

EasyEDA Tutorial

2017.09.20

EasyEDA Editor: <https://easyeda.com/editor>

EasyEDA Editor Beta: <https://beta.easyeda.com/editor>



Instruction:

- This document will be updated according to the updated EasyEDA editor.
- The latest edition please refer to <https://easyeda.com/Doc/Tutorial/>.
- The Editor beta version will release the new future and enhancement first, but maybe have some bugs, please using carefully.

Update Record:

Update Date	Editor Version	Description
2017.09.20	v4.9.3 Beta	Update export Altium Designer format description
2017.09.18	v4.9.3 Beta	Update local auto router description
2017.09.08	v4.8.5	First release, Add "Essential Check" section, add "Essential Check Before Placing a PCB Order" of PCBOrder section
NA		
NA		
NA		

Introduction to EasyEDA

What's EasyEDA

Welcome to EasyEDA, a great web based EDA tool for electronics engineers, educators, students, makers and enthusiasts.

There's no need to install any software. Just open EasyEDA in any HTML5 capable, standards compliant web browser.

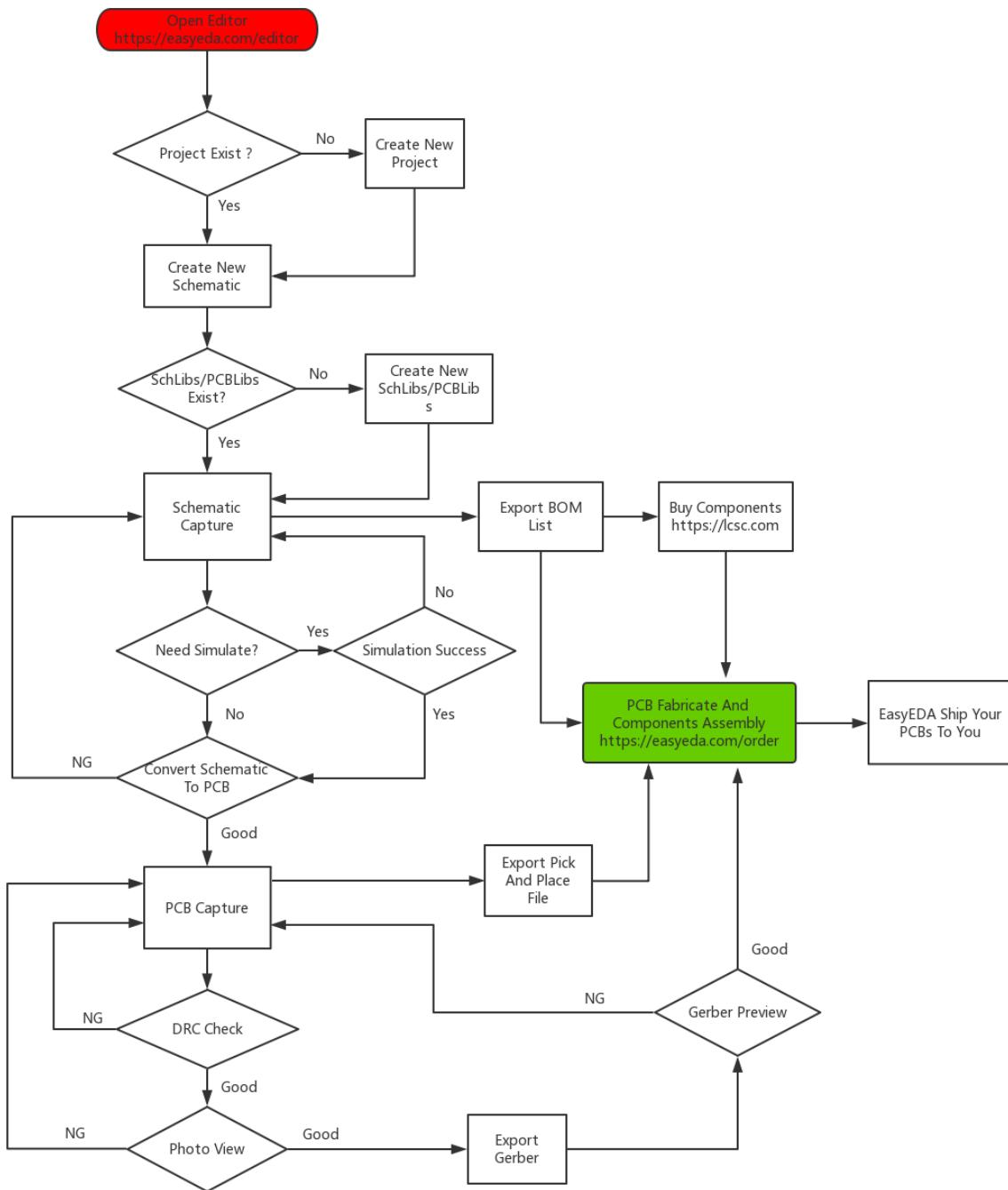
Whether you are using Linux, Mac or Windows; Chrome, Firefox, IE, Opera, or Safari. Highly recommend to use Chrome and Firefox. EasyEDA has all the features you expect and need to rapidly and easily take your design from conception through to production.

EasyEDA provides:

- Simple, Easier, Friendly, and Powerful general drawing capabilities
- Working Anywhere, Anytime, Any Device
- Real-time Team Cooperation
- Sharing Online
- Thousands of open source projects
- Integrated PCB fabrication and components purchase chain
- API provide
- Script support
- Schematic Capture
 - NgSpice-based Simulation
 - Spice models and subcircuits create
 - WaveForm viewer and data export(CSV)
 - Netlist export(Spice, Protel/Altium Designer, Pads, FreePCB)
 - Documentation export(PDF, PNG, SVG)
 - EasyEDA source file export(json)
 - Altium Designer format export
 - BOM export
 - Mutil-sheet and hierarchical schematics
 - Schematic module
 - Theme setting
 - Document recovery
- PCB Layout
 - Design Rules Checking
 - Mutil-Layer
 - Documentation export(PDF, PNG, SVG)
 - EasyEDA source file export(json)
 - Altium Designer format export
 - BOM export
 - Photo view
 - Gerber output
 - Pick and Place File output
 - Auto Router
 - PCB module
 - Document recovery
- Import
 - Altium/ProtelDXP ASCII Schematic/PCB
 - Eagle Schematic/PCB/Libraries
 - LTspice Schematic/Schematic Libraries
 - DXF
- Libraries
 - More than 500,000 Libraries(Symbol and Footprint)
 - Libraries management
 - Symbol/Subpart create and edit
 - Spice symbol/model create and edit
 - Libraries management
 - Footprint create and edit

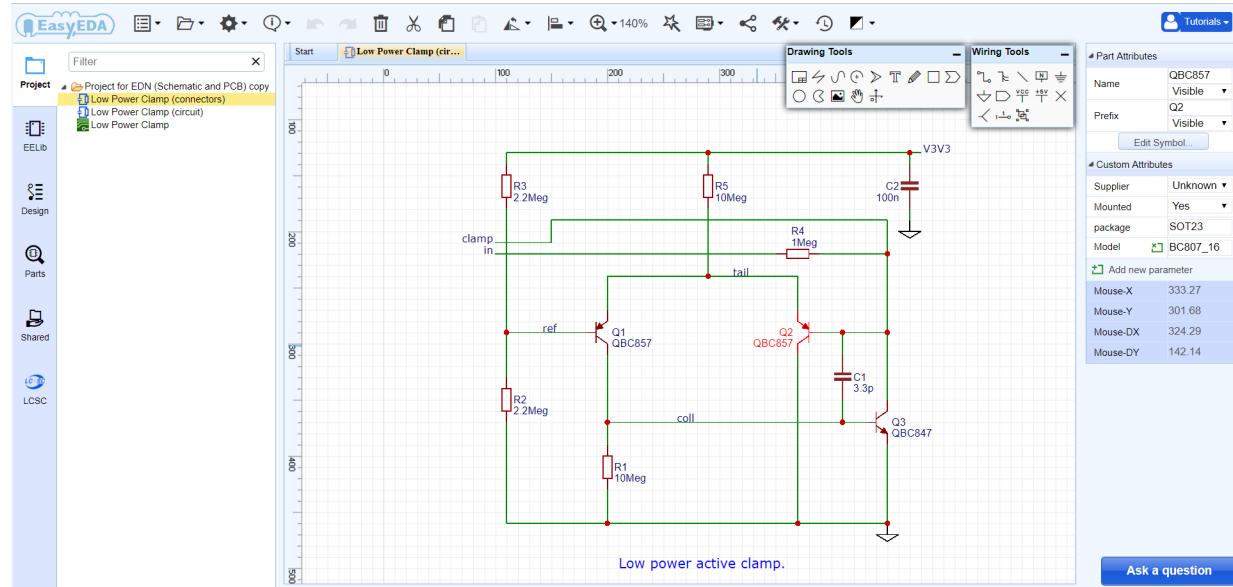
Design Flow By Using EasyEDA

You can create circuits design easily by using EasyEDA. The design flow as below:



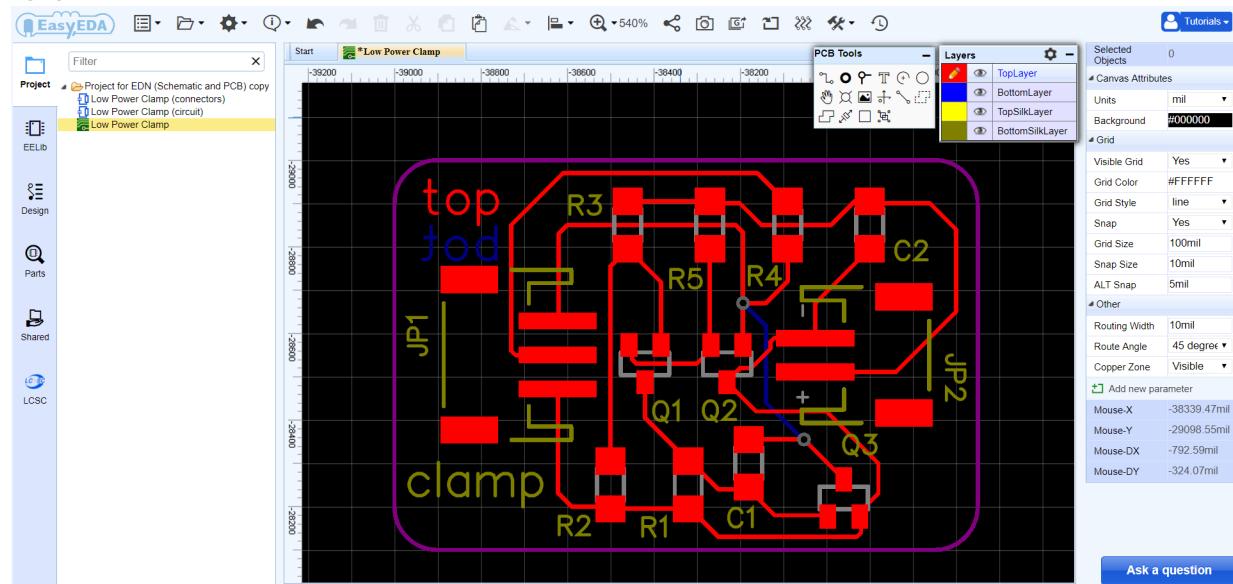
Global UI

Schematic UI



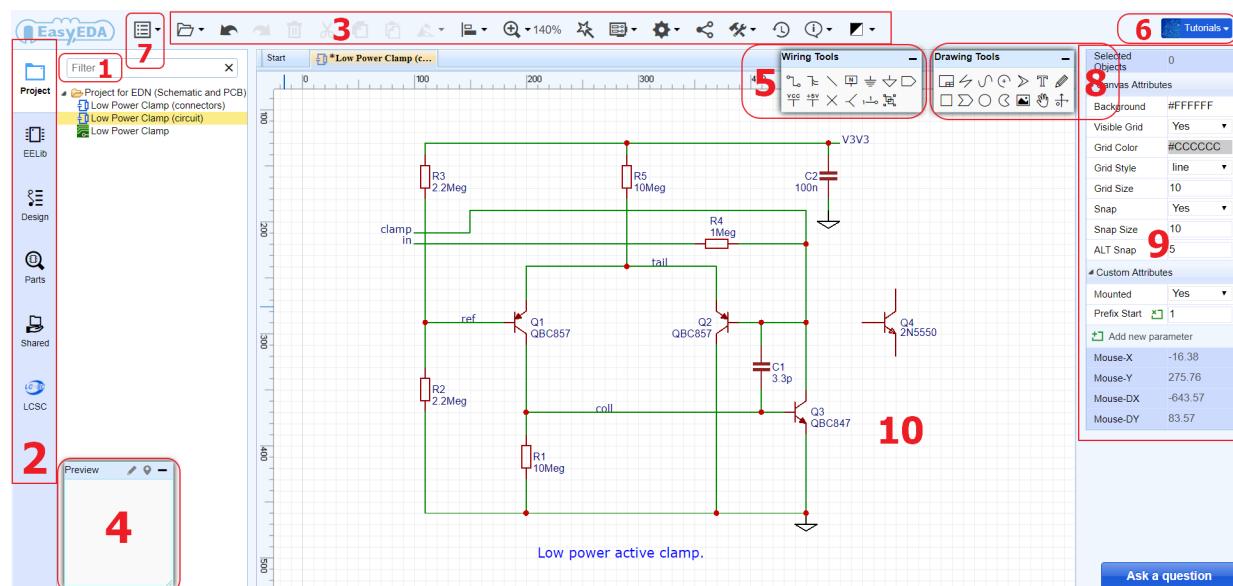
[Ask a question](#)

PCB UI



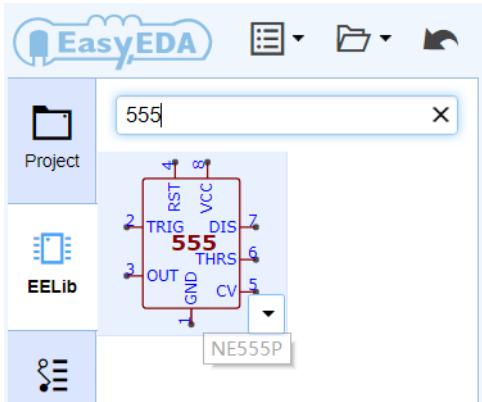
[Ask a question](#)

The Clear EasyEDA UI



1. Filter

Before using the Filter, you need to select what module you need in the left navigation panel, and then you can quickly and easily find projects, files, parts and footprints, just by typing a few letters of the title. For example, if you want to find all files containing "NE555" in the title, just type "555", it is non-case-sensitive.

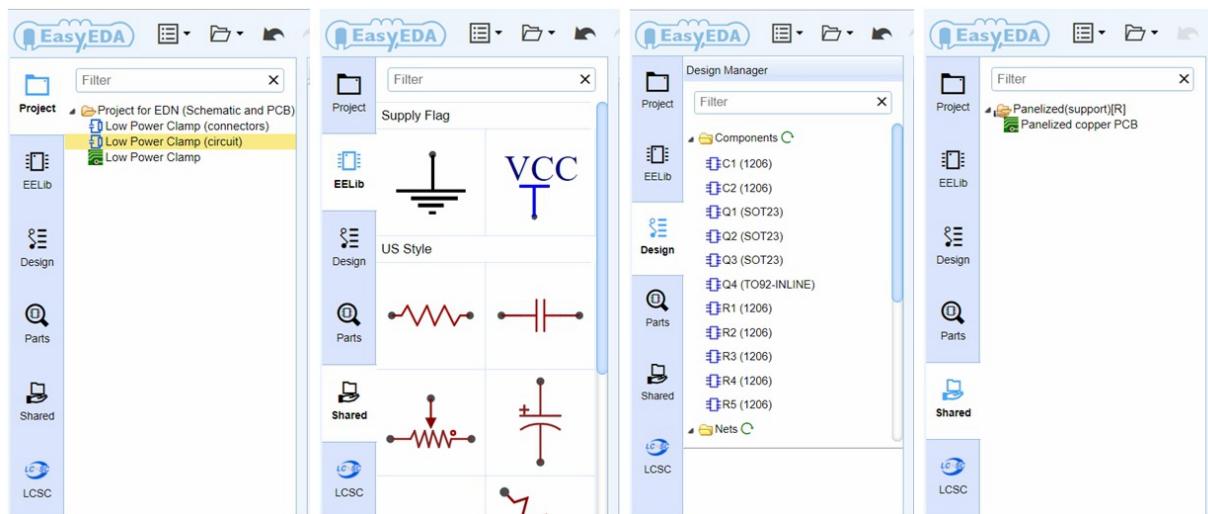


The Filter only searches project, file and part titles and names. It does not search the Description and Content fields.

Click the X to clear the filter.

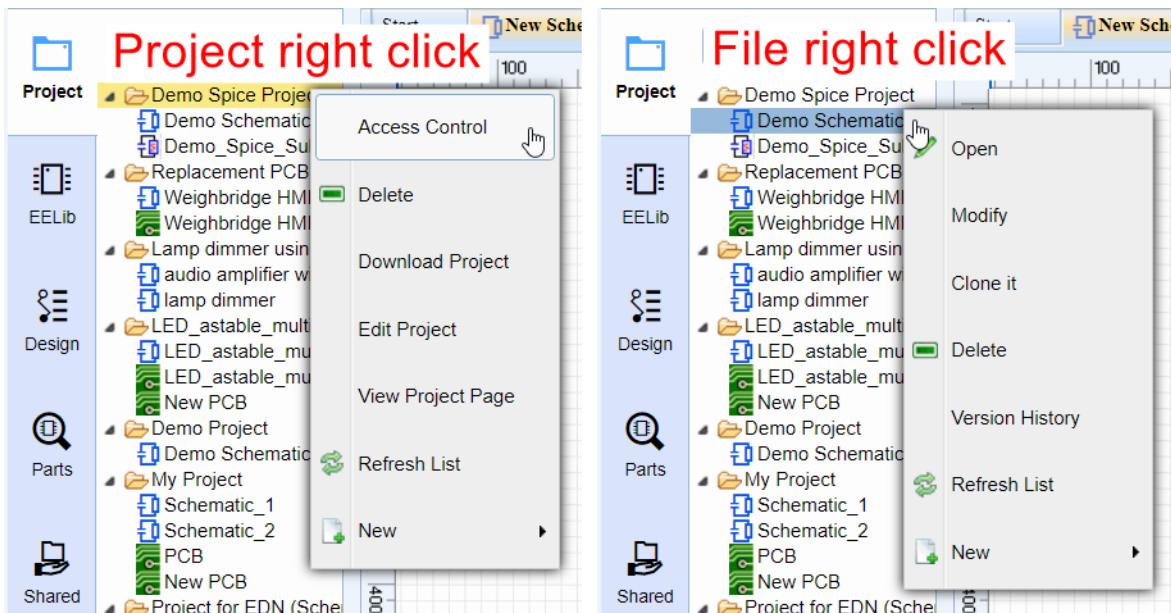
2. Navigation Panel

The Navigation panel is very important for EasyEDA: it is from here that you can find all your projects, files, parts and footprints.



Project Here, You can find all of your projects that are private or shared with the public, or fork from someone else's.

Except for System IC, these options have a content menu. For example, if you drop down to My Projects and right click an item, you will get a tree menu like :



EELib

EELib means EasyEDA Libraries, It provides lots of components complete with simulation models, many of which have been developed for EasyEDA to make your simulation experience easier.

Design

Design Manager, you can check each component and net easily, and it will provide DRC(Design rule check) to help your design better.

Parts

Contains schematic symbols and PCB footprints for many readily available components and projects. And your own libs and modules will show up here.

Shared

About this module, if your partner shared his/her private project with you by using the **Access Control** option, then the project will show here.

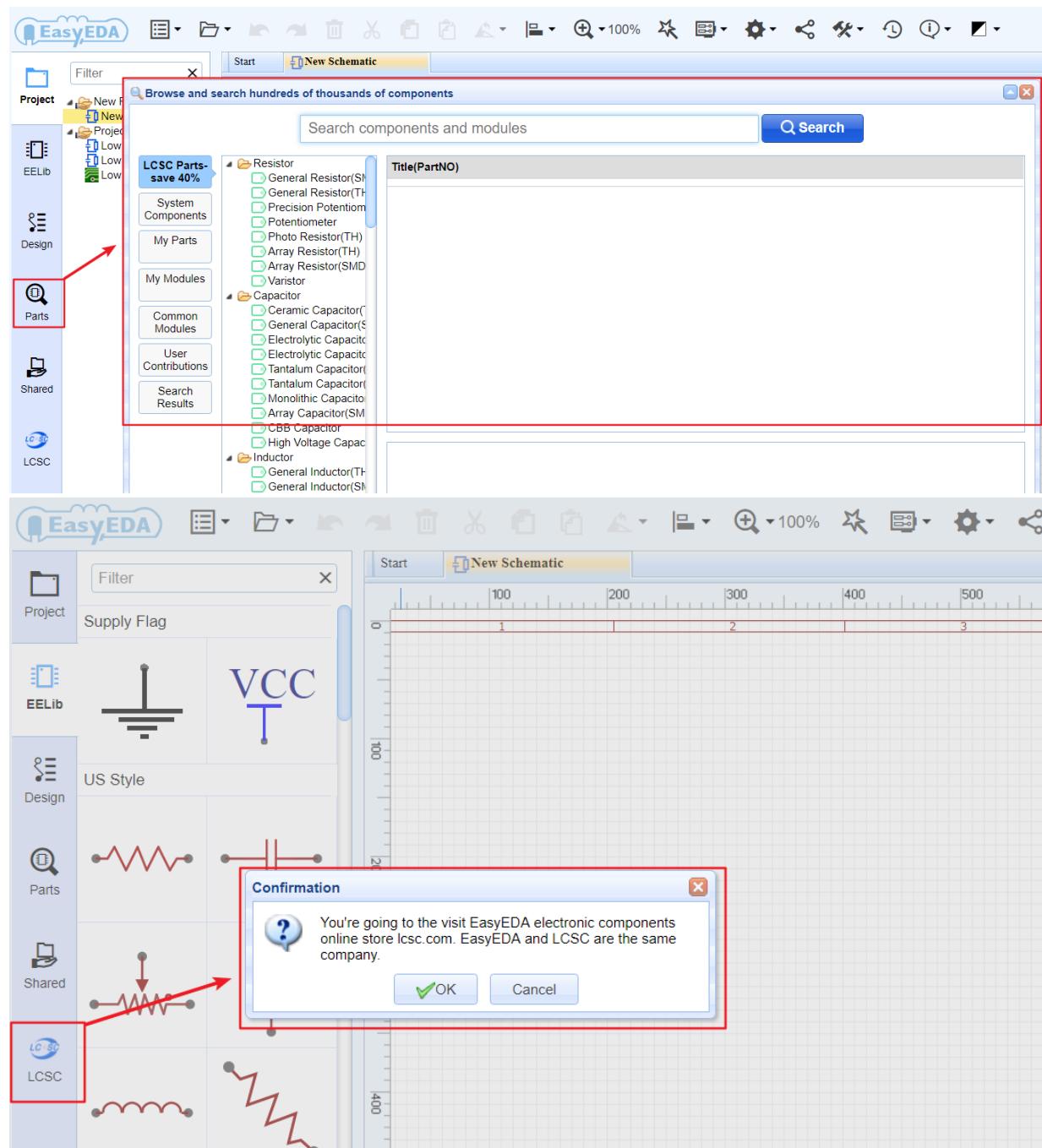
For more information you can refer to the [Access Control](#) section.

LCSC

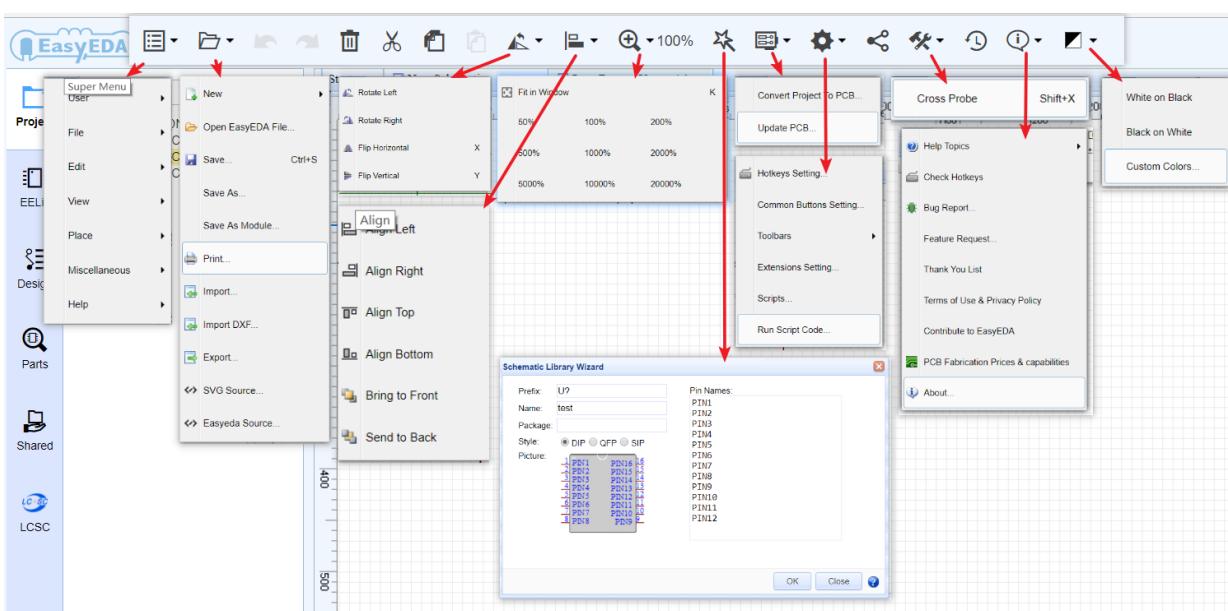
If you want to buy components to finish your PCBA, you should try the **LCSC** module, LCSC.com and EasyEDA are the same company.

EasyEDA partners with China's largest electronic components online store by customers and ordering quantity launch <https://lcsc.com>. LCSC means Love Components? Save Cost! We suggest to our users to use LCSC parts to design. Why?

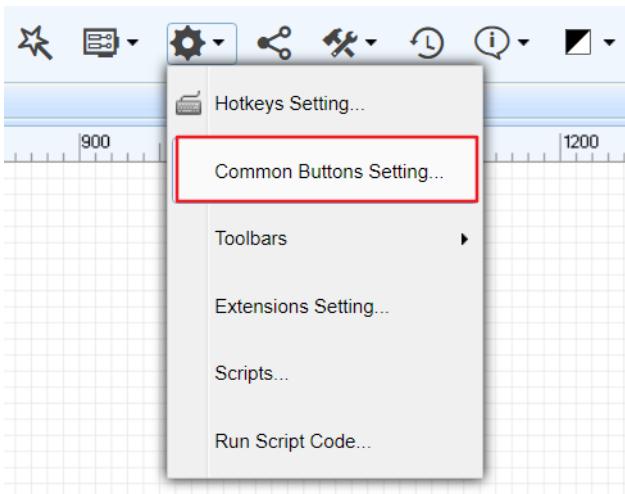
- 1. Small Quantity & Global Shipping.
- 2. More Than 25,000 Kinds of Components.
- 3. All components are genuine.
- 4. It is easy to order co after design.
- 5. You can save 40% cost at least.
- 6. You can use our symbols and package.



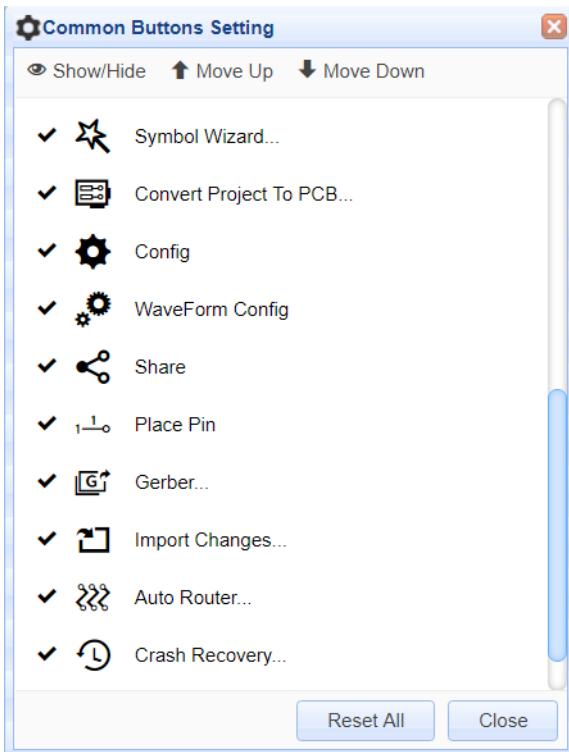
3. ToolBar



EasyEDA's toolbar can be reconfigured via Common Buttons Setting...



The configure dialog is also easy to use:



Click on a button to select it. Then you can toggle button visibility by clicking on Show/Hide or by clicking on the tick space to the left of the button icon. You can change the button position using Move up and Move Down.

Many of the buttons have been assigned hotkeys, so you can use those to replace the button actions.

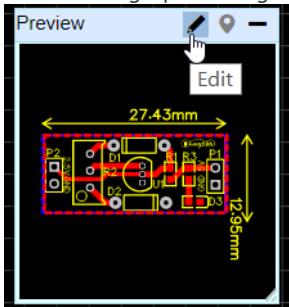
4. Preview Dialog

The Preview dialog will help you choose components and packages and can help you to identify schematics and PCB layouts.

You can close or open this dialog via:

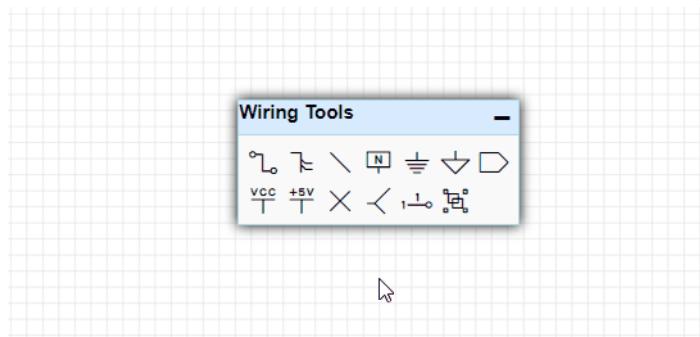
Super Menu > View > Toolbars > Preview or on the top toolbar **Config Icon** > Toolbars > Preview.

- The Preview Dialog has a resizing handle in the bottom right corner.
- The Preview Dialog can't be closed but double clicking on the top banner will roll up the panel or you can click the top right corner . Double clicking top banner again toggles it back to the selected size.



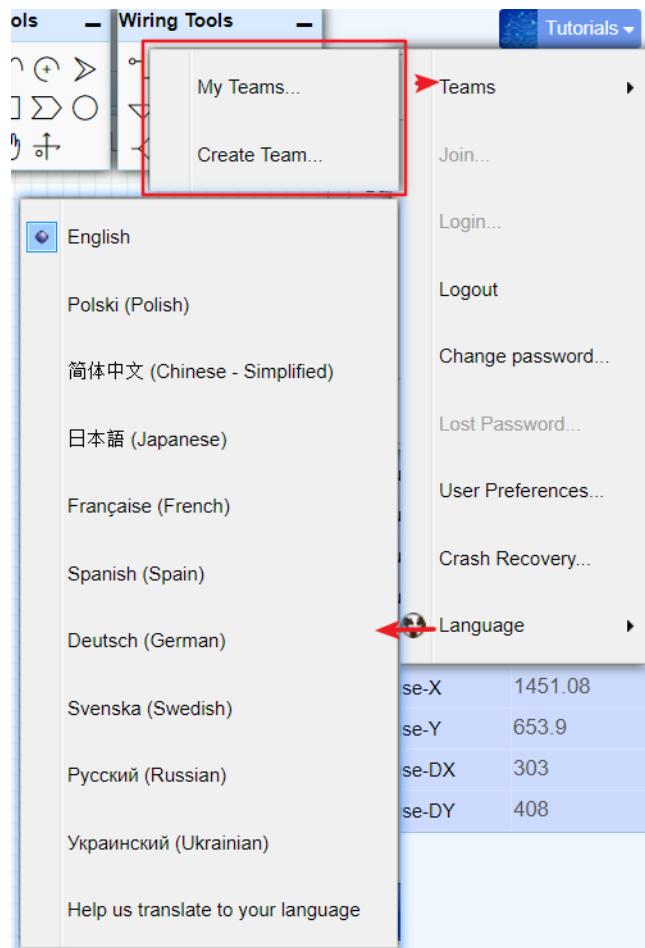
- Clicking on the little pencil edit tool opens the item in the preview for editing. Clicking on the location place tool in the top right corner of the preview dialog places the item onto the canvas. If you try to place PCB footprint into a schematic it will not provide any action and message.

5. Wiring Tools



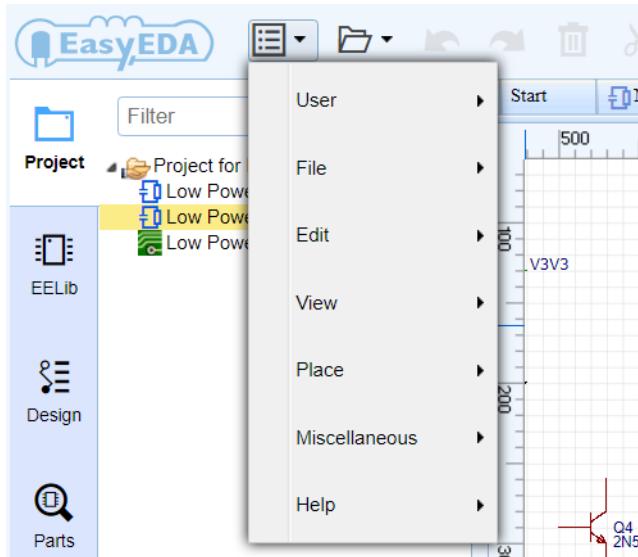
Wiring Tools are document type sensitive: different document types have different tools.

6. User management menu

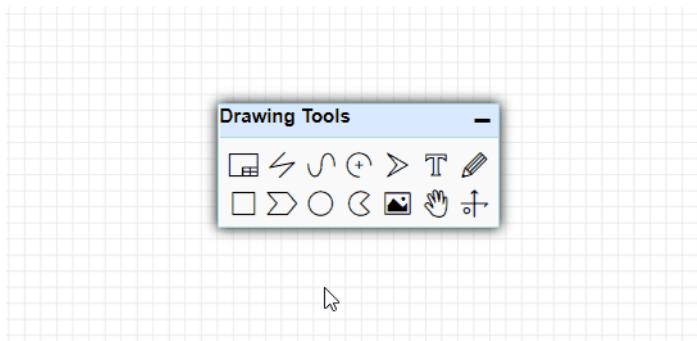


7. Super menu

All EasyEDA's menus can be found here. Most of the time, we hope you can access these options via the Hotkeys or from the top toolbar but if you find that you use some of the more specialized options from this menu frequently then may want to set them as your own hotkeys.



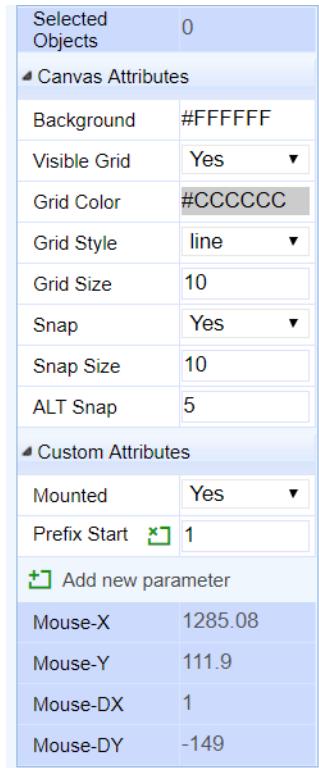
8. Drawing Tools



To keep EasyEDA's UI clean and sharp, the Wiring and Drawing tools palettes can be resized horizontally, rolled up or hidden so if you want to focus on drawing, you can roll up or hide the others to make more space and reduce the clutter.

9. Canvas Attributes

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



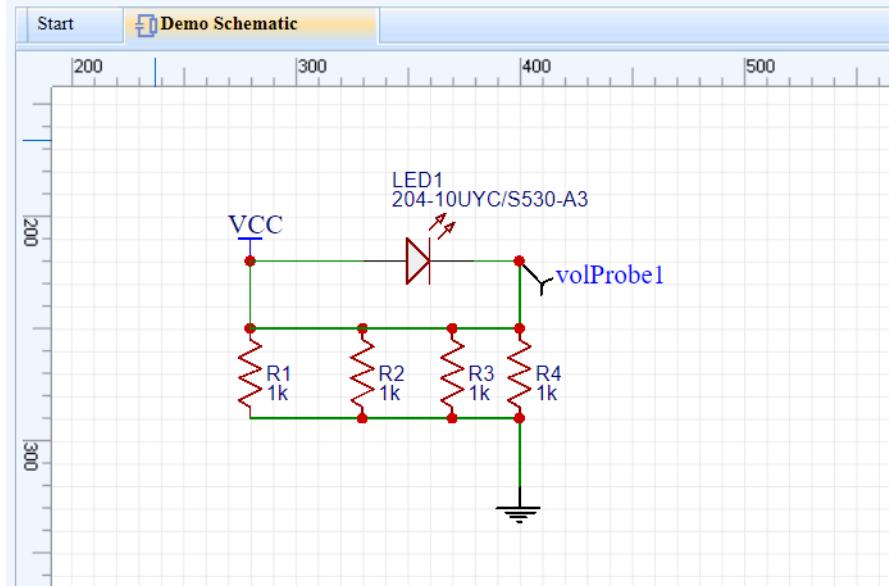
Background and grid colors and the style, size, visibility and snap attributes of the grid can all be configured.

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

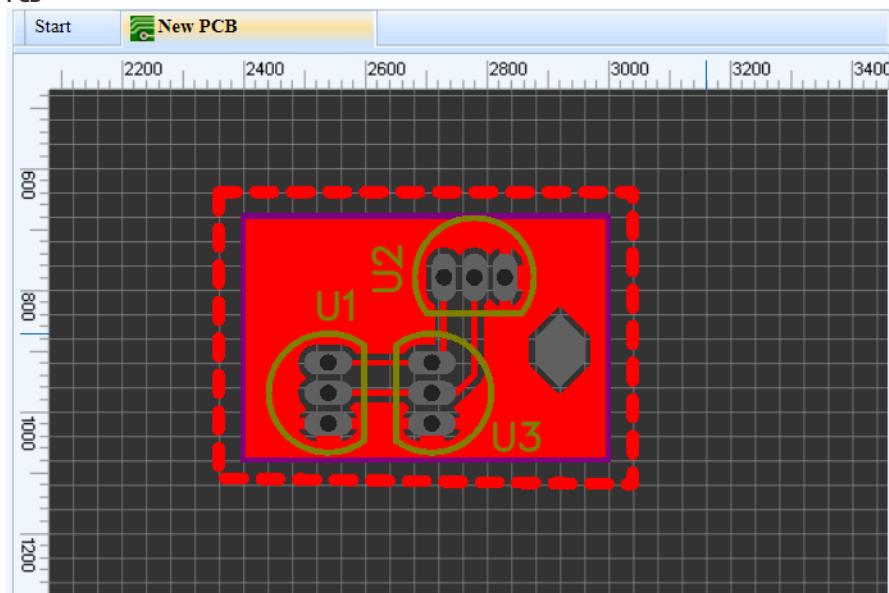
10. Canvas

This is where it all happens! This the area where you create and edit your schematics, PCB layouts, symbols, footprints and other drawings, run simulations and display WaveForm traces.

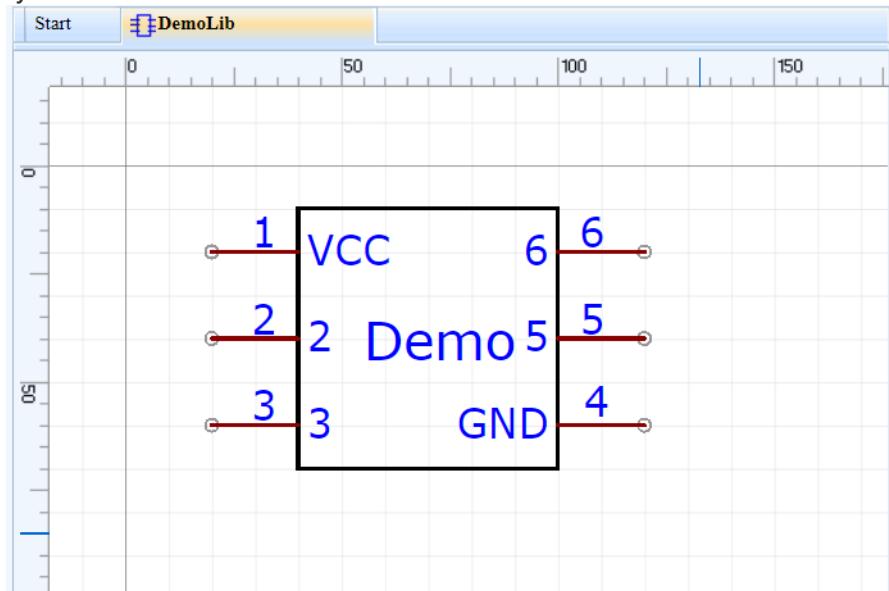
Schematic



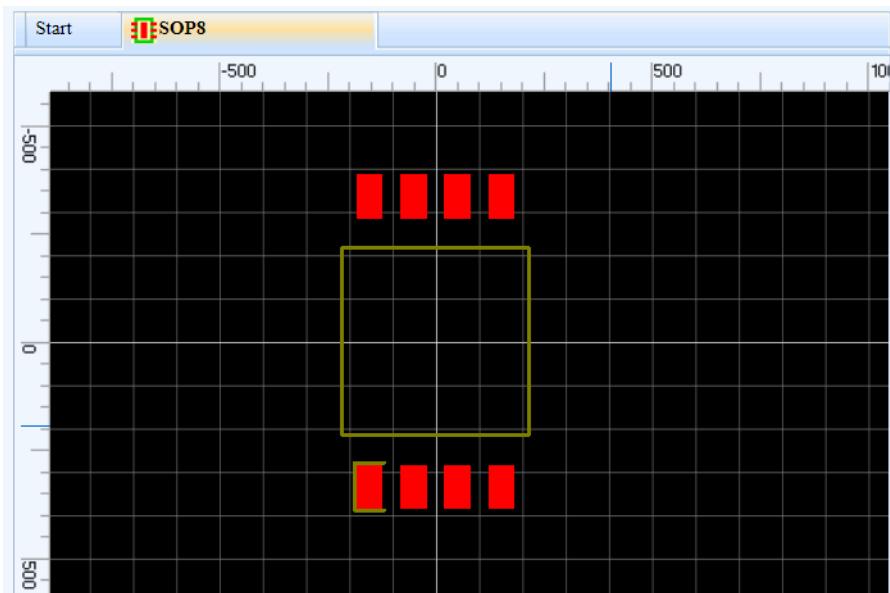
PCB



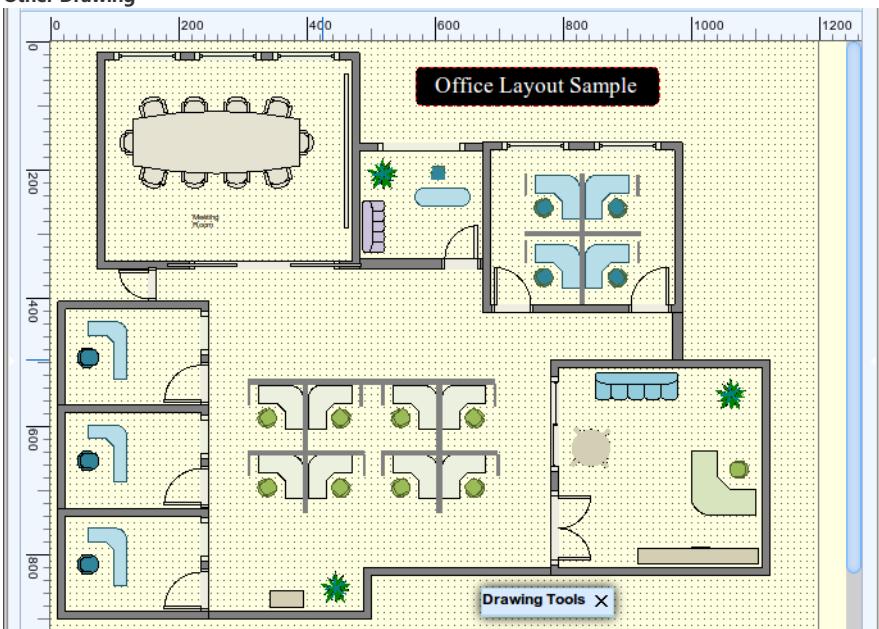
Symbols



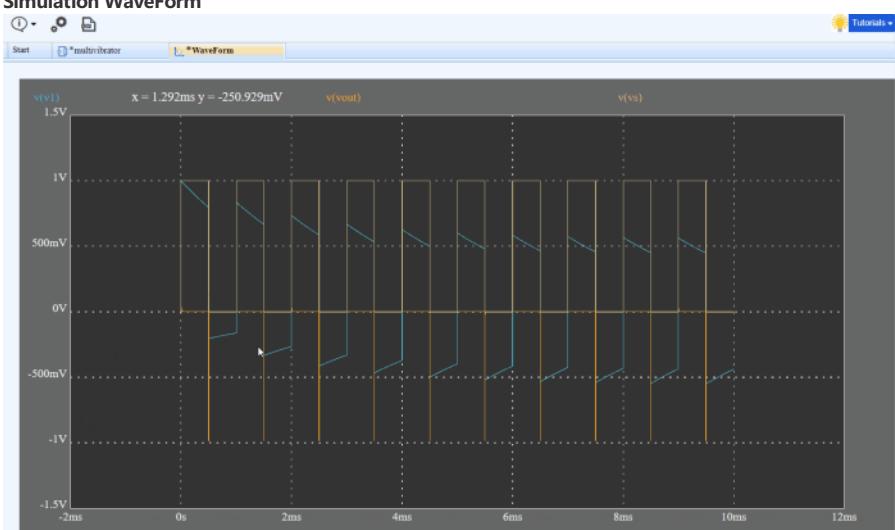
Footprints



Other Drawing

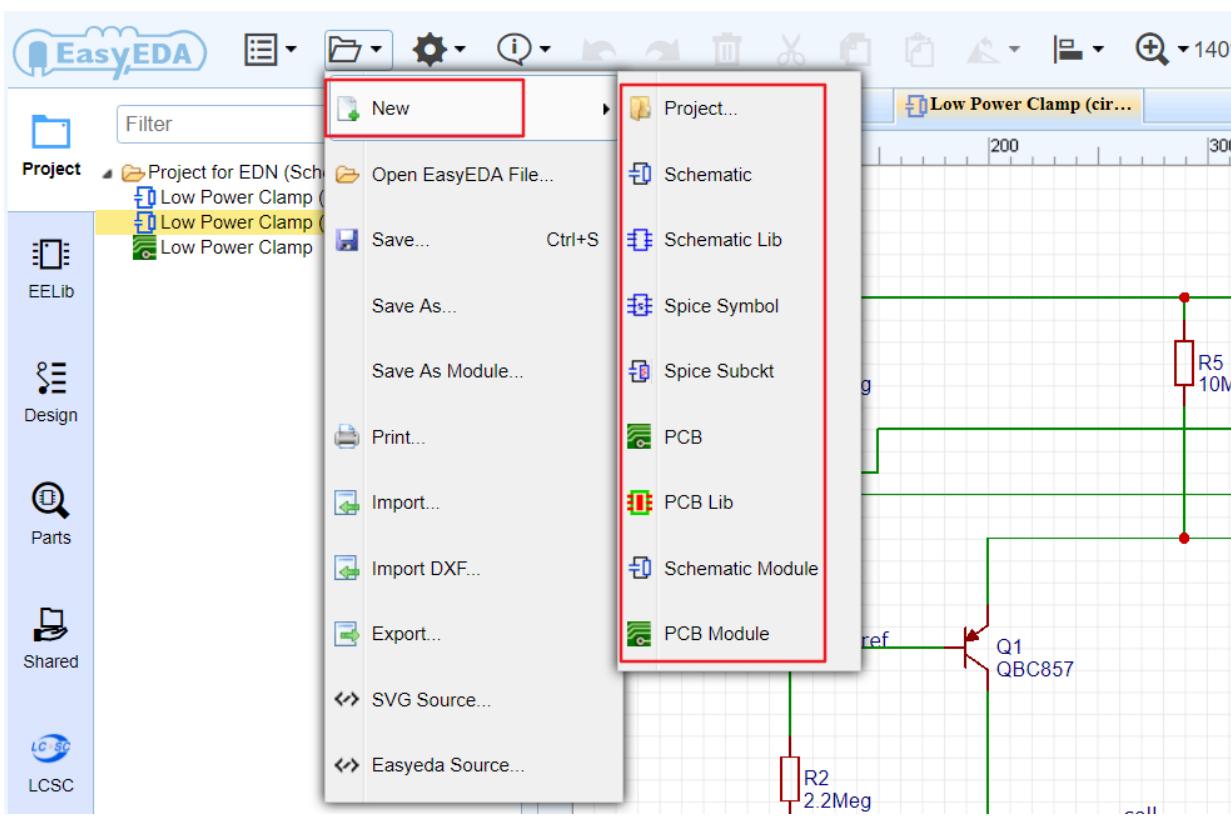


Simulation WaveForm

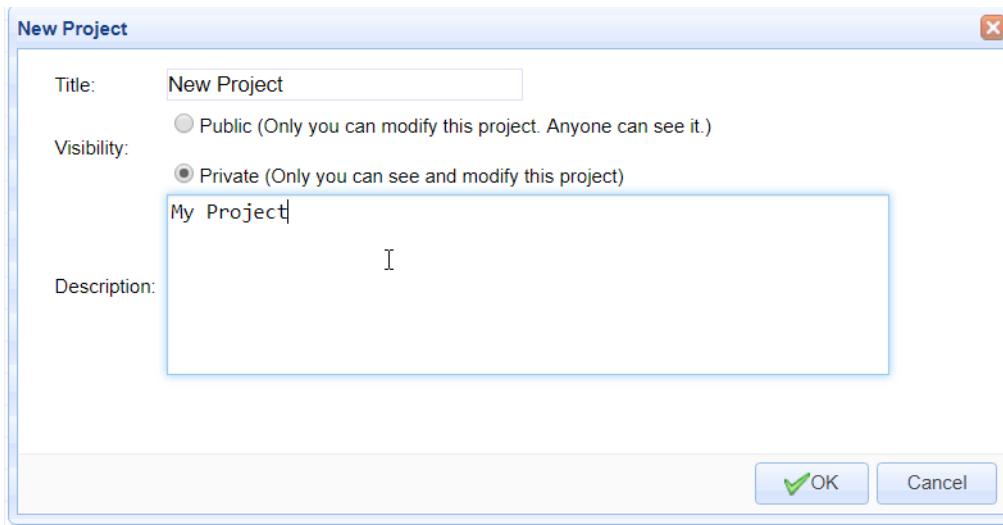


How to Create A New Project or File

After logging in, you can create a new project: Document > New > Project... > Create a new project/Schematic..etc



The Project concept is important in EasyEDA because it is the foundation of how to organise your designs.



