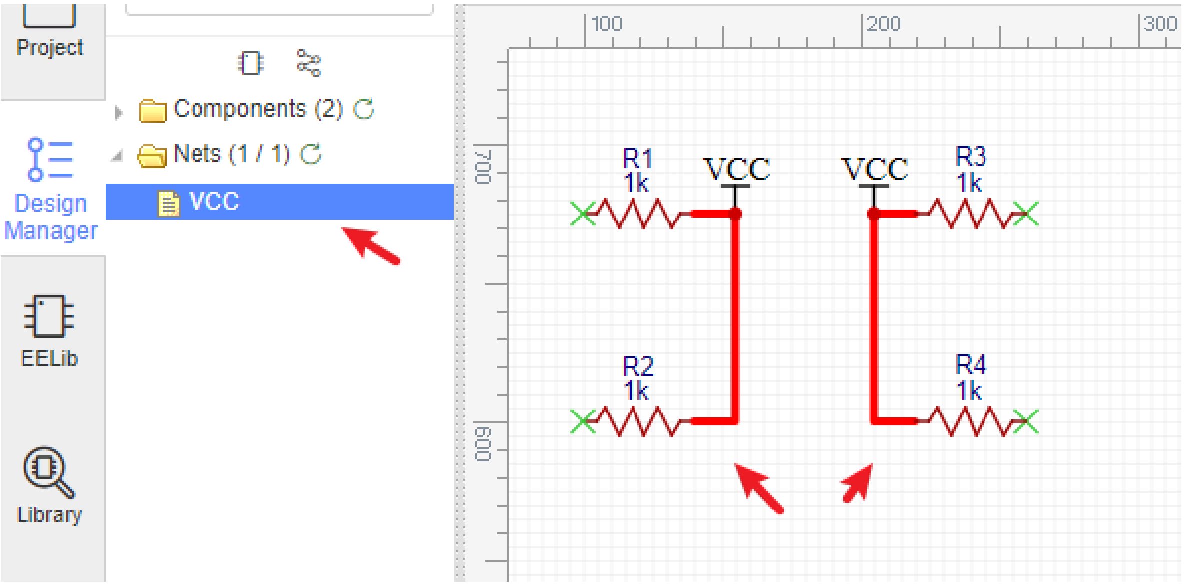
- *If wire 1 has 3 netlabels A B and C, and wire 2 has netlabel A, then wire 1 and wire 2 are the same net.*
- *Netlabel/Netflag/Netport/volprobe only support English characters and letters, and Arabic numerals.*
- *If a part prefix is P1, which has two pins, it will have two nets "P1_1" and "P1_2" by default, if you place a netlabel named P1_1 at other wire which is not connect with P1 pin1, the default "P1_1" will change to "P1_1(1)" for avoid the wrong connection with netlabel "P1_1".*

Net Flag

NetFlag is the same as NetLabel, you can find the NetFlag from the Wiring Tools palette or using the `Ctrl+G` hotkeys for **GND** or `Ctrl+Q` for **VCC**. You can also change its name, for example from **+5V** to **VDD**:



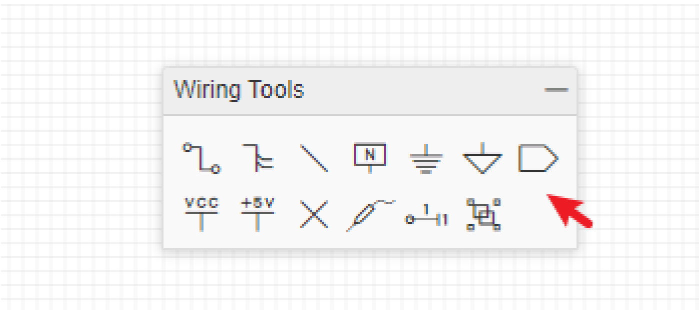
When appear two and more Netflag or Netlabels which are the same name, they will connected with each other.

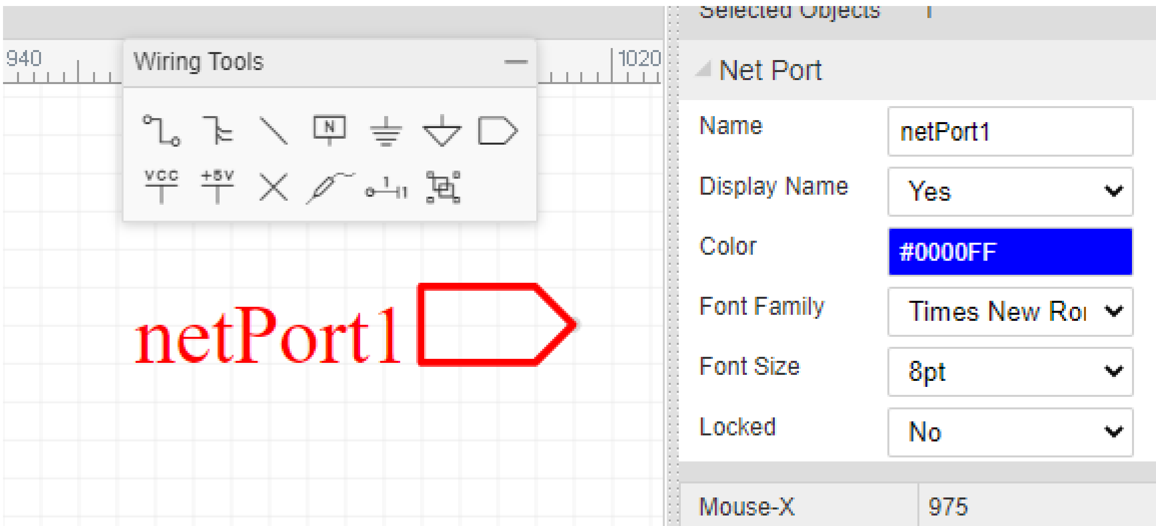


Wiring Tools palette provides NetFlag: Digital GND, Analog GND, VCC and +5V for your convenience.