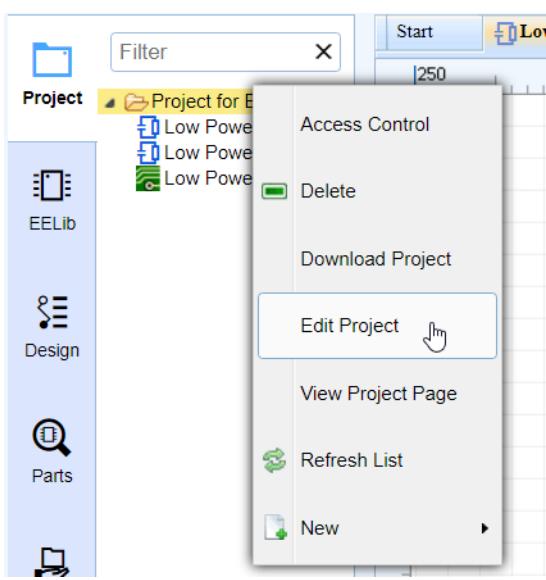
Give it a title: this will show in the project tree in the left hand panel.

You can make your project public or private by setting its Visibility.

If you choose to make your project Public, Categories allows you to select which category you want your project to be listed under on our website. If you keep your project private then the category is still applied but has no direct use in sorting your projects because this field is not searched in the Filter box in the left hand panel.

Adding a short description helps you and anyone you are sharing this project with understand what the project is about.

Once created, to modify your project, right click on it in the project tree in the left hand panel,



then will open a web page in which you can edit your project:

just test

Tags

Add Tag

Attachments

Upload Files Or drag and drop files here

Accept jpg,gif,png,jpeg,zip,rar,tar,gz,7z,doc,docx,txt,xml files. Maximum size is 8MB per file.

Public

Allow other people to see this project

Allow Comment

Allow other people to comment

License

Unknown License

Status

Not Set

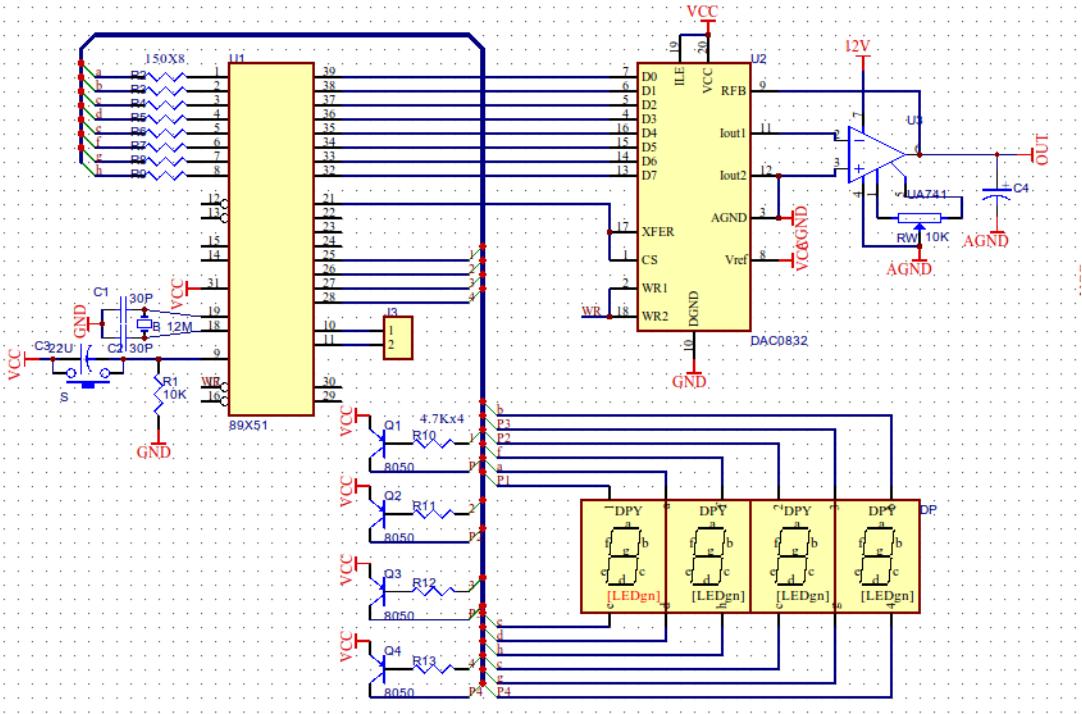
Save

From here, you can change the Visibility, allow other people to comment on your project and type a more detailed description of the project content. To help you make your project stand out or to maybe simply make a detailed description of your project easier to read, you can use Markdown syntax. If you need more information on Markdown syntax, click on [Markdown Syntax?](#) just above the Content box.

Function introduction

Schematics

EasyEDA can create highly professional looking schematics.

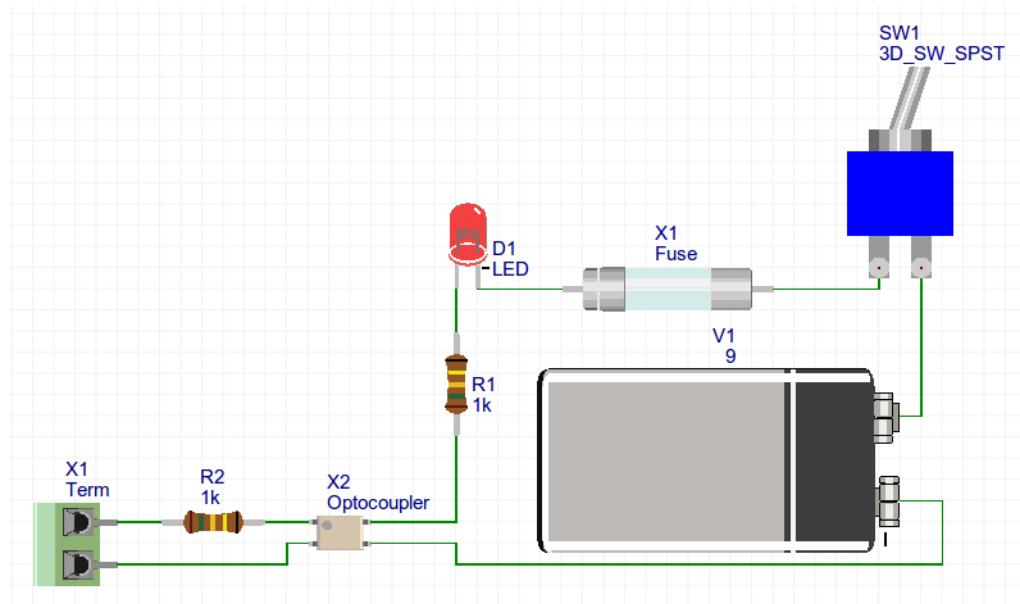


Because EasyEDA has some simple but powerful drawing capabilities, you can create your own symbols either by copying existing symbols into your own library and then editing and saving them, or by drawing them from scratch.

There is also a **Symbol Wizard** to quickly draw new symbols for **DIP**, **QFP** and **SIP**.

A feature of EasyEDA is that as well as extensive libraries of the usual simple "2D" graphical schematic symbols, it has a library of drawn **3D** component symbols, i.e. symbols that look like the physical components that they represent.

If you have enough time and patience using the drawing features to full effect in symbol creation, your schematic can be built like this:



Another powerful feature is that it is also possible to import symbols from [Kicad](#), [Eagle](#) and [Altium](#) libraries.

Libraries management

Thanks to the Free and Open Source Kicad Libs and some Open Source Eagle libs, EasyEDA now has 100,000+ components, which should be enough for most projects.

Now you can enjoy using EasyEDA without having to spend so much time hunting for or building schematic symbols and PCB footprints.

Search symbols

On the left hand Navigation panel you will find "**EElib**" and "**Parts**", just type what components you want and search.

Create symbols

EasyEDA supports creating symbols by yourself, after created you can find out your components at **Parts > My parts**, and it is easy to manage your parts.

To prepare for the final assembly stage you can create a Bill of Materials (**BOM**) via:

Super Menu > Miscellaneous > BOM Report...

and you can produce professional quality **SVG**, **.PNG** or **.PDF** output files for your documents.

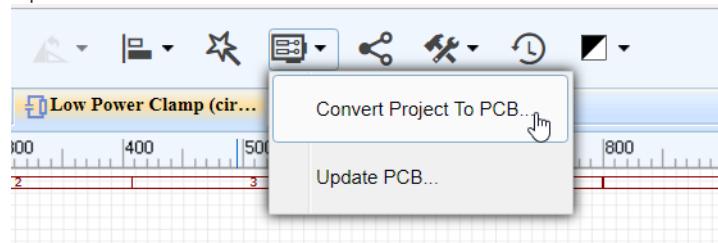
All EasyEDA Schematic Symbol and PCB Footprint libs are public, so after you have created and saved a new symbol or footprint, others will be able to find your part and you will be credited as a contributor. <https://easyeda.com/page/contribute>

PCB Design

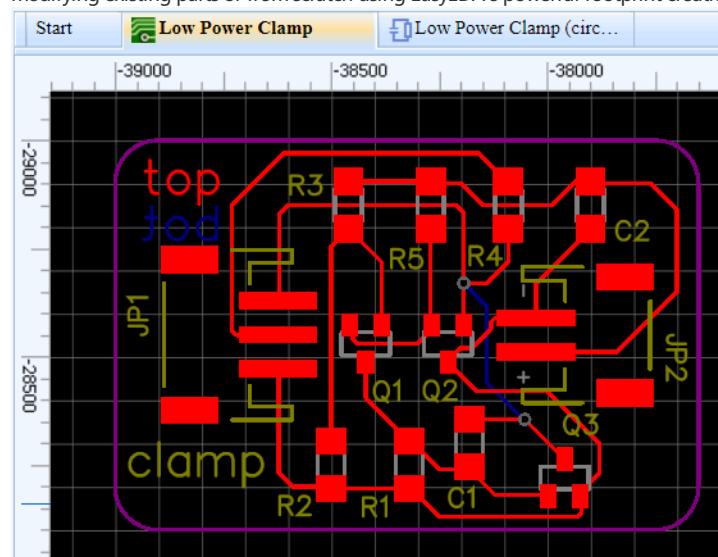
When you are satisfied with your schematic design and simulation results, you can then quickly proceed to produce your finished and populated PCB without leaving EasyEDA.

EasyEDA's PCB Design canvas helps you to quickly and easily lay out even complex multilayer designs from schematics you have already created in the Schematic canvas or directly as a layout with no schematic.

Passing an EasyEDA Schematic into the PCB Design editor is as easy as clicking a button: Just click the **Convert Project to PCB** PCB icon on the top toolbar!



EasyEDA has extensive libraries of footprints. You can also build up your own library of unusual and specialized parts by copying and modifying existing parts or from scratch using EasyEDA's powerful footprint creation and editing tools.



In a similar way as in the Schematic design canvas, to help you locate items and navigate your way around when working in the PCB Design canvas there is a PCB Design Manager.

Left Navigation Panel > Design

The PCB Design Manager is a very powerful tool for finding components, tracks (nets) and pads (Net Pads).

Clicking on any item highlights the component and pans it to the center of the window.

You can set up layers used in the PCB and their display colours and visibility using

Super Menu > Miscellaneous > Layer Options...

The active layer and layer visibility can be selected using the Layers Toolbar.

Default track widths, clearances and via hole dimensions can all be configured in the Design Rule Check dialog which is opened via:

Super Menu > Miscellaneous > Design Rule Setting...

From first setting up the Design Rule Check (**DRC**) at the start of your board layout, running a DRC is almost the last step in checking your PCB design before you generate **Gerber** and **Drill** files for board manufacture ready to place your order for a finished PCB.

The last step is to check the Gerber and Drill files using an easy to install and use Free and Open Source Software Gerber Viewer: [Gerbv](http://gerbv.geda-project.org/): <http://gerbv.geda-project.org/>

While you are waiting for your PCB to be delivered, you can create a Bill of Materials (BOM) via:

Super Menu > Miscellaneous > BOM Report...

and you can produce professional quality **.svg**, **.png** or **.pdf** output files for your documentation.

PCB Designs can be shared with colleagues and made public in the same way as Schematics.

The size of PCB that you can produce using EasyEDA is almost unlimited: designs of over 100cm * 100cm are possible ... but you might need a powerful computer for that.

EasyEDA supports up to 6 layer PCBs by default but it is capable of handling more, so if you need more layers then please contact us as shown in the section on [How to get Help?](#).

Search footprints

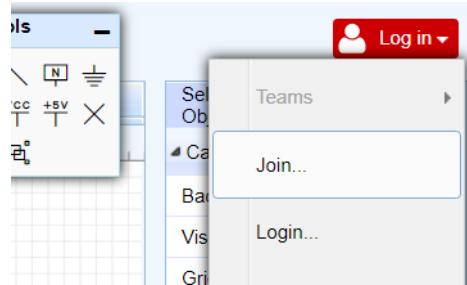
Searching footprints is the same as searching symbols by using **Parts** in the Schematic. You can place the selected footprints in the canvas after the search.

Account Management

EasyEDA is a web-based service and although you are free to use it in Anonymous mode which you can do without creating an account, you are much better off creating an account to manage your own designs and parts libraries. Creating an account is easy and gives you free access to the full power of EasyEDA for as long as you wish.

Join

Click **Join...** on the **User Management** menu:



After clicking on **Join**, a new webpage about **Create an account** opens.

One Account. One Stop Electronic Engineering Services

The image shows two side-by-side login forms. The left form is titled 'LOGIN' and has fields for 'Username or Email' and 'Password', a 'Remember Me' checkbox, and a 'LOG IN' button. It also includes social login options for Google+ and QQ, and a note about QQ login. The right form is titled 'REGISTER' and has fields for 'Username', 'Email', and 'Password', a 'I agree to the Terms of Service' checkbox, and a 'REGISTER' button. It also includes a social login option for Google+.

LOGIN

Sign in with your EasyEDA Account

Remember Me [Forgot Your Password?](#)

LOG IN

OR

Note: QQ login only applies to users who have logged in EasyEDA with QQ before.

REGISTER

Once you create your account you can use it to log in to any EasyEDA Services site.

I agree to the [Terms of Service](#)

REGISTER

OR

Just enter a username, invent a password, confirm it and type in an email address. A valid email address is needed so that we can send you a confirmation email before we create your account. This is also the address we will use to contact you with information or any questions about your PCB orders.

Login

The Login dialog image can be seen in the Join section above.

After clicking on Login, you can enter the username or email and password to login to EasyEDA. If you use a private device, you can check **Remember Me**, so you don't need to login again each time you open EasyEDA.

Alternatively, if you have a Google or Tencent QQ account, you can login in using <http://en.wikipedia.org/wiki/OpenID>; it is safe and easy.

Note: QQ login only applies to users who have logged in EasyEDA with QQ before.

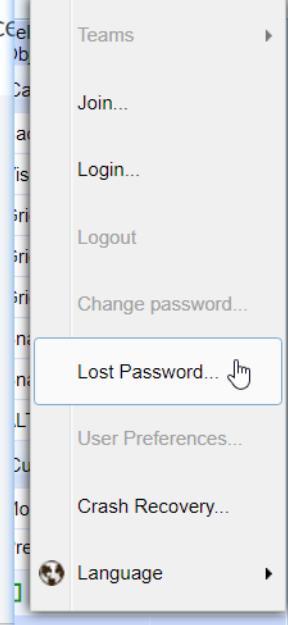
Reset Your Password

Your password is encrypted, so EasyEDA team doesn't know it, but you can reset it via your email. Click the **Lost Password** menu and type your email. If you can't get the email after 10 minutes, please send an email to us.

[Home](#)[Resources](#)[Log in ▾](#)

Forgot Password

Email *

[Submit](#)

Teams

EasyEDA provides a team feature with which you can work seamlessly with your partners. You can work as if everyone is logged in under the same account, with full access to all components, Schematics, PCBs and Projects.

How to find the team function

Under the [dashboard](#), there is a team section.

Home > Tutorials > Create Team

Create Team

Team name *

How to use team function.

Submit

How to create a team

There is a link as shown in the image above, or click <https://easyeda.com/teams/create> after you login, Or you can click **User Management > Teams > Create Team** to open this link to create a team.

Tutorials

My Teams...

Teams

Create Team...

Join...

And then you need to invite your partner(s) to join this team at Team Manage of the dashboard:

Team Manage

Members

Invite

Setting

Profile Picture

Delete Team

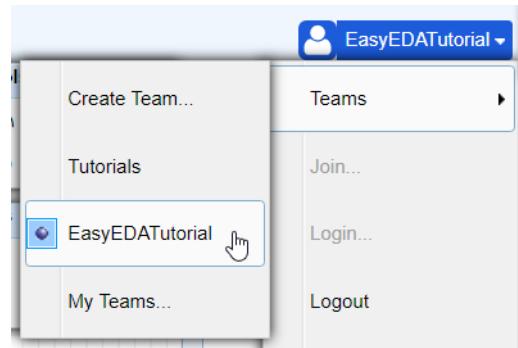
How to switch to team model

1) switch to the dashboard. After you have created a team, click the team name and the dashboard will switch to showing only the team projects, files and components.

The screenshot shows the 'Team Members' section of the EasyEDA dashboard. In the top left, there's a sidebar titled 'My Teams Create' with options like 'Tutorials' and 'EasyEDATutorial'. The 'EasyEDATutorial' option is circled in red. The main area has a table with columns 'Username' and 'Role'. One row shows 'Tutorials' in the 'Username' column and 'Owner' in the 'Role' column. Below this is a 'Team Manage' section with links for 'Members', 'Invite', 'Setting', 'Profile Picture', and 'Delete Team'.

After switching to a team, there is a team management section where you can manage your team members, invite new team members and even delete the team.

2) switch to the editor. Under your personal menu, there is a sub menu allowing you to switch to a team or to your personal account.



How to Upgrade to a team If you want to contribute all of your designs to a team, you can use this function. First you need to create a team, then click the link, shown below, under the dashboard.

Be careful !, because after you do that, **all** of your components, projects will be moved to your team.

This screenshot shows the user management menu. It includes options like 'Account', 'Invitation', 'Email', 'Avatar', 'Password', 'Subscribe', 'Upgrade to team', and 'Close'. A red arrow points to the 'Upgrade to team' option.

Tips about the team function.

1.If you switch to a team, you can't automatically use any Packages/Footprints which you have created under your personal account. You need to **Favorite** your personal package/components first.

2.You need to be aware that your team and your personal accounts are the different, separate accounts and that you can't use them both at the same time.

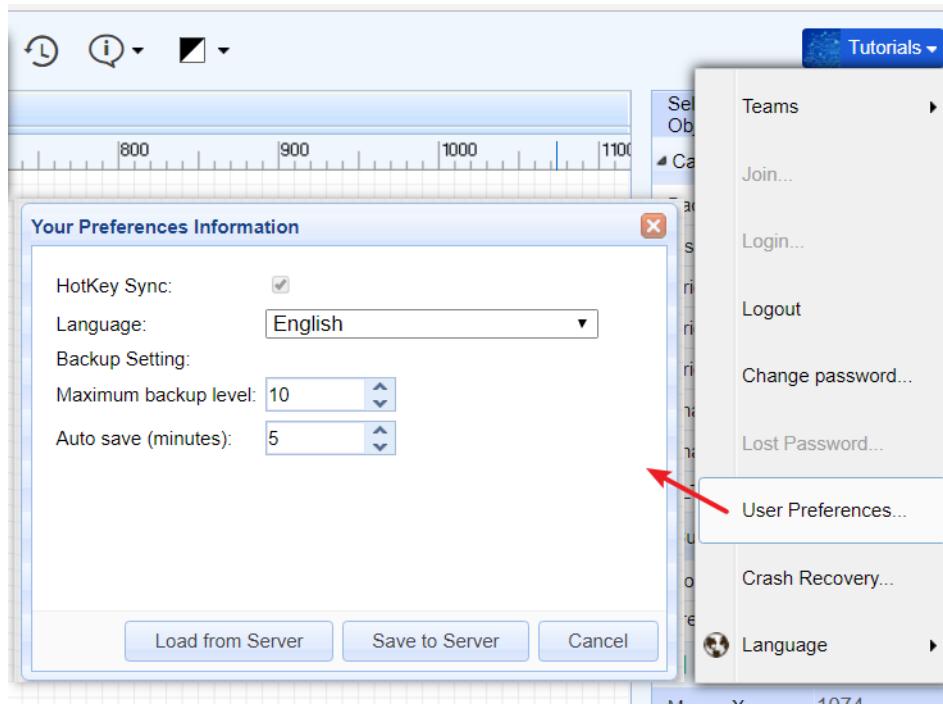
3.After making yourself the owner of a team, it is best to create any Components and Packages needed by the team under that team.

4.If you add a member, nominated to be your accountant, to your team then they can deal the team billing and invoices.

User Preferences

When EasyEDA shows up the login success popup in the bottom right of the window, your user management menu will be look like this:

Click on **User Preferences**,



Maximum backup level: every open document can be saved at up to this number of different revisions.

Auto save (minutes): this is the time interval between auto saves of all your open documents.

Save to Server: Save your preferences (Toolbar configurations, EasyEDA libs, Hotkey settings, language and so on) to the EasyEDA Server.

Load from Server: EasyEDA can't load your Preferences automatically but once you have saved them, you can load them manually. Then, when you change to a different computer or browser, you can load your preferences from the EasyEDA Server.

If you have not saved any preferences then **Load from Server** will have no effect.

Close Account

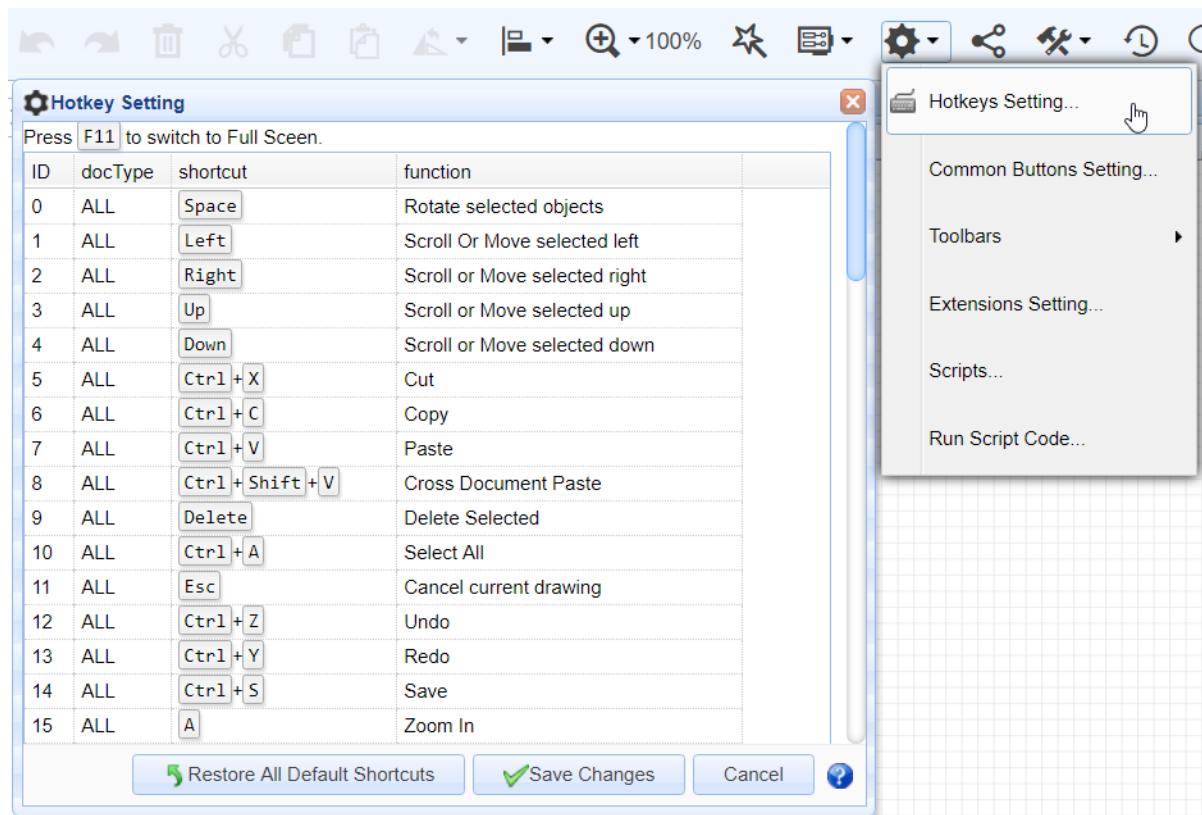
If you want to close your account, you can go to <https://easyeda.com/account/close>

Before you close your account, please let us know why, and that will make us to be better.

Hotkeys

After a while of using an EDA tool suite, clicking all over the place with a mouse gets very tedious and seriously reduces your productivity. Keyboard shortcuts or Hotkeys avoid much of that. EasyEDA not only provides lots of hotkeys, but also every hotkey can be reconfigured.

Under the Config toolbar, click the Hotkeys Setting... Menu which will open the Hotkey Setting dialog.



To change a Hotkey, click anywhere in the row for the hotkey you want to change and then press your new key.

For example, if you want to use R instead of space to rotate selected objects, click on the first row, then press R.

After you change the hotkey, don't forget to click Save Changes button.

The **docType** column describes which type of EasyEDA document each hotkey applies to. **docType** has three types:

- **ALL**: any document type in EasyEDA.
- **SCH**: schematic and schematic libs
- **PCB**: PCB and PCB libs.

The functions of some hotkeys may change between docTypes. For example, the hotkey c draws an Arc in SCH, but draws a circle in PCB.

A list of all the available default hotkeys is given below.

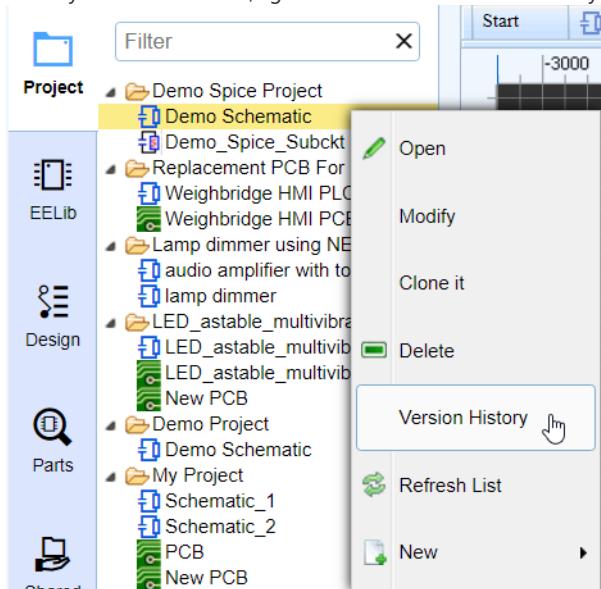
ID	docType	shortcut	function
0	ALL	Space	Rotate selected objects
1	ALL	Left	Scroll Or Move selected left
2	ALL	Right	Scroll or Move selected right
3	ALL	Up	Scroll or Move selected up
4	ALL	Down	Scroll or Move selected down
5	ALL	Ctrl+X	Cut
6	ALL	Ctrl+C	Copy
7	ALL	Ctrl+V	Paste
9	ALL	Delete	Delete Selected
10	ALL	Ctrl+A	Select All
11	ALL	Esc	Cancel current drawing
12	ALL	Ctrl+Z	Undo
13	ALL	Ctrl+Y	Redo
14	ALL	Ctrl+S	Save
15	ALL	A	Zoom In
16	ALL	Z	Zoom Out
17	ALL	X	Flip Horizontal
18	ALL	Y	Flip Vertical
19	ALL	G	Snap
20	ALL	Ctrl+F	Find Component
21	ALL	Ctrl+D	Design Manager
22	ALL	D	Drag Tool
23	SCH	W	Draw Wire
24	SCH	B	Draw Bus
25	SCH	U	Bus Entry
26	SCH	N	NetLabel
27	SCH	Ctrl+Q	NetFlag VCC
28	SCH	Ctrl+G	NetFlag GND
29	SCH	P	Place Pin
30	SCH	L	Draw Polyline
31	SCH	O	Draw Polygon
32	SCH	Q	Draw Bezier
33	SCH	C	Draw Arc
34	SCH	S	Draw Rect
35	SCH	E	Draw Ellipse
36	SCH	F	Freehand Draw

37	SCH	T	Draw Text
38	SCH	I	Edit Selected Symbol
39	SCH	Ctrl+R	Run the Document
40	PCB	W	Draw Track
41	PCB	U	Draw Arc
42	PCB	C	Draw Circle
43	PCB	N	Draw Dimension
44	PCB	S	Draw Text
45	PCB	O	Draw Connect
46	PCB	E	Draw copperArea
47	PCB	T	Change To TopLayer
48	PCB	B	Change To BottomLayer
49	PCB	1	Change To Inner1
50	PCB	2	Change To Inner2
51	PCB	3	Change To Inner3
52	PCB	4	Change To Inner4
53	PCB	P	Place Pad
54	PCB	V	Place Via
55	PCB	M	Measure
56	PCB	L	Change Route Angle
57	PCB	-	Decrease Routing Width
58	PCB	+	Increase Routing Width
59	PCB	Alt+-	Decrease Snap Size
60	PCB	Alt++	Increase Snap Size
61	PCB	H	Highlight Net
62	PCB	Shift+M	Remove All Copper Area
63	PCB	Shift+B	Rebuild All Copper Area

Basic Driving Skills.

Version History

It is easy to use this function, right click on the document for which you need the version history in like in the image below:



After clicking on the version history link, you will get a list of all of the versions like in the image below.

My Teams Create

Tutorials

Time 1 day ago

History

7 6 5 4 3 2 1

Project

Module

Component

Click the version number, you can open the saved file in the editor, if this is what you need, you can save it to your project and delete your bad file.

Note:

1. For now all of the versions are marked as number, we will allow you to add a tag soon.
2. Don't save your files too frequently, or you will get lots of versions and it will be hard to find the exact one you want.

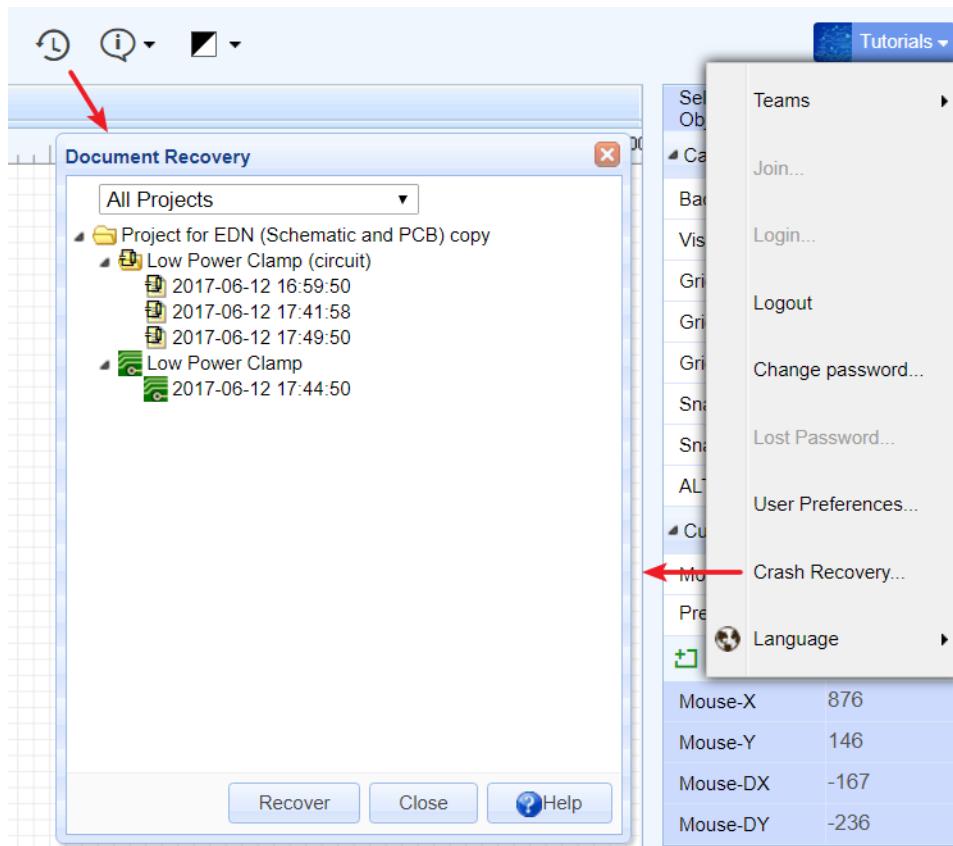
Crash Recovery

No operating system, software or network is perfect, so sometimes things can go wrong. Having your Desktop or web browser freeze or your broadband connection drop, two hours into laying out a PCB, could spoil your day.

However, with EasyEDA, your day will be just fine.

This is because EasyEDA auto saves and makes backups of all your open files to your computer so crash recovery is built into EasyEDA.

In user management menu, click on **Crash Recovery**. Or you can click **Crash Recovery** button on the top Toolbar as below:



Select the file which you would like to **recover**, then click the Recover button; your file will be opened in a new tab.

Please note:

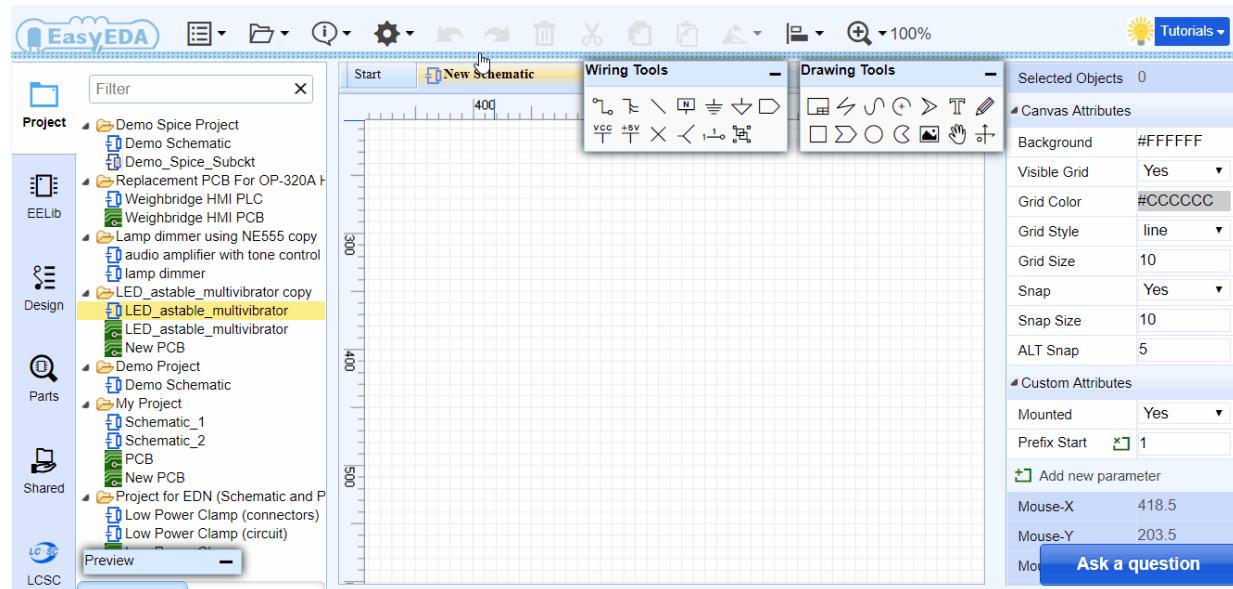
- EasyEDA saves these crash recovery files on your computer and not on the EasyEDA server. Therefore you cannot recover files from a crash on one computer or browser by changing to a different computer or browser.
- And if you cleaned your browser's cache, the recovery files will disappear. - If you make a mistake to delete a file and remove the cache already, maybe you can find your document back via : <https://easyeda.com/document/recycle>.

To use EasyEDA, you need to be familiar with a few basic terms and concepts. The best way to learn them is to open up EasyEDA, open a new schematic:

Document > New > Schematic, and play!

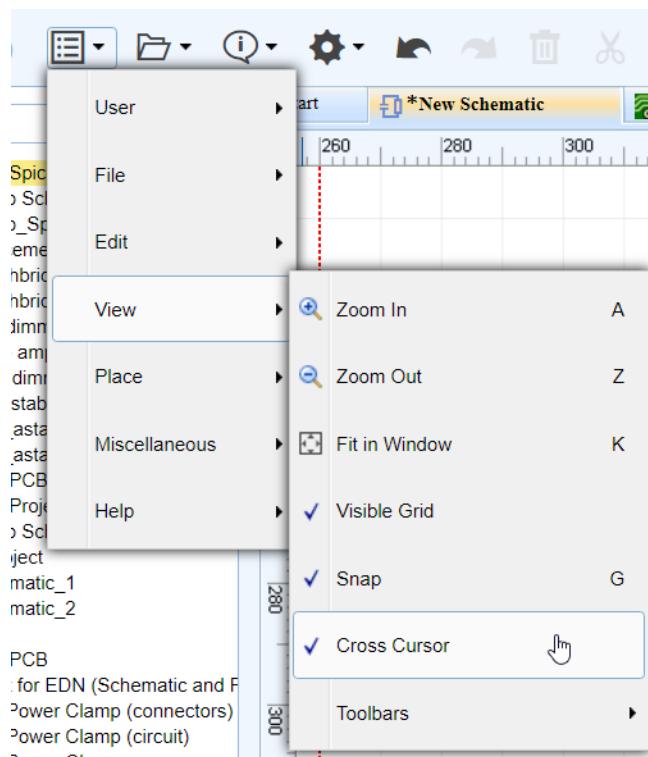
Resizing the canvas area

Hovering the mouse cursor over the areas indicated by the three green ellipses will bring up blue toolbar toggle lines. Clicking on them will toggle the visibility of their associated top, right and left toolbar areas to expand the canvas area. The vertical lines can also be dragged horizontally to resize the panels.

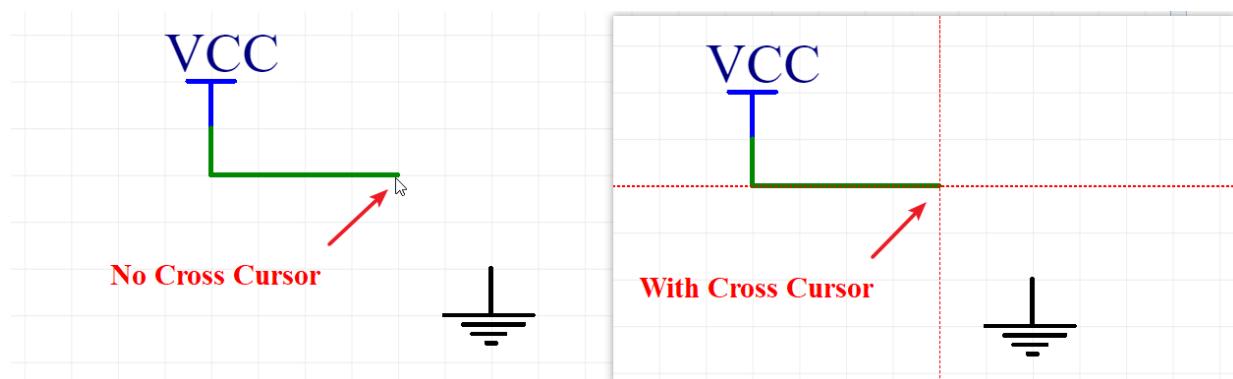


Cursor Style

Some users don't like the cross cursor, so you can change it to arrow cursor like in the image below.



These difference between these options is as below:

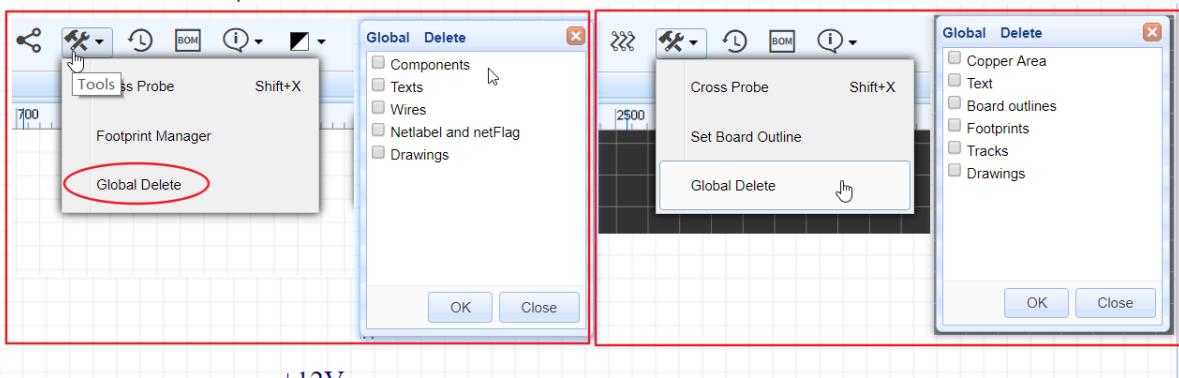


Clear

If you think your schematic or PCB looks terrible, and you want to redraw all units, you can:

- Super Menu > Edit > Clear.

- Delete this schematic and create a new one.
- Use **Global Delete** on the top toolbar Tools



Left clicking

Similar to other EDA software:

- Click on an item to select it;
- If over a selected item, click and hold to drag a selected item;
- If not over a selected item, clicking and holding while dragging creates a selection box;
- the selection box, using click and drag to the right, selects everything inside the box;
- the selection box, using click and drag to the left, selects everything inside and intersected by the box;
- Double click on a text area to edit it;
- The exact left click functionality depends on what item is being selected and in what Canvas the item exists (Schematic or PCB).

Right clicking

EasyEDA does not support right click context menus in the Schematic or PCB Canvas. Instead, right clicking executes a context sensitive command:

- When you are placing a symbol, after a right click, the active symbol will be removed;
- 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.

Ctrl+Right clicking anywhere in the Schematic, waveForm or PCB Canvas drags the canvas around within the EasyEDA window.

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).

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 roll your mouse at the same time as pressing the CTRL key. Some browsers will zoom the whole site, not just the canvas in the EasyEDA window. If this happens, just press **ctrl+0** to reset the browser zoom.*

Double clicks

Double clicking any text area opens a resizable text box to allow you edit the text inline.



Press enter to create new line. Click outside the text box to close it.

Pan

- 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.

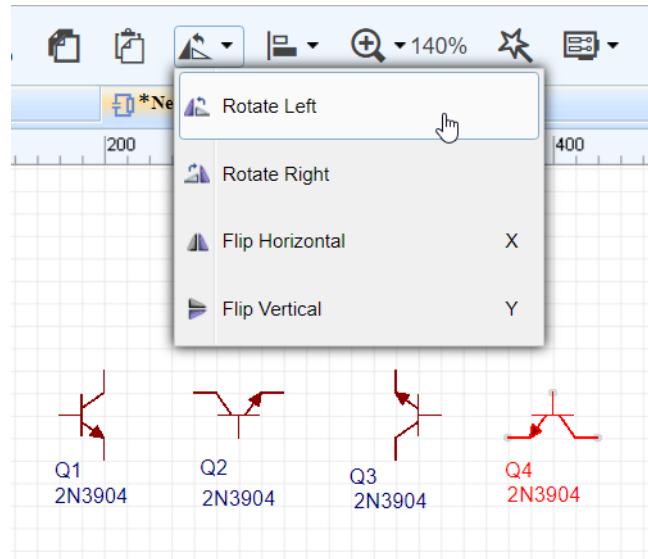
Tip: If you use Chrome, and cursor is in the canvas while pressing CTRL or ALT key and rolling your mouse, the canvas will move vertically, and when pressing SHIFT and rolling your mouse, the canvas will move horizontally.

Rotate

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

Super Menu > Edit > Rotate or click topToolBar **Rotate and Flip** > **Rotate Left** or **Rotate Right**

or by pressing the default rotate hotkey: **Space**.



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.



You can Flip one or more selected items using:

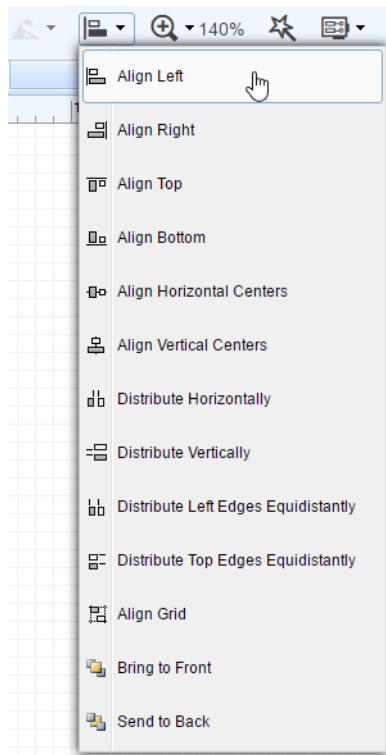
Rotate and Flip > **Flip Horizontal** or **Flip Vertical** from the toolbar,

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

Align

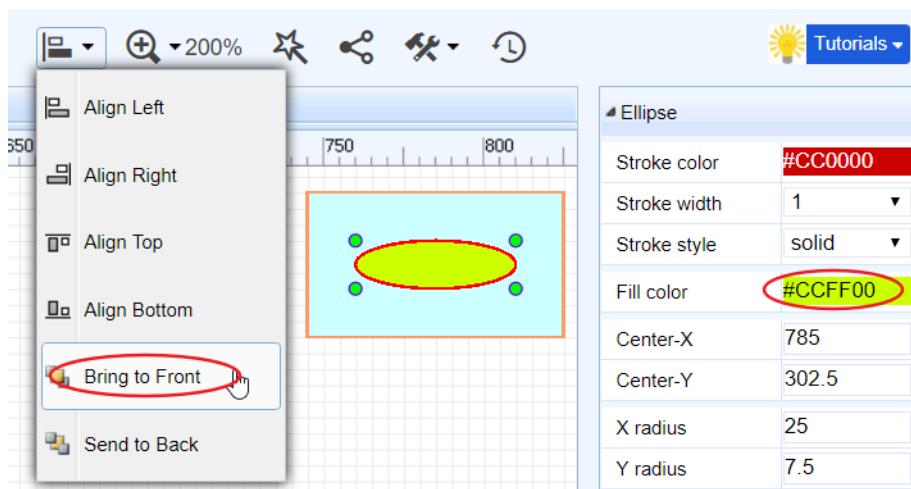
EasyEDA provides many align option features, you can align your components or footprints very easily, it include:

- Align Left
- Align Right
- Align Top
- Align Bottom
- Align Horizontal Center
- Align Vertical Center
- Distribute Horizontally
- Distribute Vertically
- Distribute Left Edges Equidistantly
- Distribute Top Edges Equidistantly
- Align Grid



Bring to Front and Send to Back

In the image below, both the rectangle and the ellipse are filled.



If you draw the ellipse before drawing the rectangle, the rectangle will overlap and therefore hide the ellipse. To reveal the ellipse, select the rectangle and then use:

Align > Send to Back from the toolbar.

To bring the rectangle to the front again, you could select it and use:

Align > Bring to Front

or select the ellipse and then use:

Align > Send to Back

Documents Tab Switch

It's easy to fit your documents tab location.



Saving Your Work Locally

Although EasyEDA saves all your files on our Server, sometimes you may want to save your work locally and EasyEDA provides a hack way to do this.

More detail you can view at [Export EasyEDA Source](#) section.

About upgrade

If you use EasyEDA online, it can seamlessly upgrade by itself. However, EasyEDA uses an App Cache technique to allow you to use EasyEDA offline ([W3C HTML5 Offline Web Applications](#)) which may delay the automatic upgrading process. Therefore, if you want to upgrade to the latest version immediately, you can follow the two simple steps below.

1. Check the About... dialog;
2. If the Built Date is older than 2017/06/01:

Close your browser open EasyEDA again.

If the Built Date is still showing older than 2017/06/01:

Close your browser and open EasyEDA again.

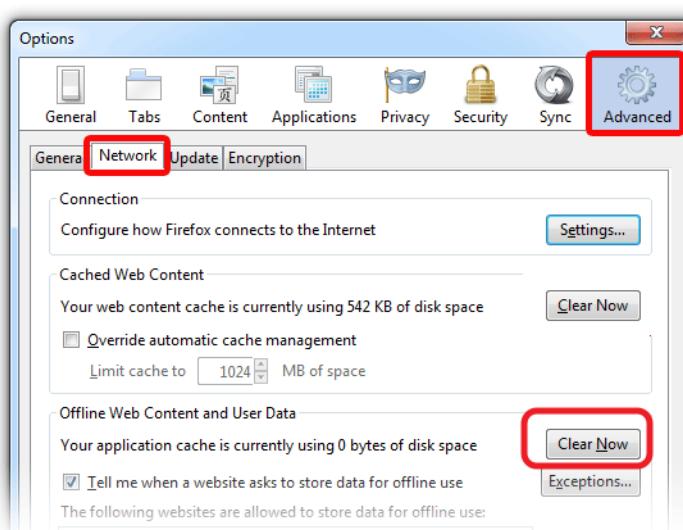
If the Built Date is at or newer than 2017/06/01, you don't need to do anything.

Note: 2017/06/01 is just an example.

If those two steps don't work, you may need to clear your browser's cache:

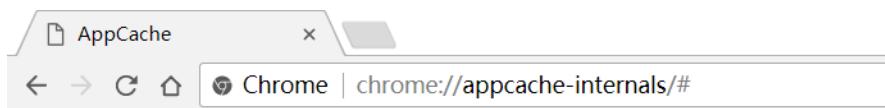
1. Mozilla Firefox

- Go to "Preferences... > Advanced > Network > Offline Storage" ,
-Click on "Clear now" ,
-Reload easyeda again.



2. Chrome

- Open the following URL: <chrome://appcache-internals/>
- Look for easyeda.com and click "Remove"
- reload easyeda again.



Application Cache

Instances in: C:\Users\AppData\Local\Google\Chrome\User Data\Default (2)

<https://easyeda.com/>

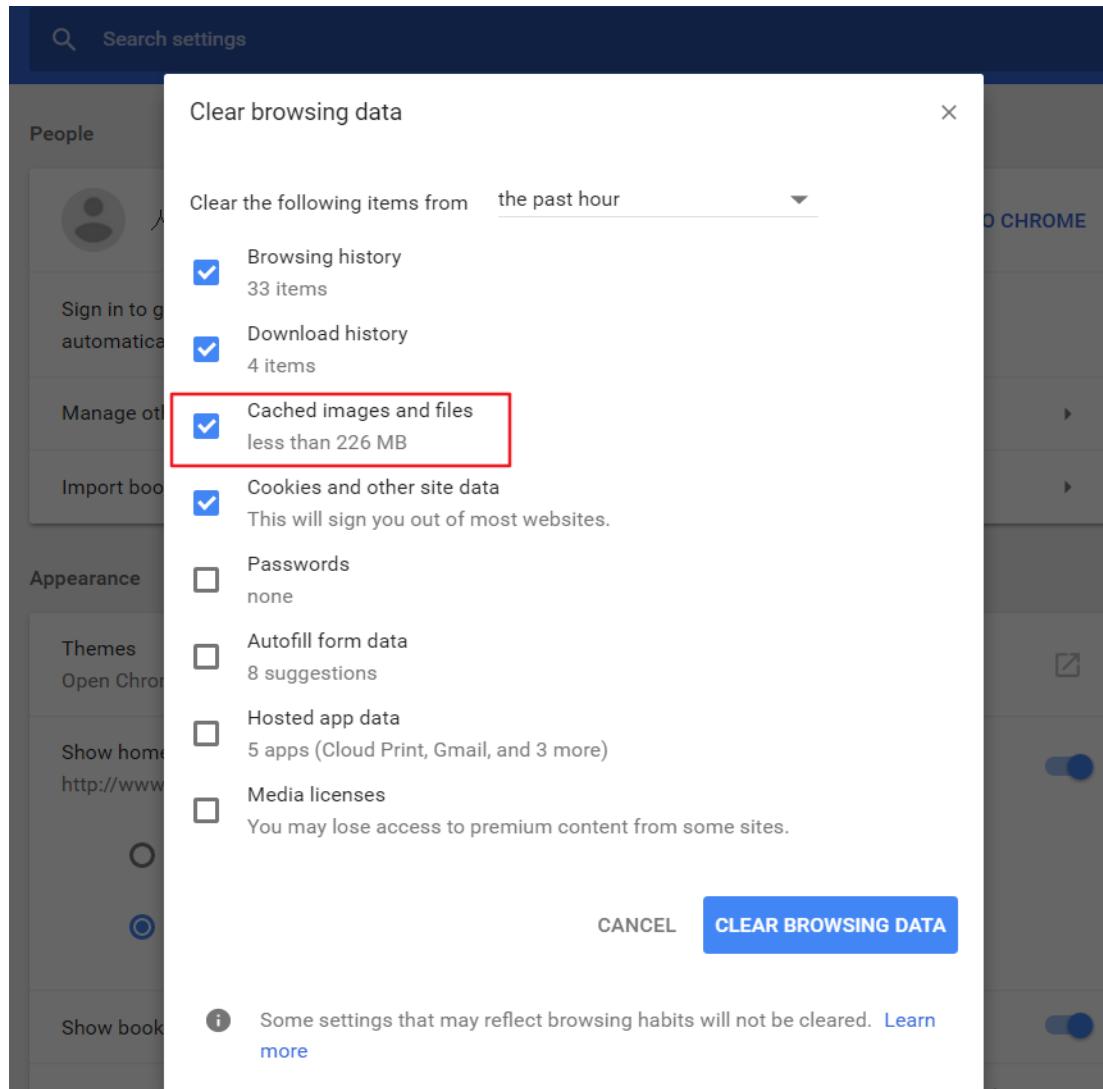
Manifest: <https://easyeda.com/editor.appcache>

Size: 3.5 MB

- Creation Time: Tue Jun 13 2017 12:43:57 GMT+0800 (中国标准时间)
- Last Access Time: Tue Jun 13 2017 14:46:52 GMT+0800 (中国标准时间)
- Last Update Time: Tue Jun 13 2017 12:43:57 GMT+0800 (中国标准时间)

[Remove Item](#) [View Details](#)

- Or you can use **Ctrl+shift+Delete** to delete Chrome caches.



How to get help

It is easy to ask for help for any problem with EasyEDA, just click on **Let's Chat**, and then complete and Submit the Support request:

The screenshot shows the 'Let's Chat' support request form. The title bar says 'Let's Chat' with a dropdown arrow and a close button. The main area contains the message 'Will email back ASAP' followed by three input fields: '* Name', '* Email', and '* Message'. At the bottom is a large blue 'Submit' button. Below the form is a small note: 'Powered by [tawk.to](#)'.

Please ask your questions in English or Chinese and don't worry if your English is not good! (Or your Chinese!)

1. You can also ask your questions directly in the [EasyEDA forum](#). We will try to respond to every post but please be patient. Maybe EasyEDA team is in a different timezone and we are a bit busy, so you may need to wait for a while.
2. If you don't want your help requests to be public then you can drop us an email to support@easyeda.com
3. If maybe you have a design that you know worked in some other EDA package and you are having problems importing it to EasyEDA, let us know and we will take a look and try to help you to fix them.

Please note that:

EasyEDA team may not have the time or resources to help you fix all your problems; we may just be able to help you to fix problems commonly encountered by newbies, such as using a drawing polyline in place of a wire, finding a spice model for a simulation or selecting the right PCB

footprint.

[1] Please note that although some browsers or plug-ins allow you to use gestures, EasyEDA does not work with gestures, so you should disable this function.

[2] Simultaneous editing is not yet fully supported: care must be taken because the last save by any collaborator overwrites all previous saves.

[3] It can also find the value text but it cannot step through multiple components with the same value.

[4] Take a few moments to think about your username because this is the name that other users will see on your designs and posts if you choose to share them or make them public. Once you have created an account, you cannot change your username.

[5] You can use upper and lower case letters, numbers and symbols to make a strong password but don't forget that the password entry is case sensitive.

[6] Except ordering of PCBs directly from EasyEDA.

[7] If you always open EasyEDA in the same browser on the same machine, your Anonymous files will appear under the Anonymous Files folder in the left hand panel but you should not rely on this as a way of keeping track of Anonymous files.

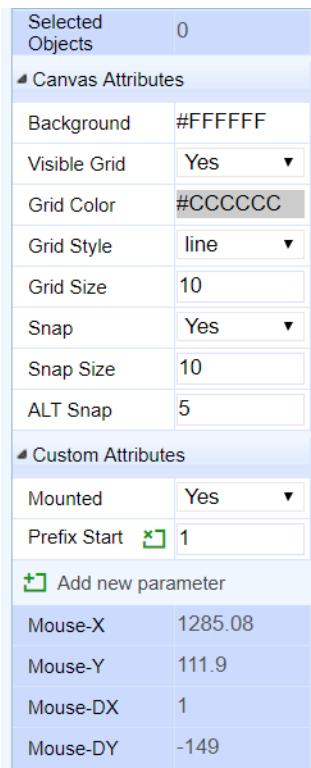
Please email support@easyeda.com when you need any help.

Creating The Schematic

During this tutorial we will guide you in using EasyEDA Schematic capture.

Canvas Settings

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

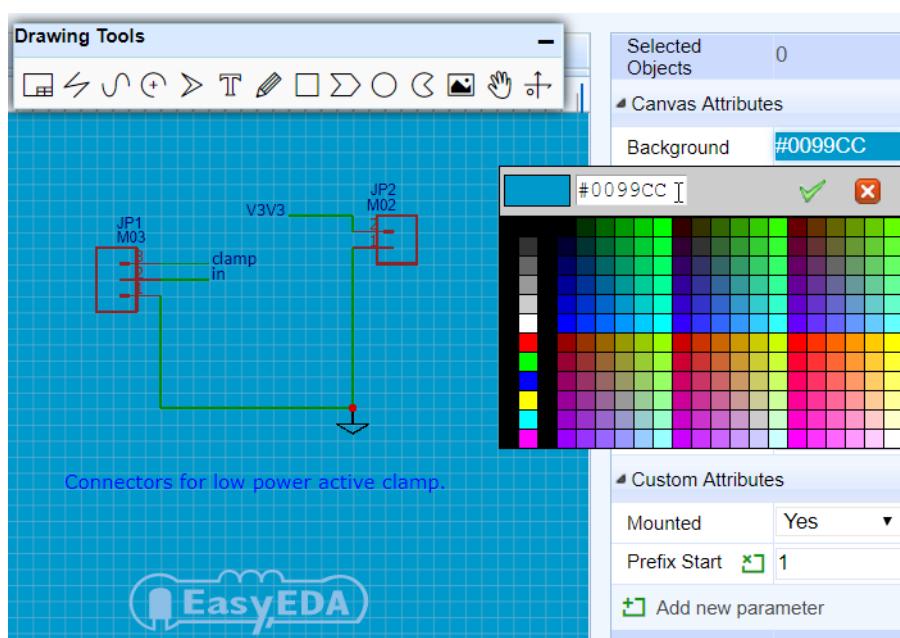
Visible Grid : Yes or No

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.

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. The default hotkey is G. 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 0.1, 1, 5.

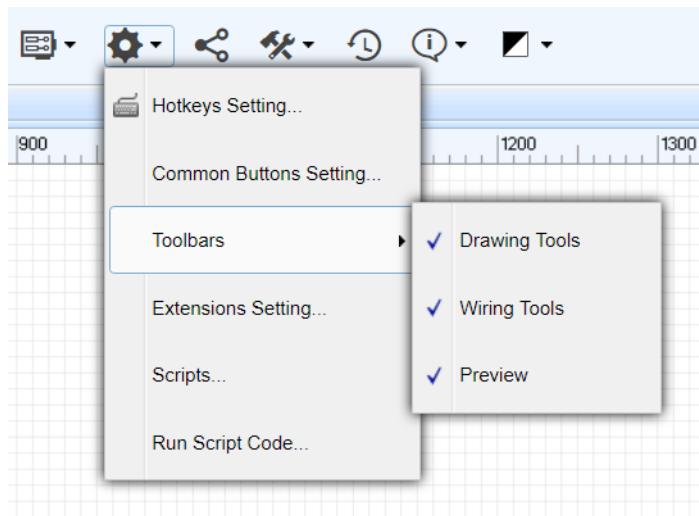
It is strongly recommended that you keep **Snap = Yes** all the time. Once items are placed off-grid it can be very difficult to reset them back onto the grid. Off-grid placement can result in wires looking as though they are joined when in fact they are not and so causing netlisting errors that can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

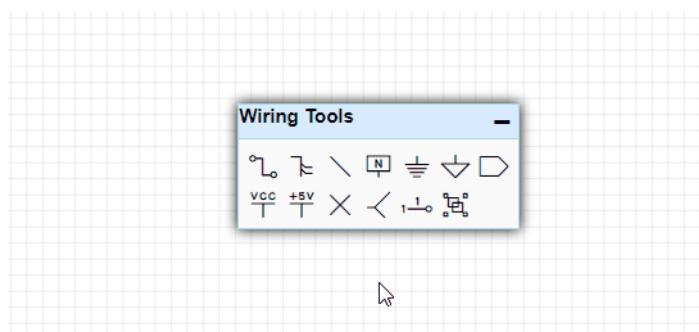
ALT Snap: Snap size when pressing the **ALT** key.

Wiring Tools

If you have hidden your tools , you can open them from here: Top toolbar **Config Gear Icon > Toolbars > ...**



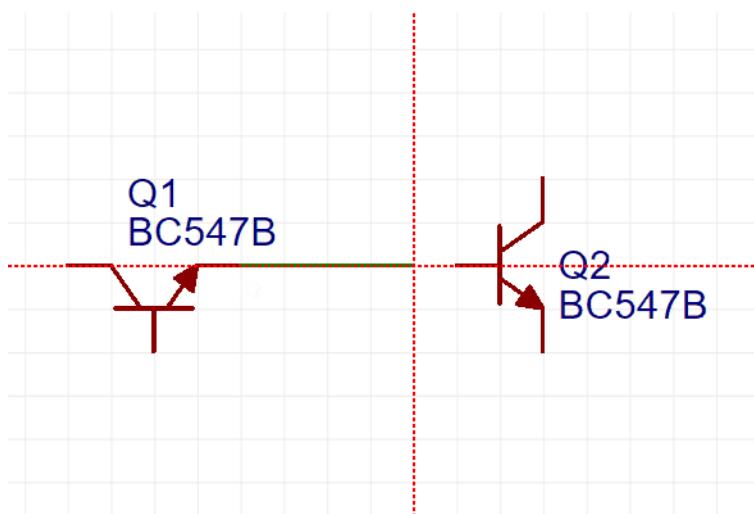
Note: All of the commands in Wiring Tools are electronics related. Don't use a wire when you just need to draw a line, shape or an arrow: use Drawing Tools instead.



Wire

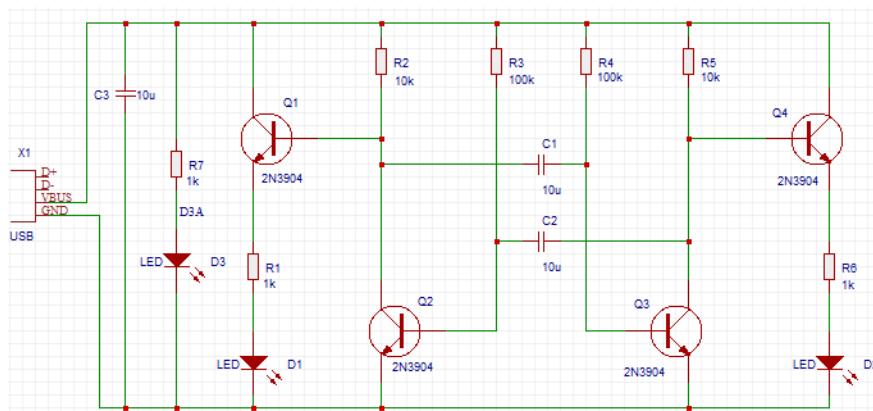
There are three ways to enter the wire mode in EasyEDA.

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.

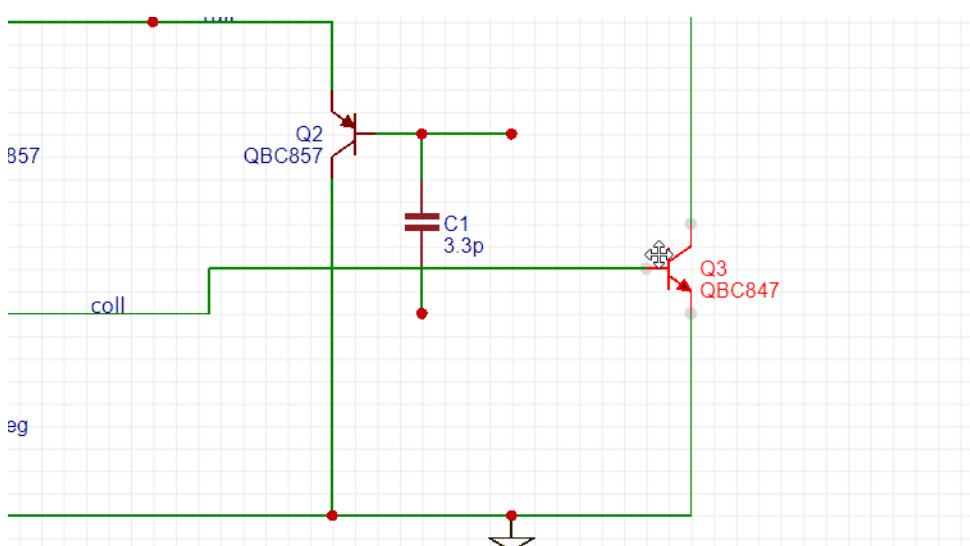
Here is a screenshot of the **Astable Multivibrator LED project schematic** after wiring:



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.

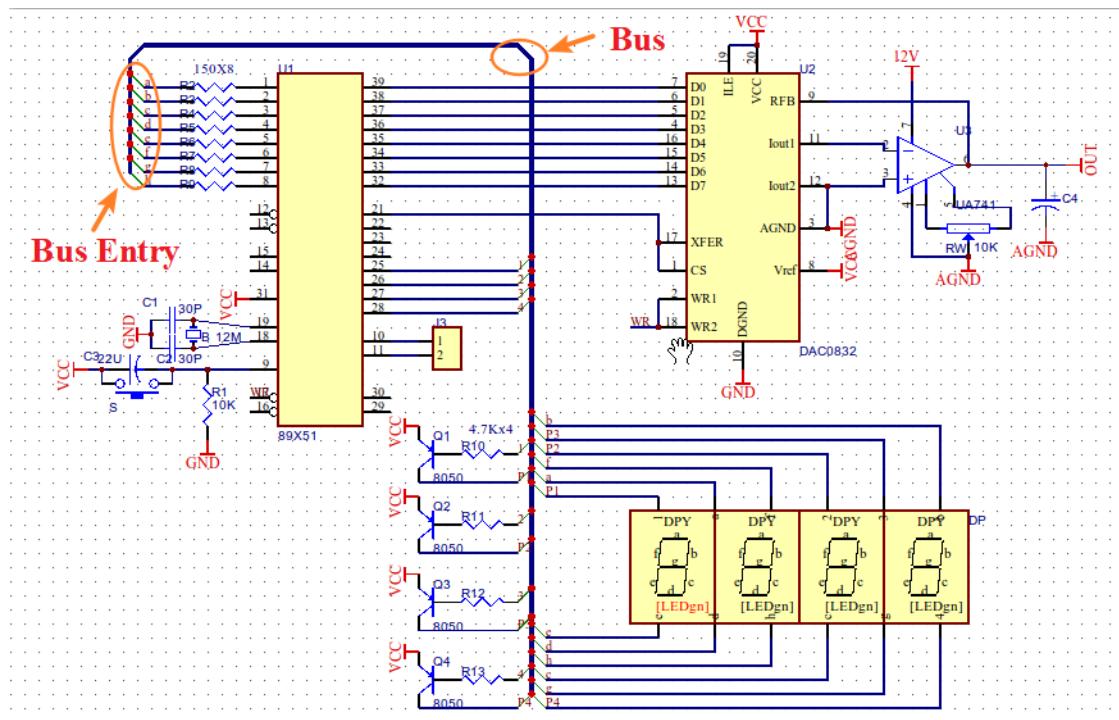


w power active clamp.

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.

Bus

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



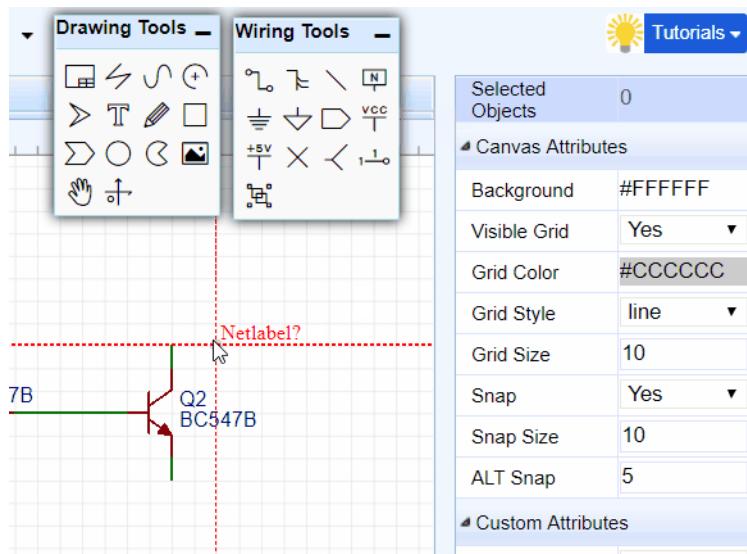
Bus Entry

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.

Net Label

NetLabel and NetFlag

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.

Net Flag

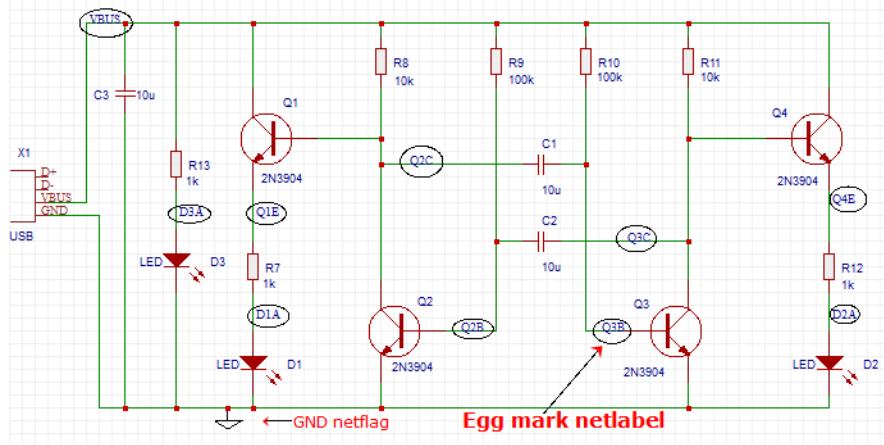
NetFlag is the same as NetLabel, you can find the NetFlag from the Wiring Tools palette or using the **Ctrl+G** hotkeys for **GND** or **Ctrl+Q** for **VCC**. You can also change its name, for example **VCC** to **VDD**:



The screenshot below is after adding NetLabels

- indicated by the little egg marks
-and a **GND** NetFlag

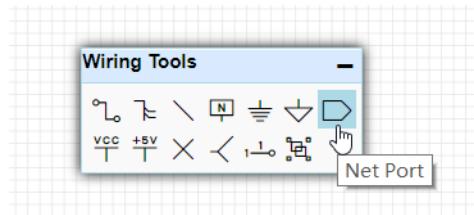
This schematic is almost finished.



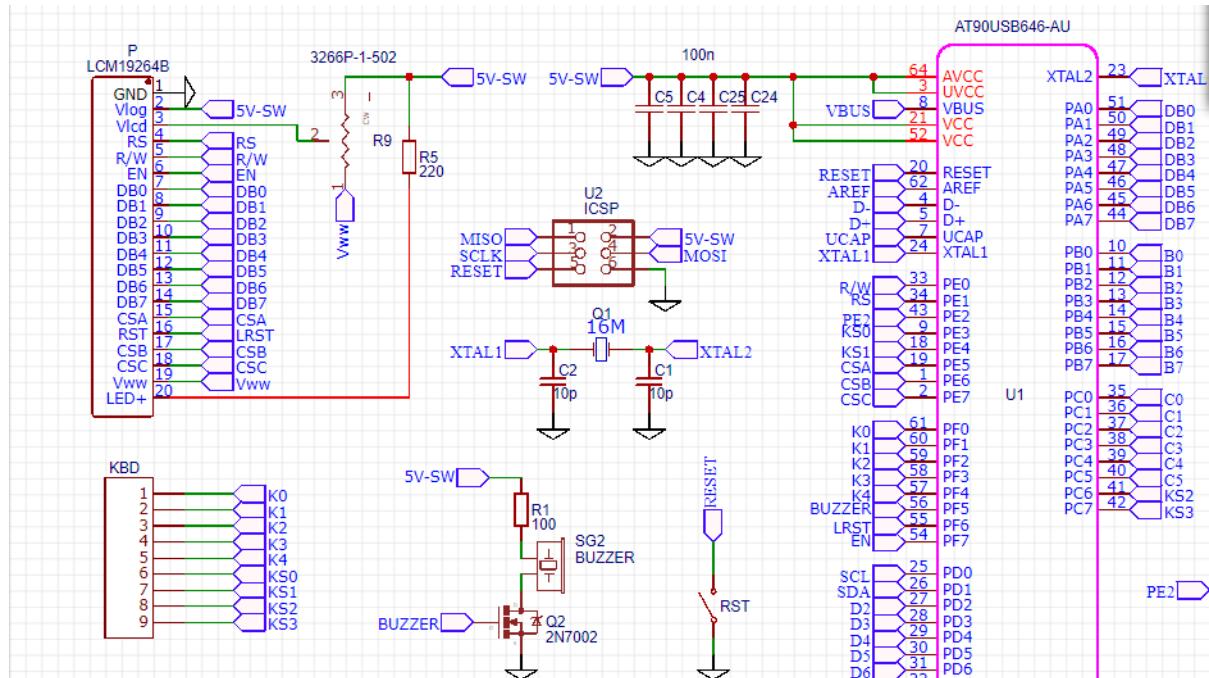
Wiring Tools palette provides NetFlag: Digital GND, Analog GND, VCC and +5V for your convenience.

Net Port

When you don't want to route too many wires, how about trying `Net Port`:

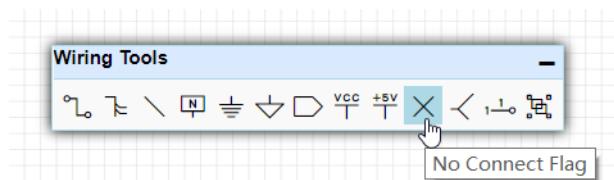


It will make your schematic look more clean, and you just need to set each Net Port a net name.

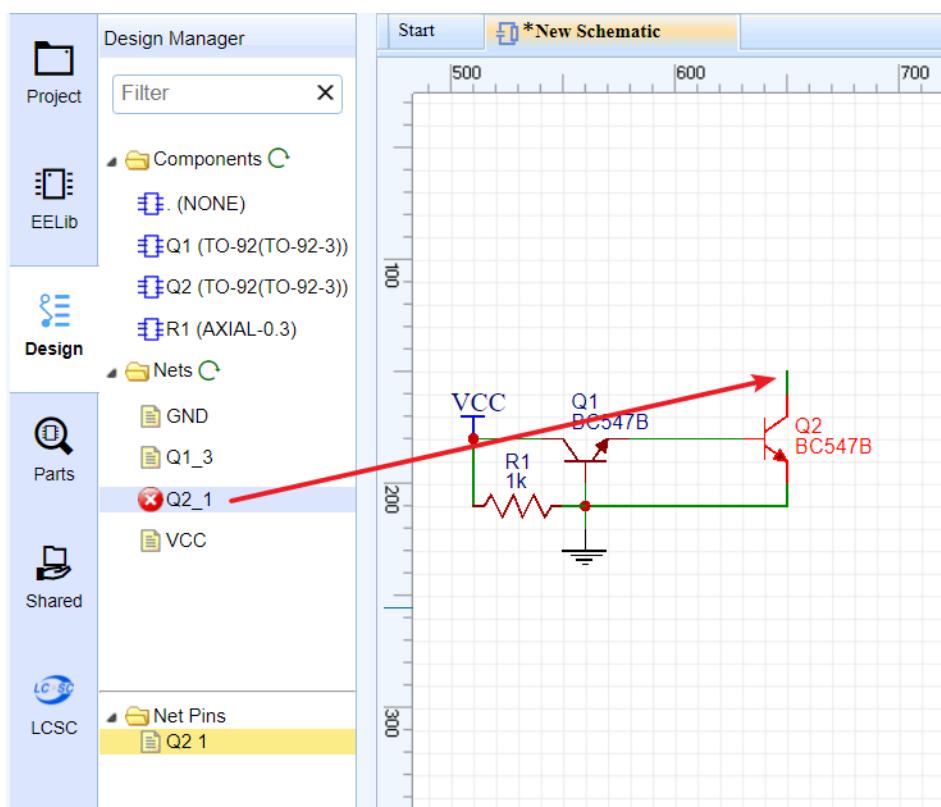


No Connect Flag

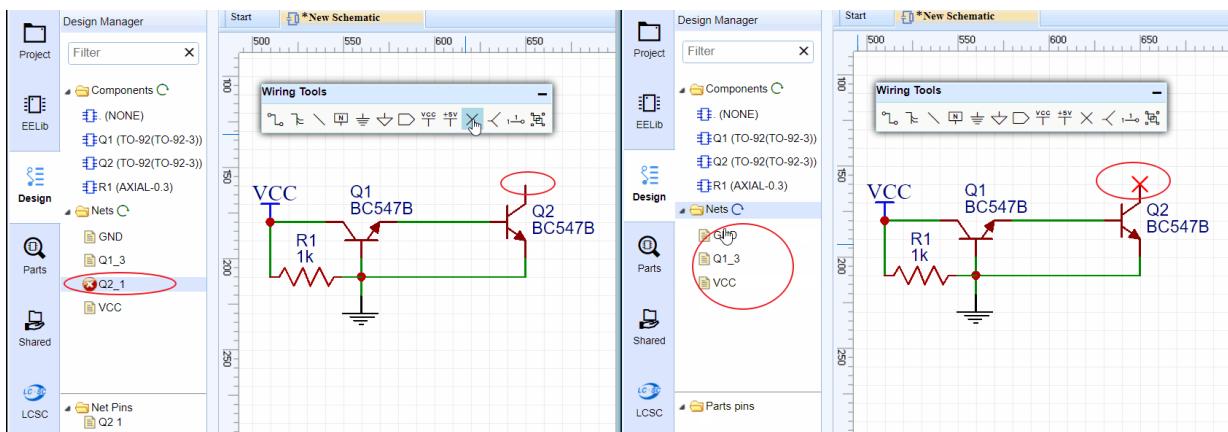
You can find the `NO Connect Flag` via wiring tool,



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.

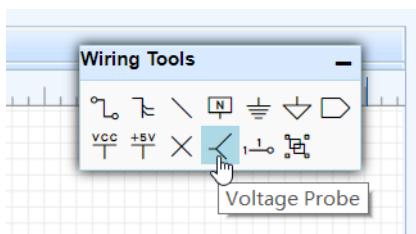


After adding a `NO Connect Flag`, the error disappears.

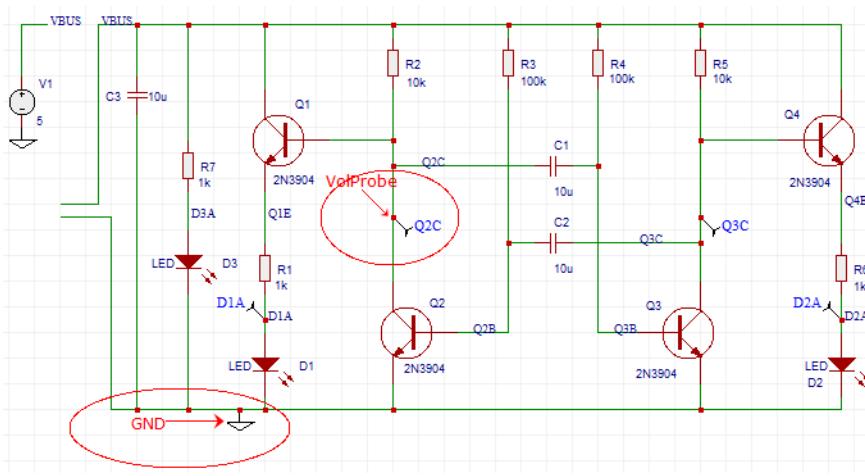


Note: `NO Connect Flag` only works on the symbol's pin directly.

Voltage Probe



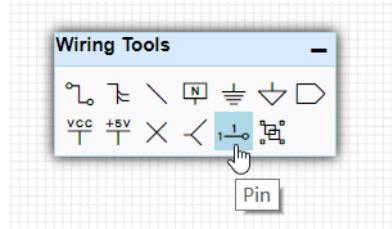
EasyEDA provides a simulation feature for the schematic. After the simulation is running, you will see the waveform where you placed the voltage probes in the circuit.



For more detail about the simulation, please check the [Simulation](#) section.

Pin

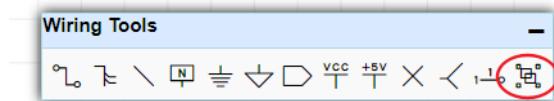
When you create a new symbol in schematic and schematic lib, you must use [Pin](#) to create pins for the new symbol, otherwise your symbol can't be wired with wires.



For more information please refer to the [Schematic Lib: Pin](#) section.

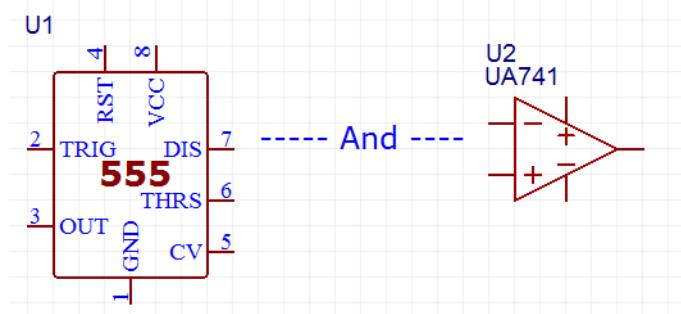
Group/Ungroup

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.

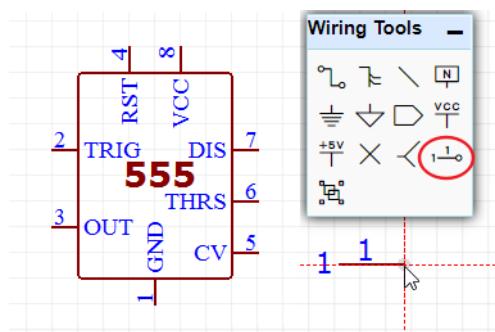
Using the [Symbol Wizard](#) you can only create generic symbols but how can you quickly and easily create symbols like these?



Here's how.

EasyEDA allows you to do something that very few other EAD tools support.

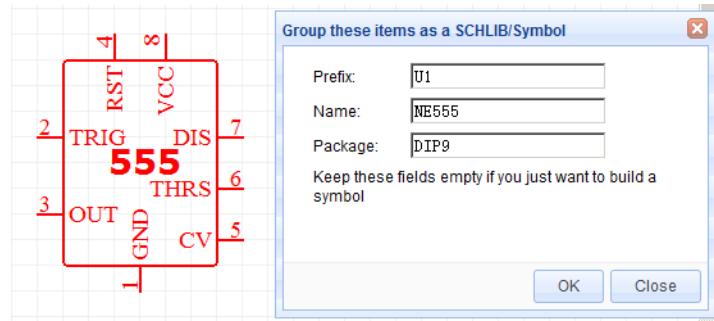
Using the [P](#) Hotkey, you are allowed to add a PIN directly onto the Schematic canvas. So you can add 8 Pins, draw a rectangle from the Drawing Tools palette and add 555 as text to form a symbol for the NE555 like the one shown below:



Now comes the clever bit.

Up to this point you have a collection of separate pins, a drawn rectangle and some text that are all separate items with no particular association with each other.

So now select all of the items and click the Group/Ungroup Symbol... button. A dialog will be opened:



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 any symbol in the schematic, easily and quickly.

How cool is that?

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

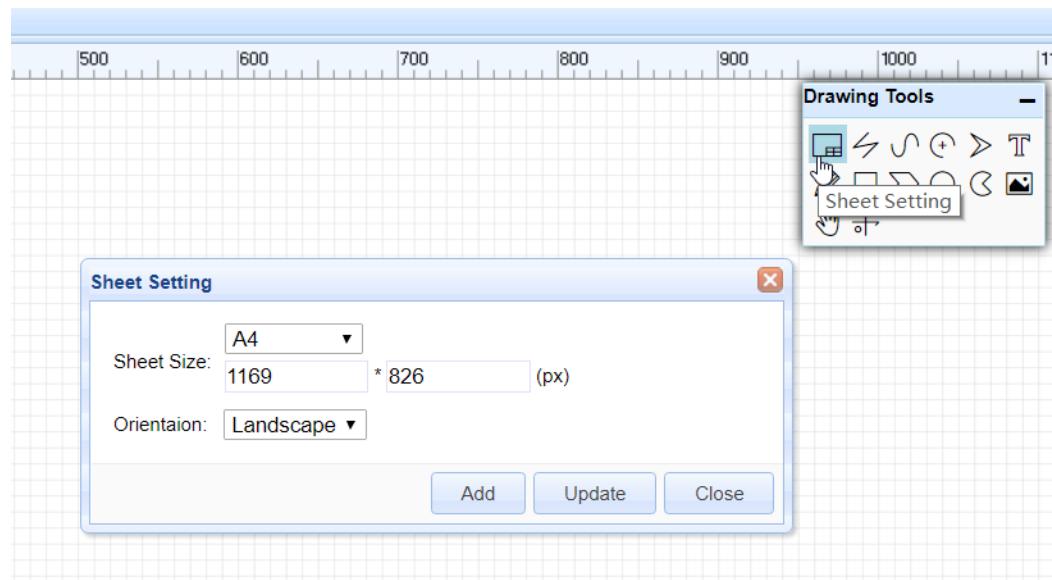
Note: The pin numbers and names cannot be moved independently of the pin.

Drawing Tools

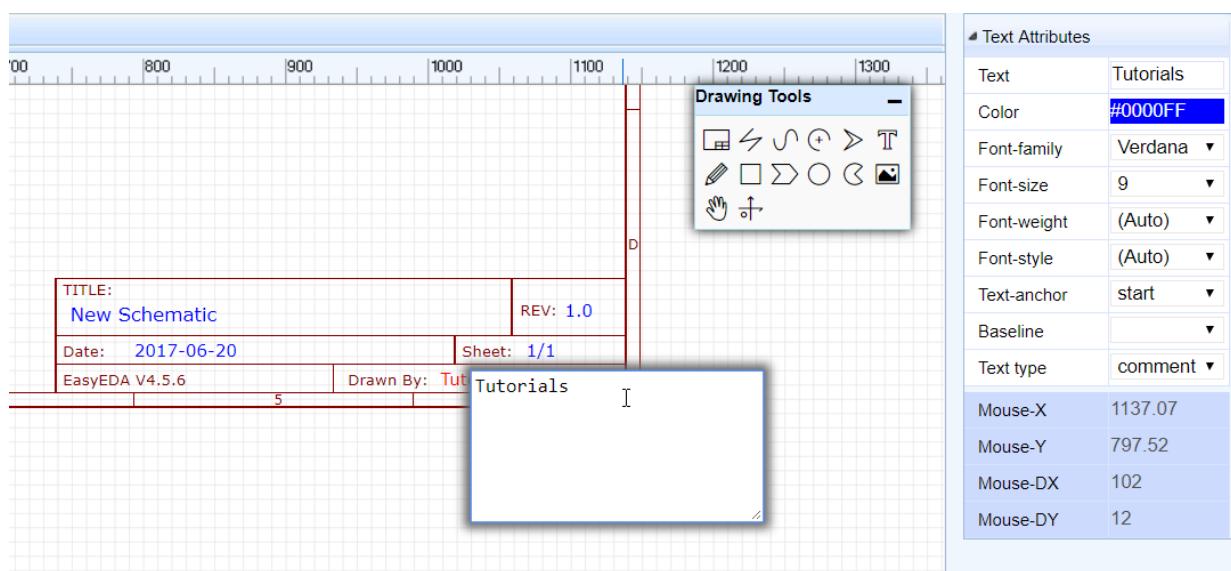
Sheet Setting

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 button like in the image below, Or via: **Super Menu > Miscellaneous > Sheet Setting**



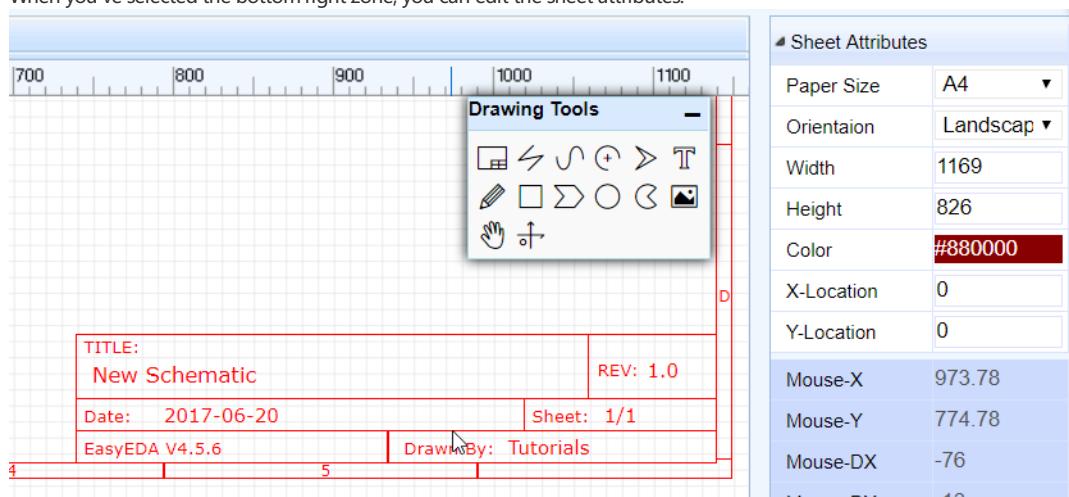
As illustrated in the image below:



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

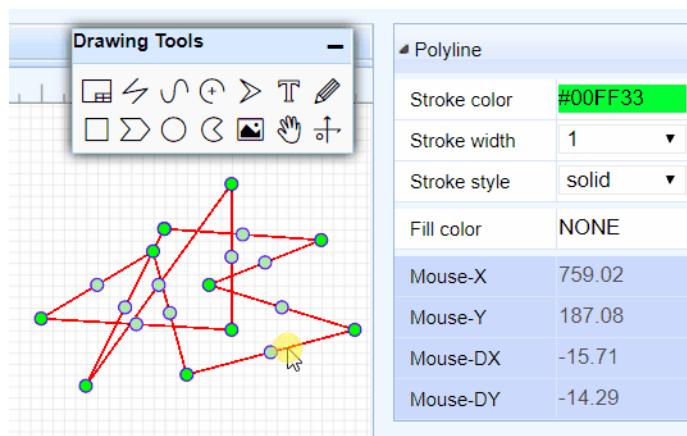
The bottom right zone can be selected and dragged or the frame can be dragged and deleted.

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



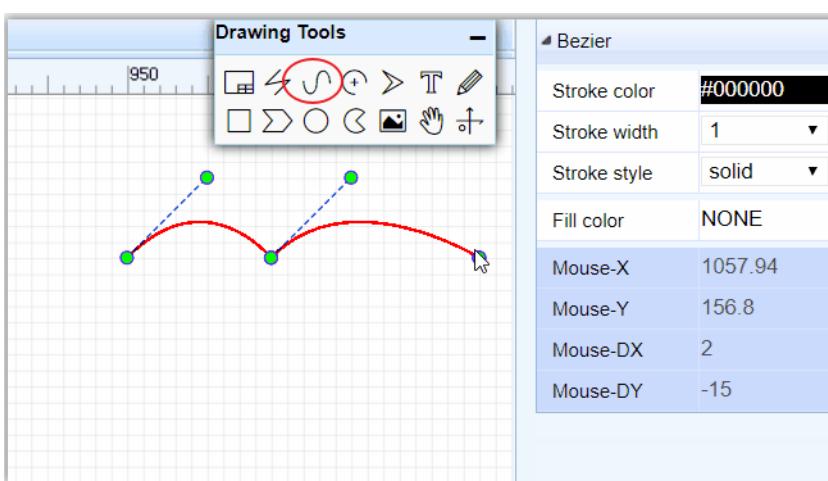
Line

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



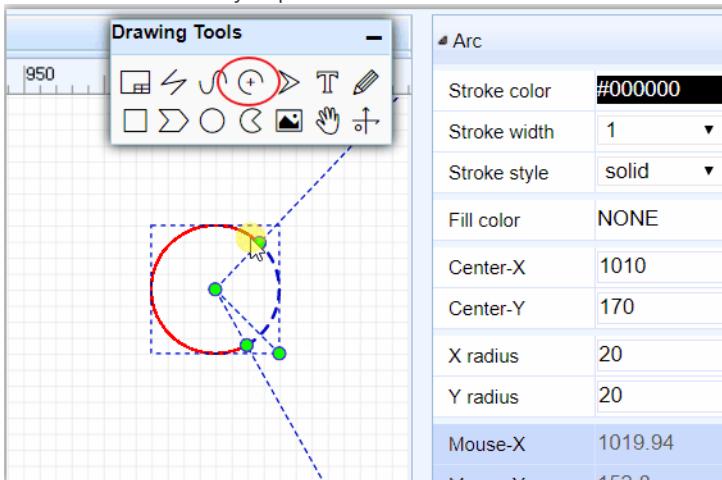
Bezier

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



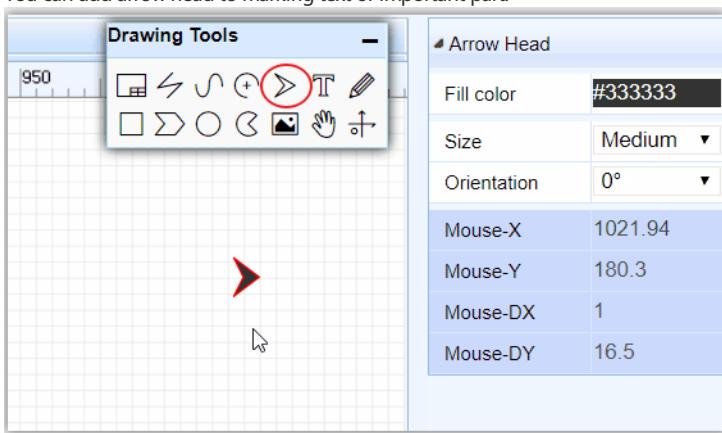
Arc

You can draw the arc of any shape.