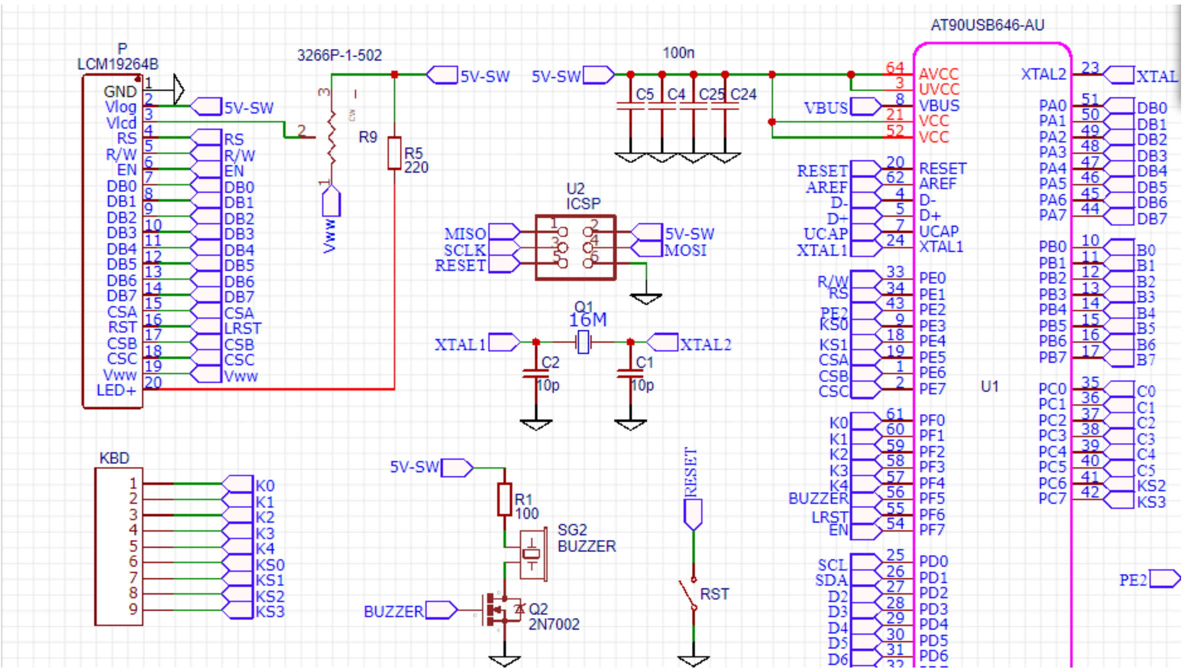
Net Port

At EasyEDA, Net Port works like Net Label, it doesn't differentiate the input and output net port. When you don't want to route too many wires, how about trying `Net Port` :



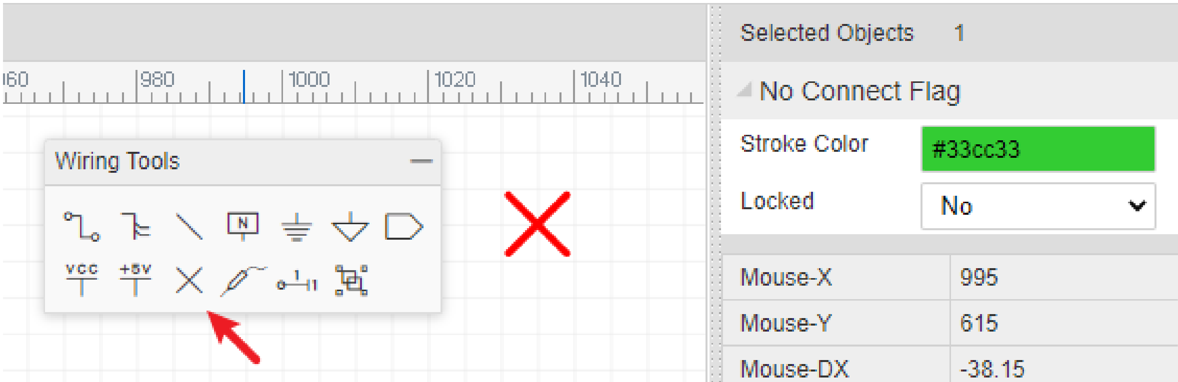


It will make your schematic look more clean, and you just need to set each Net Port a net name.

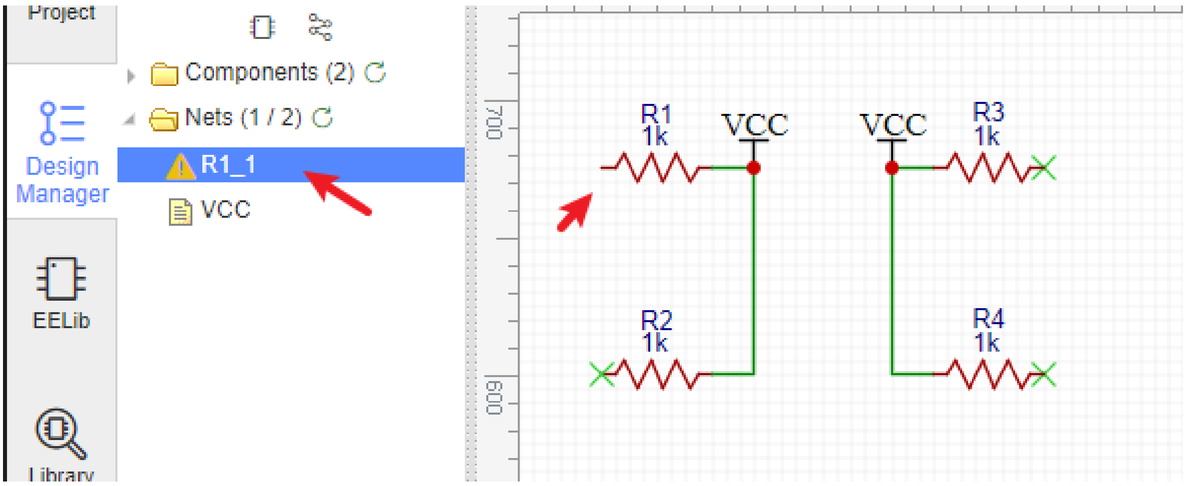


No Connect Flag

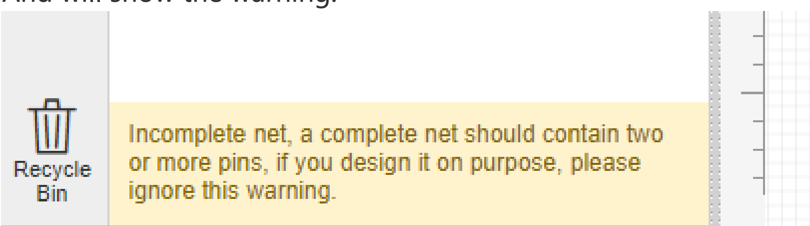
You can find the `NO Connect Flag` via wiring tool,



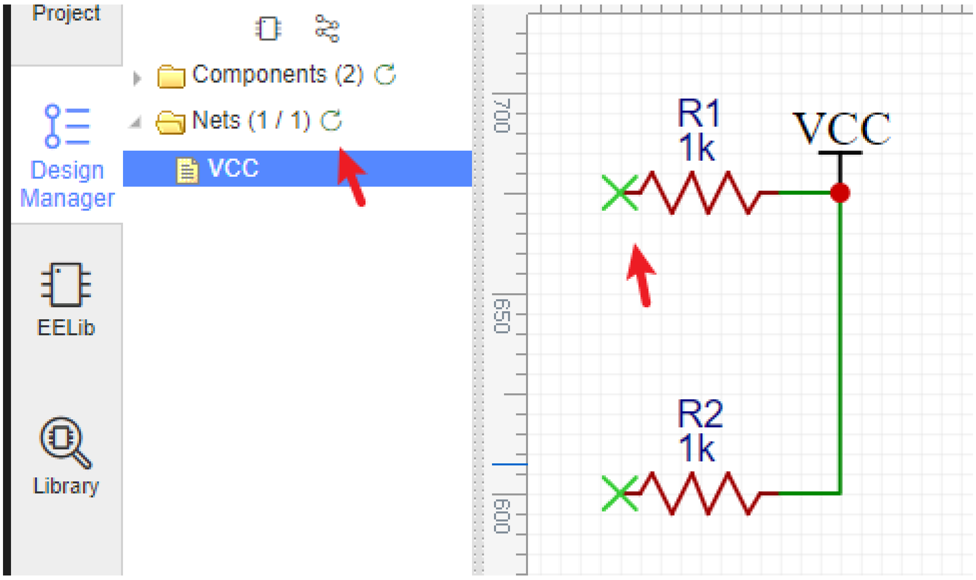
In the below schematic, if you don't add a `NO Connect Flag`, there is an error flag in the nets collection of the design manager.



And will show the warning:

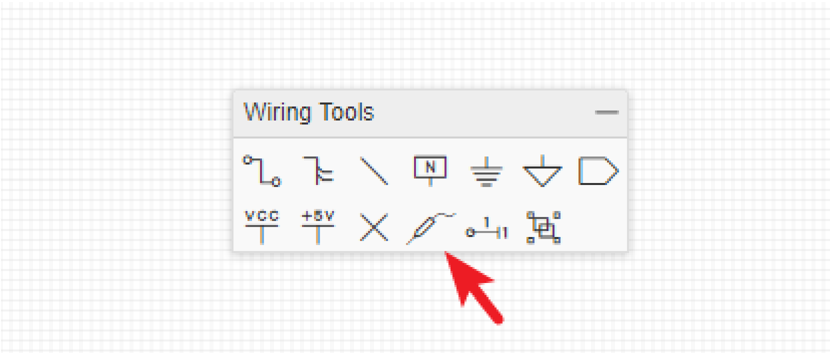


After adding a `NO Connect Flag`, and then refresh the Nets folder, the error disappears.

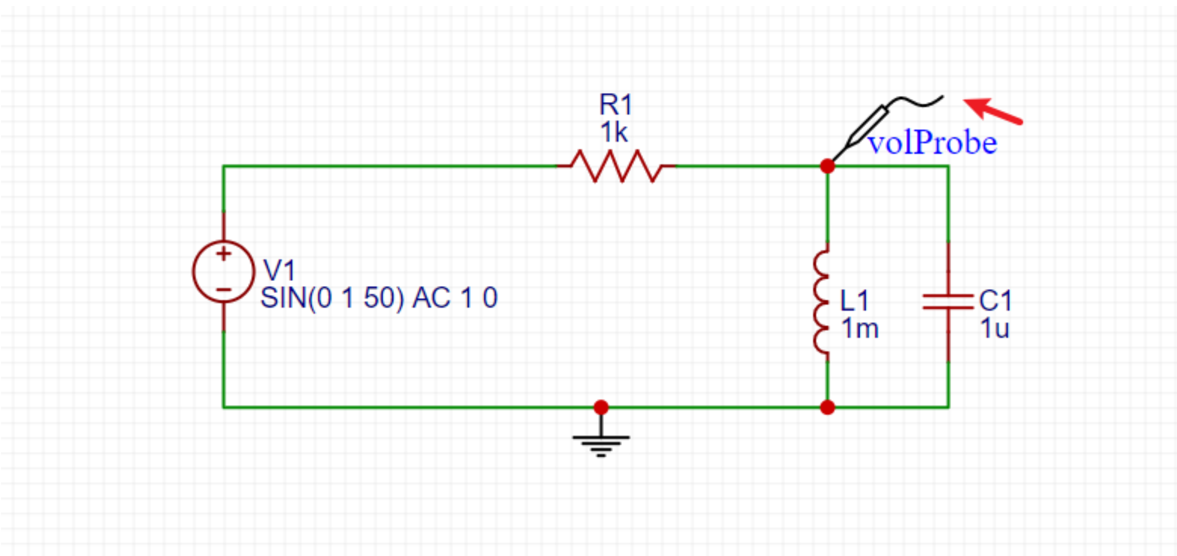


Note: *`NO Connect Flag` only works on the symbol's pin directly.*

Voltage Probe



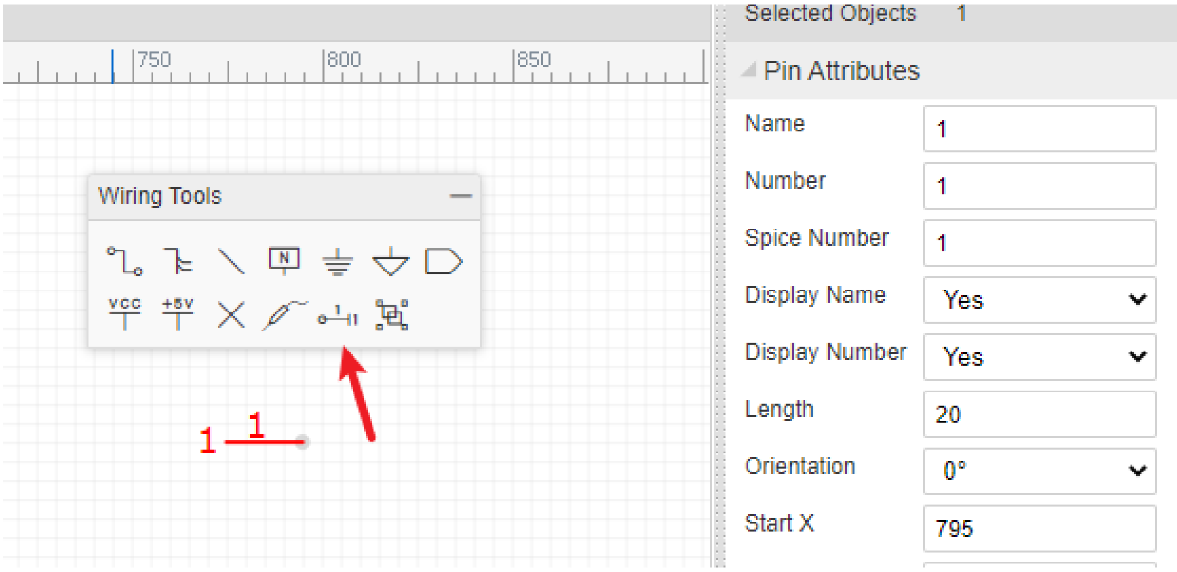
EasyEDA provides a simulation feature for the schematic. After the simulation is running, you will see the waveform where you placed the voltage probes in the circuit.