Arrow Head

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

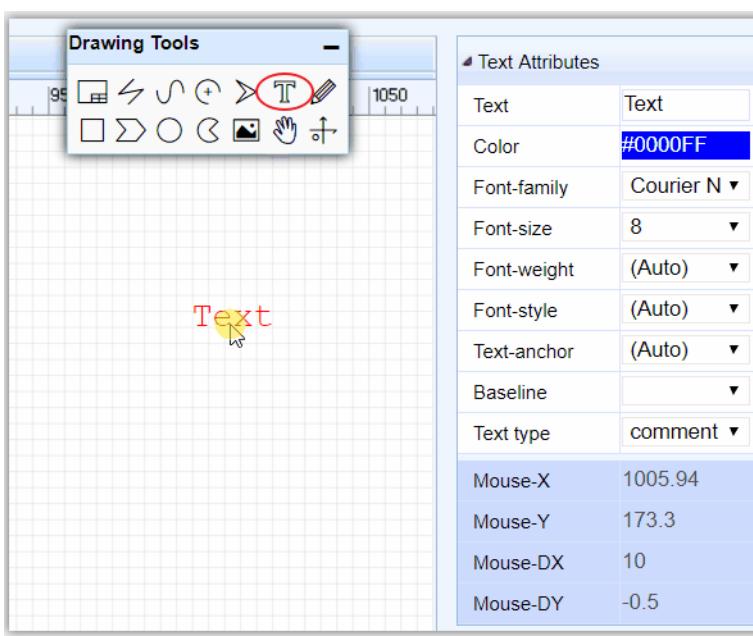


Text

Text attributes provide many parameters for setting:

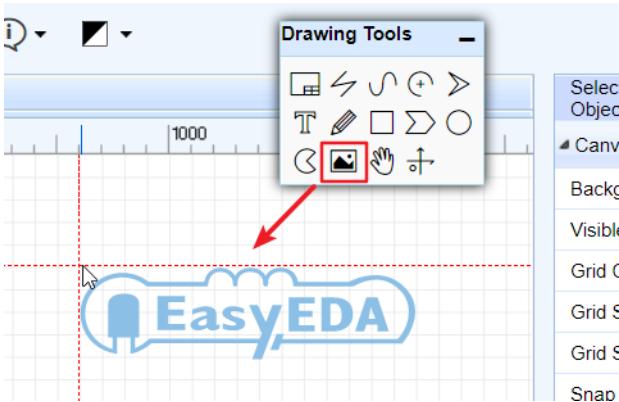
- Text:** You can change text in inner box or double click the text. For every new text, the default text is `Text`. **-Color:** Defines text color.
- Font-family:** It provides 12 fonts for choosing. **-Font-Size:** Defines Text size. **-Font-weight:** Defines Text weight. **-Font-Style:** It contains `(auto)`, `normal`, `italic`.
- Text-anchor:** It contains `(auto)`, `start`, `middle`, `end`, `inherit`.
- Baseline:** It contains `(auto)`, `use-script`, `no-change`, `reset-size` ... and so on.
- Text type:** types include `comment` and `spice`.

The editor will remember your last text parameters.

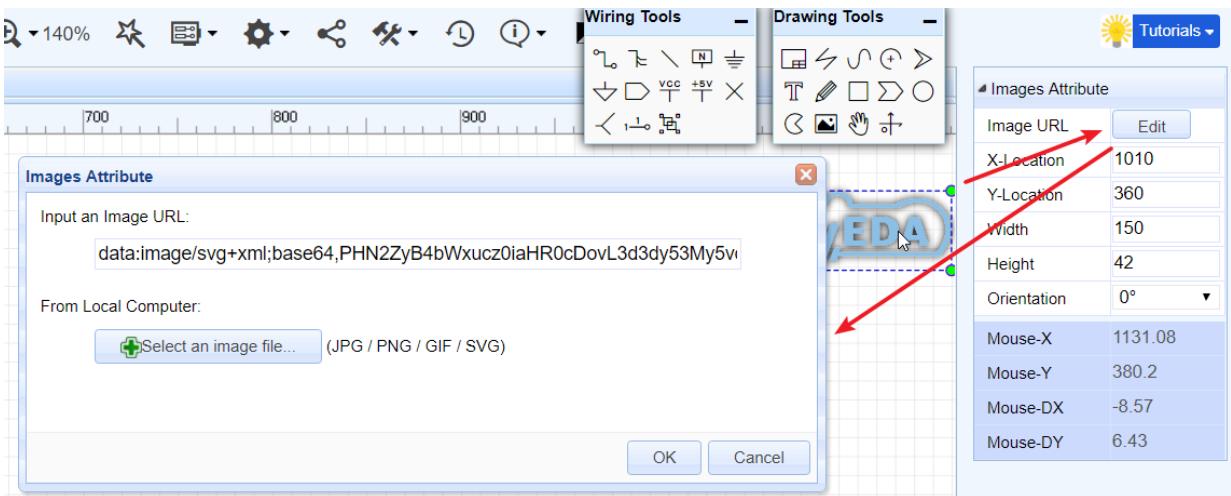


Image

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



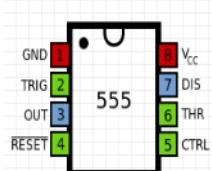
Select the place holder, so you can see the image's attributes in the right hand Properties panel:



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

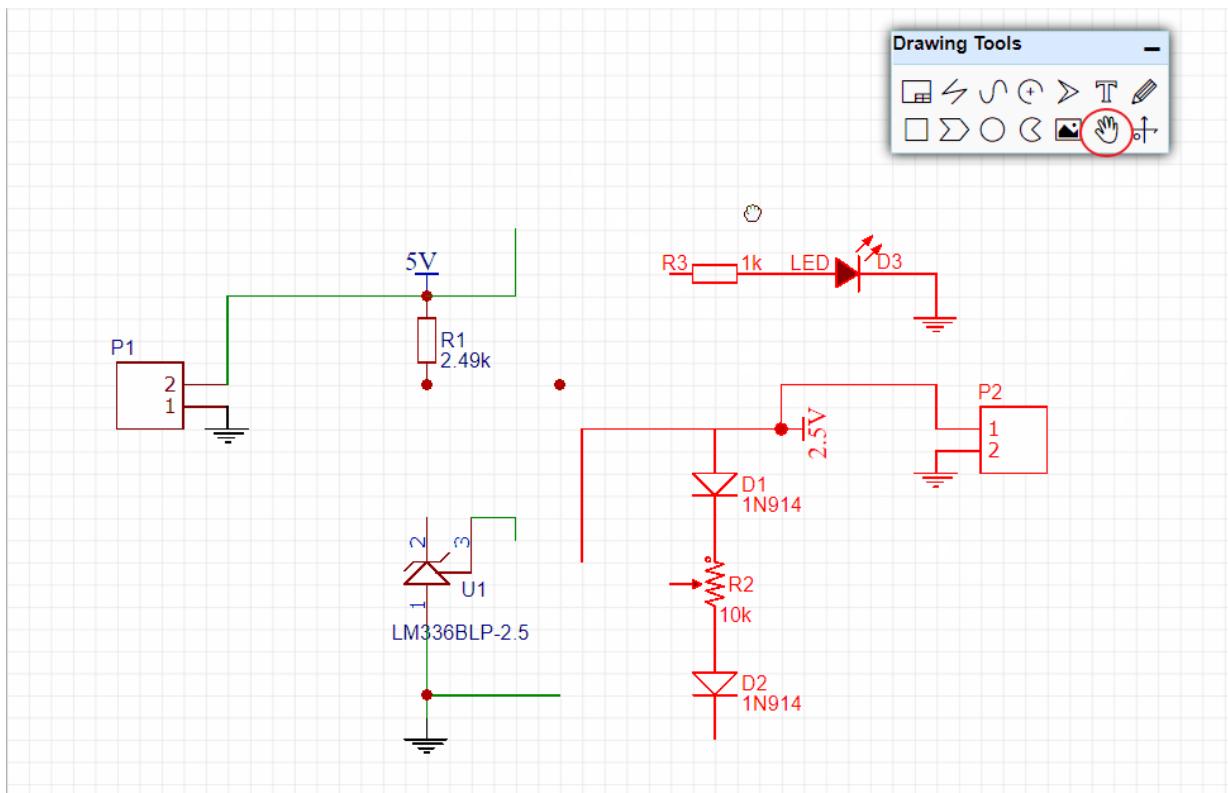
will make your image look like this:



Please note: at present, EasyEDA cannot host images, so you need to upload your images to an image sharing site such as <http://www.imgur.com>.

Drag

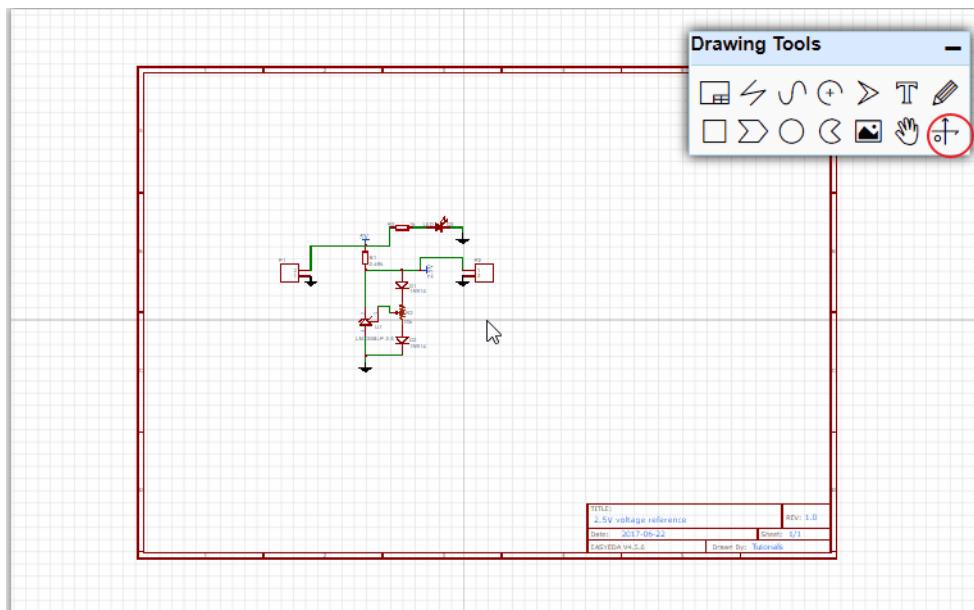
If you want to move some kind of parts and wires, you can use drag.
Or you can select the parts and wires area first and move them.



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.

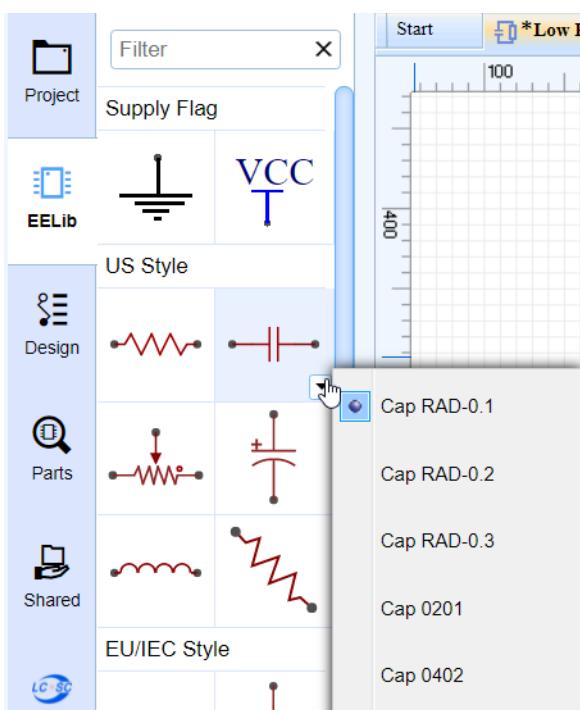
For another way to set canvas origin, you can try **Super Menu > Miscellaneous > Canvas Origin**.



Search symbols

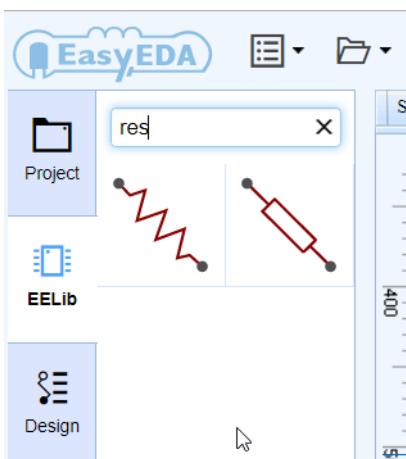
On the left hand Navigation panel you will find "**EELib**" and "**Parts**" ,

1) **EELib** contains ready made symbols for a wide range of components and which can be simulated.



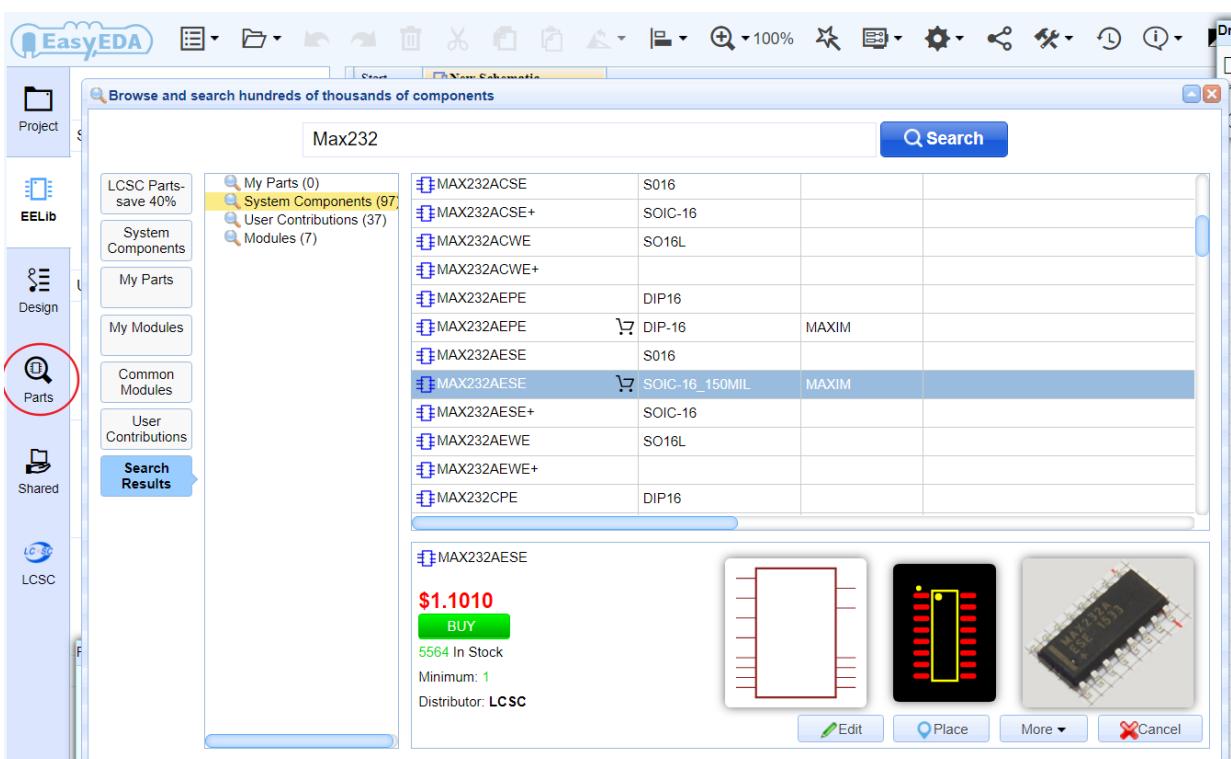
Many of these components have optional US and EU style symbols, we split them, so you can select those you like. Click on the drop down list or right click to popup the context menu, it contains many packages or parameters. EasyEDA will remember your choices for the next time.

Don't forget to use Filter to locate a component fast. For example, you just need to type `res` to find all of resistors:



2) **Parts**, or press the hotkey combination `shift+F`.

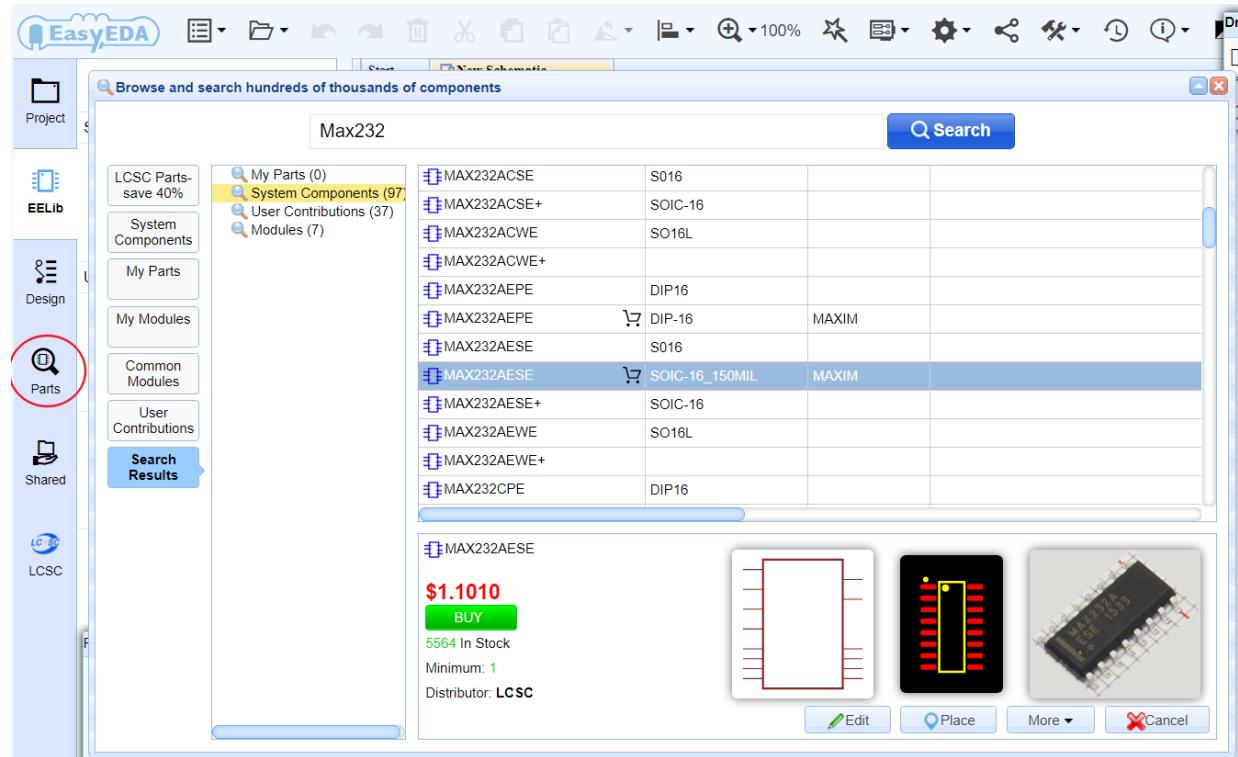
then you will see a dialog as shown in the image below. Simply type your part number or symbol's name to Search.



and then click the "Table of contents" to open the categories list to choose your components.

From there you can scroll up and down to browse parts from each category.

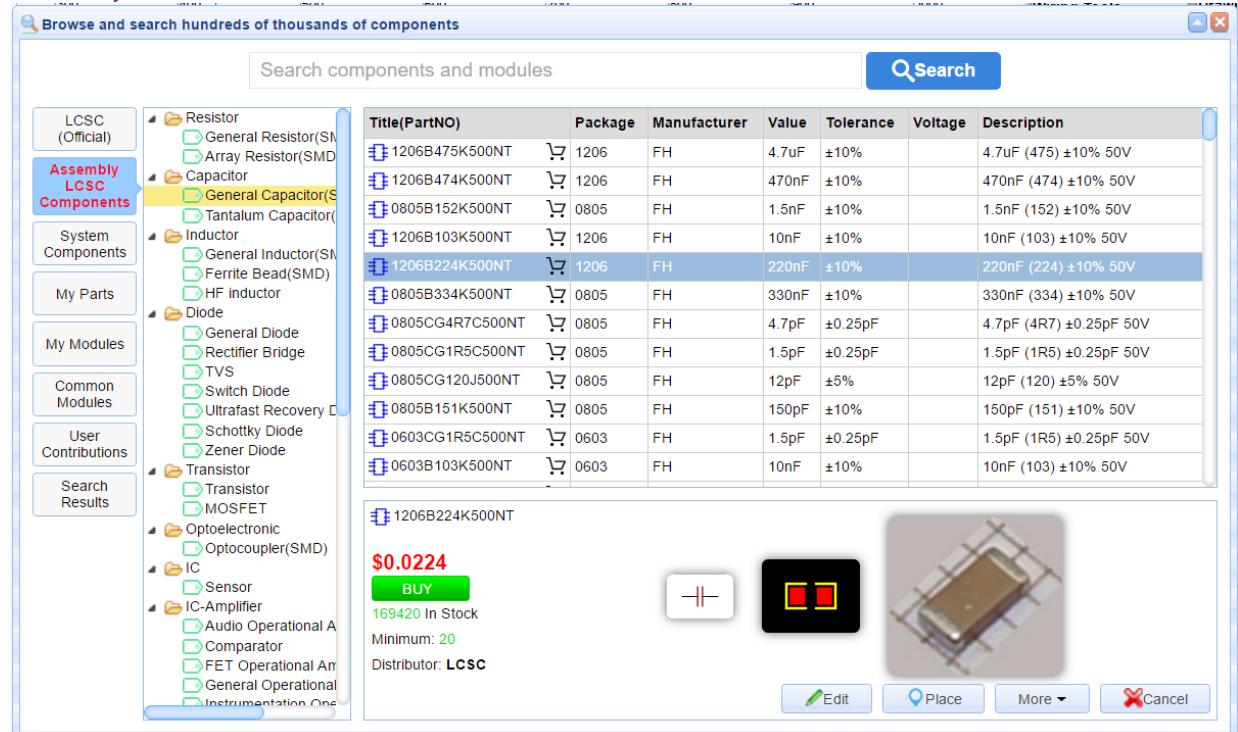
Suppose you wanted to find the **MAX232** (which converts signals from an **RS-232** serial port to signals suitable for use in **TTL** compatible digital logic circuits). Simply type **Max232** into the Search box and press Enter:



When you hover the mouse over the picture of the Schematic symbol or PCB footprint, you will find a toolbar with "Edit", "Place", "More" buttons.

LCSC Assembly Components

We add an LCSC Assembly Components option of the Parts, It's easy to choose which component can be assembled by LCSC. Yes, We provide the assembly service.



Place: For parts you use infrequently, you don't need to Favorite them; just Place it into your canvas directly.

Note:

- EasyEDA supports multi-documents so please make sure that you are placing the part into the right (active) document. The active document is the one with the highlighted tab.
- You can't place a Schematic symbol into a PCB file, or a PCB Footprint into a schematic.

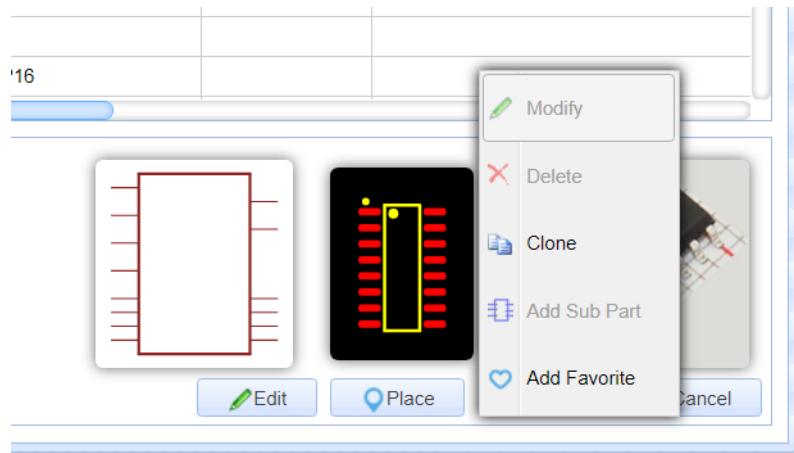
Edit: If you want to create your own version of a symbol or footprint then you can open an existing part from the library to use as a template, edit it and then save it to your local **My Parts** library in **Parts** of the Navigation Panel.

More: We can't promise that every component in the library is free of errors so please check all symbols and footprints carefully before you commit to a PCB order.

If you do find a mistake in a component, please [let us know](#)(mail to support@easyeda.com) so that we can fix it.

Components with sub parts (multi-device packages).

When you find a component with sub-parts, you can't Place or Edit it, but you can Favorite and Clone it as your own part, which you can then edit.

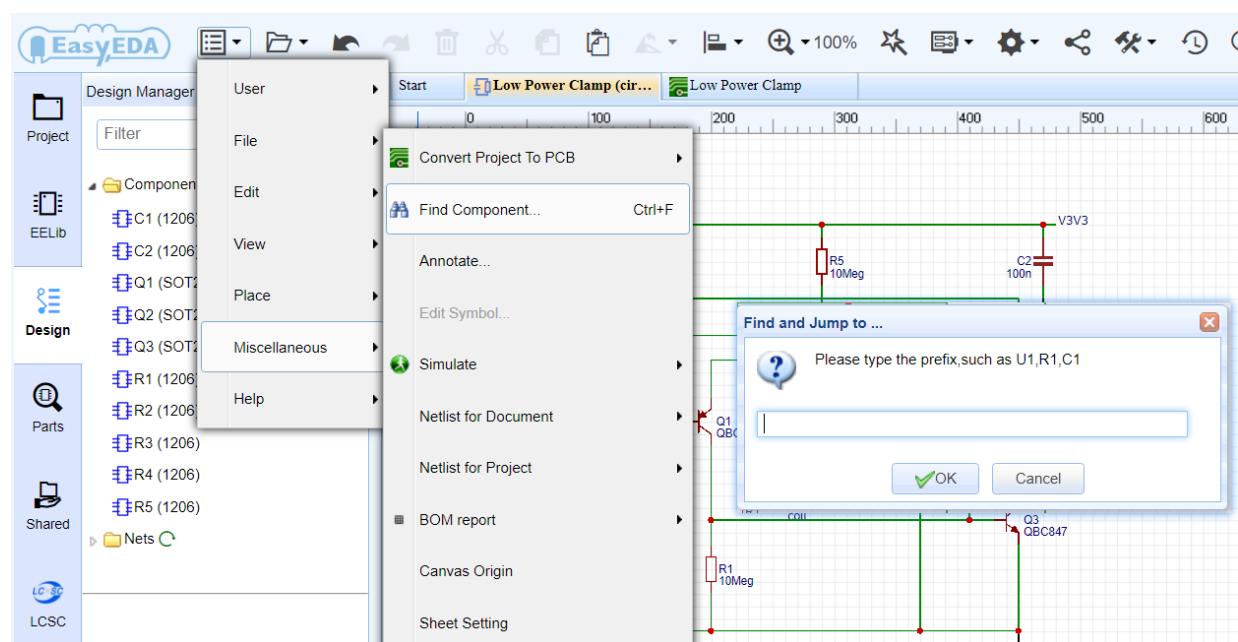


Find Components In The Schematic

Finding individual **components** in a dense schematic can be very time consuming. EasyEDA has an easy way to find and jump to components:

Super Menu > Miscellaneous > Find Component...

(or **Ctrl+F**)



Note: You have to click OK in this dialog or use the Enter key.

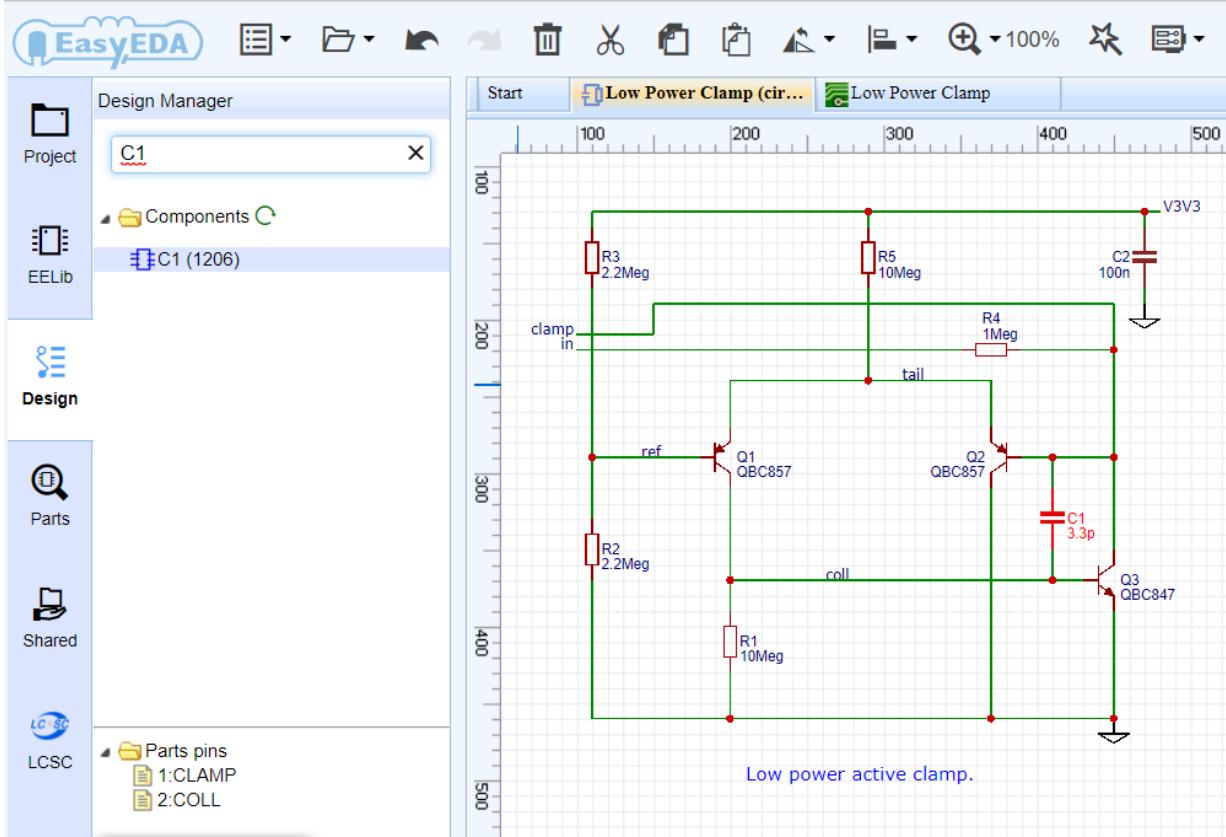
This feature will find, highlight and center in the window, parts by their Prefix (or reference designator). However, it cannot be used to find net names or other text in a schematic.

This is where the Design Manager comes in.

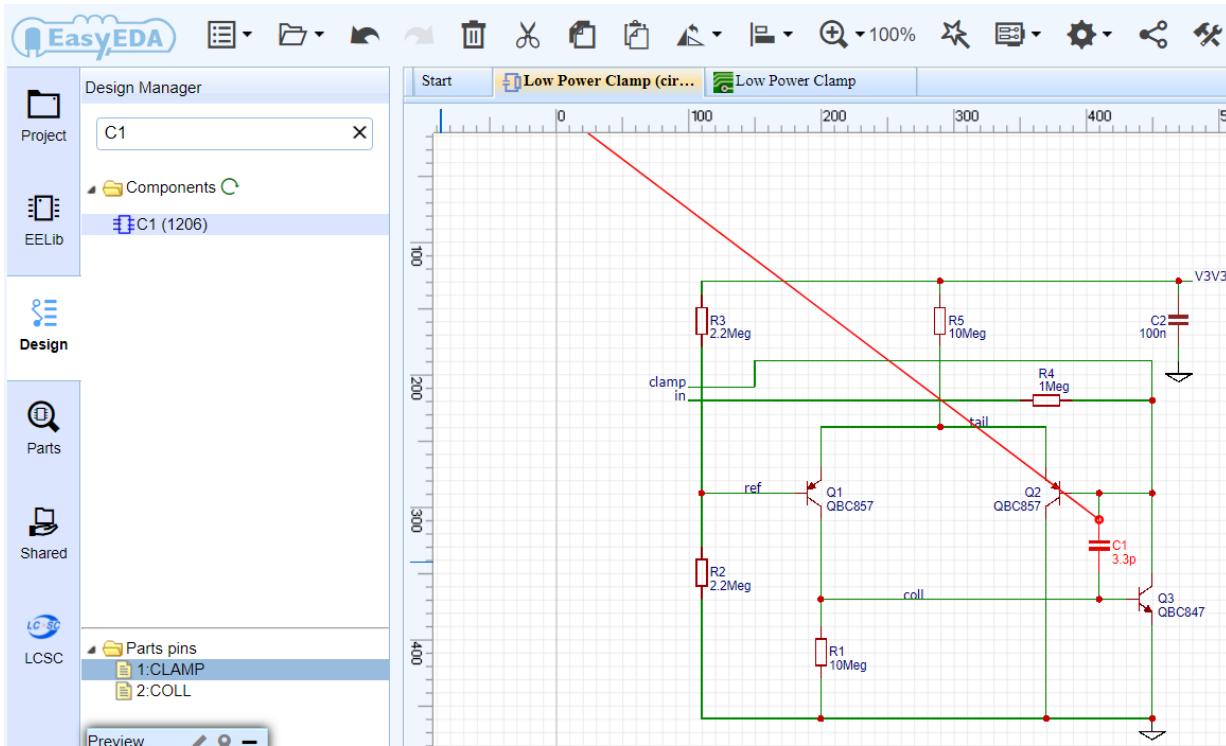
Left Navigation Panel > Design, or use hotkey **ctrl+D**.

The Schematic Design Manager is a very powerful tool for finding **components**, **nets** and **pins**.

Clicking on a Component item **highlights** the component and pans it to the center of the window.



Clicking on a Part pins item brings up a temporary pointer:



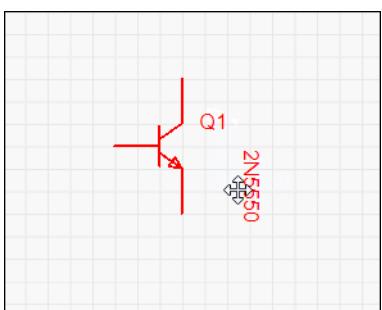
Placing Components

Find the component which you plan to place to your schematic, then move your mouse to the canvas and left click. If you want to add more, just left click again. To end the current sequence of placements, right click once or press **ESC**.

Don't try to Drag and Drop a component to the canvas: EasyEDA team thinks that Click-Click to place components will be easier to use than a Click-Drag mode.

Rotating the Prefix and Value (Name) of components

The default Prefix and Value (or name) of EasyEDA components are horizontal. To change them to vertical like this...

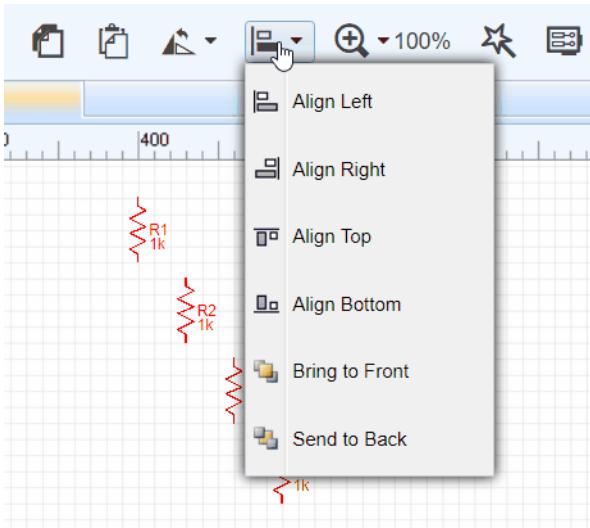


...Left click the prefix or value and when it is highlighted in red color, then press the rotation hotkey **Space** and you're done.

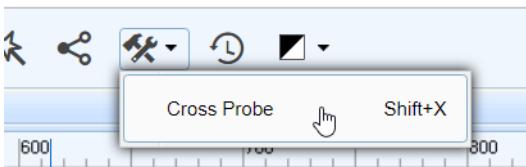
Adjusting Components

About adjusting components you can:

1. Move components with your mouse
2. Move components with the arrow keys.
3. Find components with the Design Manager via the **CTRL+D** hotkey: select the component in the Design Manager to pan it to the centre of the canvas and then move it with your mouse.
4. Align the components:

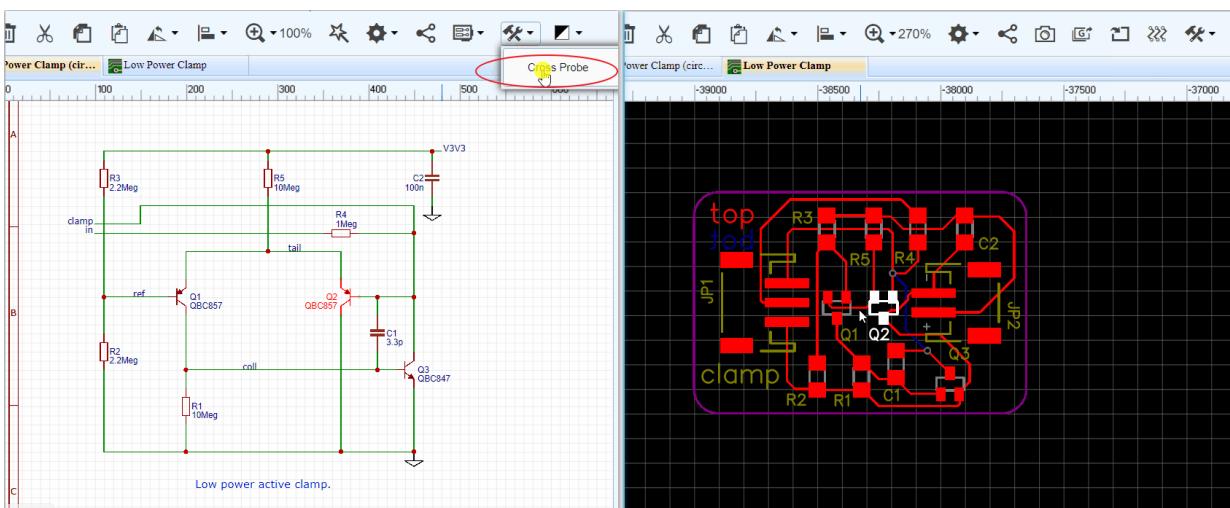


Cross Probe



This tool is used to cross probe from chosen objects on the current schematic to its corresponding counterparts in the PCB, or from PCB Footprints to corresponding counterparts in the schematic.

Note: You don't need to open PCB first before using cross probe in the schematic. Editor will open the PCB automatically.
And don't forget to use the hotkey **SHIFT+X**.

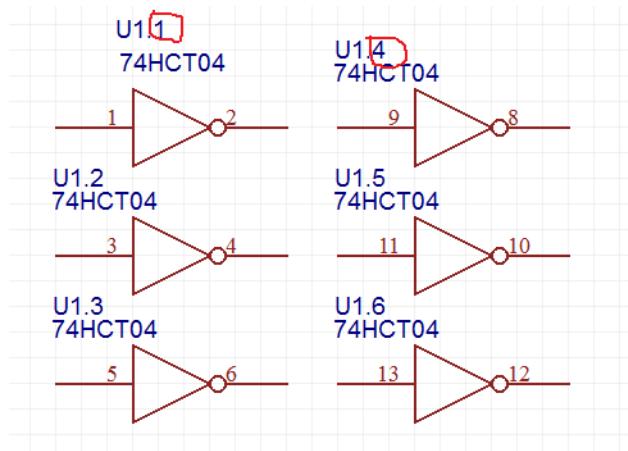


Multi-part Components

The number of pins on some components can be quite large. That's why it's easier to divide such a component into several parts or functional blocks. As a simple example, there are six gates in the 74HC04 Hex Inverter component. To avoid clutter in the schematic, GND and VCC pins of such components are usually served by a separate part of the component. This is really convenient as it doesn't interfere the working process with logical parts. The NetLabel names of VCC and GND Pin are usually hidden.

When placing the 74HC04 on a schematic, it will look like the screenshot below.

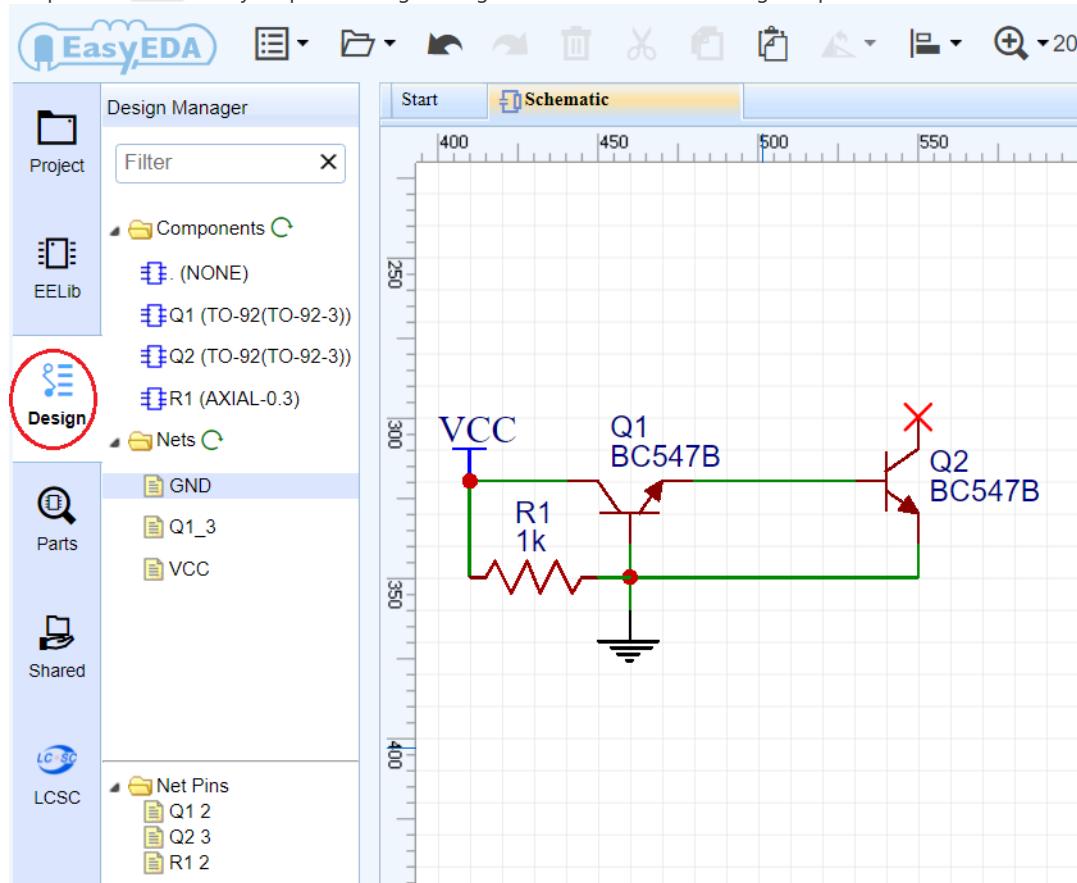
Note: The component Prefix will be in form of: U?.1, U?.2 etc.



Design Manager

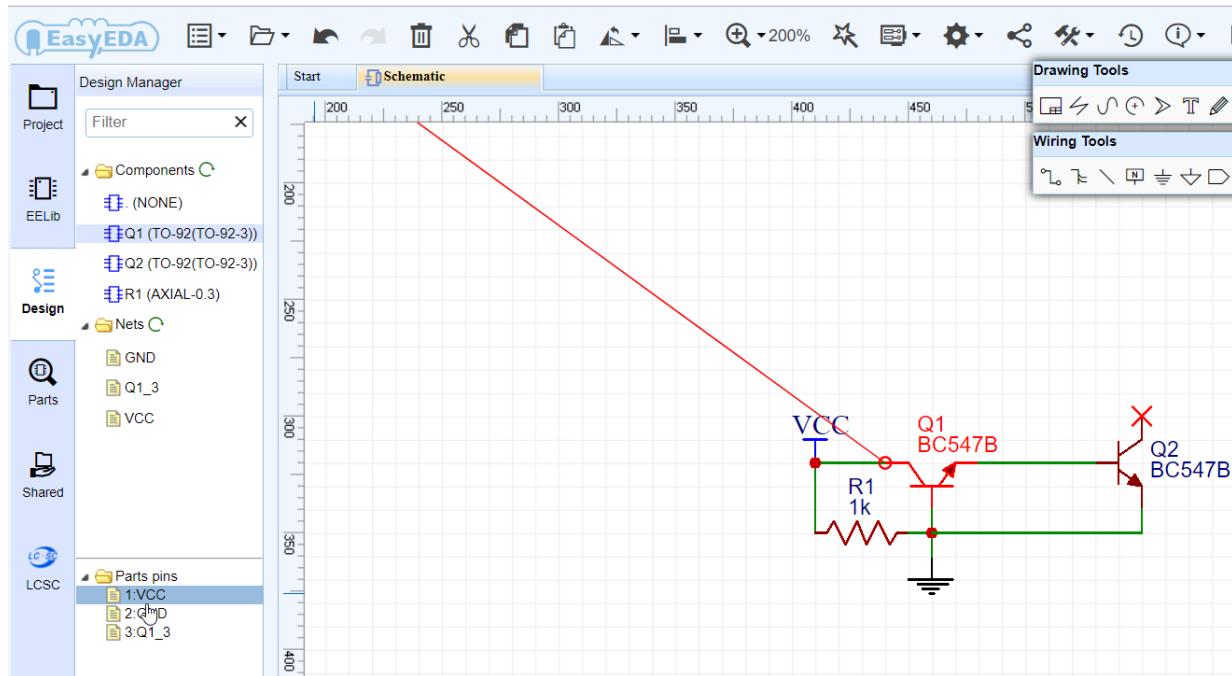
With large schematics it can be hard to find the components quickly. Sometimes, you may make a mistake such as wiring to a wrong component pin. So you need a tool to help you out. **Design Manager** is just the tool.

Just press the **CTRL+D** hotkey to open the Design Manager. or click it via on the left navigation panel:



1. **Filter:** You can find your components or net name easily: for example, if you want to find all capacitances, you just need to type **c**;
2. **Components:** Lists all the components in this schematic. Clicking on a Component item highlights that component and pans it to the center of the window.
3. **Nets:** Lists all the nets in this schematic. A net must connect at least two Pins, or the net name will be marked as a red error.
4. **Net Pins/Parts Pins:** Lists all the pins of the selected net name or components.

If you click the **Q1 Pin 1:VCC**, EasyEDA will show you where it is with a temporary marker from the top left of the canvas:

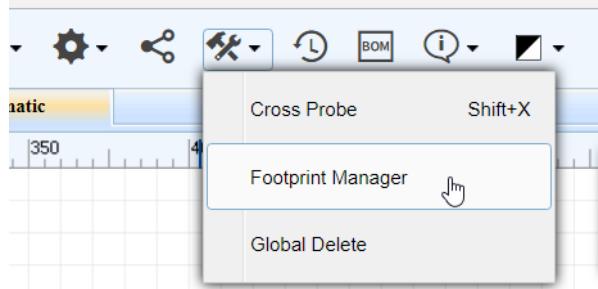


Footprint Manager

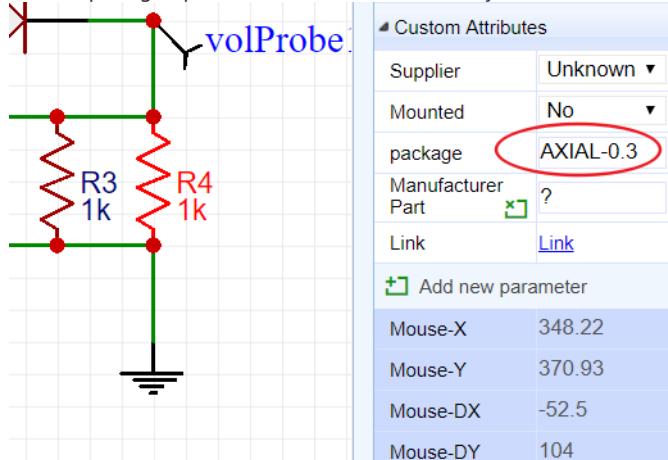
Want to batch modify components? Can't identify the corresponding relationship between component pins and footprint pins? Don't worry, EasyEDA can do this.