For more detail about the simulation, please check the Simulation section.

Pin

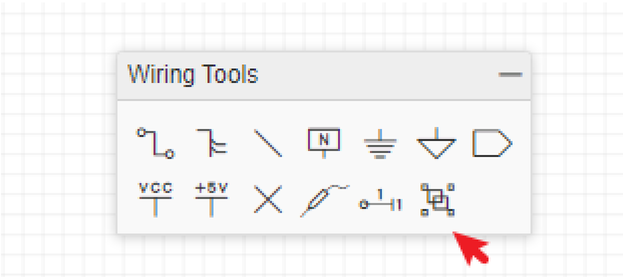
When you create a new symbol in schematic and schematic lib, you must use `Pin` to create pins for the new symbol, otherwise your symbol can't be wired with wires.



For more information please refer to the **Symbol Library - Create Symbol** section.

Group/Ungroup Symbol

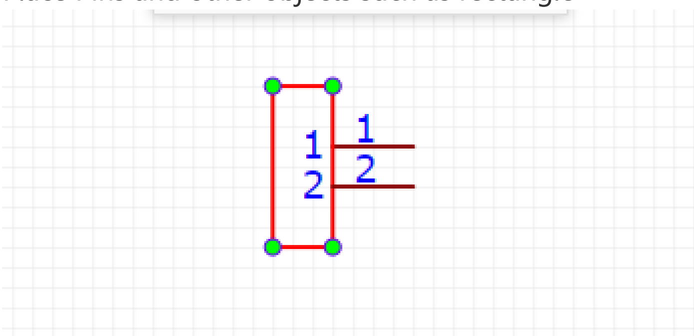
On the **Wiring Tools** palette there is the **Group/Ungroup Symbol...** button.



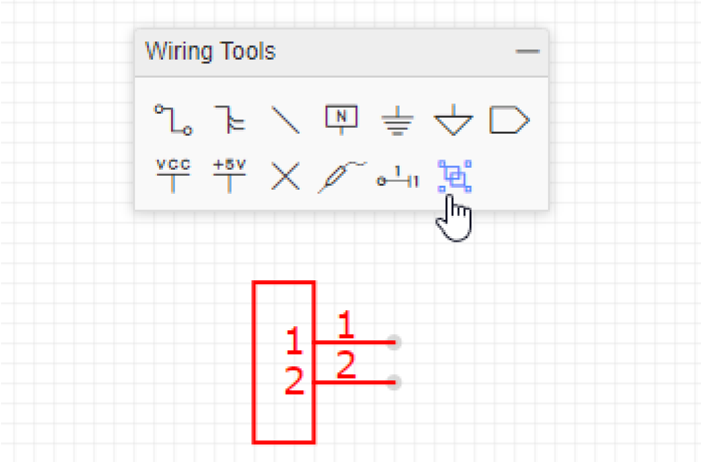
Just like the **Symbol Wizard**, this tool is also for you to quickly create schematic library symbols.

Here's how.

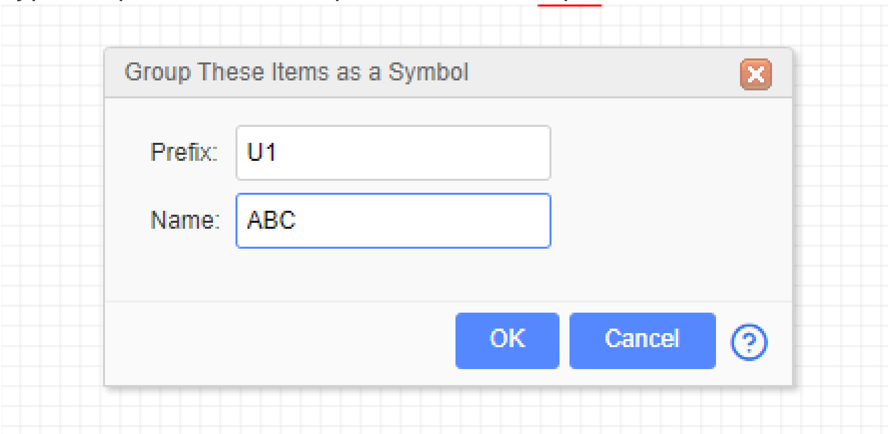
- Place Pins and other objects such as rectangle

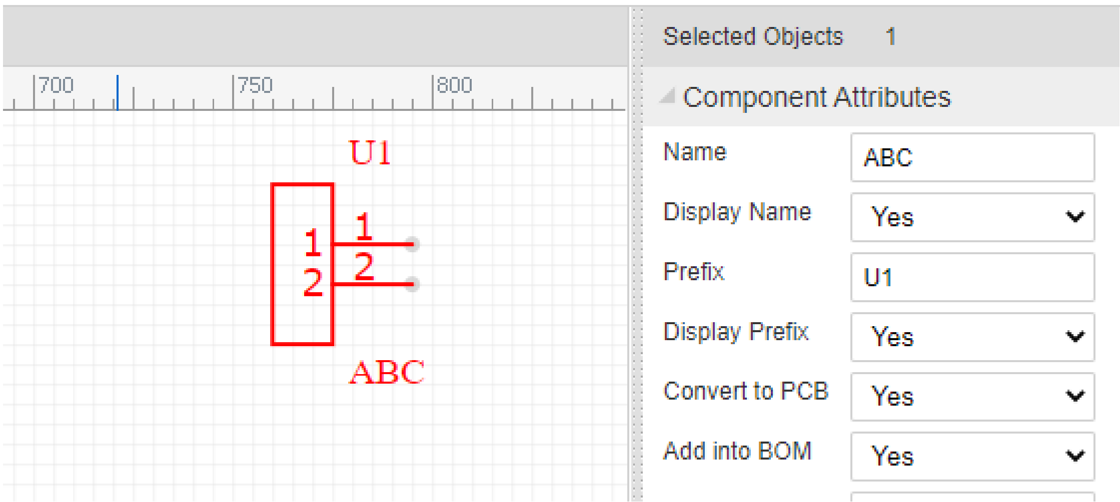


- Select them, and click the "Group/Ungroup Symbol" icon



- Type the prefix and name, press OK, done. A part is created.





So what does Ungroup do? Try selecting a symbol and then click the Group/ungroup command to see what happens!

Note:

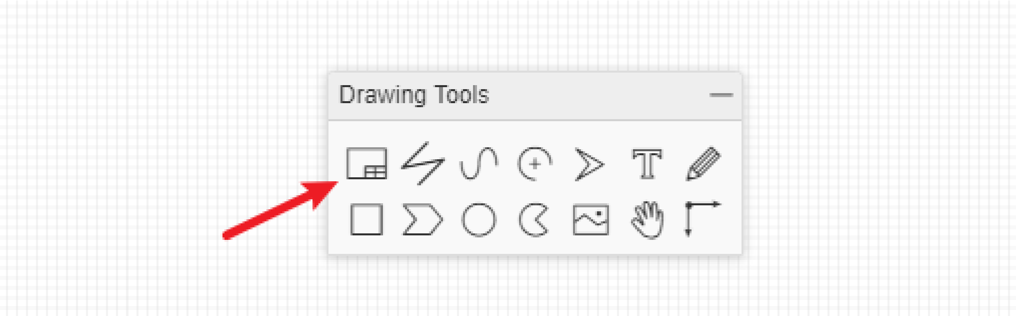
- The symbol you created in the schematic will not be saved in the personal libraries, if you want to use it repeatly, please create a Symbol via: Top Menu - File - New - Symbol.

Drawing Tools

Sheet Setting

It is now possible to add design notes to the frame and the frame selection, for example A4, which can assist in aligning and improve the look of printed schematics and PCB designs.

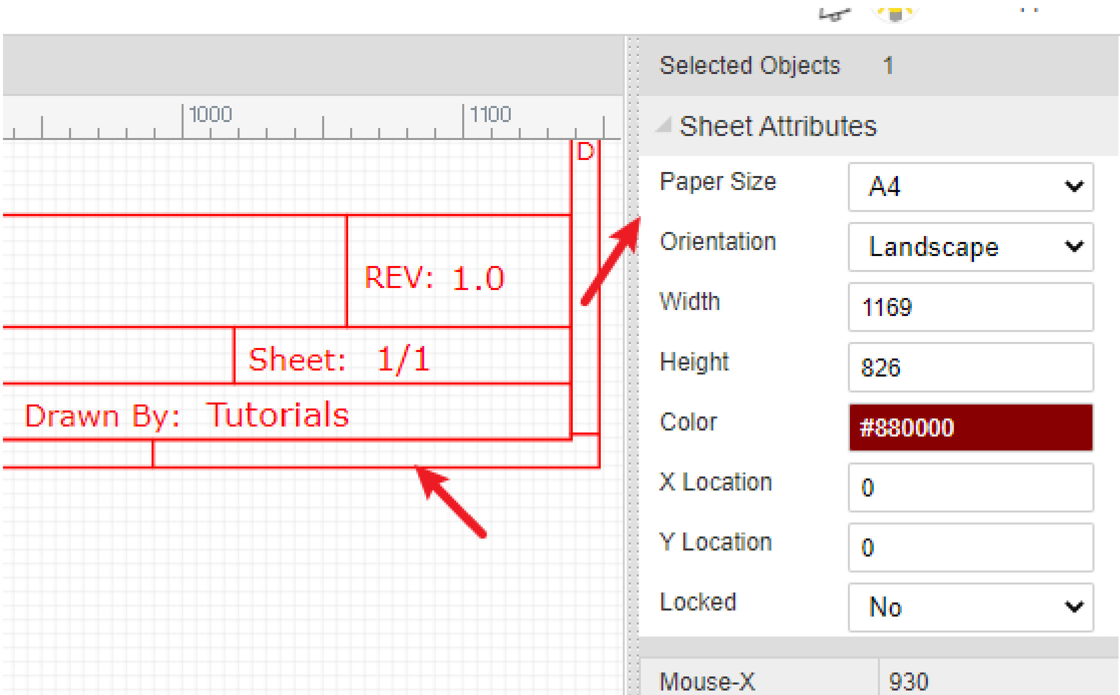
Click the frame/drawing/document button like in the image below:



And you can edit the blue text when you've selected the text attributes or double clicked it.

The bottom right zone can be selected and dragged or the frame can be dragged and deleted.

When you've selected the bottom right zone, you can edit the sheet attributes:

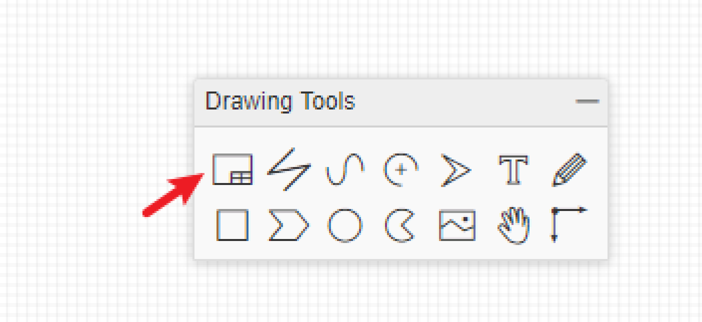


Custom Sheet

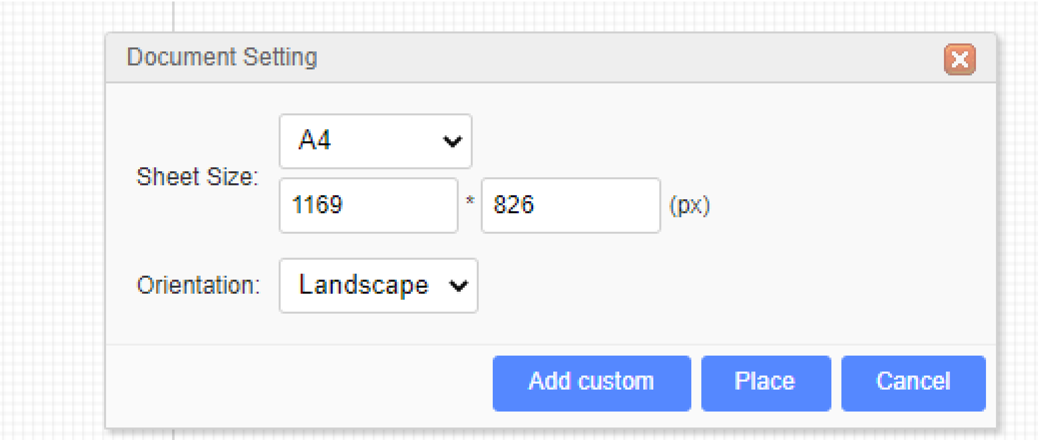
EasyEDA supports the schematic diagram drawing frame required by custom. At present, custom drawings need to be placed manually, and automatic reference of custom drawings is not supported when creating new schematic diagram.