There are two ways to open the footprint manager:

- Click top toolbar Tools icon:



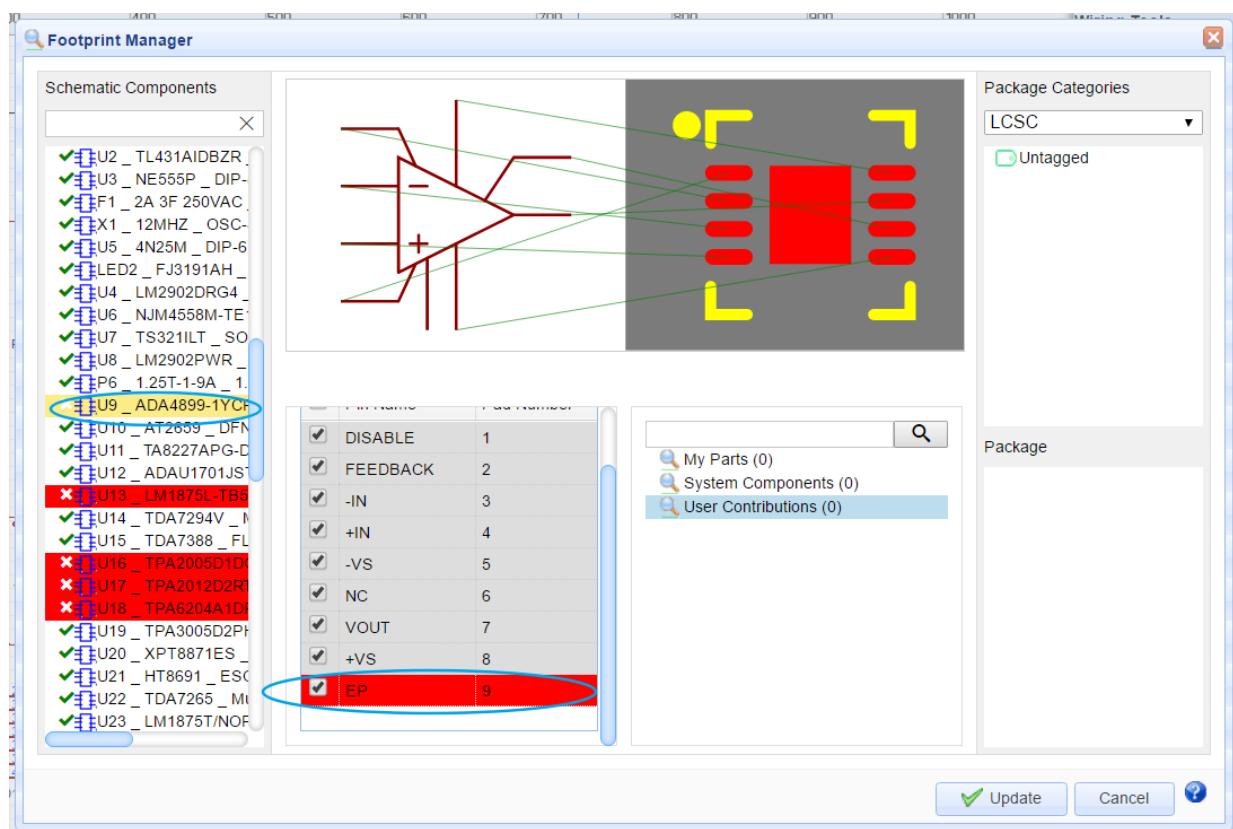
- Click the package input box of custom attributes when you've selected a component:



1. Footprint manager will check your parts package correct or not automatically. If the part without the package or this package doesn't exist in EasyEDA Libraries, or if the part's Pins doesn't correspond the package's Pads correctly, the footprint manager will show the red alert.
For example, If your part U1 has 2 pins, pin number are 1 and 2, pin name is A and B, but you assigned a footprint has 2 pads, **pad number** are A and B, but the part's pin number doesn't match the pad number, so the the footprint manager will alert red, in order to solve this:

- method 1: change part's pin number as A and B.
- method 2: change package's pad number as 1 and 2.
- method 3: find an other package and update.

2. In the preview area, you can zoom in, zoom out and pan with mouse.



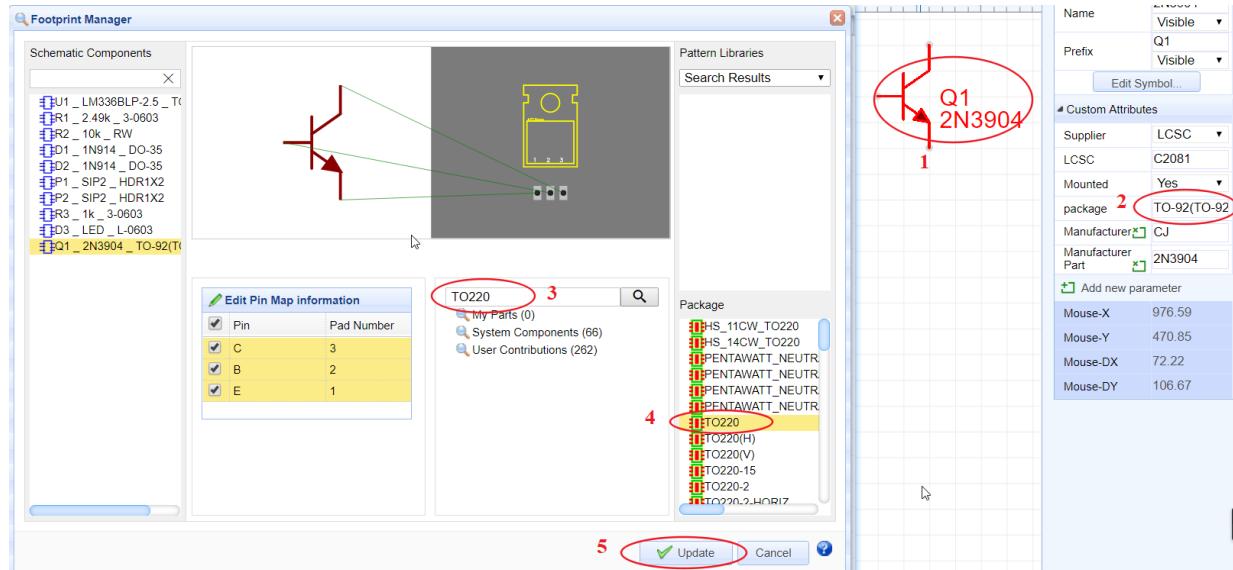
Update Package

If you want to change the Package, for example, select a component such as Q1, from **TO-92** TO **TO220**, you just need to click in the package input box. EasyEDA will popup the footprint manager dialog. You can follow the instructions.

- Type **TO220** into the search box and search,
- Select the **TO220** package,
- Verify it in the preview box,
- then press the **Update** button.

After that you will find you have changed the package to **TO220**.

Note: To ensure that you use a package type that is already in the EasyEDA libraries, it is recommended that you use this technique to change component packages rather than just typing a package type directly into the package text box.



Batch Update/Modify Packages

If you want to batch modify components' packages, in the footprint manager dialog you can press **CTRL** and select all components you want. If your schematic has many components, you should filter them first with package name. Such as in the below .gif which will show you how to batch modify resistors' packages from **AXIAL-0.3** to **0603**.

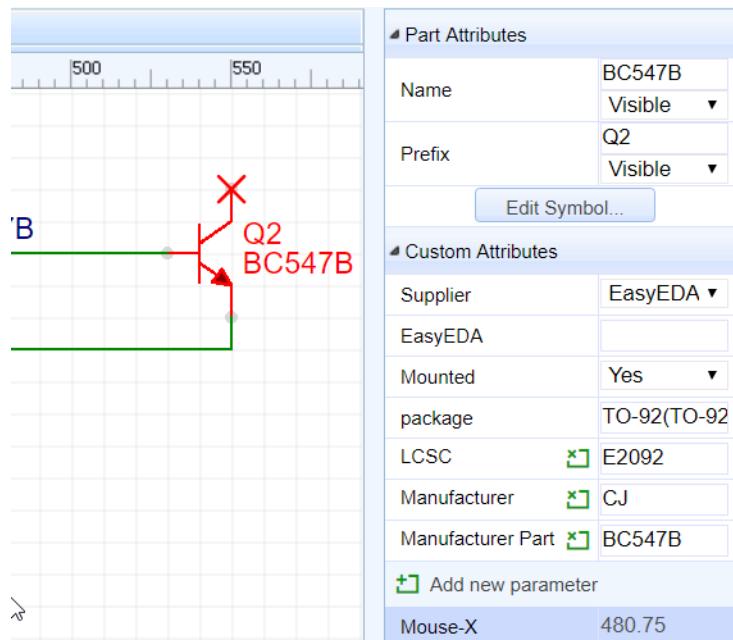
If you want to use your own packages, you can select **My Parts** on Pattern Libraries area.

Modify Pin Map Information

And you can modify component's pin map information in here.

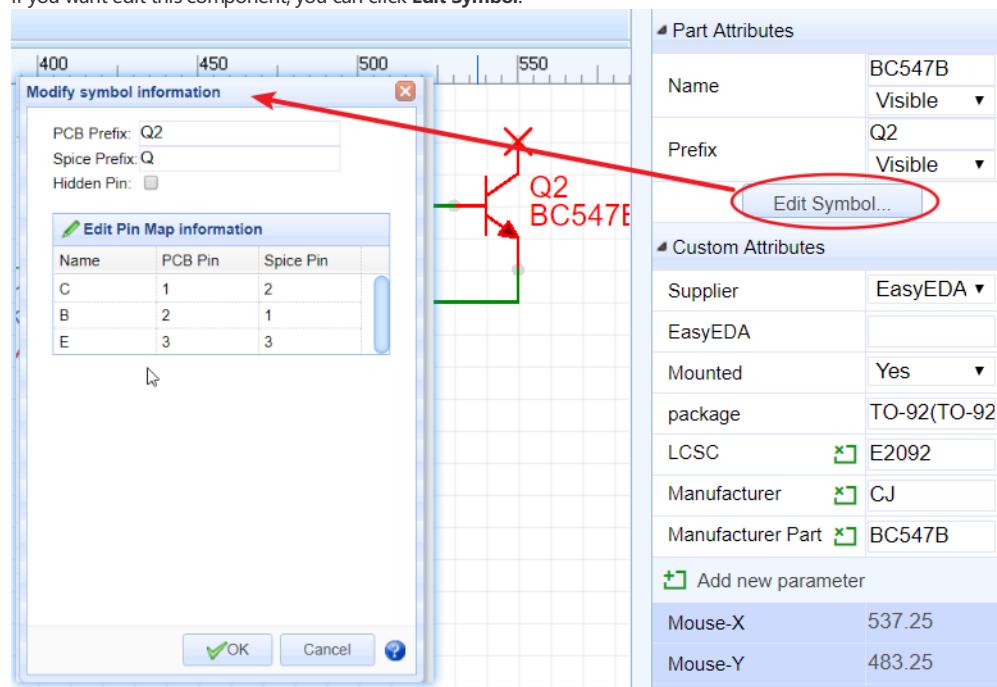
Component Attributes

After selecting a component, you can find the component's attributes in the right hand Properties panel.



1. Part Attributes: You can change the **Prefix** and **Name** here , And make them **visible** or **invisible**.

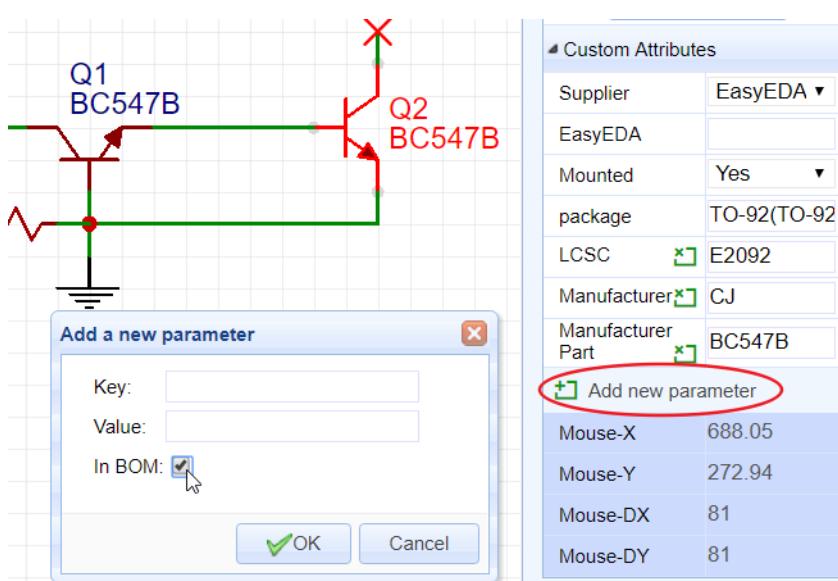
If you want edit this component, you can click **Edit Symbol...**.



2. Custom Attributes: You can change *component's supplier, mounted or not, change package, and add new parameter*.

Define BOM Parameters

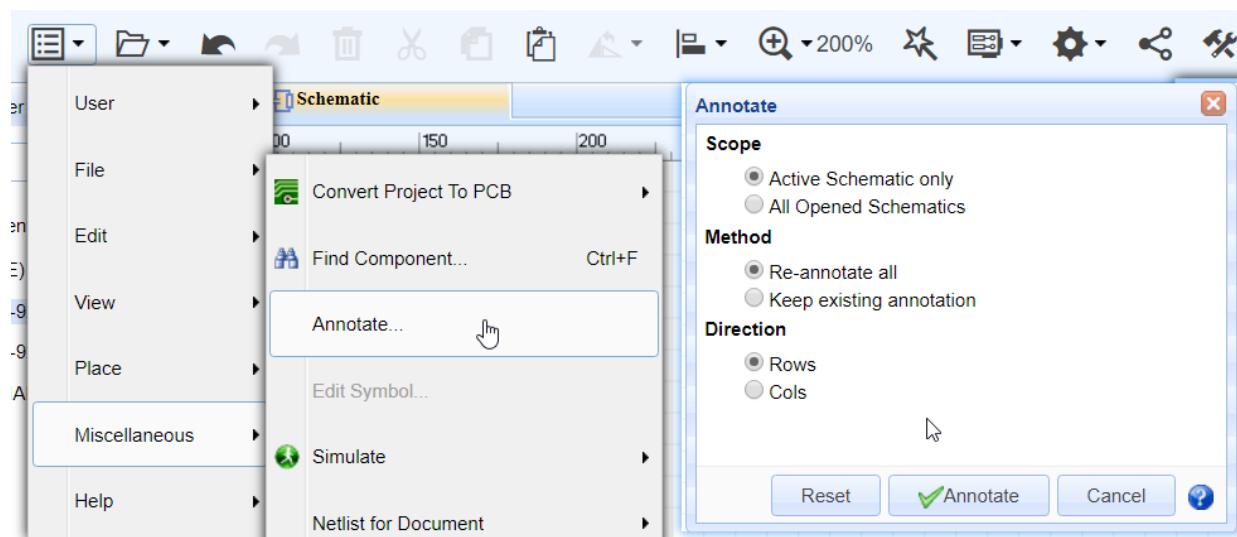
After selected a schematic symbol, you can add a parameter, and you can mark it as `In BOM`, when you export a BOM file, you can find this parameter in CSV file.



Annotate

After creating a schematic, it is quite likely that you have component Prefixes (reference designators) that are in no particular order on the canvas. You may also have duplicates. You can automatically renumber all the components using the **Annotate** function.

Super menu > Miscellaneous > Annotate



Various Annotate possibilities are available:

- **Active Schematic only:** applies annotation actions to the current schematic only.
- **All Opened Schematics:** applies annotation actions to all Schematics that are open in EasyEDA.

Note: This option applies even if the opened schematics are from different Projects! If the project that you want to annotate has more than one schematic, you should open all of them and close any schematics that are open from other Projects.

- **Re-annotate all:** resets all existing annotation and then annotates all components again from scratch;
- **Keep existing annotation:** annotates new components only (i.e. those whose reference designator finishes with ? like R? or U?).
- **Direction:** Rows annotates across the schematic in a raster pattern from top left to bottom right; Cols annotates down the schematic in a raster pattern from top left to bottom right.
- **Annotate:** applies the selected annotation actions.

Note: Annotation cannot be undone! if you do not accept the result: close all of the affected schematics without saving. If you do accept the result: make sure you save all of the affected schematics.

- **Reset:** if you want to reset all the reference designators to end with '?', just click the Reset button. After that, R1 will be R?, U1 will be U?, etc.

Note: Reset does not reset annotation back to where it was before pressing the Annotate button.

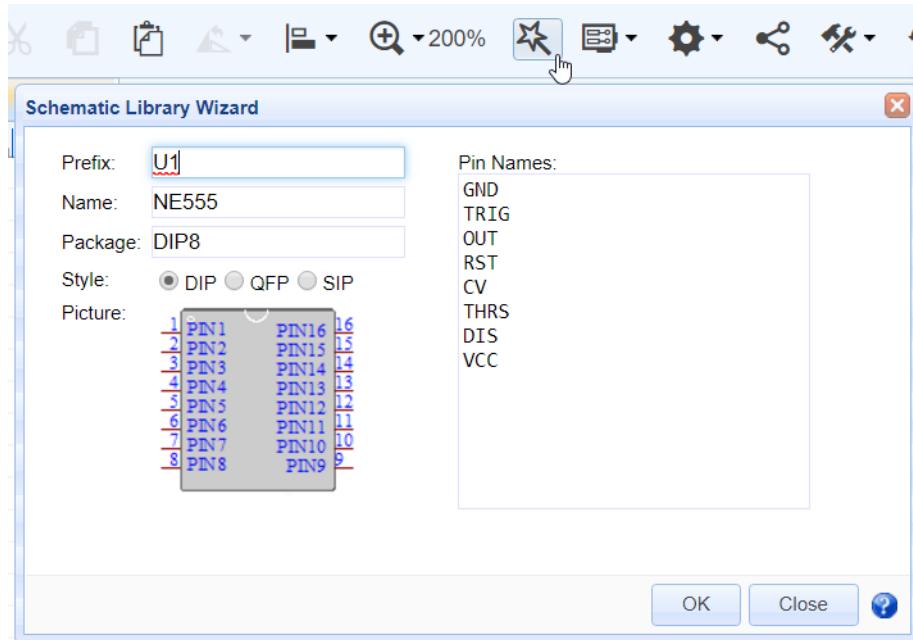
Symbol Wizard

How many times have you hit a schematic capture roadblock because you couldn't find a component symbol?

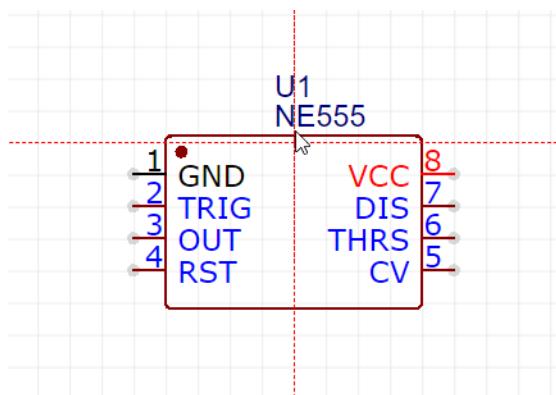
Well, in EasyEDA that would be never because the **Symbol Wizard** provides a quick and easy way to create a general schematic library symbol.

The **Symbol Wizard...** command can be found in the top toolbar.

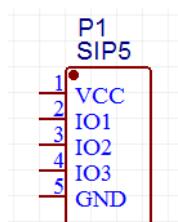
Or **Super Menu > Miscellaneous > Schematic Library Wizard** in a new schematic lib document.



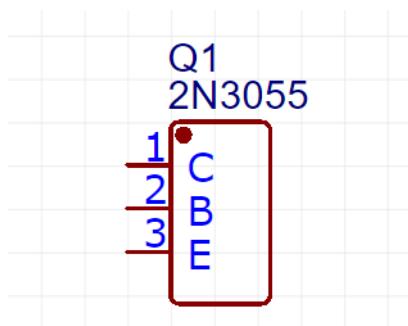
Using the **NE555** timer as an example: this device is available in a **DIP8** package so select **DIP**. Then enter the NE555 pin names into the **Pin Names** text box separated by new line or space, Then press **OK**. Abracadabra! As if by magic, you will find a perfectly formed dual in line 8 pin symbol for the NE555 attached to your mouse cursor, ready to be placed! You just need a few seconds to build a NE555 symbol, quickly and easily.



The EasyEDA Schematic Symbol Wizard allows you to create DIP, QPF or SIP styles symbols. If you are designing Arduino Shields then you will need lots of SIP symbol, so you can create a SIP symbol like the one shown below in a few seconds.



If you are not too worried that the symbols may not look quite the way people might expect and that they may not look anything like the **Package** type you enter, then of course you can use the wizard to create symbols for any component:

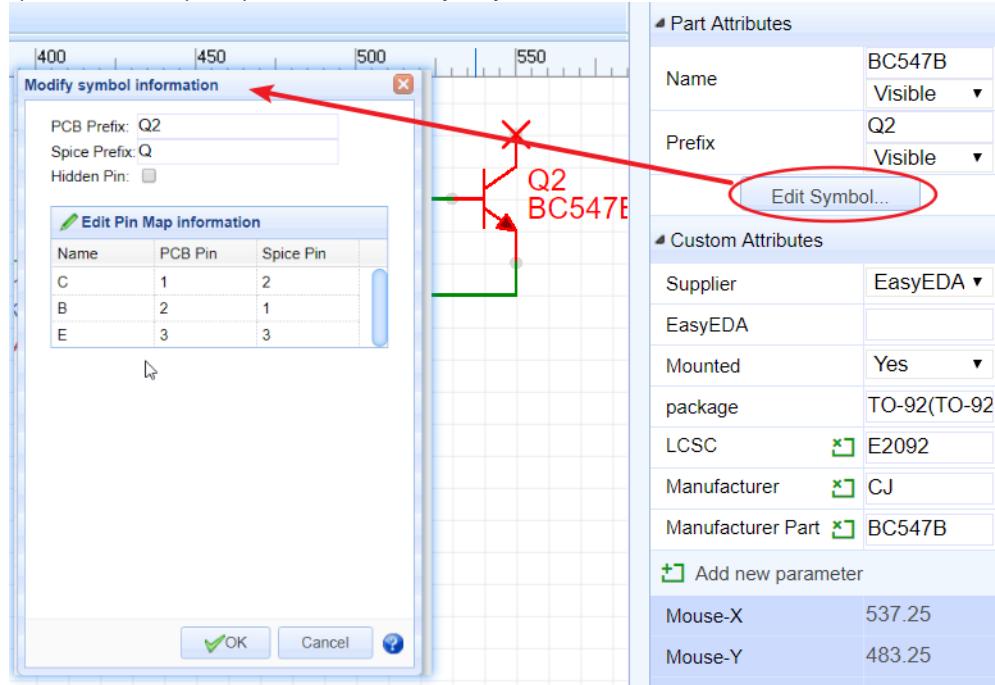


Pinmap Modify symbol information

When you select a component, for opening the Modify symbol information dialog, you can do:

- 1.Super menu > Miscellaneous > Edit Symbol...;
- 2.Or press the **I** hotkey;
- 3.Or click the **Edit Symbol** on the Parts Attributes on the left panel.

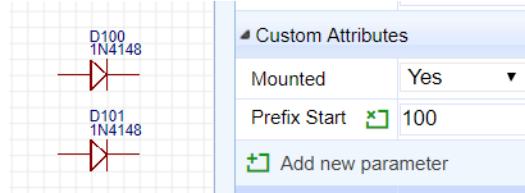
Using this dialog you can edit the pin names and numbers, for example, to suit a different package or device variant. You can also enter a Spice Prefix and swap the spice Pin order to make your symbol usable in simulation.



More detailed description of PCB and Spice Prefixes and pin numbers at next section.

Prefix Start

Every **NEW** schematic file has a `Prefix Start` custom parameter, some users would like use **multi-sheet designs**, but they hate every prefix start by 1, they hope one schematic start by 1, next start by 100, 200, 300. So you can use this solution.



Prefixes And Pin Numbers

Device and subcircuit (or hierarchical block) symbols created for use in schematics that are intended to be run as spice simulations, in addition to having a PCB Prefix that is used for the reference designator in the schematic, also have a **Spice Prefix**. They also have two sets of pin numbers: PCB pins and Spice pins.

PCB And Spice Prefix

The rules on the assignment of the PCB Prefix or reference designator of a schematic symbol are somewhat dependent on the EDA tool and on the user's preferences. Depending on how a device is graphically represented by its schematic symbol it may have a different **PCB Prefix** or** reference designator**. For example, a single discrete MOSFET device may have a PCB Prefix of Q, M or perhaps TR, whereas if it is part of a monolithic multiple transistor array it may have a PCB Prefix of U or IC.

The rules on the assignment of the **Spice Prefix** of a schematic symbol are strict. This is because the Spice Prefix is used to tell the simulator which circuit element the symbol represents and therefore which simulation model it is to use.

Simulation models for most of the spice circuit elements are in the form of a single-line .model statement however some of them may be in the form of a multi-line .subckt subcircuit definition. For example, some MOSFETs may be described by a .model statement in which case their Spice Prefix is **M** but many MOSFETs are described by a .subckt and so their Spice Prefix is **X**.

Therefore, irrespective of the PCB Prefix chosen for a schematic symbol, the Spice Prefix for a schematic symbol representing a given circuit element must match the type of model required to simulate that instance of that circuit element in your schematic.

For example, if you have two different n-channel MOSFETs in a schematic; Q1, a BSS123 which is modelled by a .model statement:

BSS123*SRC=BSS123;DI_BSS123;MOSFETs N;Enh;100V 0.170A 1.00ohms Diodes Inc.

MOSFET

```
.MODEL DI_BSS123 NMOS( LEVEL=1 VTO=1.00 KP=6.37m GAMMA=1.24
+ PHI=.75 LAMBDA=625u RD=0.140 RS=0.140
+ IS=85.0f PB=0.800 MJ=0.460 CBD=19.8p
+ CBS=23.7p CGSO=36.0n CGDO=30.0n CGBO=124n )
* -- Assumes default L=100U W=100U --
and Q2, a BSS127S which is modelled by a .subckt:
BSS127S*----- BSS127S Spice Model -----
.SUBCKT BSS127S 10 20 30
* TERMINALS: D G S
M1 1 2 3 3 NMOS L = 1E-006 W = 1E-006
```

```

RD 10 1 84.22
RS 30 3 0.001
RG 20 2 29
CGS 2 3 1.958E-011
EGD 12 0 2 1
VFB 14 0 0
FFB 2 1 VFB 1
CGD 13 14 2E-011
R1 13 0 1
D1 12 13 DLIM
DDG 15 14 DCGD
R2 12 15 1
D2 15 0 DLIM
DSD 3 10 DSUB
.MODEL NMOS NMOS LEVEL = 3 VMAX = 8E+005 ETA = 1E-012 VTO = 3.419
+ TOX = 6E-008 NSUB = 1E+016 KP = 0.127 U0 = 400 KAPPA = 1.044E-015
.MODEL DCGD D CJO = 1.135E-011 VJ = 0.9232 M = 0.9816
.MODEL DSUB D IS = 2.294E-010 N = 1.601 RS = 0.1079 BV = 65
+ CJO = 1.956E-011 VJ = 1.514 M = 0.8171
.MODEL DLIM D IS = 0.0001
.ENDS
*Diodes BSS127S Spice Model v1.0 Last Revised 2012/6/6

```

then even though both have the same PCB Prefix of **Q**: **Q1** must have a Spice Prefix of **M** and **Q2** must have a Spice Prefix of **X**.

A list of Spice Prefixes and their associated circuit elements is given in the table below.

Spice Prefix	Element description	Comment
A	XSPICE code model	analogue, digital, mixed signal
B	Behavioral (arbitrary) source	
C	Capacitor	
D	Diode	
E	Voltage-controlled voltage source (VCVS)	linear, non-linear
F	Current-controlled current source (CCCS)	linear
G	Voltage-controlled current source (VCCS)	linear, non-linear
H	Current-controlled voltage source (CCVS)	linear
I	Current source	
J	Junction field effect transistor (JFET)	
K	Coupled (Mutual) Inductors	
L	Inductor	
M	Metal oxide field effect transistor (MOSFET)	
N	Numerical device for GSS	
O	Lossy transmission line	
P	Coupled multiconductor line (CPL)	
Q	Bipolar junction transistor (BJT)	
R	Resistor	
S	Switch (voltage-controlled)	
T	Lossless transmission line	
U	Uniformly distributed RC line	
V	Voltage source	
W	Switch (current-controlled)	
X	Subcircuit	
Y	Single lossy transmission line (TXL)	
Z	Metal semiconductor field effect transistor (MESFET)	

For more information on circuit elements in Ngspice, please refer to:

<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.21.2>

PCB and Spice pin numbers

The two sets of pin numbers are:

- **PCB pin number:** these are the numbers for the real, physical device pins in its package. They are required so that the pins of a device symbol in a schematic can be mapped onto the physical pins of a PCB footprint. In other words, so that the connections shown in the schematic, end up connected properly by copper on the PCB.
- ** Spice pin number or pin order:** these are the numbers that map the pins on the symbol to their respective functions in the spice model or subcircuit.

Actually the spice pin ordering has a slightly deeper meaning.

Spice has no concept of component symbols: they are a construct of the schematic editor.

When a spice netlist is generated, the symbol in the schematic editor is either - in the case of model defined devices such as resistors, capacitors, inductors, diodes, transistors and sources - mapped directly to the relevant models (defined by the device prefix such as R, C, L, D, Q and so on), or in the case of a subcircuit, converted into a subcircuit call statement.

The spice pin ordering for the majority of built-in models such as resistors, capacitors, inductors, diodes, transistors and sources are defined and generally taken care of by the schematic editor, more care has to be taken with the spice pin ordering of subcircuits.

This can be illustrated by a simple opamp with 5 pins: inverting and non-inverting inputs; output and positive and negative supply pins but the principle applies to all spice subcircuits.

The subcircuit call for this opamp might look like this in the spice netlist:

```
X1 input feedback vpos vneg output opamp_ANF01
```

where:

X1 is the name of the subcircuit in the top level (i.e. the calling) circuit;

input feedback vpos vneg output are the netnames in the circuit calling (i.e. containing) the subcircuit and

opamp_ANF01 is the name of the subcircuit being called.

Pay special attention to the order of the netnames in the subcircuit call.

The spice pin ordering for the majority of opamp subcircuits is like that shown

in the example below:

```
*
* opamp_ANF01
*
* Simplified behavioural opamp
*
* Node assignments
*
*          noninverting input
*
*          |    inverting input
*
*          |    |    positive supply
*
*          |    |    |    negative supply
*
*          |    |    |    |    output
*
*          |    |    |    |
*
* spice pin order:  1   2   3   4   5
*
*          |    |    |    |
.
.subckt opamp_ANF01 inp inn vcc vee out ; these are the netnames
*
*          used internally to the
*
*          subcircuit.
B1 out 0
+
V=(TANH((V(inp)-V(inn))*{Avol}*2/(V(vcc)-V(vee)))*(V(vcc)-V(vee))
+
+(V(vcc)+V(vee))/2
*
.
.ends opamp_ANF01
*
```

Note: The spice pin order of the subcircuit call is in exactly the same order as that of the subcircuit.

Although the physical pin numbering of any device is critical for successfully mapping the pins on a schematic symbol onto a physical package footprint when laying out the PCB, because spice only knows about single devices and does not care about how they are physically packaged, each instance of any device in a spice schematic has to be mapped onto its own copy of the spice model or subcircuit, irrespective of where it is in any physical package.

Therefore, for the physical, package pin numbering of the four opamps in a quad opamp in say, a **SOIC14** or a **DIP14** package, as shown below, to work with the example subcircuit above, the spice pin ordering would be:

Opamp A	pin number	spice pin order
OUT	1	5

IN-	2	2
IN+	3	1
V+	4	3
V-	11	4
Opamp B	pin number	spice pin order
OUT	7	5
IN-	6	2
IN+	5	1
V+	4	3
V-	11	4
Opamp C	pin number	spice pin order
OUT	8	5
IN-	9	2
IN+	10	1
V+	4	3
V-	11	4
Opamp D	pin number	spice pin order
OUT	14	5
IN-	13	2
IN+	12	1
V+	4	3
V-	11	4

The physical package pin numbering reflects that of each opamp in the package.

The spice pin ordering is the same for each instance of the individual opamps.

Of course there is only one physical instance of each supply pin on the schematic symbol for the quad opamp in this example but each spice subcircuit must have the supply pins explicitly defined.

Exactly how this is handled is at the schematic symbol level depends on how the schematic capture package handles symbols for multiple devices with shared supply pins but the generation of a spice netlist from the schematic will always generate the complete set of pins required in the subcircuit calls.

In cases where the subcircuit is built by the user as opposed to where it is supplied by a vendor for a particular device, exactly the same rules apply except that it is up to the user to specify the subcircuit pin order and to construct the symbol appropriately.

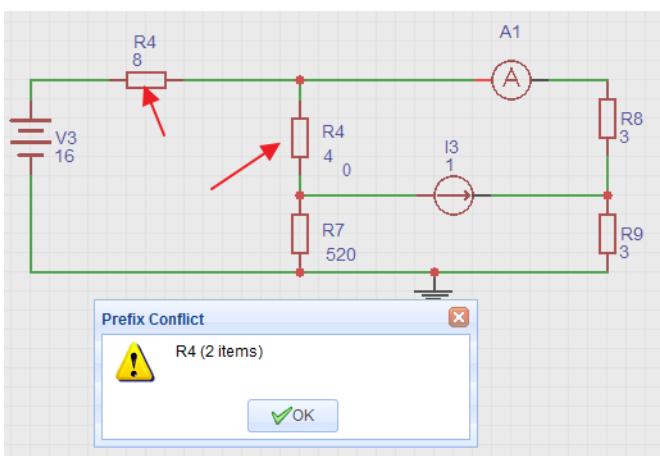
Although as described earlier, built in spice models usually have defined spice pin orders, not all subcircuits have the same spice pin numbering. Therefore if your spice circuit throws errors - especially if there are warnings about pin numbers or pin names - it is worth remembering to check that the pin order of the symbol that is netlisted to form the calling statement matches that of the subcircuit that is being called!

[8] In Debian based distributions gerbv is listed under Electronics in the package management system. The version in the repositories may be an earlier version but some users may find it easier to install than the SourceForge archive.

[9] As is the opamp_ANF01 example above

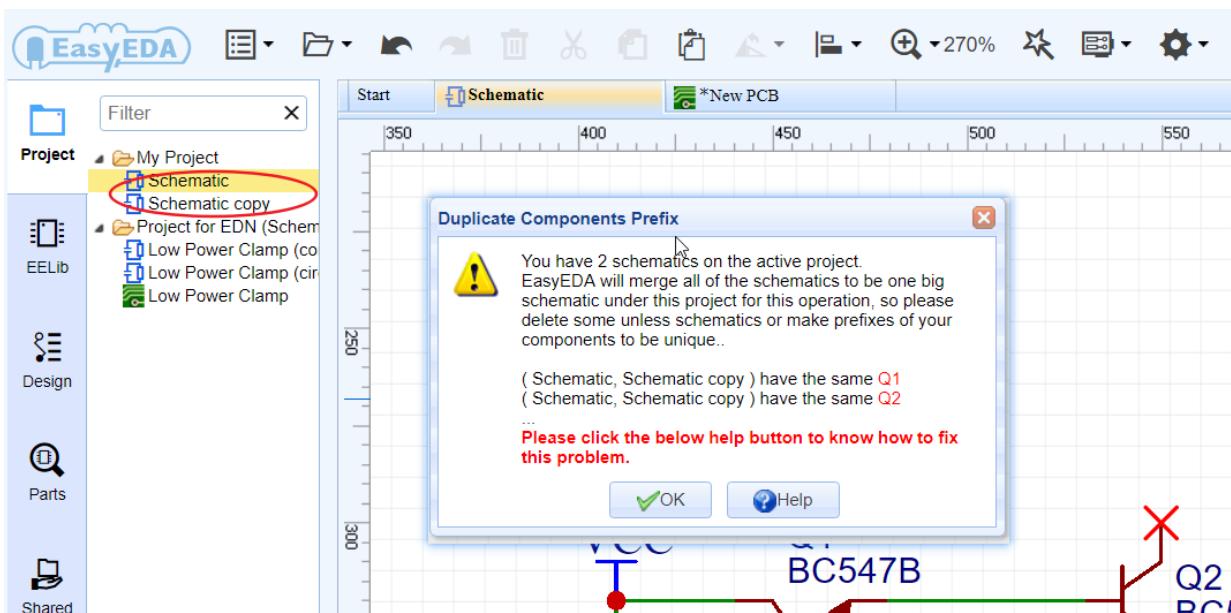
Prefix Conflict Error

Sometimes, when you convert a project to PCB, open the Design manager or run a simulation, you will get a Prefix Conflict error message.



In this schematic, you will find two components with the R4 reference designator, so you just need to change one to Rx where x is a unique number in that schematic.

It may be tempting to backup a schematic into the same project as the original, however, if an attempt is then made to do Convert Project to PCB, you will get the Prefix Conflict error for every component.

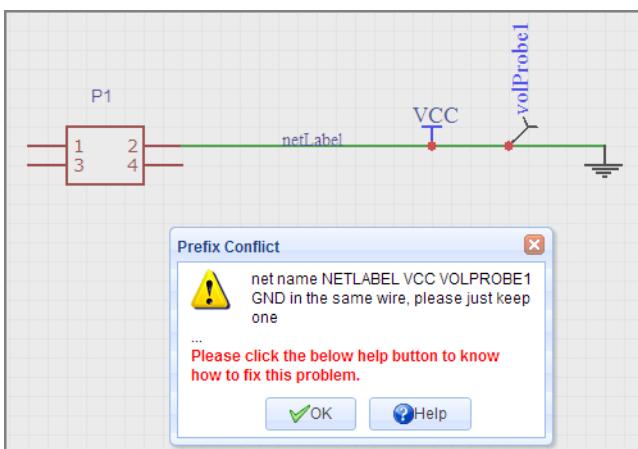


In the above image, you can find the two identical copies of the same schematic, which when you Convert Project to PCB, EasyEDA will try to merge into a single schematic, so every item will have 2 copies.

To fix this, you just need to create a backup project and remove or better still save backup copies of your schematics to that project.

Net Name Conflict Error

Sometimes, when you convert project to PCB, open the Design manager or run a simulation, you will get a **Net Name Conflict** error message.



In this schematic, you will find four net label/net flag (EasyEDA takes volprobe, GND VCC as netlabel too) in the same wire, So you must remove the others.

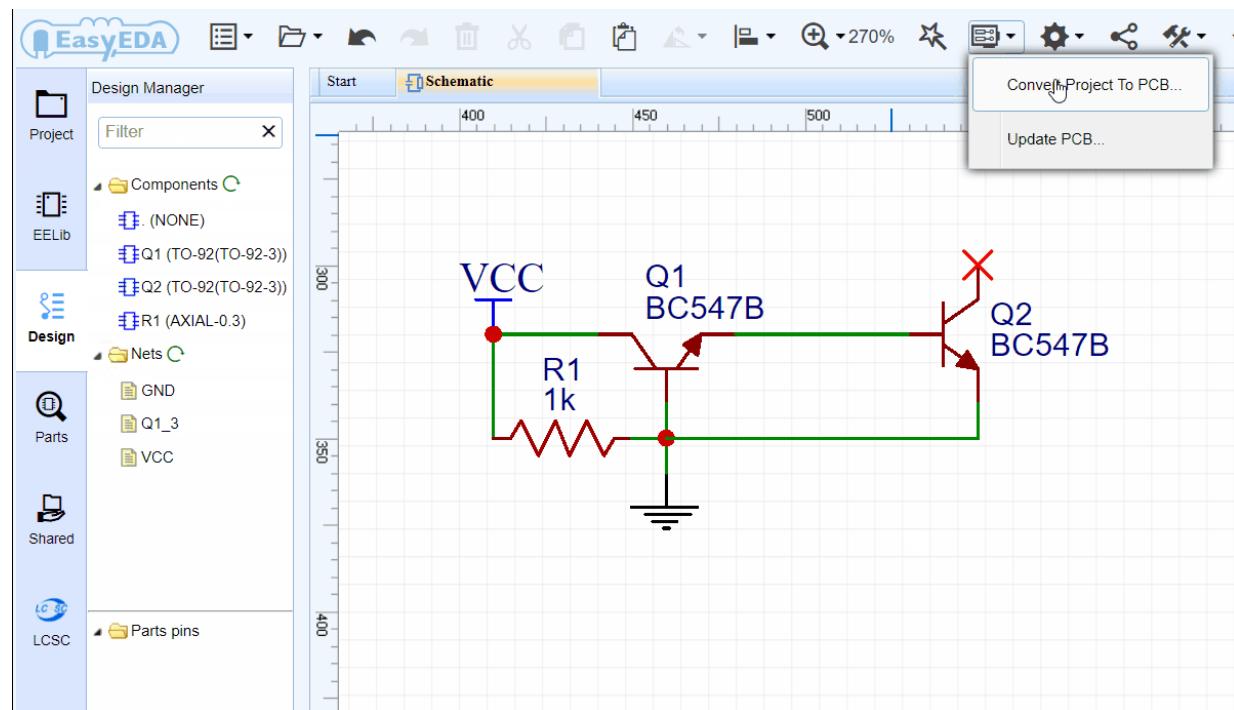
If you would like to probe a GND, you can use [Probe command](#)

Converting Schematics To PCB

Before using "Convert to PCB", "Update PCB" in Schematic and "Import Changes" in PCB, please read [Essential Check](#) section.

Most of the time, schematics are created with the aim of producing a PCB. So how do you convert your schematic to a PCB in EasyEDA? You just need to click the PCB icon on the toolbar with the title **Convert project to PCB**.

Note: Before converting, you need to use the Design Manager and Footprint Manager to check all the components, nets(connection) and packages/footprints to ensure no errors exist.



PCB Libs search order

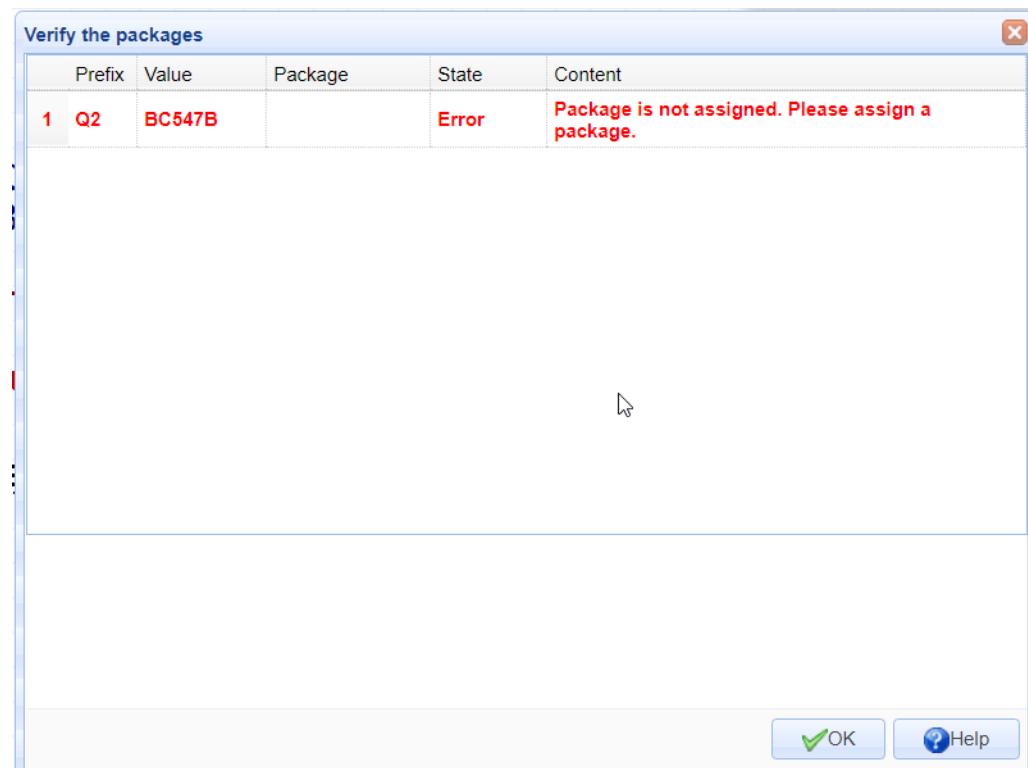
To find PCB footprints to match the package type attributes of your Schematic symbols, EasyEDA will search the available PCB libraries.

EasyEDA will search your own PCB Libs from the **My Parts** section first. If it doesn't find a matching footprint there then it will search in the System PCB Libs.

So, for example, if your symbol calls up a "TO220" package, you have a TO220 package in your My Parts section and there is a "TO220" package in the system PCB Libs, then EasyEDA will use the "TO220" package in your My Parts and ignore the system PCB Lib.

Verify Packages and Build PCB

After clicking the **Convert project to PCB** button, if the project has errors the following dialog will open:

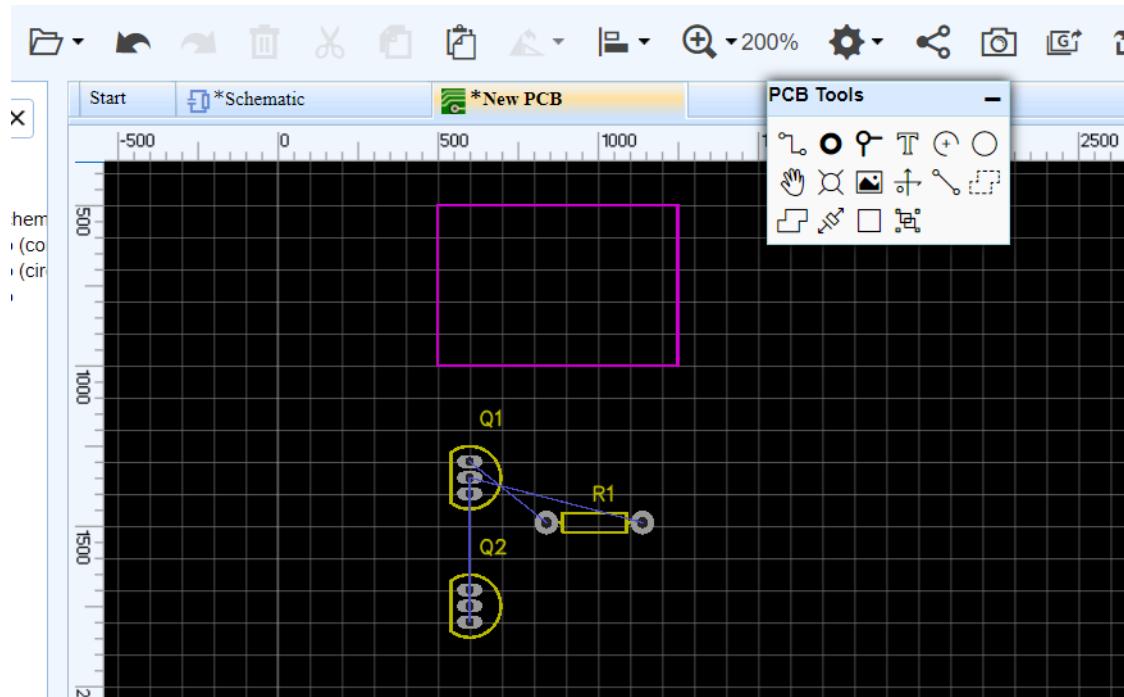


The row in red indicates that EasyEDA can't find a PCB footprint matching the Package that the schematic symbol is calling for.

This could be because you have made an error entering the package attribute in the symbol's Properties or maybe you haven't yet created a PCB footprint for the package that your symbol is calling for.

In this case the package should have been **TO-92(TO-92-3)** but instead it is empty. To correct it you can click on the row and change it to **TO-92(TO-92-3)**.

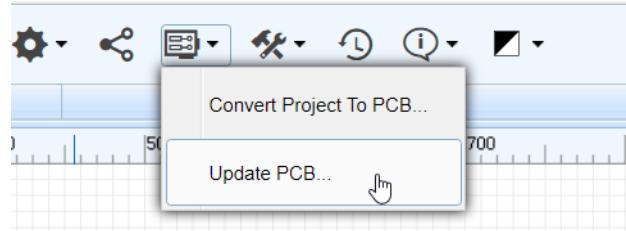
After making any necessary corrections, click the **Convert project to PCB** button and EasyEDA will automatically load all the package PCB footprints into the PCB editor as shown in the image below.



This shows the footprints placed in arbitrary positions with the connections between them shown as blue Rat lines.

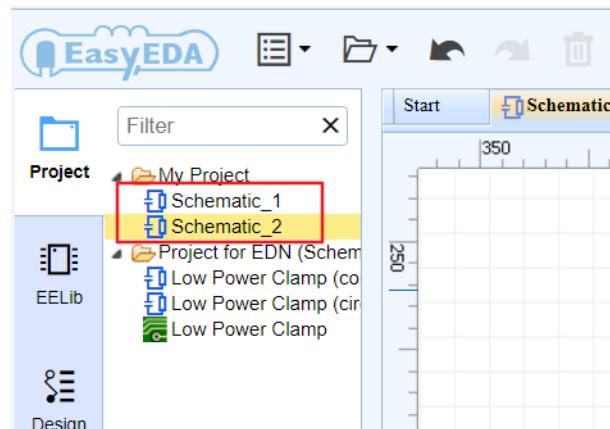
Update PCB

Converting a schematic to PCB can be done using the [Convert Project to PCB...](#), but if you do modifications to the schematic, by using the [Update PCB](#) button you can immediately be passed forward to update the selected PCB without having the PCB editor window already open or without creating a new PCB file.

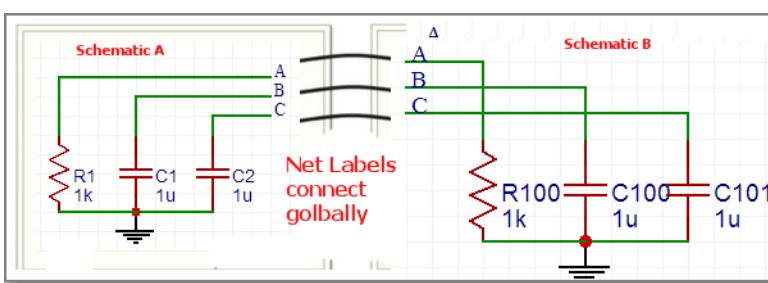


Hierarchy

EasyEDA does not support true hierarchical designs but it does support **multi-sheet designs**. You can put several schematics in one project with connections between made by netlabels. All nets in EasyEDA are global so if you create a netlabel DATA0 in schematic A and then create a netlabel DATA0 in schematic B, when Schematic A and schematic B are in the same project, they will be connected.



Multi-sheet designs(equivalent to a circuit spread over several pieces of paper), all schematics under the same project will be merged into one when be converted to PCB connecting in **netlabe**, **netflag**.



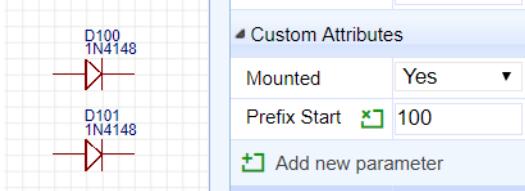
Note:

Please make all of the prefixes unique, if the Schematic A has a R1, and the Schematic B has a R1, then you will get a [Prefix Conflict Error](#) on above section.

Tip:

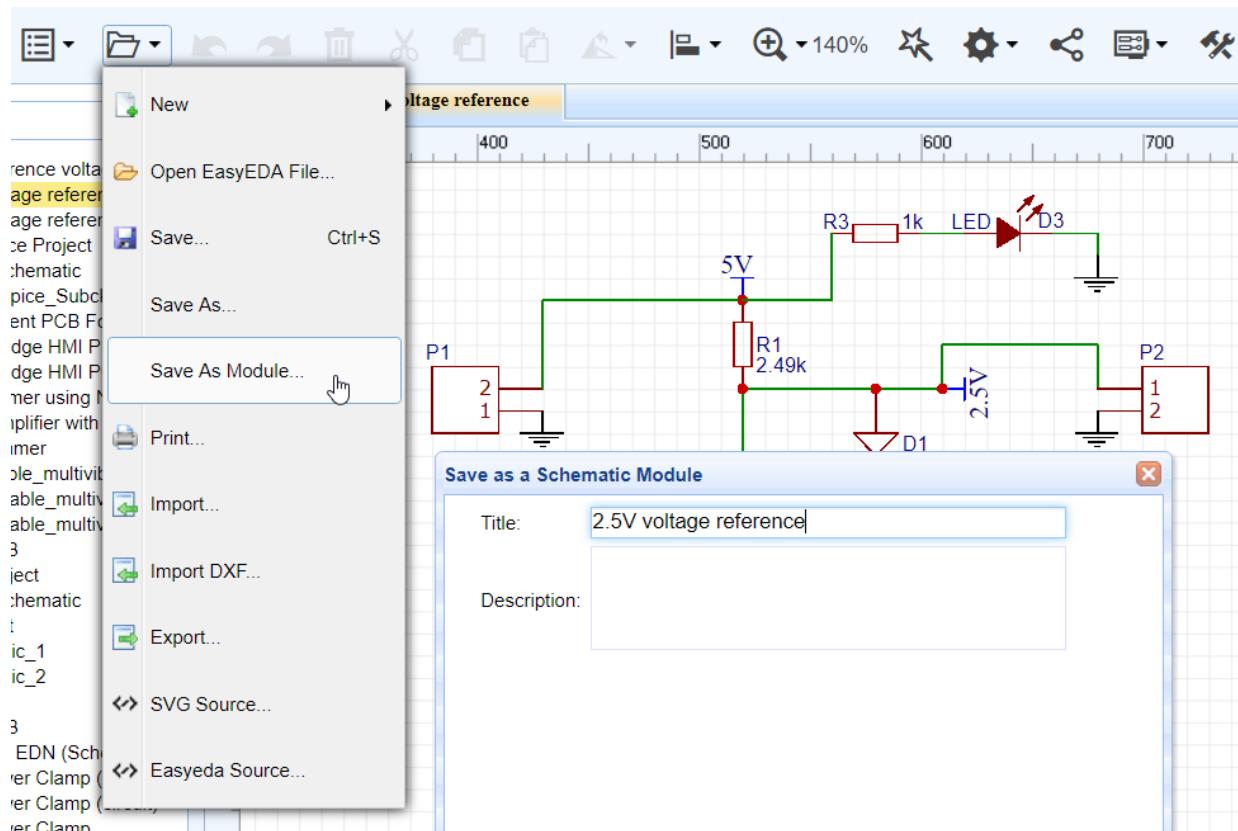
Every schematic's prefix will start from 1, such as R1, C1, U1 etc.

1. you can use [Annotate](#) to fix prefix.
2. You can set the prefix start to 100, then your components will start from R100, C100.

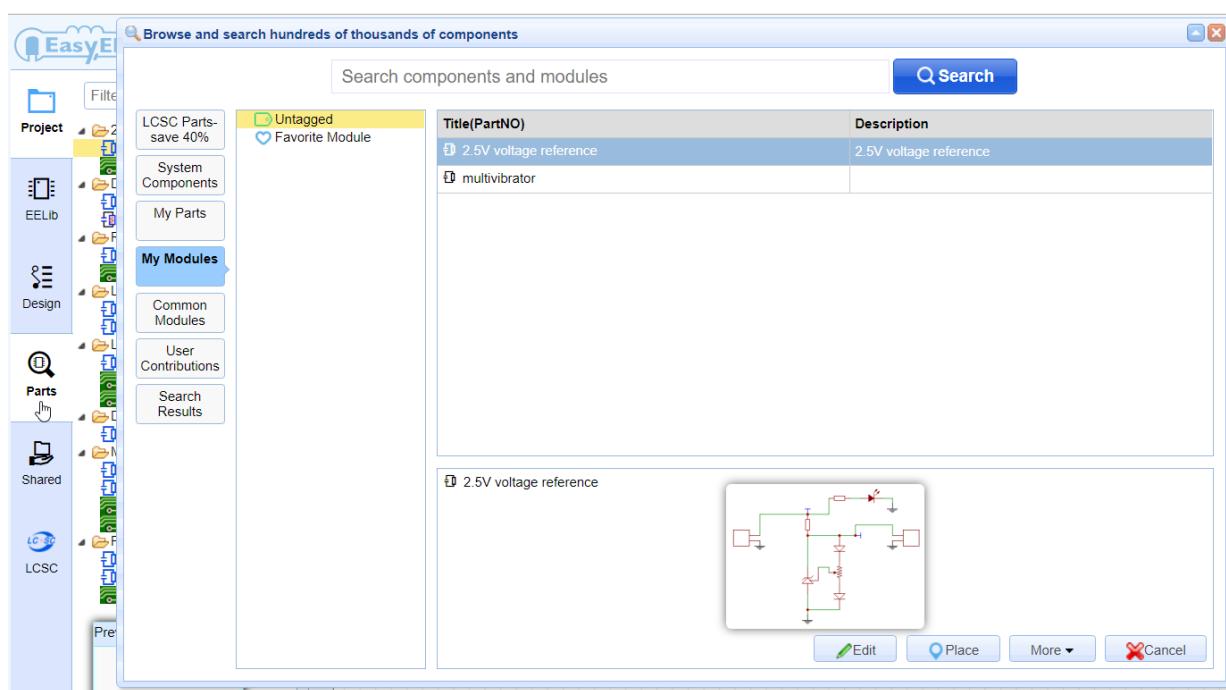


Modules

Copying codes is an easy job for coders, now copying and reusing a schematic or PCB is easy. Take a power supply unit for example, you can save this unit as a schematic module.



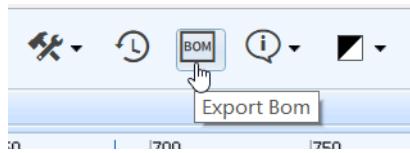
After saving, you can find it at the **Parts > My Modules** section. And you can place the whole block to your schematic.



BOM/Netlist

Export BOM

You can **export** the Bill of Materials (BOM) for the active schematic (Document) and PCB or for the active project (i.e. the BOM for all the sheets in the project) as shown below , click the top toolbar BOM icon:



After clicking the BOM export option, the dialog below will open.

In this dialog , you can assign LCSC part's order code for your components.

ID	Value	LCSC Part #	Supplier	Price(USD)	Quantity	Package	Components	Manufacturer Part	Manufacturer
13	DS1034-09MUNSI44	C75752	LCSC	\$0.2976	1	DSUB9-2	J1	DS1034-09MUNSI44	CONNFLY
14	RJ11	C45827	LCSC		1	6P4C	RJ1	RJ11	LCSC
15	RJ45	C36373	LCSC		1	RJ45-3.68	RJ2	RJ45	LCSC
16	Audio-PJ001	C3792	LCSC	\$0.0304	2	AUDIO-PJ001	J2,J4	Audio-PJ001	LCSC
17	USB-A-2	C2345	LCSC	\$0.0315	1	USB-A-2	USB1	USB-A-2	LCSC
18	SWITCH-6x6x5_TH	C69330	LCSC	\$0.0161	1	SWITCH-6x6x5_TH	SW1	SWITCH-6x6x5_TH	LCSC
19	VDG-02HG-R	C3661	LCSC	\$0.1691	1	VDG-02HG-R	DIP1	VDG-02HG-R	LCSC
20	SRD-03VDC-SL-C	C24585	LCSC	\$0.3281	1	RELAY-SL-SRD	RELAY1	SRD-03VDC-SL-C	LCSC
21	1N4148	C14516	LCSC	\$0.0063	1	DO-35	D1	1N4148	ST
22	204-10UYC/S530-A3	C73643	LCSC	\$0.0433	1	LED-3MM/2.54	LED1	204-10UYC/S530-A3	EVERLIGHT
23	MBR0520LT1G	C23848	LCSC	\$0.0399	1	SOD-123	D2	MBR0520LT1G	ON
24	PESD5V0S1BA	C19224	LCSC	\$0.0465	1	SOD-323	D3	PESD5V0S1BA	NXP
25	2W10	C3064	LCSC	\$0.0731	1	BRIDGE-WOB	D4	2W10	LCSC
26	2N3904	C2081	LCSC	\$0.02	1	TO-92(TO-92-3)	Q1	2N3904	CJ

After clicking on the assign icon , the components and packages search dialog will pop up, and you can choose which component you want to assign.

Browse and search hundreds of thousands of components

Search components and modules

LCSC (Official)

- Resistor
 - General Resistor(SI)
 - General Resistor(TH)
 - Precision Potentiometer
 - Potentiometer
 - Photo Resistor(TH)
 - Array Resistor(TH)
 - Array Resistor(SMD)
 - Varistor
- Capacitor
 - Ceramic Capacitor(
 - General Capacitor(SI)
 - Electrolytic Capacitor
 - Electrolytic Capacitor
 - Tantalum Capacitor(
 - Tantalum Capacitor(
 - Monolithic Capacitor
 - Array Capacitor(SM)
 - CBB Capacitor
 - High Voltage Capacitor
- Inductor
 - General Inductor(TH)
 - General Inductor(SI)
 - Radial Inductor(TH)
 - RJ45 Transformer
 - Ferrite Bead(SMD)
 - Power Inductor(SMI)
 - HF inductor
- Diode
 - General Diode
 - Rectifier Bridge

Title(PartNO)	Package	Manufacturer	Value	Tolerance	Power	Description
MFR03SF1003A10	AXIAL-1.0	UniOhm	100KΩ	±1%	3W	100KΩ (1003) ±1%
MFR03SF1000A10	AXIAL-1.0	UniOhm	100Ω	±1%	3W	100Ω (1000) ±1%
MFR03SF1000A10	AXIAL-1.0	UniOhm	100Ω	±1%	3W	100Ω (1000) ±1% 25ppm
MFR03SF100JA10	AXIAL-1.0	UniOhm	10Ω	±1%	3W	10Ω (10R0) ±1%
MFR03SF1001A10	AXIAL-1.0	UniOhm	1KΩ	±1%	3W	1KΩ (1001) ±1%
MFR03SF2003A10	AXIAL-1.0	UniOhm	200KΩ	±1%	3W	200KΩ (2003) ±1%
MFR03SJ0200A10	AXIAL-1.0	UniOhm	20Ω	±5%	3W	20Ω(200)±5%
MFR03SF2203A10	AXIAL-1.0	UniOhm	220KΩ	±1%	3W	220KΩ (2203) ±1%
MFR03SF2320A10	AXIAL-1.0	UniOhm	232Ω	±1%	3W	232Ω (2320) ±1% 25PPM
MFR03SF200KA10	AXIAL-1.0	UniOhm	2Ω	±1%	3W	2Ω(2R00)±1%
MFR03SF3000A10	AXIAL-1.0	UniOhm	300Ω	±1%	3W	300Ω(3000)±1%
MFR03SF330JA10	AXIAL-1.0	UniOhm	33Ω	±1%	3W	33Ω (33R0) ±1%

\$0.0318

BUY

0 In Stock Out of Stock

Minimum: 10

Distributor: LCSC

Assign **Cancel**

When you click "Export BOM at LCSC", we will help you to list all the components of your BOM. If you want to buy the components from LCSC, and you just need to put them to the cart and check out.

Export BOM

ID	Value	LCSC Part #	Supplier	Price(USD)	Quantity	Package	Components	Manufacturer Part	Manufacturer
13	DS1034-09MUNSI44	C75752	LCSC	\$0.2976	1	DSUB9-2	J1	DS1034-09MUNSI44	CONNFLY
14	RJ11	C45827	LCSC		1	6P4C	RJ1	RJ11	LCSC
15	RJ45	C36373	LCSC		1	RJ45-3.68	RJ2	RJ45	LCSC
16	Audio-PJ001	C3792	LCSC	\$0.0304	2	AUDIO-PJ001	J2,J4	Audio-PJ001	LCSC
17	USB-A-2	C2345	LCSC	\$0.0315	1	USB-A-2	USB1	USB-A-2	LCSC
18	SWITCH-6x6x5_TH	C69330	LCSC	\$0.0161	1	SWITCH-6x6x5_TH	SW1	SWITCH-6x6x5_TH	LCSC
19	VDG-02HG-R	C3661	LCSC	\$0.1691	1	VDG-02HG-R	DIP1	VDG-02HG-R	LCSC
20	SRD-03VDC-SL-C	C24585	LCSC	\$0.3281	1	RELAY-SL-SRD	RELAY1	SRD-03VDC-SL-C	LCSC
21	1N4148	C14516	LCSC	\$0.0063	1	DO-35	D1	1N4148	ST
22	204-10UYC/S530-A3	C73643	LCSC	\$0.0433	1	LED-3MM/2.54	LED1	204-10UYC/S530-A3	EVERLIGHT
23	MBR0520LT1G	C23848	LCSC	\$0.0399	1	SOD-123	D2	MBR0520LT1G	ON
24	PESD5V0S1BA	C19224	LCSC	\$0.0465	1	SOD-323	D3	PESD5V0S1BA	NXP
25	2W10	C3064	LCSC	\$0.0731	1	BRIDGE-WOB	D4	2W10	LCSC
26	2N3904	C2081	LCSC	\$0.02	1	TO-92(TO-92-3)	Q1	2N3904	CJ

Export BOM at LCSC **Cancel**

102	<ul style="list-style-type: none"> value: AT91SAM9260B-QU package: PQFP-208_28x28x05P Manufacturer Part: AT91SAM9260B-QU Supplier: LCSC More Search	 <p>AT91SAM9260B-QU Package: PQFP-208_28x28x05P LCSC Part #: C22665 Mfr.Part #: AT91SAM9260B-QU Mfr: ATMEL</p> <p> </p>	<p>1+ \$ 7.8352 1 <input type="text" value="1"/> </p> <p>136 in stock</p>
103	<ul style="list-style-type: none"> value: ATGM336H-5N-3X package: 9.7x10.1MM Manufacturer Part: ATGM336H-5N-3X Supplier: LCSC More Search	 <p>ATGM336H-5N-3X Package: 9.7x10.1mm LCSC Part #: C90770 Mfr.Part #: ATGM336H-5N-3X Mfr: ZHONGKEWEI</p> <p> </p>	<p>1+ \$ 4.5290 1 <input type="text" value="1"/> </p> <p>90 in stock</p>
104	<ul style="list-style-type: none"> value: 3A/250V package: 3.6x10 Manufacturer Part: 3A/250V Supplier: LCSC More Search	 <p>3A 250V Slow break Package: 3.6x10 LCSC Part #: C30449 Mfr.Part #: Glass tube fuse Slow break 3A/250V Mfr: ReliaPro</p> <p></p>	<p>10+ \$ 0.0512 1 <input type="text" value="10"/> </p> <p>3189 in stock</p>
105	<ul style="list-style-type: none"> value: 5S15A 250V package: 5*20MM-15A Manufacturer Part: 5S15A 250V Supplier: LCSC More Search	 <p>Slow break 5S15A 250V Package: 5*20mm 15AWithout lead LCSC Part #: C48473 Mfr.Part #: Slow break 5S15A 250V Mfr: ReliaPro</p> <p></p>	<p>5+ \$ 0.0626 1 <input type="text" value="5"/> </p> <p>114 in stock</p>

BOM **GO TO CART & CHECK OUT**

And Click the "BOM" button to download the BOM file. You can open it in any text editor or spreadsheet.

	A	B	C	D	E	F	G	H	I	J
1	id	value	quantity	package	components	Manufacturer Part	Manufacturer	Supplier	LCSC	price
2	1	150	2	AXIAL-0.3	R1,R4	25121WFJ020KT4F	UniOhm	LCSC	C45278	\$0.02
3	2	22k	2	AXIAL-0.3	R2,R3	25121WF300LT4F	UniOhm	LCSC	C16074	\$0.03
4	3	22u	2	CAP-D3.0XF1.5	C1,C2	1812B225K500NT	FH	LCSC	C28503	\$0.28
5	4	204-10UYC/S53I	2	LED-3MM/2.54	LED1,LED2	67-21S/KK3C-H2727QAR3LED EVERLIGHT	LCSC	C73540		\$0.04
6	5	2N3904	2	TO-92(TO-92-3)	Q1,Q2	MURA220T3G	ON	LCSC	C37995	\$0.17
7										

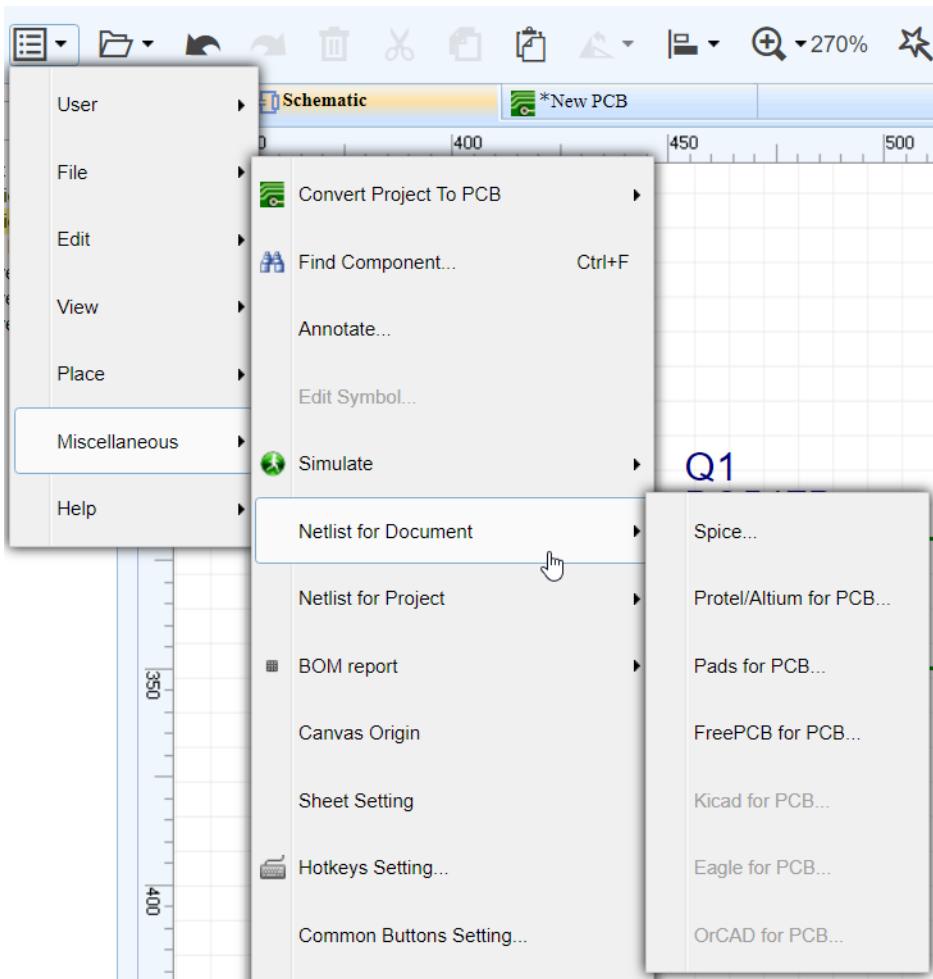
Export Netlist

EasyEDA can export the netlist for the active schematic (Document) and/or for the whole active project:

Super menu > Miscellaneous > Netlist for Document or Netlist for Project

EasyEDA can export a netlist in a variety of formats:

- **Spice**: this is a Spice3f5 compatible netlist generated by the simulation engine of EasyEDA, [Ngspice](#). It is not normally used as the basis for a PCB layout.
- **KiCad**: a PCB netlist in a format that can be imported straight into Pcbnew, the PCB layout tool part of the free, open source cross-platform EDA suite.
- **Altium Designer**: a PCB netlist in a format that can be imported straight into Altium Designer and its predecessor, Protel.
- **Pads**: a PCB netlist in a format that can be imported straight into Pads PCB layout tools.
- **FreePCB**: a PCB netlist in a format that can be imported straight into FreePCB, a free, open source PCB editor for Windows.

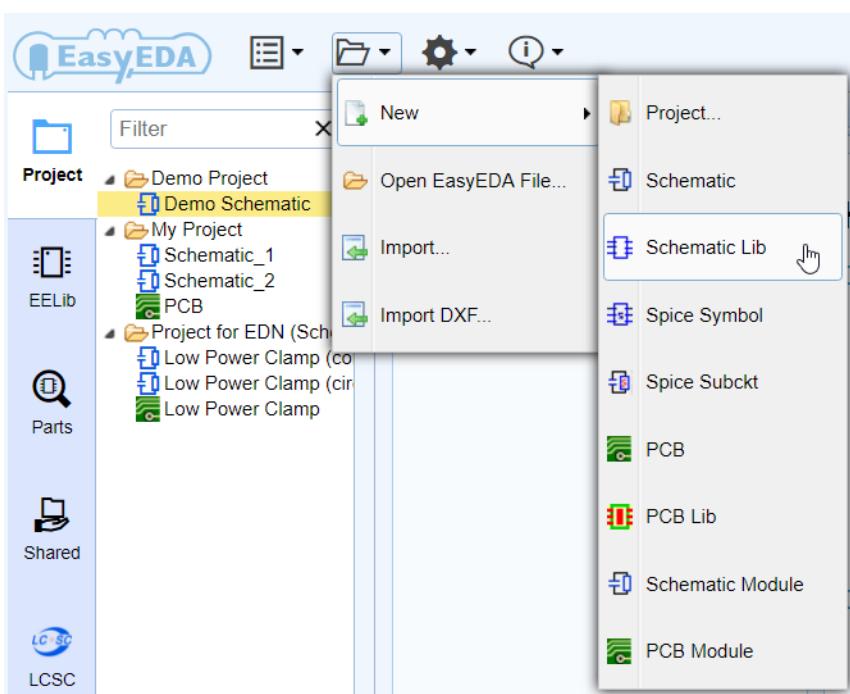


Creating the Schematic Libs

Using **Symbol Wizard** and **Group/Ungroup...** is a quick way to create schematic symbols but they are placed directly into the schematic that they are built in. It is possible to reuse them by copying them (**CTRL+C** hotkeys) from the schematic they were created in and then cross-document-pasting them (**CTRL+SHIFT+V** hotkeys) into a different schematic but this quickly gets messy if you need to copy symbols that were created in several different schematics. OK, you could keep copying new symbols into a dedicated "symbol library" schematic sheet to save searching for them but EasyEDA offers you an easier way to create and manage your symbols in a library.

Start a new Schematic Lib as shown below or by doing:

Document > New > Schematic Lib



This opens the New Schematic Lib symbol editor.

You can now create a symbol using Symbol Wizard as before or draw it using the Drawing Tools palette and add pins using the **P** hotkey (except that you no longer need to use **Group/Ungroup...**).

Then you can edit the pin map using:

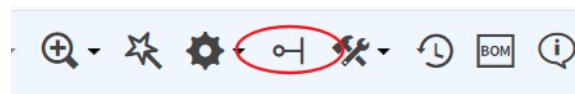
Super menu > Miscellaneous > Pin Map...

Note the Origin Point. To simplify rotating and flipping your symbols when they are placed into a schematic, make sure all of your symbols are created as near as possible centered around that point.

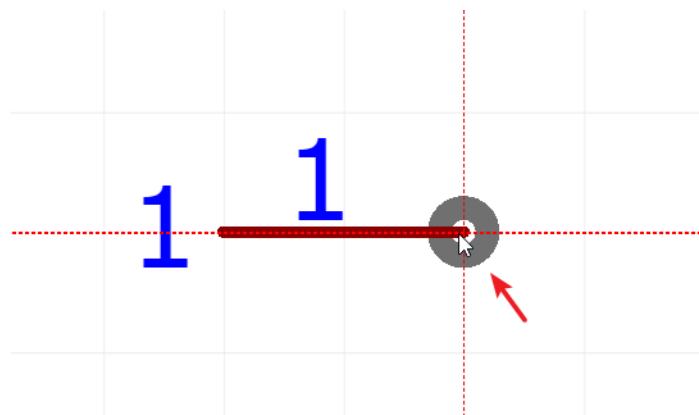
Pins

Symbols pins are the most important part of any Schematic Lib symbol. They are the things that allow wires to be attached to symbols to connect up your circuit.

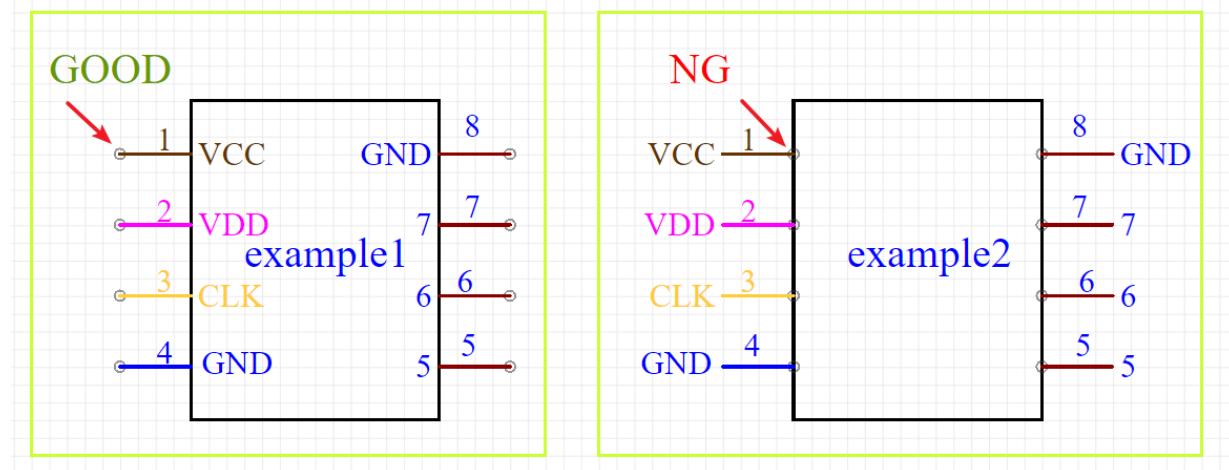
You can use the **P** hotkey to add a Pin or from the toolbar:



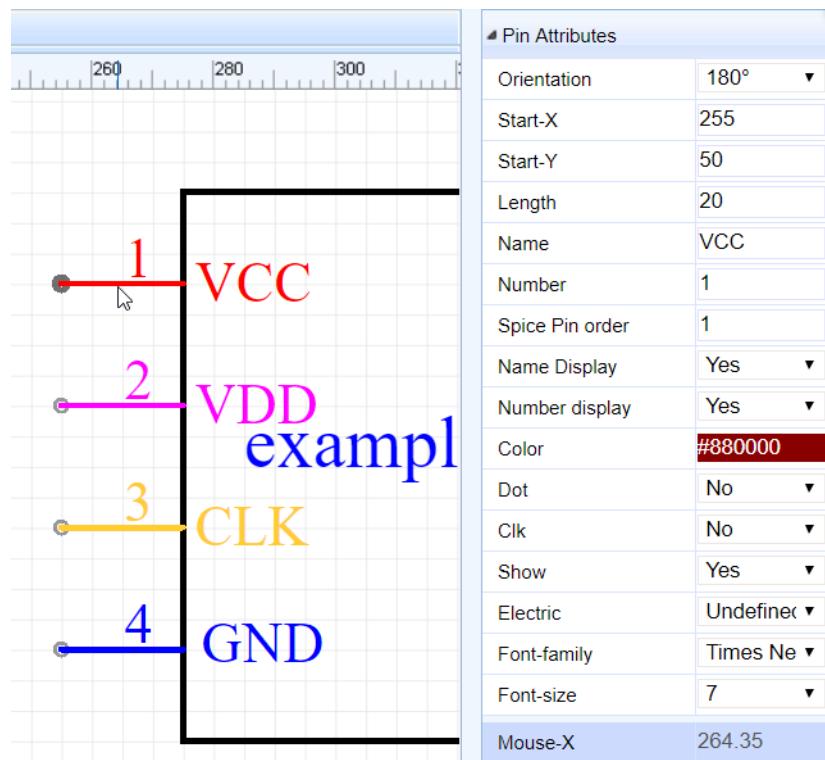
Before placing it on the canvas, you can use the rotation hotkey or rotate and flip from the menu to rotate it to the right orientation. Make sure the **Pin Dot**(black dot) is in the right position. The **Pin Dot** will be used to connect your wires or netlabels. Whenever a PIN is either placed as directly onto the canvas or as part of a symbol, the mouse has to point to the **Pin Dot** position to automatically start the Wire mode or to join a wire to it.



Whenever a Pin is placed as part of a symbol, the **Pin dot** should be **outside** of — and pointing away from — the symbol like in example 1(correct position), inside or pointing towards the symbol as shown in example 2(wrong position).



When you select a single Pin, the **Pin attributes** will be shown in the right hand **Properties** panel:



Orientation: 0°, 90°, 180° and 270°. If you want to create a 45° pin, you need to set it length as 0, and draw a line with 45°.

Start-X and Start-Y: The pindot position. Sometimes it may be difficult to move the pin to the desired position using the mouse, so you can move the pin via Start-X and Start-Y.

Length: Pin length.

Name: In this example, *VCC* is the name of the Pin.

Number: In this example, *1* is the number of the Pin. This number is the pin number of the device in a physical package and so will be the pin number used in the device footprint for that device in your PCB lib.

Note that you can use alphanumeric identifiers such as; A1, B1, C1, A2, B2 and so on as the Number.

Spice Pin order: These are the pin numbers used to connect your symbol to the corresponding pins defined by the .model or .subckt used to simulate your device. The pin numbers of the simulation model may be different from the physical package pin numbers and - unless the model is specifically created to model multiple devices in a single package - do not change for different instances of a device in a multi-device package. The Spice Pin order must be **numerals** only.

For more information about Spice Pin order please see the section on [Prefixes And Pin Numbers](#).

Name Display: If you don't want to show *VCC*, switch it to NO.

Number Display: If you don't want to show *1*, switch it to NO.

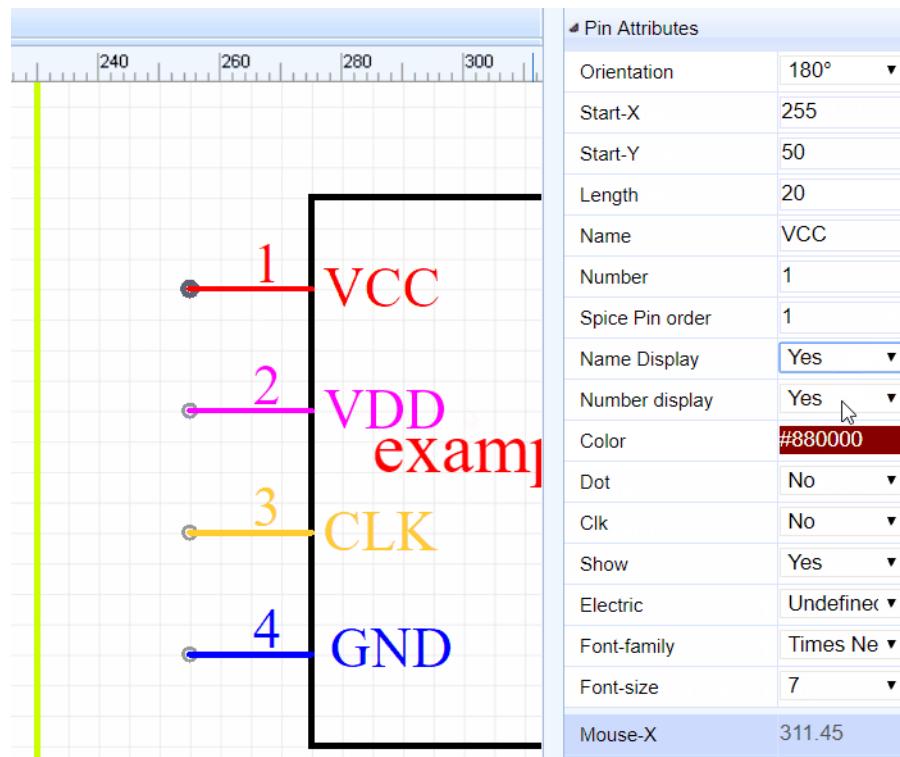
You can adjust the Name or Number position using your mouse but note that rotate and flip applies to the whole pin including the name and pin number; these items cannot be rotated and flipped independently of the pin itself.

Note also that rotate and flip actions do not result in upside down or mirrored pin number or names.

Color: You can set the Pin to different colours, such as *PIN3:CLK* as orange and *PIN4:GND* as blue. In this example, the PIN1 is set as color #880000, but it shows as red, because it is selected. After deselecting it, the pin will appear color #880000.

Dot: adds a circle to the inside end of the pin to indicate logical (or analogue) inversion.

Clk: adds a ▷ to the inside end of the pin to indicate that the pin is logical clock input.



Show: YES/NO. Allows you to hide the pin. When set it to NO, this Pin will be hidden when the symbol is placed on the schematic editor canvas.

Note that the pin is not hidden here in the Schematic Lib symbol editor canvas because if it was, it would disappear from view and so how would you find it to make it visible again? For the same reason this option has no effect in symbols made using Group/Ungroup...

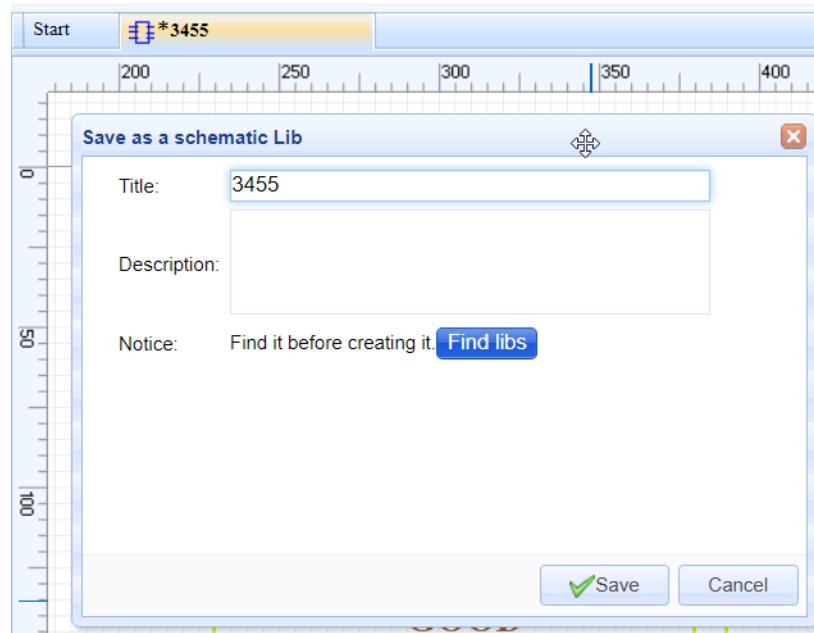
We may not have thought of everything in EasyEDA but we do try. :)

Electric: [Undefined, Input, Output, I/O, Power]

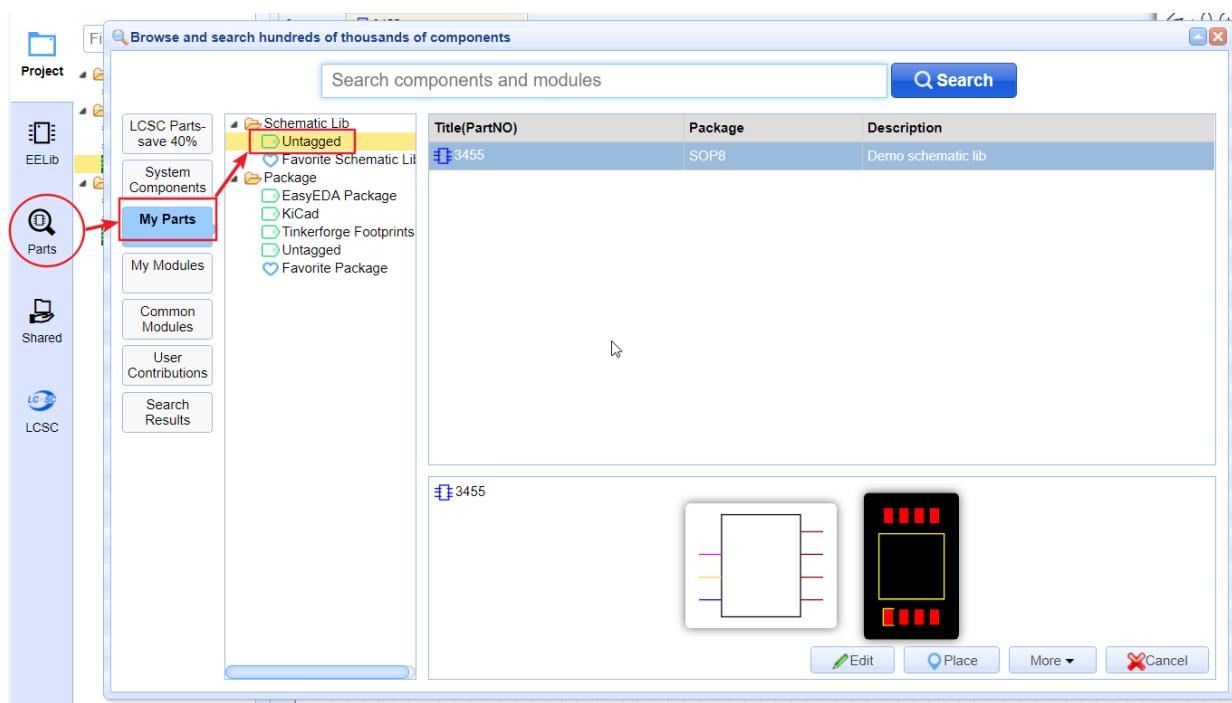
EasyEDA provides Electrical Rules Checking (ERC) right now, But you still need to set electric of your Schematic libs.

If you set the PIN as Power and set the pin to be hidden, then the Pin will be connected by Name which is the NetLabel. If the Name is VCC, it will be connected to the net in your circuit with the NetLabel or NetFlag VCC. This helps to keep the schematic clear and uncluttered when using Multi-part Components.

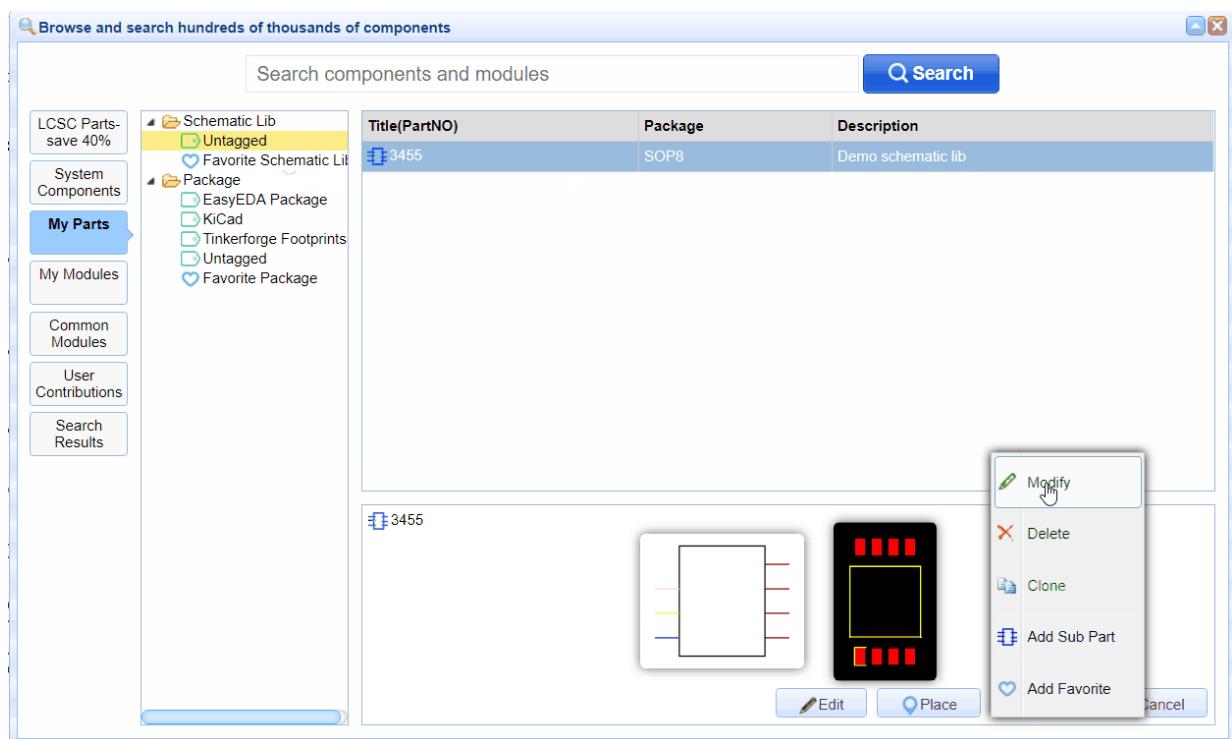
After created the Lib, use **CTRL+S** will open the save dialog:



After clicking **Save**, you will see it appears in **Parts > My Parts > Schematic Lib** of the left hand Navigation panel.



You can add a tag for your new symbol: **Parts > My Parts > Schematic Lib > Select New Lib > More > Modify**, otherwise it will appear on **Untagged**.



Subparts

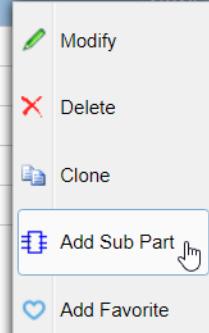
We have already touched on how EasyEDA can support **Multi-part Components** but how do you create **multi-part components**?

EasyEDA provides a sub parts facility to do this.

After creating a part, you can right click the part in the My Parts section to pop up the content menu.

Suppose you have created your own symbol for a 74HCT04 hex inverter.

Title(PartNO)	Package	Description
74HCT04		74HCT04
74HCT04.1		
74HCT04.2		
74HCT04.3		
74HCT04.4		
74HCT04.5		



Right Click **Add sub part** and that will add 74HCT04.1,

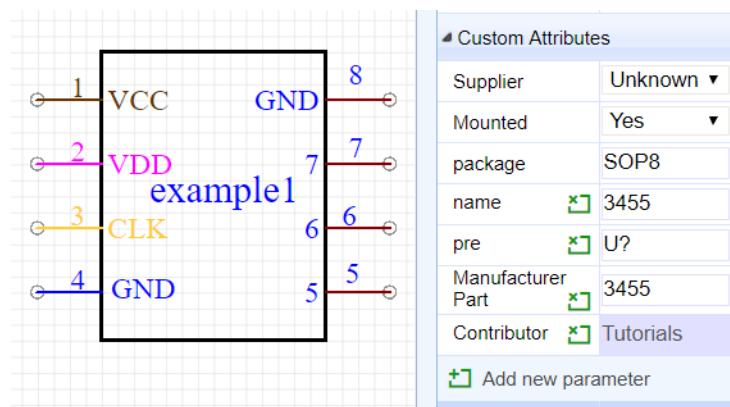
Click again to add 74HCT04.2, up to 74HCT04.6.

Then double click on each sub part in turn to modify the Pin Name and Number attributes.

Easy or what?

Custom Attributes

In the Schematic Lib editor's canvas Properties panel, you will find a **Custom Attributes** section:



Mounted

You can control this part mounted or not on the PCB. If you choose No, this component will not appear in the BOM report.

Package

If you would like to built a PCB, you need to assign a package for your schematic Lib symbol. Although there are other ways to do this in EasyEDA, here is the right place to do it. When you set a package , the package's pad numbers must match the schematic Lib's pin number, otherwise, when you convert the schematic to PCB , there will miss several nets.

Click in the **Package** input box, and the **Footprint Manager** dialog will open as used to do this task in the Schematic Editor.

Prefix

The default Schematic symbol Prefix is **U?** If you create a resistor, you can set the Prefix to **R?**

Name

You can change the schematic lib's name here, it is can be different from the part's file name.

Contributor

This is your registered user name. Other EasyEDA's users will remember your contributions!

Edit SchematicLibs

When you feel the Schematic Libs can not be satisfy for you, you can edit it.

Via "Parts" > "Search Part/My Parts/LCSC Parts/System Components/User Contributions" > Select Schematic Lib > Edit

The screenshot shows the Easyschematic software interface. On the left, there's a sidebar with categories like Project, EELib, Shared, and LCSC. Under LCSC, there's a 'Parts' section with a circled 'Edit' icon. The main area has a search bar at the top. Below it is a table of component results. One row for a resistor is highlighted with a yellow background. To the right of the table is a detailed view of a component, showing its value (\$0.0028), buy button, stock information (61395 in stock, minimum 100), distributor (LCSC), and a 3D model. At the bottom right of this detail panel is another 'Edit' button, which is also circled.