How to create:

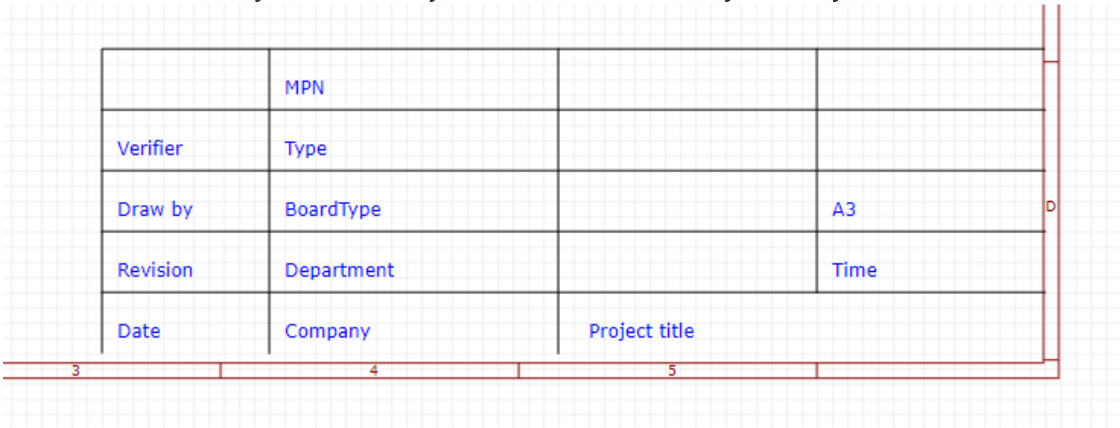
1. Click the "Sheet Setting" button at "Drawing Tool".



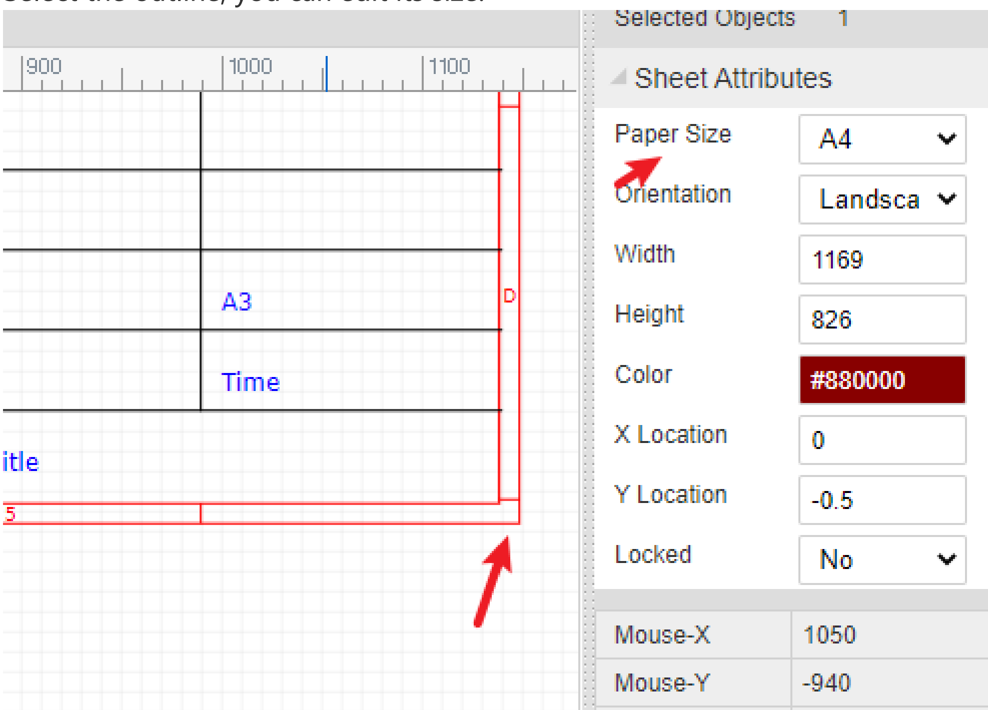
2. Click "Add Custom" button.



3. It will create a new symbol editor, you can edit the table by line as you want, as below:



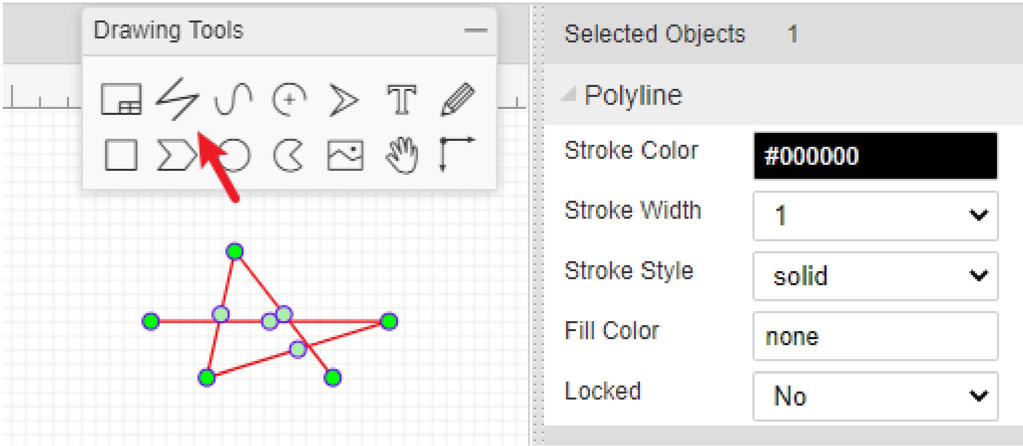
4. Select the outline, you can edit its size.



5. Save it. You can place it in schematic such as a part at "Library".

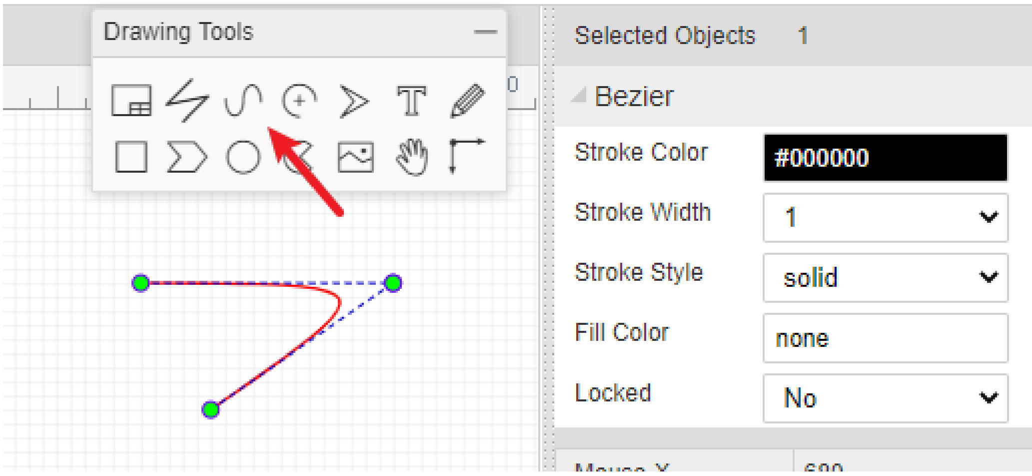
Line

In the Schematic editor, you can draw a line with any direction. You can change its attribute as in the image below:



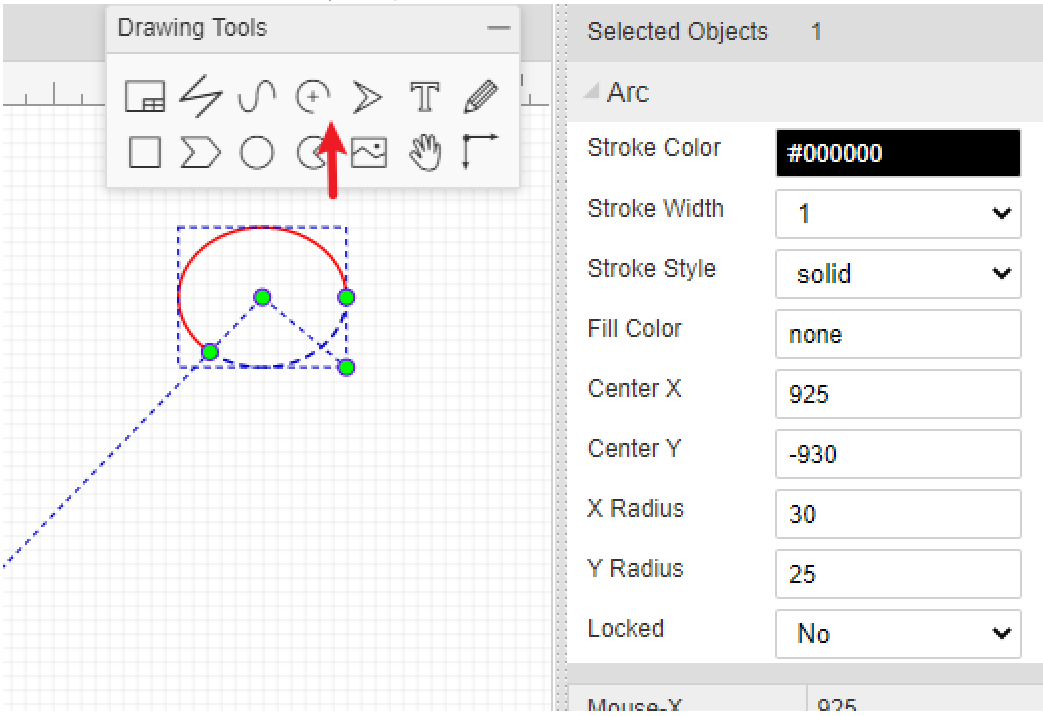
Bezier

With this tool, you can draw a pretty cool pattern.



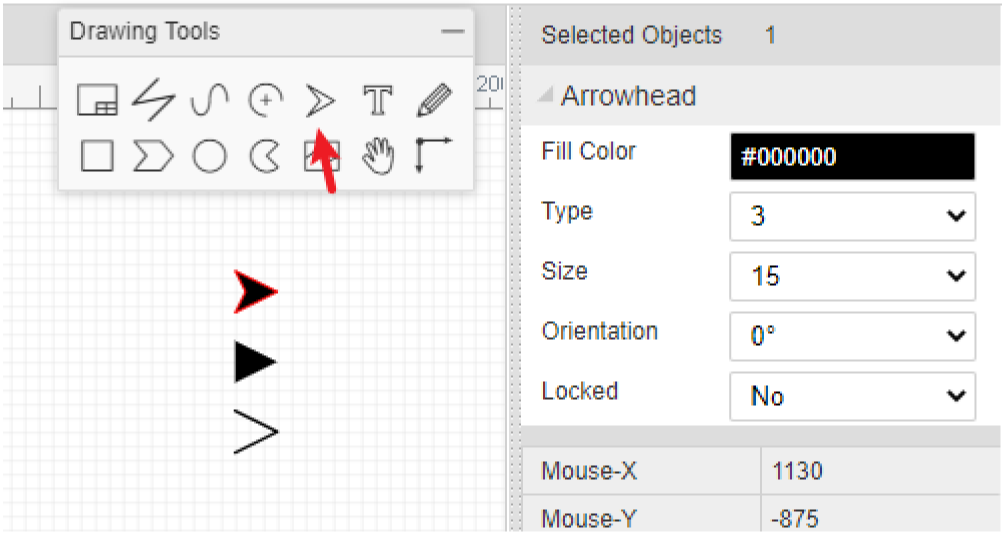
Arc

You can draw the arc of any shape.



Arrow Head

You can add arrow head to marking text or important part.

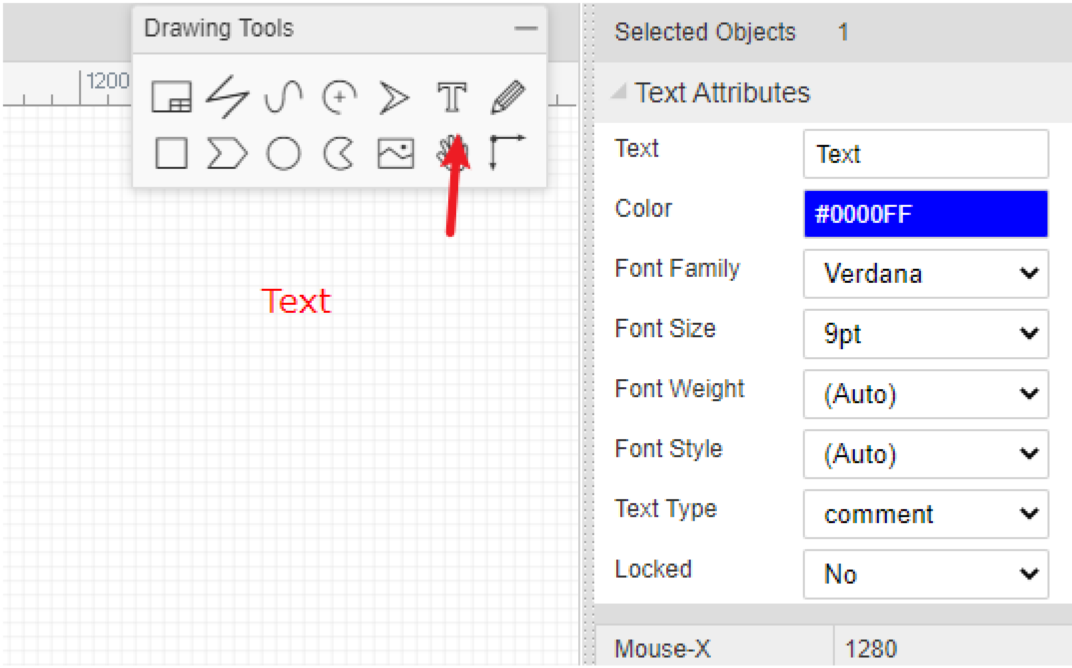


Text

Text attributes provide many parameters for setting:

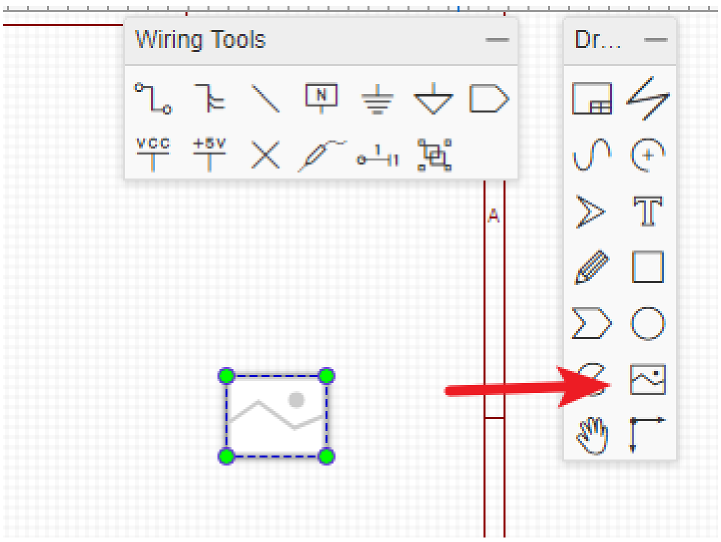
- **Text**: You can change text in inner box or double click the text. For every new text, the default text is `Text`.
  -**Color**: Defines text color.
  -**Font-family**: It provides 12 fonts for choosing.
  -**Font-Size**: Defines Text size.
  -**Font-weight**: Defines Text weight.
  -**Font-Style**: It contains (auto), normal, italic.
  -**Text type**: types include comment and spice.

The editor will remember your last text parameters.

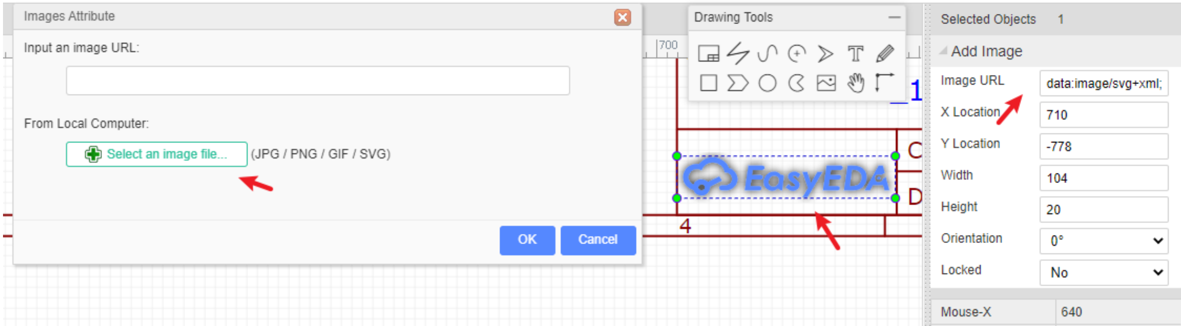


Image

When you select Image from the Drawing Tools palette, an image place holder will be inserted into the canvas:

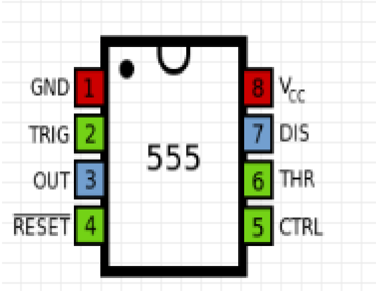


Select the place holder, so you can see the image's attributes in the right hand Properties panel:



Set the URL of your image. For example, setting the URL to:

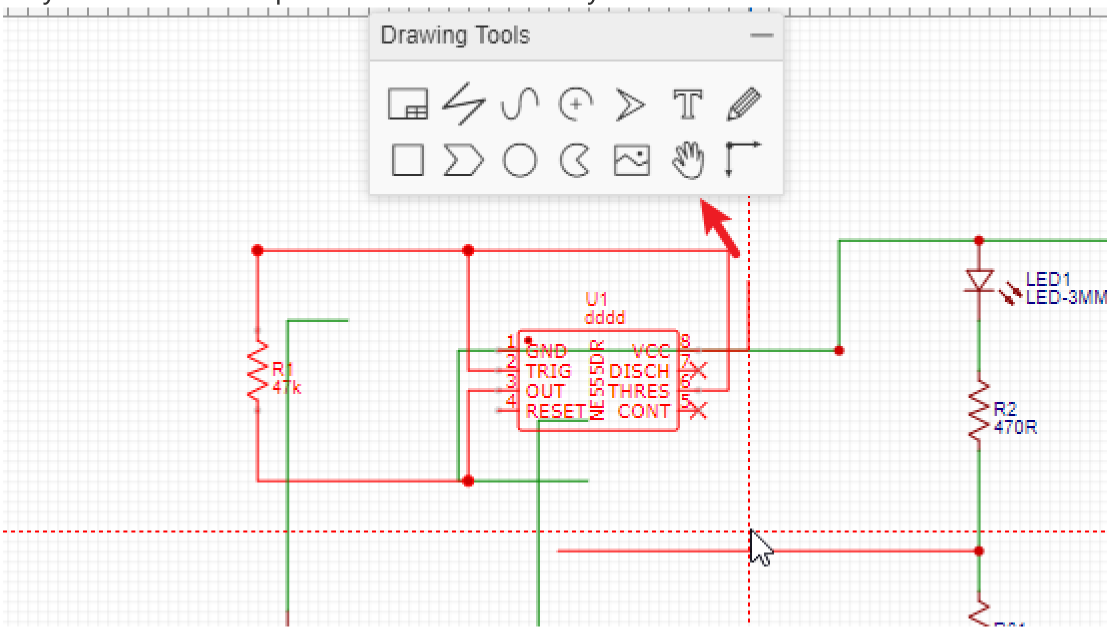
http://upload.wikimedia.org/wikipedia/commons/thumb/c/c7/555_Pinout.svg/220px-555_Pinout.svg.png will make your image look like this:



Please note: at present, EasyEDA cannot host images, so you need to upload your images to an image sharing site.

Drag

If you want to move some kind of parts and wires, you can use drag, hotkey D.
Or you can select the parts and wires area firstly and move them.



Canvas Origin

Canvas origin default is set at left top corner of the schematic sheet, but you can set it where you want via Canvas Origin.

For another way to set canvas origin, you can try **Top Menu> Place > Canvas Origin**.