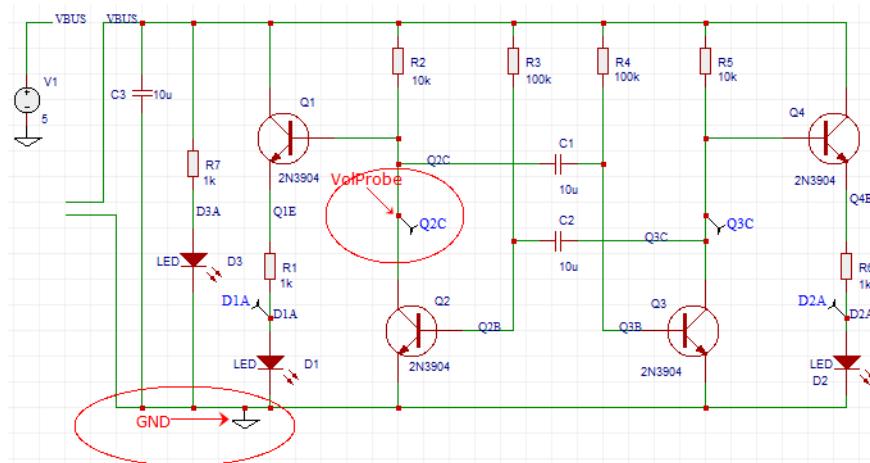
when you finish and save , it will be saved to your personal libraries "My Parts" and become your personal libraries.

Spice Simulation

Build the circuit

To simulate your circuit, at least until you become more familiar with the idea and concepts of simulation, first you should build a circuit as described in the chapter on [Creating The Schematic](#).

The circuit below is the **simulation schematic** for the Astable Multivibrator LED project:



Now, to make your circuit simulatable, you should note that:

1. You do not have to draw the whole schematic again from scratch. You can **CTRL+C** copy the schematic you have already drawn for the PCB layout and **CTRL+SHIFT+V** paste it into a new schematic canvas and then save it into a new Project folder (with maybe the same name but with "simulation copy" or something added to the end to avoid the Design Manager flagging up duplicates or you accidentally editing the original);
2. You should then remove anything from the schematic that you do not want to include in the simulation. Connectors and mechanical items such as heatsinks and often manually operated switches can be removed. (Although there are situations where these items may need to be included in the simulation; that is getting off the topic and into advanced simulation territory so we will leave it there for the moment..)
3. You may want to replace a simple battery or - to simplify the simulation, reduce the size of the simulation output file and reduce the simulation time, a complicated power supply - with a simple SPICE V voltage source. You will almost certainly need to add some sort of voltage or current signal input source such as a simple SIN or PULSE source or maybe something more complex such as the EasyEDA electret microphone model, a guitar pickup model, a photodiode or an optoisolator. If you are simulating a power supply or a power amplifier of some sort, you may also need to add a representative load of some sort. Unless you are specifically simulating the effects of wiring impedances for Power Integrity, you can usually omit any power supply decoupling capacitors hung directly off the supply rails: they have no effect, clutter up the circuit, generate lots of useless output data and add to the simulation time;
4. Your circuits **must** have a GND net. You can use a NetLabel or NetFlag to add one. You can call this net GND or 0 (the numeral zero);
5. You should use the Schematic Design Manager to help verify that your circuits are wired correctly. It can be hard to debug wiring errors from the Simulation Results... dialog error messages;

6. As you draw a schematic, EasyEDA assigns default net names to all the wires. Any section of wires that are joined will be assigned the same net name. This is how EasyEDA "knows" that those wires are joined together.

These default net names are usually of the form N001, N002, etc.

Adding NetLabels to name nets (wires or nodes) which you would like to observe (probe) makes it much easier to identify traces when the simulator shows them in WaveForm. Remember that in any circuit, you may want to probe the voltages on nets other than the obvious Input or Output nodes that may be the nets you first think you will want to probe.

Probing voltages and currents

To probe voltages, you can add some Voltage Probes which can be found in the Wiring Tools palette. These will appear on your schematic auto-numbered as volProbe1, volprobe2, etc.

When you place a voltage probe onto a net, the name you give the voltage probe will overwrite any name that is already assigned to that net. So if you place a voltage probe called foo on a net call bar, that net is renamed to foo.

Therefore it is strongly recommended that you change the name of the voltage probe to be identical to the name of the net onto which you then place that probe (except for the letter case which is ignored).

If this net name is used somewhere else in the simulation - for example in an expression for an arbitrary source - then it is possible that the voltage probe name overwriting the net name will break the expression and so the simulation will give unexpected results or throw errors. Giving voltage probes identical names to their target nets avoids this problem.

It is also recommended that you name all nets because if you have used an EasyEDA-assigned default net name in an expression then, if you edit the schematic, say to insert an extra resistor or a current probe, then EasyEDA will reassign the default net names to different nets. This breaks the expression and so the simulation will give unexpected results or throw errors.

To probe the current in a wire you can place an instance of the Ammeter component, from the EasyEDA Libs, in series with the wire.

For an alternative method of probing voltages on nets and currents through the EasyEDA Ammeter component, see [Probe](#).

Checking models and subckts

You then need to check that all the devices in the simulation schematic have the necessary and the correct spice models and/or subckts.

Missing spice models and subckts will be indicated in the Simulation Results... dialog after attempting to run a simulation but it is much easier to do this before you try to run a simulation.

Simple components such as resistors, capacitors and inductors do not pull models into the netlist because their models are built-in to Ngspice at a very low level but almost all other components will pull in either a .model statement or a set of line enclosed in the .subcktends spice keywords.

By looking at the spice netlist that is generated as a simulation schematic is being created,

Super menu > Miscellaneous > Netlist for Document > Spice...

or

Super menu > Miscellaneous > Netlist for Project > Spice...

it is easier to check that each component in the schematic has pulled into the netlist an associated .model statement or .subcktends block of lines.

In the astable example spice netlist below, Q1 - Q4 are 2N3904 devices which all pull in - and share - the .model 2N3904 statement.

Similarly, D1 - D3 are the same LED device and pull in the shared .model LED statement.

Astable Multivibrator simulation copy

```
.param pi = 3.141593
V1 VBUS GND 5
R7 D3A VBUS 1k
R6 D2A Q4E 1k
R5 Q3C VBUS 10k
R4 Q3B VBUS 100k
R3 Q2B VBUS 100k
R2 Q2C VBUS 10k
R1 D1A Q1E 1k
Q4 VBUS Q3C Q4E 2N3904
Q3 Q3C Q3B GND 2N3904
Q2 Q2C Q2B GND 2N3904
Q1 VBUS Q2C Q1E 2N3904
D3 D3A GND LED
D2 D2A GND LED
D1 D1A GND LED
C3 GND VBUS 10u
C2 Q2B Q3C 10u
C1 Q2C Q3B 10u

.MODEL 2N3904 npn
+IS=1.26532e-10 BF=206.302 NF=1.5 VAF=1000
+IKF=0.0272221 ISE=2.30771e-09 NE=3.31052 BR=20.6302
+NR=2.89609 VAR=9.39809 IKR=0.272221 ISC=2.30771e-09
+NC=1.9876 RB=5.8376 IRB=50.3624 RBM=0.634251
```

```

+RE=0.0001 RC=2.65711 XTB=0.1 XTI=1
+EG=1.05 CJE=4.64214e-12 VJE=0.4 MJE=0.256227
+TF=4.19578e-10 XTF=0.906167 VTF=8.75418 ITF=0.0105823
+CJC=3.76961e-12 VJC=0.4 MJC=0.238109 XCJC=0.8
+FC=0.512134 CJS=0 VJS=0.75 MJS=0.5
+TR=6.82023e-08 PTF=0 KF=0 AF=1
.MODEL LED D
+ IS=661.43E-24
+ N=1.6455
+ RS=4.8592
.control
probe V(D1A) V(D2A) V(Q2C) V(Q3C)
quit
.endc
.END

```

In fact the astable example circuit has no elements defined by subcircuits but the principle is the same as for .model statements.

The example below of a simple 555 timer based monostable, includes a .model statement for a type of 2N7002 MOSFET and a subcircuit for the 555 timer which in turn, calls up .model statements for the bipolar transistors, QN and QP and the diode DA that are used within the subcircuit.

It is quite possible to call one subcircuit from within another subcircuit but let's not get too carried away just yet ... 555 monostable

```

.param pi = 3.141593
XU1 GND XU1_2 OUT VCC XU1_5 XU1_6 XU1_6 VCC 555
VGATE GATE GND PULSE(0 9 0 10u 10u 10m 300m) AC 0
VBATT VCC GND 9
R4 XU1_2 VCC 2k
R1 XU1_6 VCC 100k
M1 XU1_2 GATE GND GND DI_2N7002K
C4 VCC GND 1u
C2 XU1_5 GND 10n
C1 XU1_6 GND 1u
*****
* Bipolar 555 timer model
**
* Rfix added to stop V(out) exceeding V(vcc)
* with no external load on OUTPUT pin.
**
* Last edited 140111
**
*      GND
*      | TRIGGER
*      | | OUTPUT
*      | | | RESET
*      | | | | CONTROL
*      | | | | | THRESHOLD
*      | | | | | | DISCHARGE
*      | | | | | | | VCC
*      | | | | | | |
.SUBCKT 555 34 32 30 19 23 33 1 21
*****
Q4 25 2 3 QP
Q5 34 6 3 QP
Q6 6 6 8 QP
R1 9 21 4.7K
R2 3 21 830
R3 8 21 4.7K
Q7 2 33 5 QN
Q8 2 5 17 QN
Q9 6 4 17 QN
Q10 6 23 4 QN
Q11 12 20 10 QP
R4 10 21 1K
Q12 22 11 12 QP
Q13 14 13 12 QP
Q14 34 32 11 QP
Q15 14 18 13 QP
R5 14 34 100K
R6 22 34 100K
R7 17 34 10K
Q16 1 15 34 QN
Q17 15 19 31 QP
R8 18 23 5K
R9 18 34 5K
R10 21 23 5K
Q18 27 20 21 QP
Q19 20 20 21 QP
R11 20 31 5K
D1 31 24 DA
Q20 24 25 34 QN
Q21 25 22 34 QN
Q22 27 24 34 QN
R12 25 27 4.7K
R13 21 29 6.8K
Q23 21 29 28 QN
Q24 29 27 16 QN
Q25 30 26 34 QN

```

```

Q26 21 28 30 QN
D2 30 29 DA
R14 16 15 100
R15 16 26 220
R16 16 34 4.7K
R17 28 30 3.9K
Rfix 30 0 1G
Q3 2 2 9 QP
.MODEL DA D (RS=40 IS=1.0E-14 CJO=1PF)
.MODEL QP PNP (level=1 BF=20 BR=0.02 RC=4 RB=25 IS=1.0E-14 VA=50 NE=2)
+ CJE=12.4P VJE=1.1 MJE=.5 CJC=4.02P VJC=.3 MJC=.3 TF=229P TR=159N)
.MODEL QN NPN (level=1 IS=5.07F NF=1 BF=100 VAF=161 IKF=30M ISE=3.9P NE=2
+ BR=4 NR=1 VAR=16 IKR=45M RE=1.03 RB=4.12 RC=.412 XTB=1.5
+ CJE=12.4P VJE=1.1 MJE=.5 CJC=4.02P VJC=.3 MJC=.3 TF=229P TR=959P)
.ENDS
*SRC=2N7002K;DI_2N7002K;MOSFETs N;Enh;60.0V 0.300A 2.00ohms Diodes Inc. MOSFET
.MODEL DI_2N7002K NMOS( LEVEL=1 VTO=2.50 KP=32.0m GAMMA=3.10
+ PHI=.75 LAMBDA=104u RD=0.280 RS=0.280
+ IS=150f PB=0.800 MJ=0.460 CBD=98.8p
+ CBS=119p CGSO=60.0n CGDO=50.0n CGBO=390n )
* -- Assumes default L=100u W=100u --
.control
tran 500u 500m
probe V(GATE) V(OUT)
quit
.endc
.END

```

Whoa! I thought this was supposed to be easy?

At this stage you might be forgiven for feeling a sense of panic at the sudden complexity of what should be a simple job of checking that all the symbols in your simulation schematic have the necessary and correct models associated with them.

Well, to quote the Hitchhikers Guide to the Galaxy:

Don't Panic!

All you have to do is check that every different type of device - not every instance - in your simulation schematic has a corresponding .model or .subckt statement associated with it.

If it hasn't then the first thing to check is that you have got all the device names right.

If you still haven't pulled in a .model or a .subckt then it probably means that a simulation model for that device is not available in the EasyEDA libraries. This may be because we haven't been able to find a copyright unrestricted model, we haven't had time to build our own or we just haven't caught up with entering all the thousands of possible models yet ...

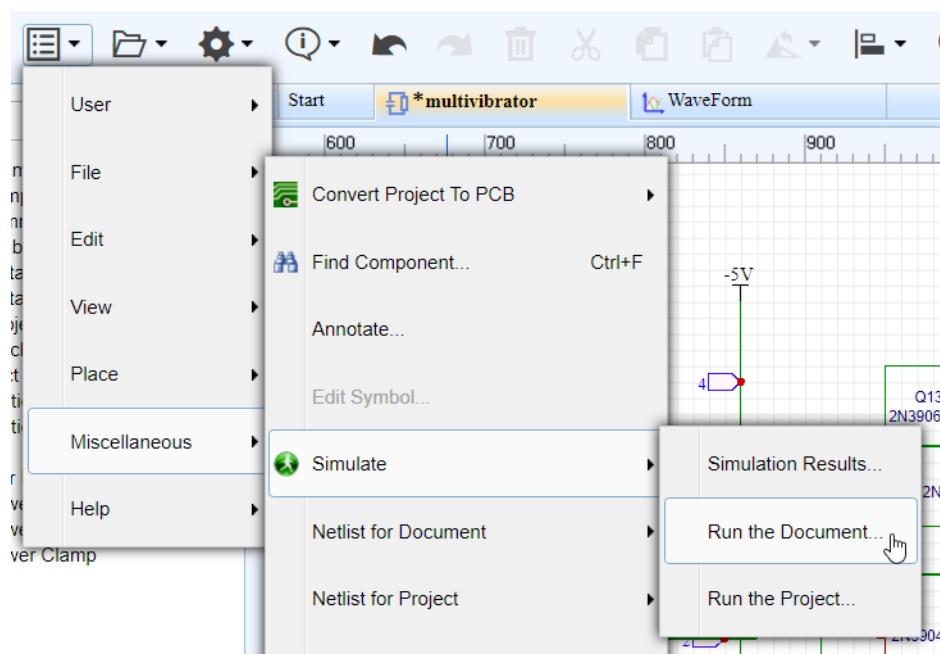
If you're desperate then EasyEDA gives you several ways that you can include third party models in your simulation but more of that later.

If you're really desperate then if you ask us nicely we just might find or even build one for you. Please see the section on [How to get help?](#)

Once you are satisfied that you have done everything to pull in the right models then you can save and then run the simulation, but don't worry, EasyEDA will still tell you if you have made any mistakes in the Simulation Results.. dialog. It's just that until you are familiar with using simulation it really is easier if you do the checking before you run a simulation because the error reporting from Ngspice may include warnings and error messages about other things besides just missing models and that can make it very confusing for beginners.

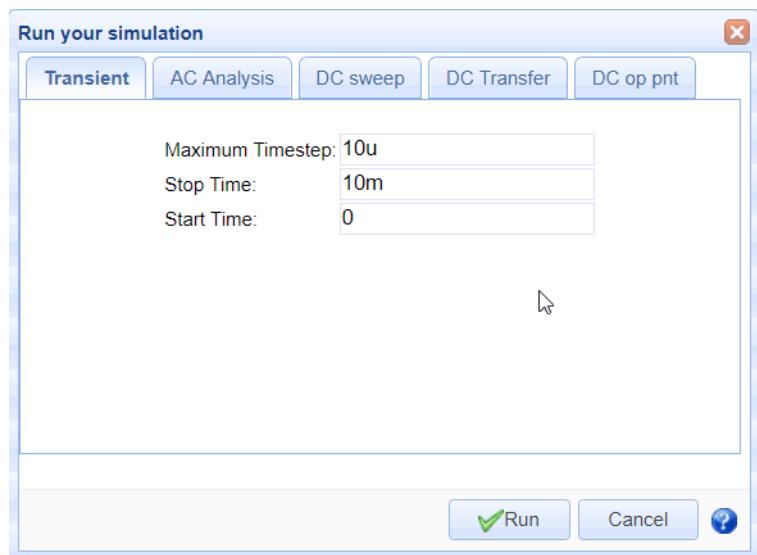
Run Simulation

Your schematic is ready, so now you can run it. **SuperMenu > Miscellaneous > Simulation > Run The...**



Run the Document: Just for the active schematic, you can also open this dialog using the **CTRL+R** hotkeys.

Run the Project: EasyEDA will merge all the schematics in the project to one, and simulate them.



EasyEDA provides the following simulation analyses:

- Transient: the time domain response of the circuit;
- AC Analysis: the frequency domain response of the circuit (including an experimental FFT);
- DC sweep: the DC response of the circuit as a voltage or current source or a component or parameter is swept between user specified limits;
- DC Transfer: computes the DC small-signal value of a transfer function (ratio of output variable to input source), input resistance, and output resistance of the circuit;
- DC op simulation: computes the dc operating point of the circuit with inductors shorted and capacitors opened.

For more information about these analyses, please refer to:

<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.1.2.1>

<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.1.2.2>

<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.1.2.3>

Please note that although using Ngspice for its simulation engine, at present (140218) EasyEDA does not support all the possible analysis modes available in Ngspice.

Note that for transient simulations, at present (140218):

the maximum value of (Stop Time-Start Time)/(Maximum Timestep) = 1000

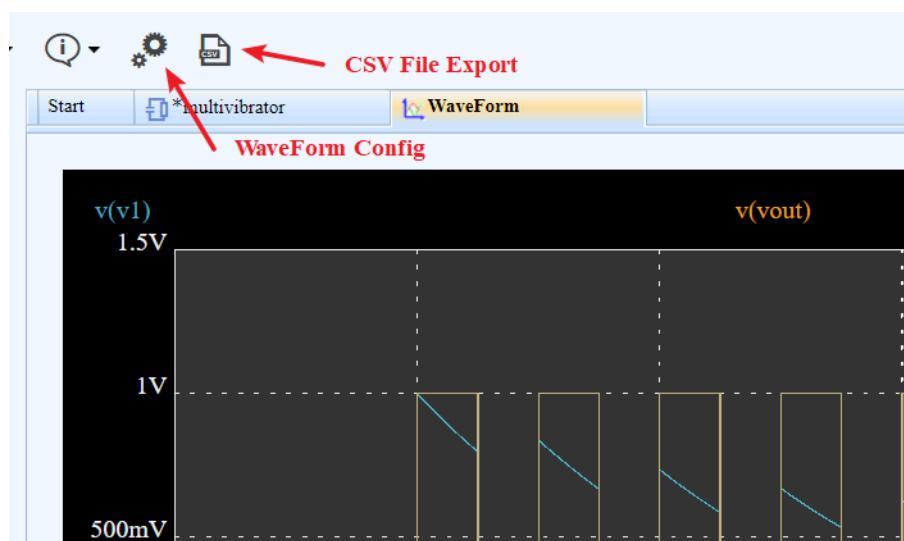
See also [CTRL+R to Run Simulation Immediately](#)

WaveForm

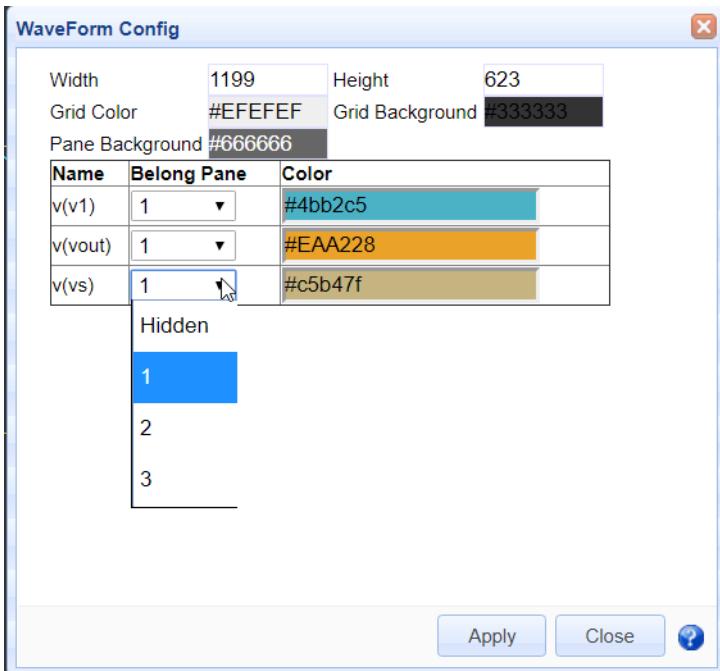
EasyEDA's WaveForm display is super easy but to make sure you don't miss any of the features it supports, we've put some detailed information into this tutorial.

Transient, AC Analysis and DC Sweep simulation results are shown in the WaveForm trace viewer.

After you run a spice simulation which should plot some traces, EasyEDA will automatically open a WaveForm tab like the image below.



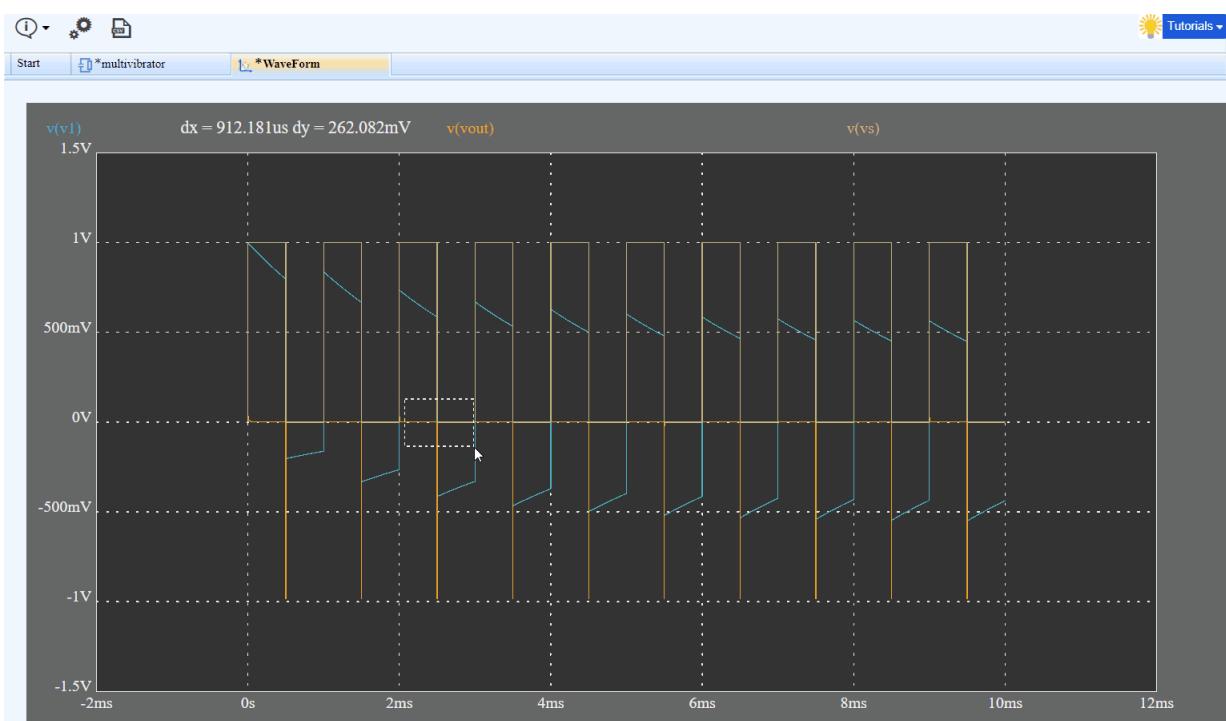
The Waveform window width and height, trace, grid and background colours and the placement of traces in up to three panes can all be configured using the WaveForm Config dialog. To open the WaveForm Config dialog, click the Config button on the toolbar above the Waveform window.



WaveForm allows the display of traces in any selection of up to three vertically stacked plot panes. The Y axes automatically scale to fit the units and the range of the traces being displayed. Traces can be hidden but at least one trace must be visible.

X and Y trace data can be seen on-screen just by moving the mouse cursor around the plot area of a pane with the readout adapting to the Y axes in each pane.

Delta X and delta Y trace data can be seen on-screen using a Left-Click and Drag select box, with the readout adapting to the Y axes in each pane. Returning the cursor to within a small radius of the starting point of the select box -without releasing the Left-Click - returns the readout to X and Y trace data.



Left-Click, Drag and then releasing the Left-Click zooms all plots, synchronised across all panes, horizontally. Double clicking anywhere in the WaveForm window resets the zoom.

Vertical plot zoom is not supported but traces are dynamically autoscaled to fit the available pane height as the horizontal zoom is changed.

The window can be moved around within the EasyEDA window using the horizontal and vertical scroll bars or using Right-Click and Drag.

WaveForm plot data can be exported in CSV format for further analysis and manipulation in external programs such as LibreOffice Calc, Scilab or Excel, however a particular feature of EasyEDA is that the WaveForm window can not only be saved in an EasyEDA Project but that the plots in a saved WaveForm window can be viewed and manipulated in exactly the same way as when they first appear as a result of a simulation. This makes it easy to compare the results from several simulations.

	A	B	C	D
1	time	v(v1)	v(vout)	v(vs)
2	0.000000000000	0	0	0
3	0.000000000010	0	0	0
4	0.000000000020	0	0	0
5	0.000000000040	0	0	0
6	0.000000000080	0	0	0
7	0.000000000160	0	0	0
8	0.000000000320	0	0	0
9	0.000000000640	0	0	0
10	0.000000001000	0	0	0
11	0.000000001064	0.064	0.06382	0.064
12	0.000000001192	0.192	0.19121	0.192
13	0.000000001448	0.448	0.44463	0.448
14	0.000000001768	0.76816	0.75904	0.76816
15	0.000000002000	1	0.985	1
16	0.000000002064	1	0.98325	1
17	0.000000002192	1	0.97978	1
18	0.000000002448	1	0.97293	1
19	0.000000002960	1	0.95953	1
20	0.000000003985	0.99999	0.93393	1
21	0.000000006034	0.99999	0.88714	1

Once saved in a Project, a WaveForm window can be exported as a .pdf, .png or .svg file into your browser window. This can then be saved to your device so it is easy to create professional quality documentation.

Build Your Own Simulation Component

There are several reasons why you may want to build your own simulation component.

- You may have downloaded a spice model in text form for a device for which EasyEDA has no symbol;
- Perhaps you have designed a simulation schematic of a circuit for which there is no readily available spice model and you need to create your own symbol for it;
- You have a subckt for a device and EasyEDA already has a symbol for it but you want to use your subckt in place of the one already attached to the EasyEDA symbol.

EasyEDA gives you three ways to build your own components so that you can simulate them:

1. From a model in text form

1. If you already have a spice subcircuit in text form, for example one that you have downloaded from a component manufacturer's website but you haven't got a spice symbol for it, then you can create a spice symbol and attach a .subckt definition to it.
2. First make a note of the exact name given in the .subckt line. Spice names are case insensitive but can only be made up from alphabetical, numeric and underscore characters.

For example: LM741EE_demo would be a valid name and would be seen as identical to lm741ee_Demo but **LM741EE-demo** and **LM741EE~demo** are **invalid** names because they contain invalid characters.

In this example we shall assume that you have a .subckt with the name: *Demo_Spice_Symbol*

3. Next, create your symbol.

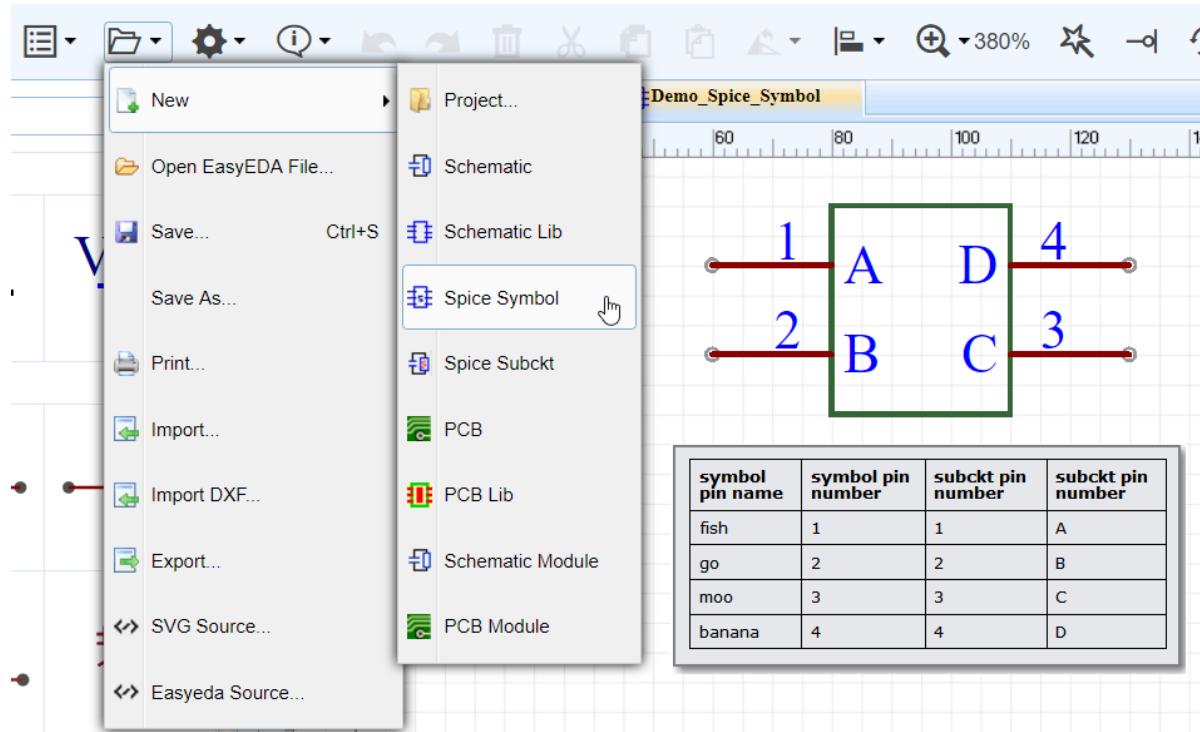
You need to do this using:

Document > New > Spice Symbol... instead of: **Document > New > Schematic Lib...**

because that option does not support attaching a spice model to a schematic symbol.

Using **Document > New > Spice Symbol...** also automatically sets the Spice Prefix of the symbol to X which is essential for a .subckt definition to attach to your symbol.

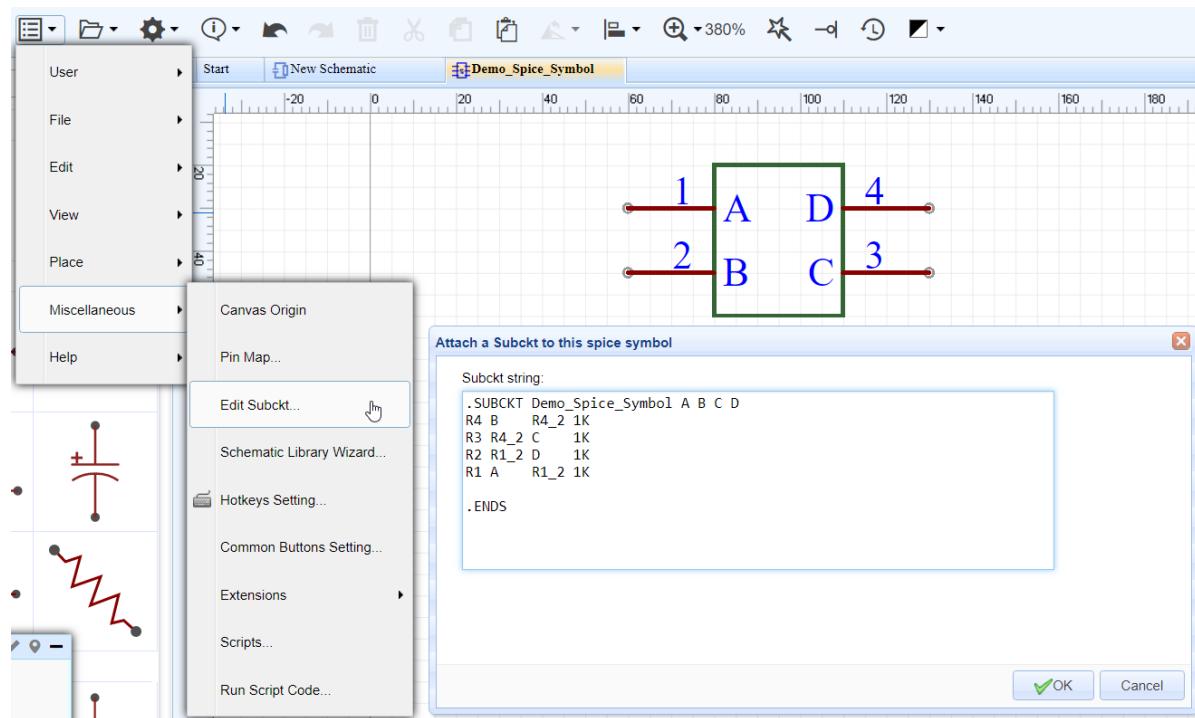
Remember that the Spice Pin names assigned to the symbol **must** be numbered in the same order that they appear in the .subckt. So, if there are four pins named A, B, C and D in the order 1, 2, 3 and 4 in the subckt, then the corresponding pins on the symbol must be in the same number order. They don't have to have the same names: you could have symbol pins named; fish, go, moo and banana but if they correspond, in the same order, to the .subckt names A, B, C and D then they must be numbered as:



4. You are now ready to attach your subcircuit to the symbol by opening the attached this spice symbol with subckt dialog using:

Super menu > Miscellaneous > Edit Subckt...

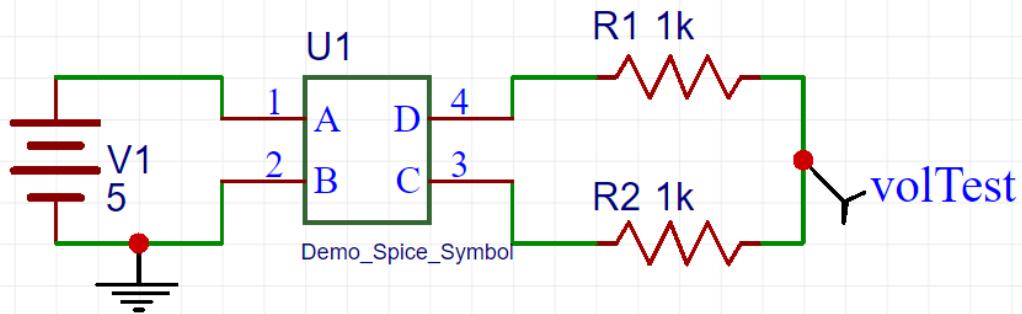
and then pasting the .subckt definition that you wish to use into the Subckt string: text area.



5. Click OK and save the symbol but remember: the symbol name must be identical to the name of the subckt:

.SUBCKT Demo_Spice_Symbol A B C D

6. Lastly, add your new spice symbol to a schematic and run a simulation.

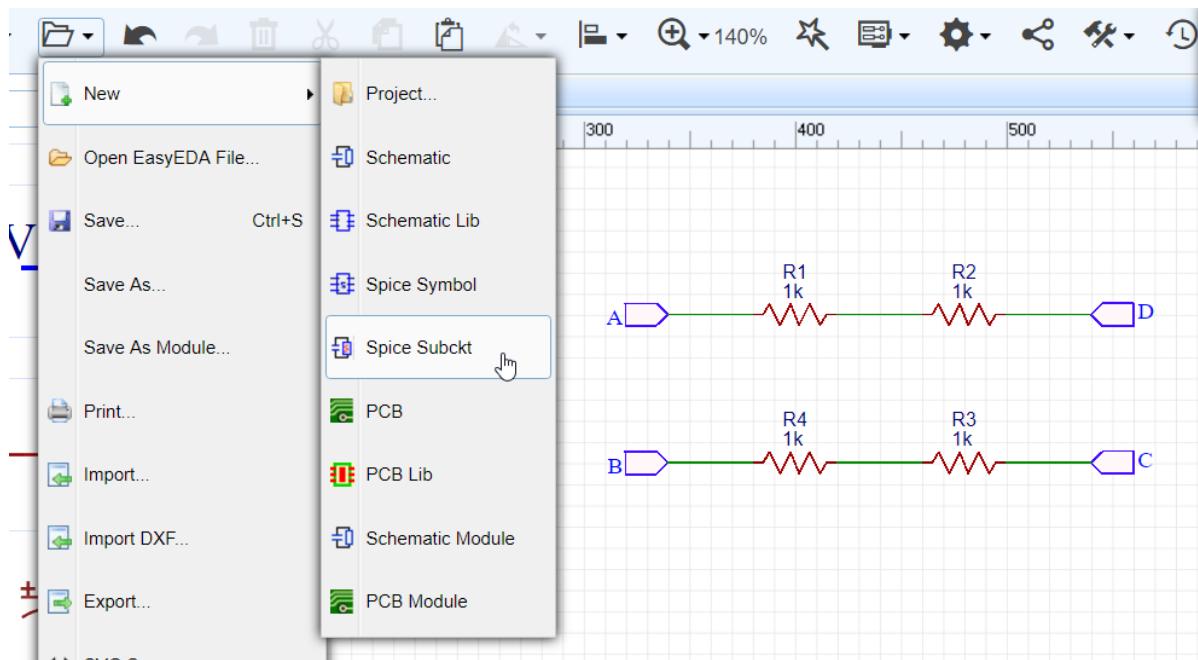


7. If you run a DC op simulation on this example, the result, shown in the Simulation Results... window, should be 2.5V

2.From a subcircuit in schematic form

1. Create a spice symbol and subckt circuit.
2. The same as (1) above, create a spice symbol.
3. Next create a spice subckt as a schematic:

Document > New > Spice Subckt...



Draw the schematic that you want EasyEDA to turn into a subckt and attach to your symbol.

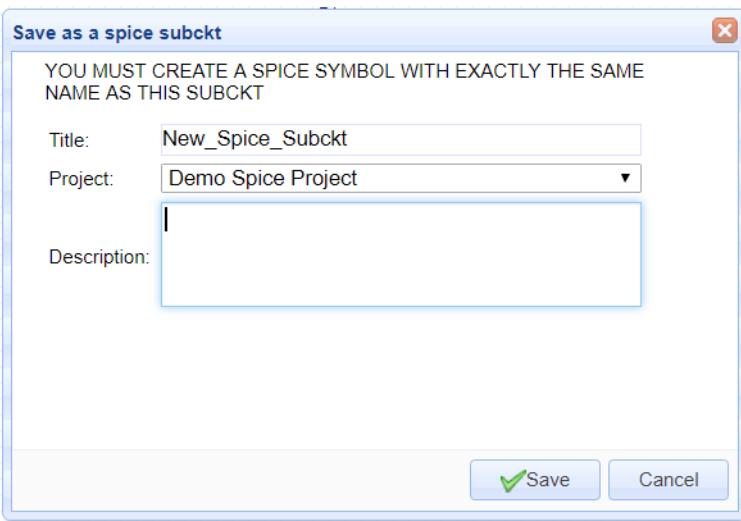
To connect your simulation schematic to your symbol there must be a net in the circuit that is to be attached to each pin of the symbol. Each of these connecting nets in your circuit must have the same name as that of the symbol pin to which it connects. For example if your symbol has four pins called A, B, C and D then your simulation schematic must have exactly four connecting nets; one called A, one called B, one called C and one called D.

To attach these nets in the schematic to the pins in the symbol you must name them using NetPort from the Wiring Tools palette.

Do not use NetLabel or NetFlag.

NetPort is used to distinguish those subckt nets that are to connect to symbol pins from all other nets named using EasyEDA default net names and those added using NetLabel or NetFlag.

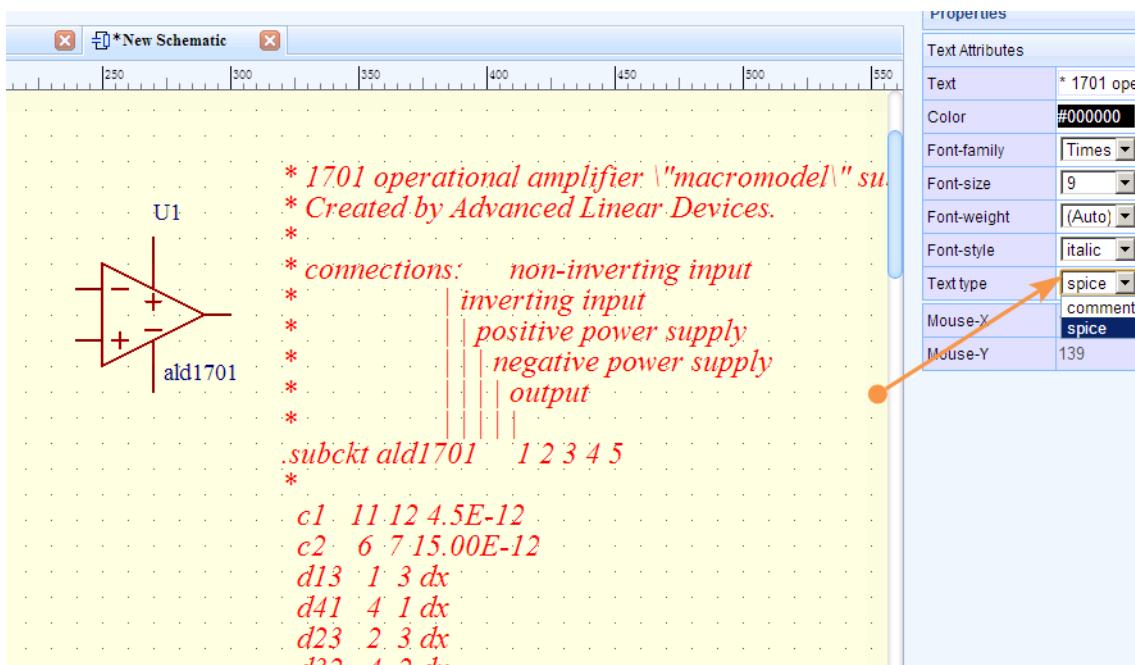
4. Save your spice subckt with exactly the same name as your spice symbol.



5. Lastly, add your new spice symbol to a schematic the same as in (1) above and run a simulation.
6. If you run a DC op simulation on this example, the result, shown in the Simulation Results... window, should be 2.5V.

3.From a spice directive in a schematic

1. When you already have a spice symbol which has a subckt attached to it - for example, an opamp symbol from the EasyEDA Libs - but you want to use a subckt for a different device which is not already in the EasyEDA Libs, then you can use this method to easily attach a subckt to a symbol directly in your schematic.
2. Paste your .subckt text into the schematic.
3. Next, select the pasted text and, in the right hand Properties panel, change the Text type from comment to spice. This will create a spice directive which the simulator will then incorporate into the spice netlist.
4. Next, select the symbol and, either directly in the symbol or in the right hand Properties panel, edit the Model text to exactly the same name as in your pasted subckt.
5. Check that the Spice pin order of the symbol matches that of the pasted .subckt and edit it if necessary (not all subckts for a given type of device use the same Spice Pin order!).
6. Save your schematic and then you can run your simulation.



4.Using .models instead of .subckts

1. All three techniques can be used to attach .model statements to symbols in exactly the same way as .subckts but after placing the symbol in your schematic, you must use:
Super menu > Miscellaneous > Edit Symbol... to set the symbol's Spice Prefix to the appropriate letter for the device model you are using.
2. You also have to know the spice pin order for the type of .model statement you are using because, unlike .subckts, the .model statement does not show this explicitly.

The Spice Prefixes and Spice Pin names and orders for the most commonly used devices for which you may want to use different models are listed below:

Spice Prefix	Device description	Spice pin order
D	Diode	Anode = 1 Cathode = 2
J	Junction field effect transistor (JFET)	D = 1 G = 2 S = 3
M	Metal oxide field effect transistor (MOSFET)	D = 1 G = 2 S = 3
Q	Bipolar junction transistor (BJT)	C = 1 B = 2 E = 3
X	Subcircuit	Depends on subckt
Z	Metal semiconductor field effect transistor (MESFET)	D = 1 G = 2 S = 3

Actually there is a way to save the symbol with the required Spice Prefix so that you don't have to edit it every time you place a new instance of the symbol into a schematic ... but that needs a bit more insight into editing the EasyEDA Source for the symbol so will be left for the moment.

We did say that EasyEDA Source enables some powerful ways to manipulate schematic and spice files and symbols! :)

Advance Tips

EasyEDA uses [Ngspice](#) as the simulation engine, so once you get more familiar with it you can use many [other commands and features of Ngspice](#) that are not directly available via the EasyEDA UI.

The lists below show which Ngspice commands are currently supported by EasyEDA and which are not.

Ngspice Commands Whitelist

EasyEDA allows these commands:

```
let define option options unlet op tf tran pss ac dc pz sens disto noise fft fourier meas alter run while repeat dowhile foreach if else end break
continue label goto linearize print probe echo
```

Ngspice Commands Blacklist

EasyEDA does not currently allow these Ngspice commands:

```
reshape snsave snload circbyline alias deftype display destroy setplot setcirc setscale transpose xgraph gnuplot wrdata wrs2p hardcopy
asciplot write compose print eprint codemodel load cross undefine listing edit dump psd spec show showmod sysinfo altermod resume state
stop trace save iplot altermod status delete step remcirc reset aspice jobs rspice bug where newhelp tutorial help oldhelp removecirc quit
source shift unset unalias history shell rusage cd version diff rehash cdump mdump mrdump settype strcmp devhelp inventory source
```

Probe

An alternative to using the volProbe element to probe voltages in a circuit - which avoids the possibility described in [Probing voltages and currents](#) of overwriting net names and consequently corrupting any expressions that use them - is to use the Probe command.

For example, to probe the voltages on two nets named in and out all you have to do is enter this text into the schematic:

```
Probe V(out) V(in)
```

and then, in the Properties panel, set the Text type to spice to set it to be included in the spice netlist as a spice directive.

You can also use the Probe command to probe a current in your circuit.

To measure the current in a wire you insert an Ammeter, from the EasyEDA Libs, in series with the wire you wish to probe. EasyEDA then inserts a small subckt comprising a 0V, zero resistance, voltage source in series with the wire and then probes the current in that voltage source. Hence although an ammeter in an EasyEDA schematic is shown with an A prefix, it is spice netlisted with an X prefix (for a subckt call) followed by V (for the voltage source).

For example, to add the current in an Ammeter, named A/loadcurrent1, to the command probing the two voltage probes above, you would change the Probe command in your schematic to:

```
Probe V(out) V(in) I(XVA_load_current1)
```

It is also possible to use expressions in a Probe command. In the example above, if we assume that V(out) is connected directly to a grounded load then, to plot the power dissipation of the load, you can add this expression:

```
V(out)*I(XVA_load_current1)
```

the Probe command list:

```
Probe V(out) V(in) I(XVA_load_current1) V(out)*I(XVA_load_current1)
```

Note that your probe list can be as long as you like but all entries in a Probe command list must be entered as a single line of text with no returns.

A useful feature allowing you to easily switch between different sets of probe points is that any number of Probe commands, each with their own list of probe points, can be included in a schematic by setting the Text type of only one at a time to spice and setting all others to

comment.

But this is just the tip of the iceberg ...

Using CTRL+R to Run Simulation Immediately

As described in [Run Simulation](#), using:

CTRL+R

will open the

Run the Document

or

Run the Project

simulation control dialog.

That approach is a great way for you to quickly and easily set up and Run any of the most commonly used simulation analyses types but EasyEDA gives you a way to harness the real power of **Ngspice**.

Simply by entering your simulation control commands as text, directly into the schematic and setting the Text type to spice, you can set up powerful spice analyses. You can run these straight away, without needing the Run your simulation dialog just using the **CTRL+R** hotkeys.

Using this method it's quick and easy to create and run more advanced simulation analyses and to make automated measurements on your circuit.

Here's a quick insight into how it works but you can skip this if you like and just get into how to make this amazing feature work for you!

EasyEDA automatically embeds the simulation commands set up in the Run your simulation dialogs within a control section. You can see this in the spice netlist for any circuit that has been through a simulation run at least once via:

Simulate > Simulation Results... > Download netlist

The control section starts with the .control command and ends with the .endc command. All commands between these delimiters are run in an Ngspice interactive simulation control mode.

Now, you don't need to worry about these two commands because EasyEDA automatically inserts them in the netlist in the right place to enclose your commands so all you need to do is to enter a list of commands as text, anywhere in the schematic canvas and then, in the Properties panel, set the Text type to spice for it to be included in the spice netlist as a **spice directive**.

The following examples show some of the things you can do using **spice directives**.

Run a **transient simulation** with the following parameters:

Maximum Timestep: 10u **Stop time:** 11ms **Start Time:** 1ms

just add this text anywhere on the schematic canvas:

```
tran 1u 11m 1m
```

set Text type to **spice**

then type:

CTRL+R

Run an **AC Analysis** with the following parameters:

Type of Sweep: Decade

Number of points: 100 (per decade)

Start Frequency: 1k

Stop Frequency: 1Meg

just add this text anywhere on the schematic canvas:

```
ac dec 100 1k 1Meg
```

set Text type to **spice**

then type:

CTRL+R

Run a **DC Sweep** with the following parameters:

(And, yes, you can sweep component values, not just sources!)

Source to Sweep: R1

Start Value: 1k

Stop Value: 2k

Increment: 100

just add this text anywhere on the schematic canvas:

```
dc R1 1k 2k 100
```

set Text type to **spice**

then type:

CTRL+R

A couple of more advanced examples:

Run a **Fourier** analysis:

```
tran 2u 2m 0
fourier 1K V(volout)
run
probe V(volout)
```

set Text type to spice

then type:

CTRL+R

For more information on Fourier Ngspice, see:

<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.17.5.24>

Run an **FFT** analysis:

```
tran .1m 2s 0
run
linearize
fft v(out)
probe db(mag(v(out)))
```

set Text type to spice

then type:

CTRL+R

For more information on **FFT** in Ngspice, see:

<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.17.5.25>

Run a DC op pnt analysis and Print the power in the load into the Simulation results window:

```
op
print V(out)*I(XVA_load_current1)
```

set Text type to spice

then type:

CTRL+R

Measure the gain and 3dB bandwidth of an amplifier.

This prints the gain and bandwidth values of this x1 and x10 amplifier example:

https://easyeda.com/file/viewFind-gain-and-bandwidth_8GE0KRFDn.htm

in the Simulation Results... window:

```
* This is a control block.
* Note: variables in a control block must start with
* a letter.

* Set up an AC analysis:

ac dec 100 1k 10Meg

* Define a 3dB value:

let neg3dB = 20*log10(sqrt(2)/2)

* Convert the outputs of both amplifiers into dB:
```

```

let x1gain_dB = DB(v(x1Avcl))
let x10gain_dB = DB(v(x10Avcl))

* Find the low frequency gain of each amplifier
* (look at the Bode plots in WaveForm and choose
* a frequency where the gain is level; i.e. well
* below any possible hf gain peaking):

meas ac x1_lfgain_dB find x1gain_dB at=1k
meas ac x10_lfgain_dB find x10gain_dB at=1k

* Subtract 3dB from the lf gains to find
* a value 3dB down from the lf gain:

let x1_3dBdown = x1_lfgain_dB + neg3dB
let x10_3dBdown = x10_lfgain_dB + neg3dB

* Find the frequencies at which the outputs
* are 3dB down from the lf gains:

meas ac x1_f3dB when x1gain_dB = x1_3dBdown
meas ac x10_f3dB when x10gain_dB = x10_3dBdown

```

set Text type to spice

then type:

CTRL+R

For more information on the meas statement, see:

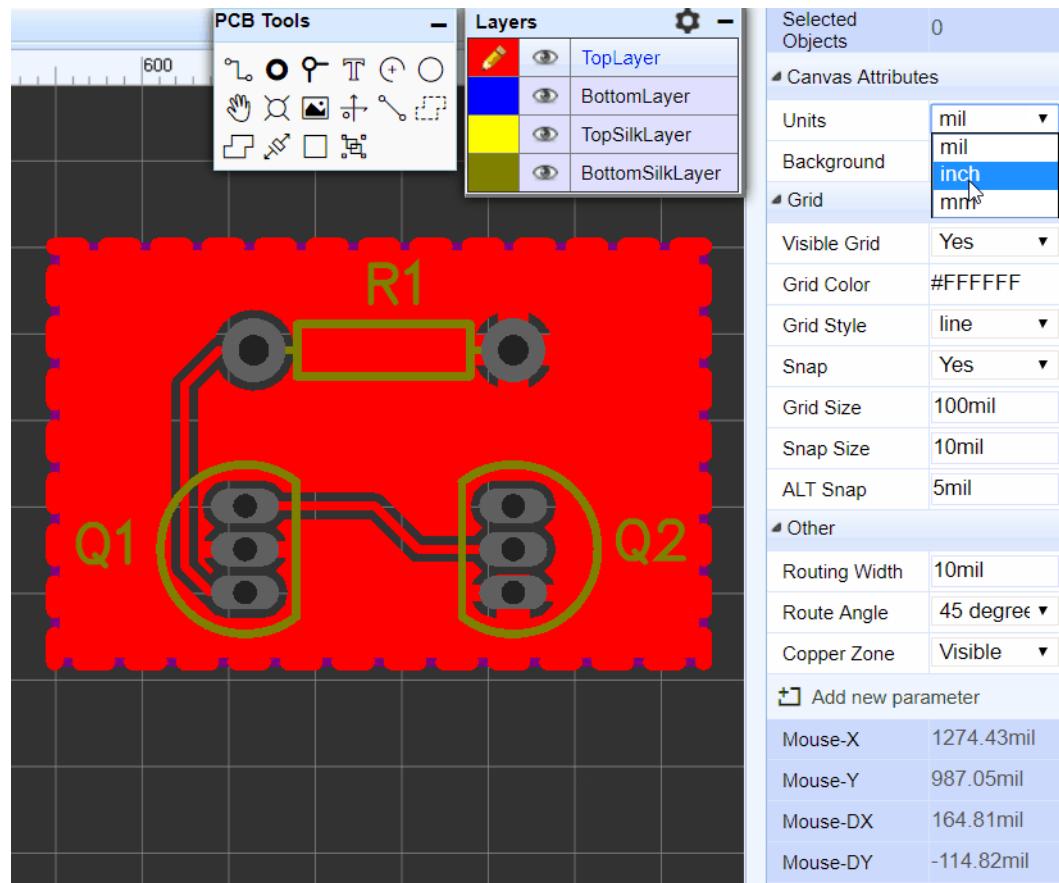
<http://ngspice.sourceforge.net/docs/ngspice-manual.pdf#subsection.17.5.37>

PCB Design Editor

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

Canvas

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



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.



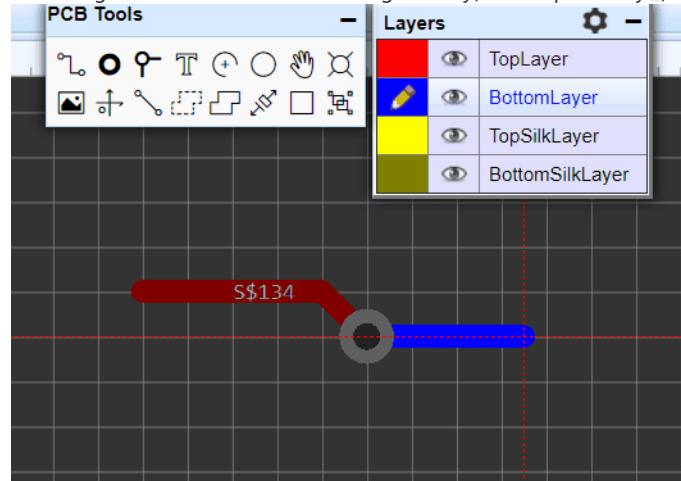
Track

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!).

Some Tips about Track.

1. 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.

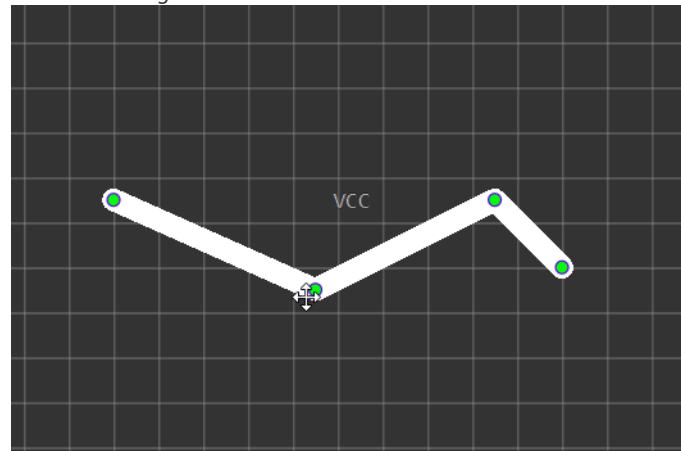
2. Drawing a track at the same time as using a hotkey(for example hotkey **B**) for changing the active layer will automatically insert a Via:



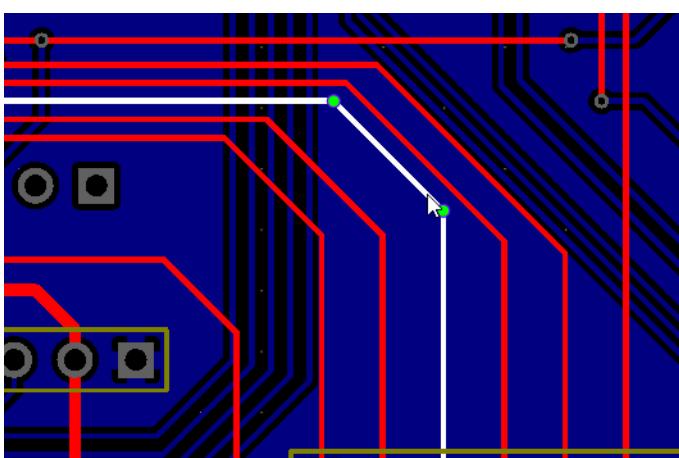
If you start drawing a track on the top layer - you will see it drawn in red - then press the **B** key to change to bottom layer and you will see EasyEDA insert a grey via and then the track will continue being drawn but now on the bottom layer in blue.

3. Pressing the **+** or **-** Hotkeys when drawing the track will change the width of the track on the fly.

4. 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.



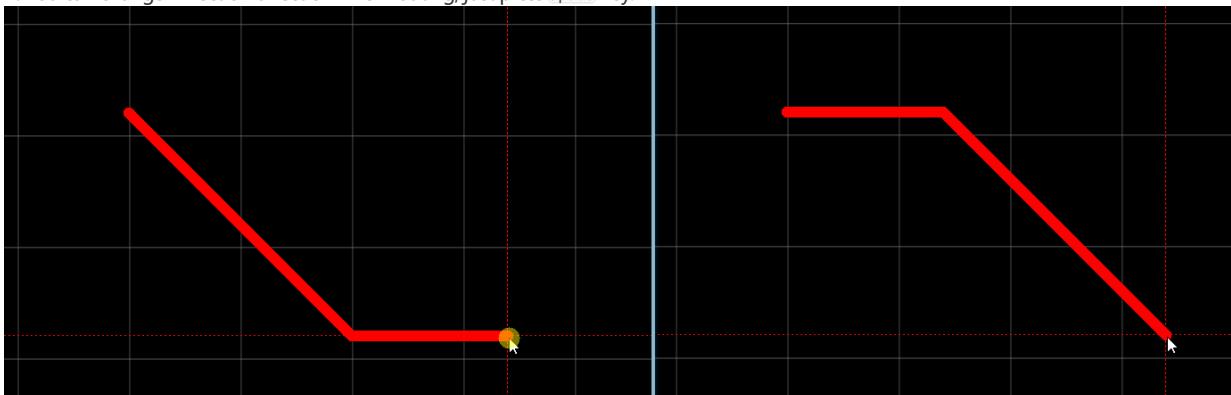
5. Click to select the track and then Click and Drag on a segment of the track to adjust the segment between vertices.



6. 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.

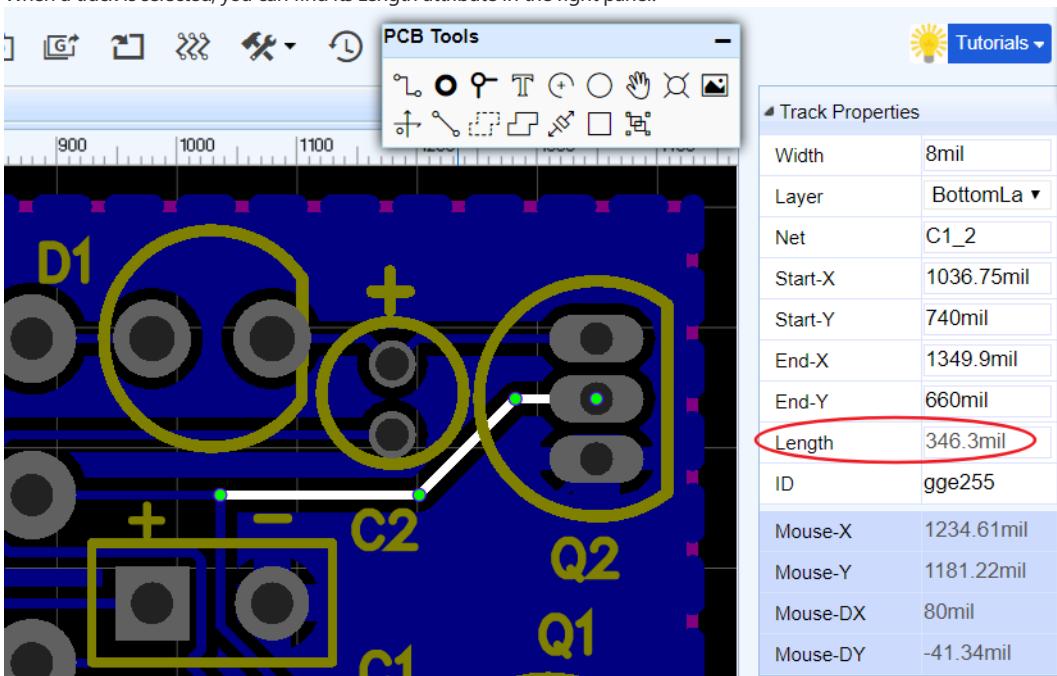


7. You can change inflection direction when routing, just press **Space** key.



Track Length

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



Delete a Segment from a Track

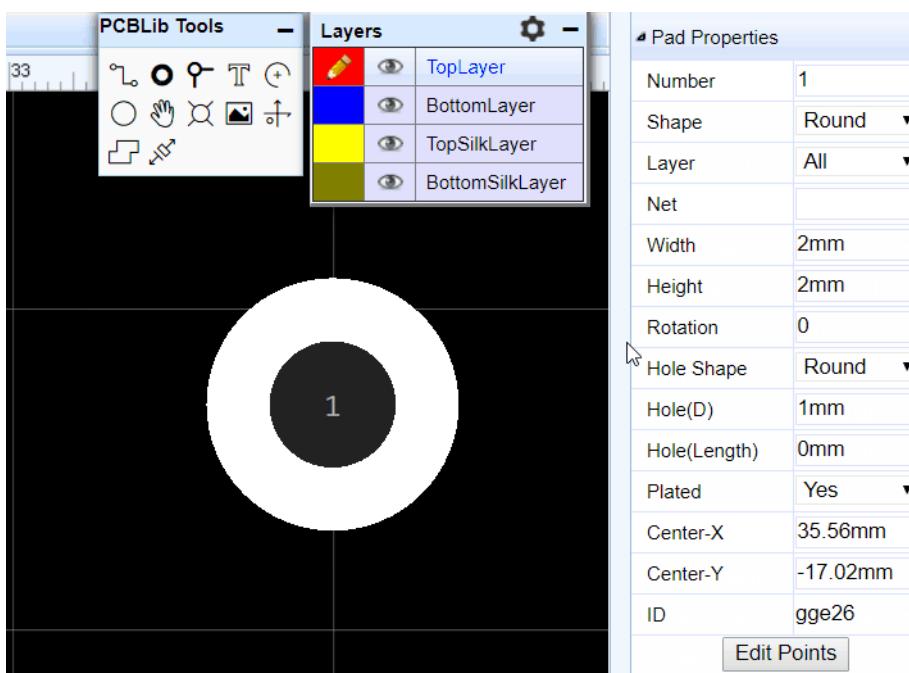
In lots of other EDA tools, the track is segment line, but in EasyEDA, the track is polyline. Sometimes, if we want to delete a segment, we must delete the whole track and route again. Now we provide a better way to do this. Move your mouse to the segment which you want to delete, click it, then hold **SHIFT** and **double click it**, the segment will be removed.



Pad

You can add pads using the Pads button from the PCBLib 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.



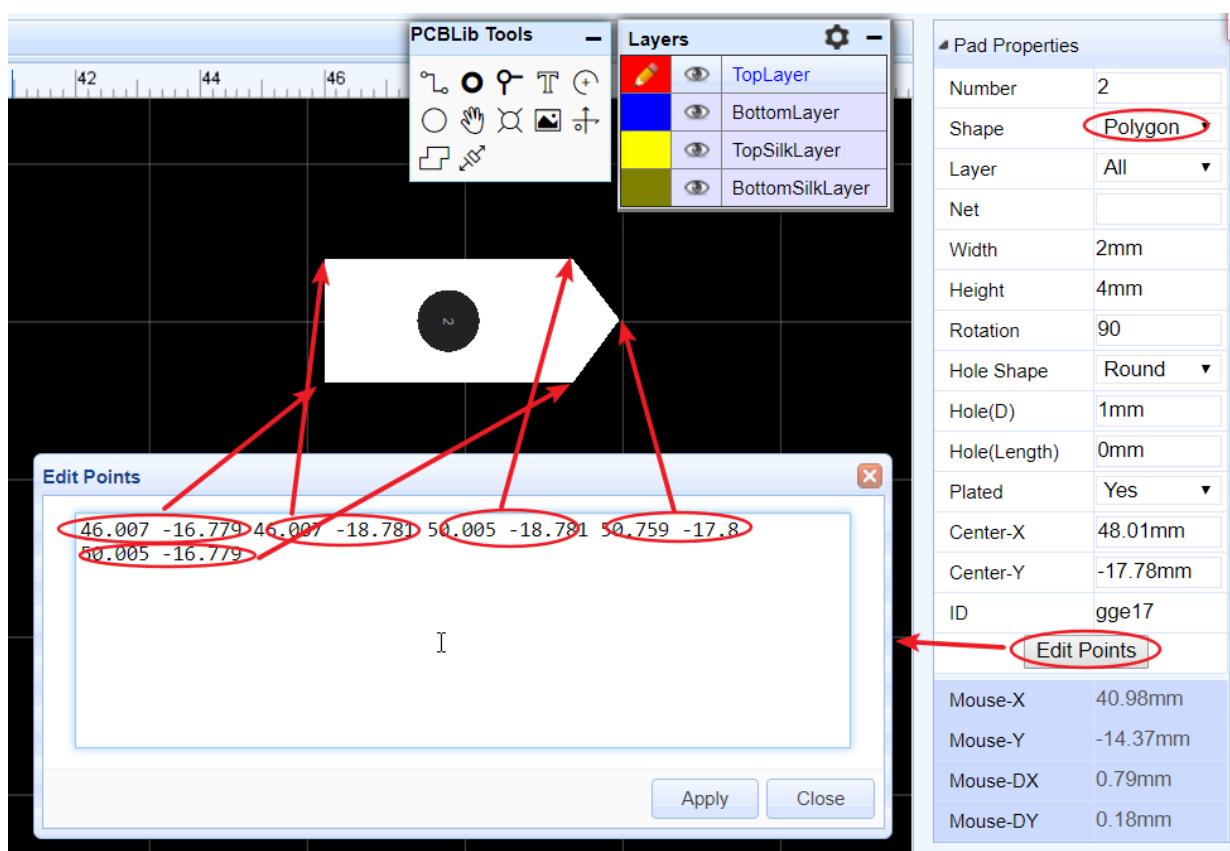
Number: Remembering the pin numbers you set in the schematic symbol in your Schematic Lib: to connect those schematic symbol pins to the pads in your PCB footprint, the pad numbers you set here in the PCB Lib footprint must be the same.

Shape: Round , Rectangular , Oval and Polygon.

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

- **OVAL** PAD will give you 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 **All**.

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 this to **zero**.

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

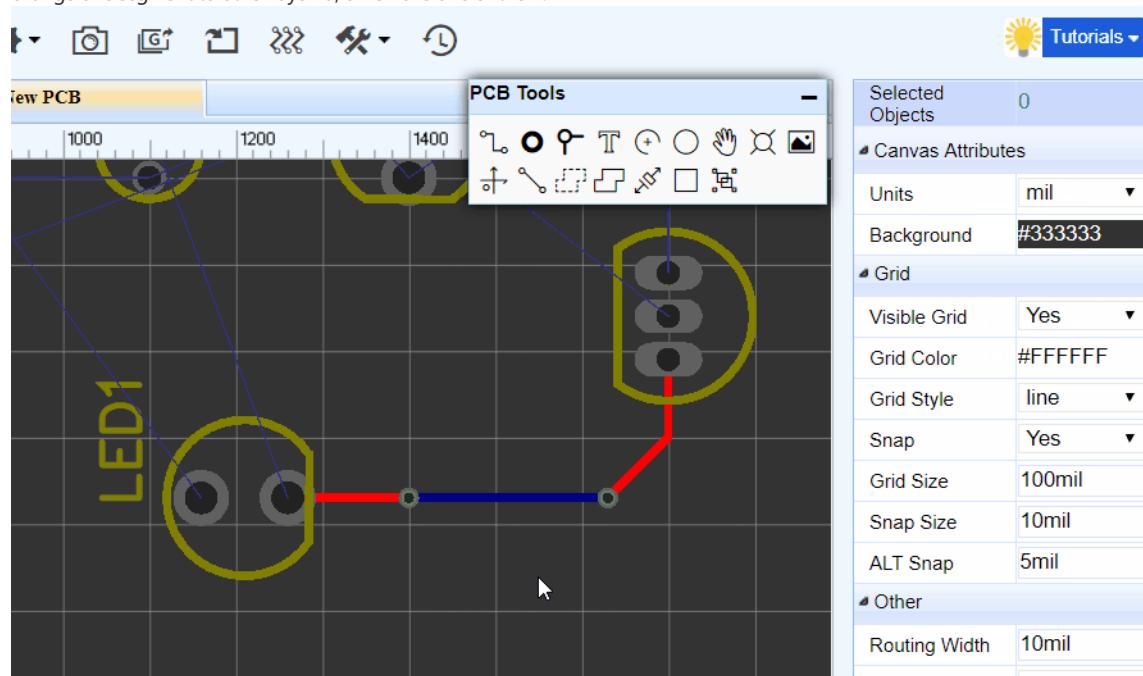
Plated : Yes or No.

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

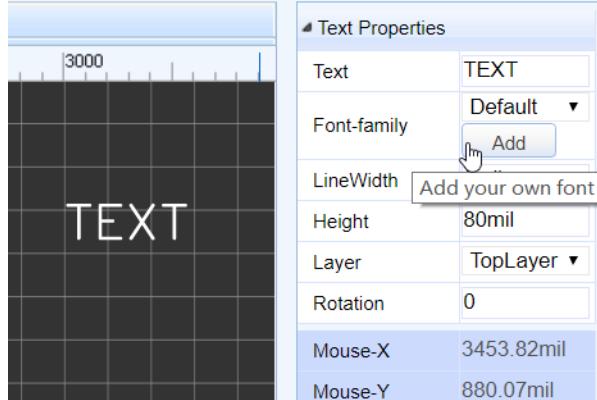
When placing a **via** on a track, the track will be cut to two segments. Placing two vias on a tracks, you will get three segments, then you can change one segment to other layer id, or remove one of them.



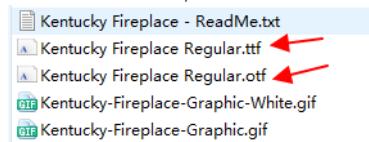
Text

You can add more fonts from your computer or download some [free fonts](http://www.1001freefonts.com):www.1001freefonts.com.

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`.

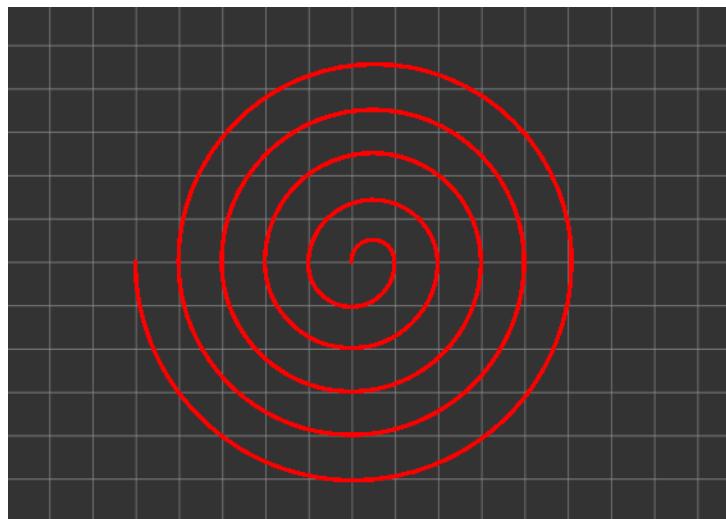


So you can add any fonts by yourself. EasyEDA doesn't cache the font on our server, so if you close the editor, you need to add the font again by yourself.

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

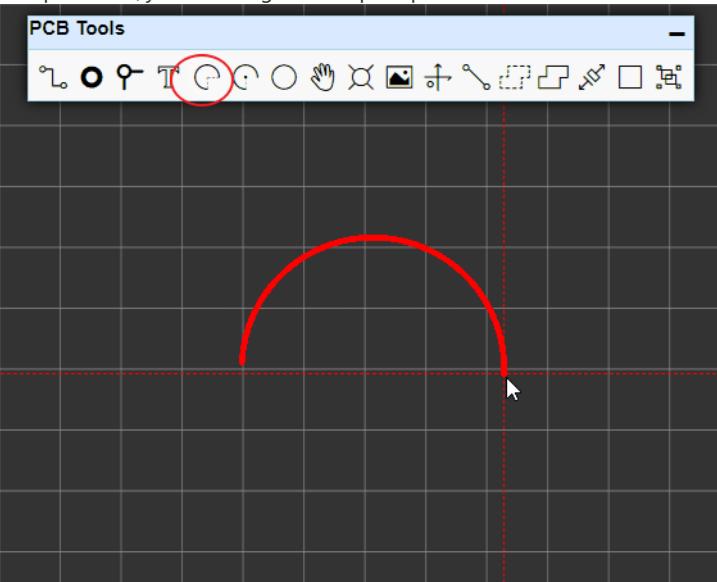
Arc

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

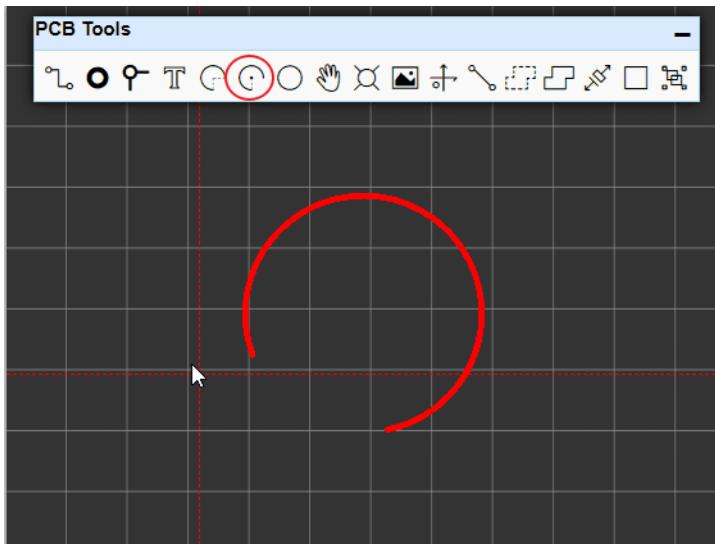


EasyEDA provides two Arc tools:

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



- Center point fixed, you can change the radius.



Circle

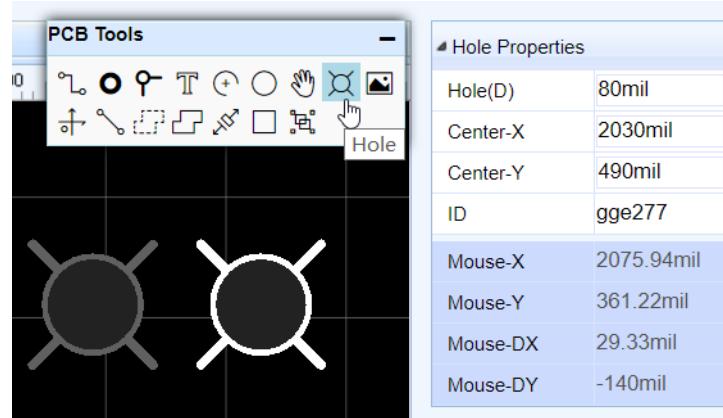
You can draw a circle in PCB , but it can only be drawn at SilkLayer and Document Layer. If you want to draw a circle at TopLayer or BottomLayer, please use Arc.

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.

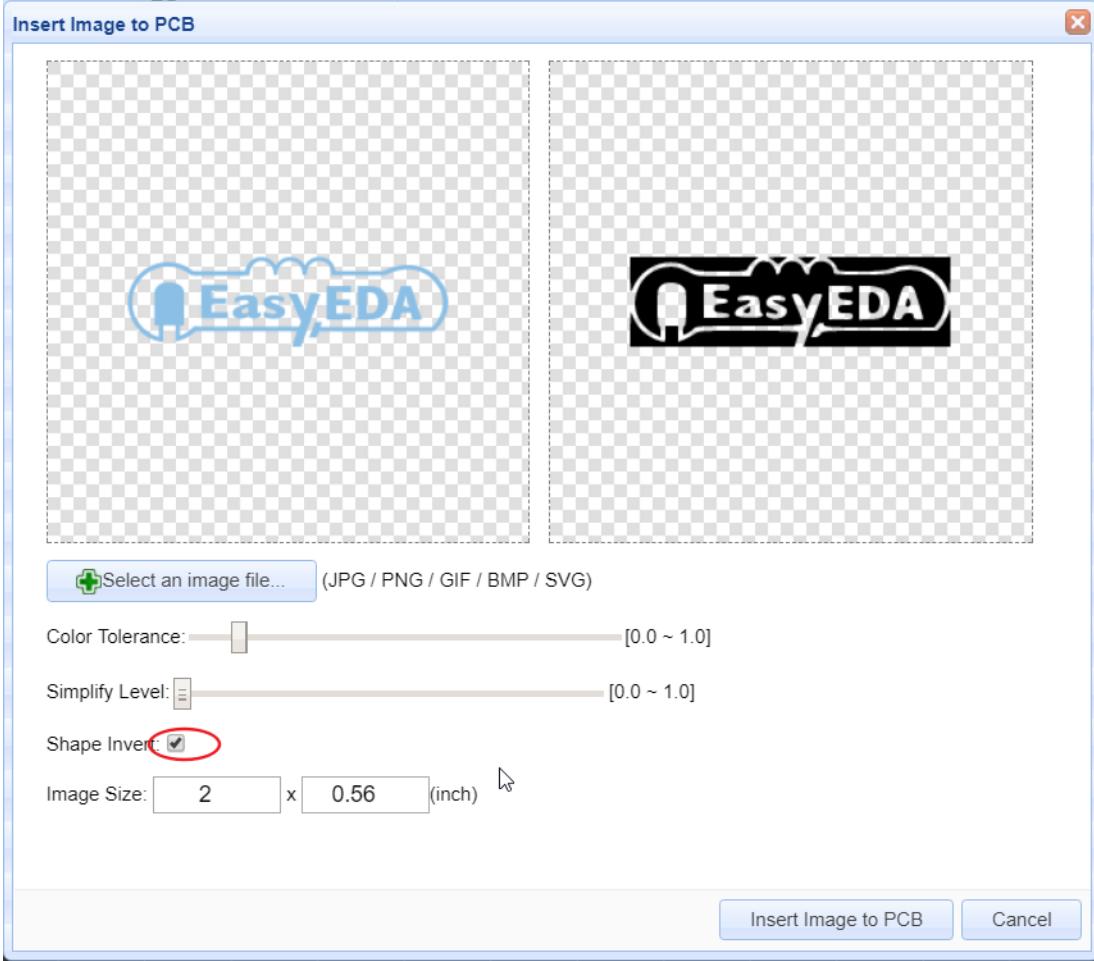


Image

On PCB and PCB Lib 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.

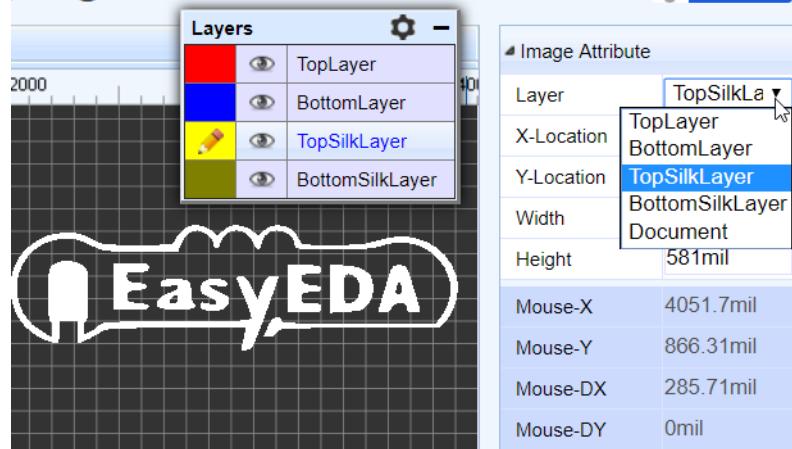


In this dialog, you can choose your favorite image, EasyEDA support **JPG**, **BMP**, **PNG**, **GIF**, and **SVG**. Unlike some other EDA tools which only support a Monochrome Bitmap image, EasyEDA supports full color, but Monochrome Bitmap is welcome.

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.

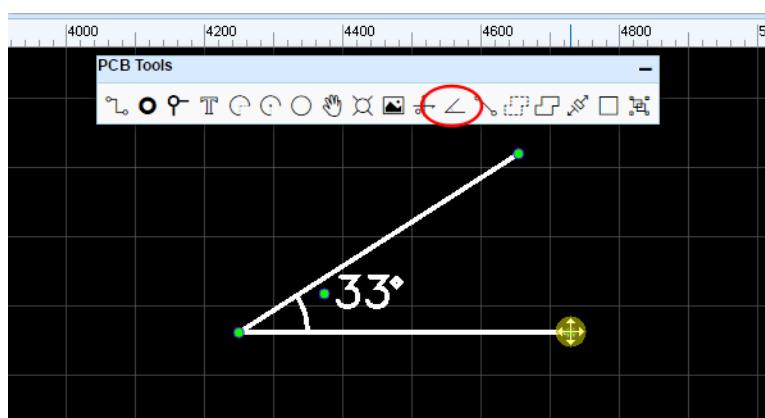


Canvas Origin

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

Protractor

We provide a protractor for PCB tools.

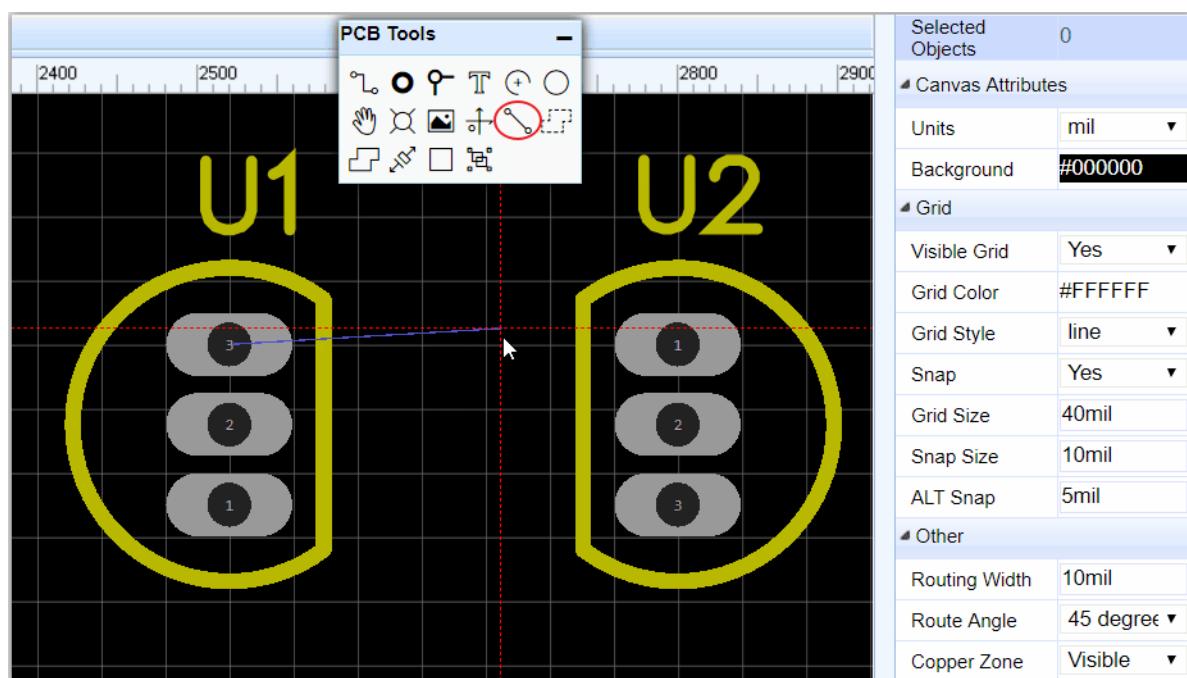


Connect Pad to Pad

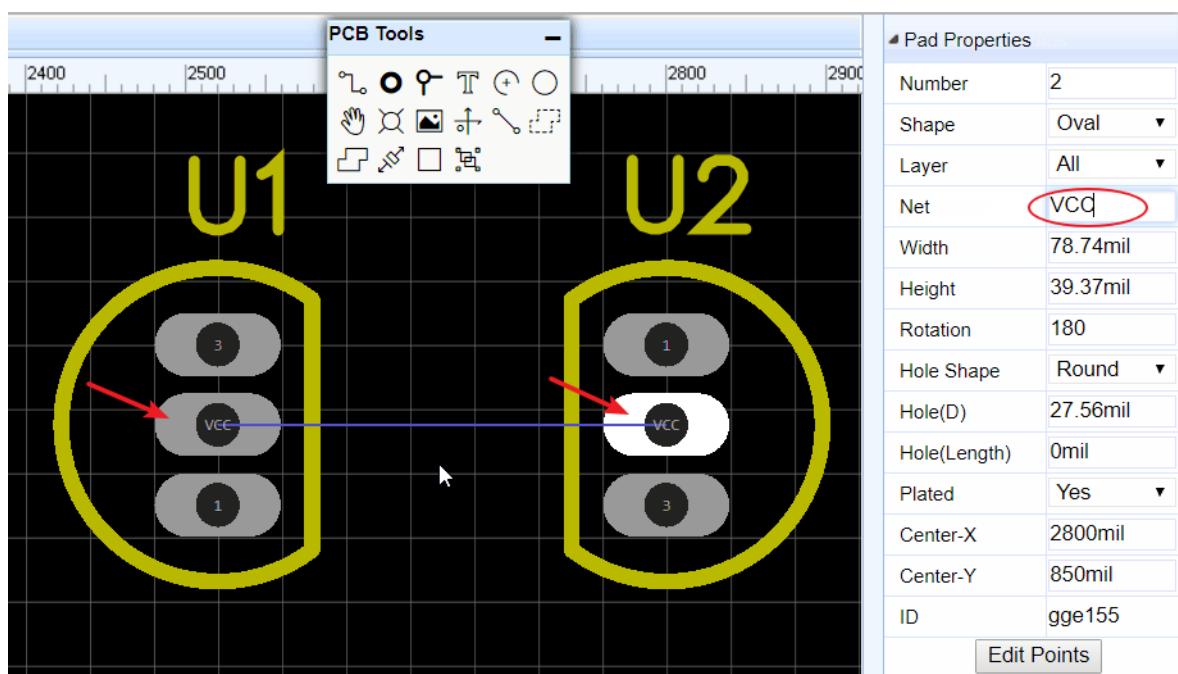
When creating a PCB without a Schematic, none of the pads on the Footprints have nets connecting them so there will be no ratlines.

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.

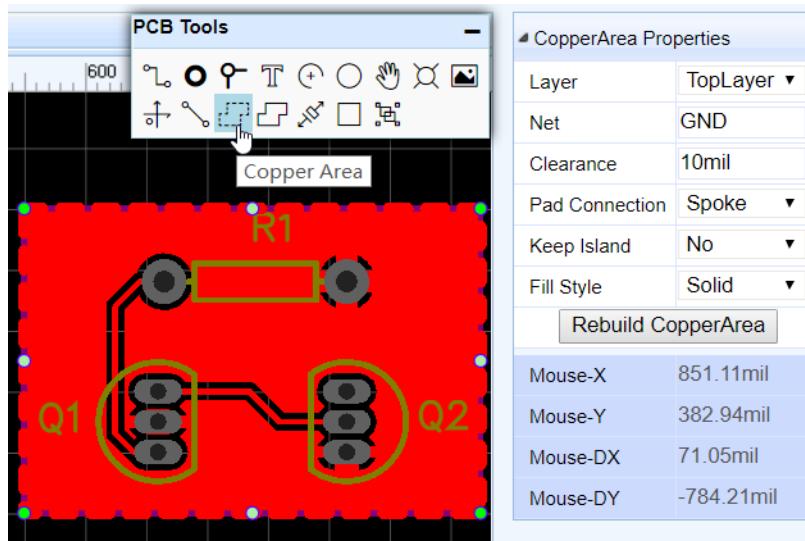


For more information about Ratline you can refer to the [Ratline](#) section.

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.



Layer: Bottom, Top, Inner1, Inner2, Inner3, Inner4;

Net: the net that the copper area is connected to;

Clearance: clearance of the copper area from other nets and floods;

Pad Connection: direct or spoke (i.e. a cross shaped heat shunt);

Keep Island: Yes/No. This keeps or removes any isolated areas of copper created as part of the flooding process. It is usually good practice to removes these unless you really need them to maintain a more even spread of copper (copper balance) on your PCB;

Fill Style: No/Filled. No removes the fill so that you can see the tracking more clearly;

After drawing the copper area, set the net it is to be connected to (floating copper areas are not recommended because they can cause EMC and Signal Integrity (SI) problems).

Lastly, don't forget to click the button Rebuild Copper Area to **rebuild** the flood.

Two Tips:

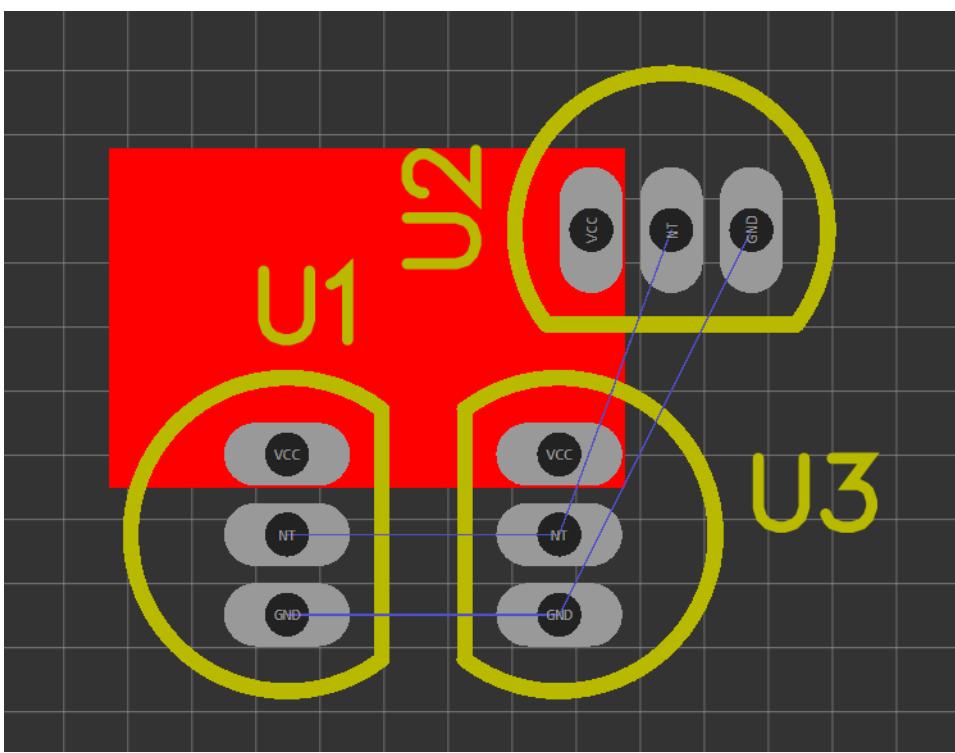
1. Hotkey **Shift+B** to build all of the copper areas.
2. Hotkey **Shift+M** to clear all of the copper areas.

Solid Region

EasyEDA has added a new tool Solid Region for PCB design

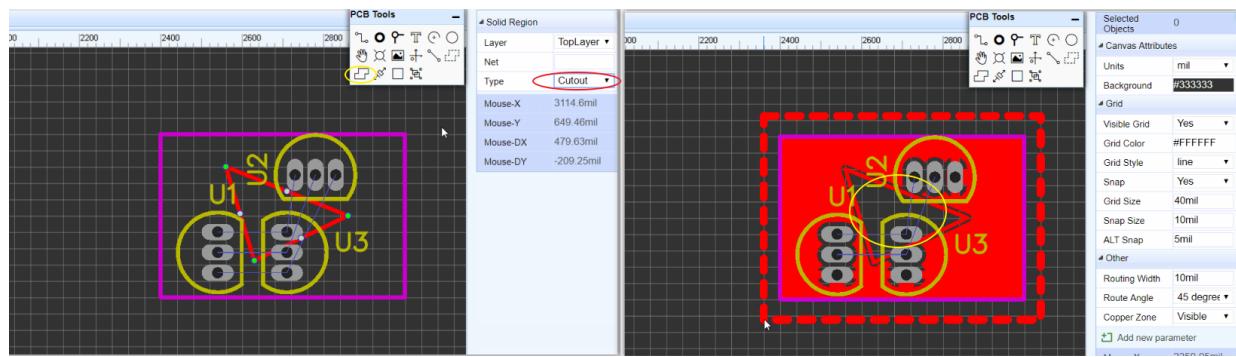


This is a very useful, quick way to connect Pads. You can draw a Solid Region to include all of these pads with same net name, then set the region to the same net name as the pads. It is like Copper Area but easier to use for small areas. To use Solid Region like this, set the Type attribute (in the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



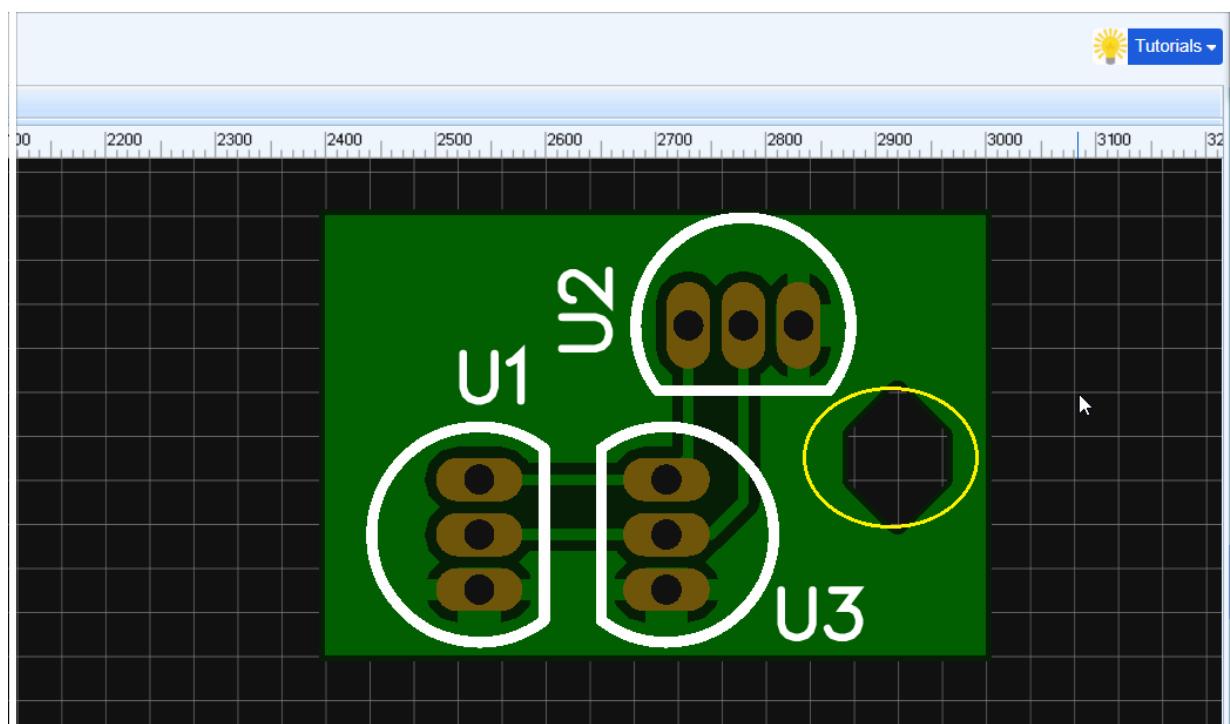
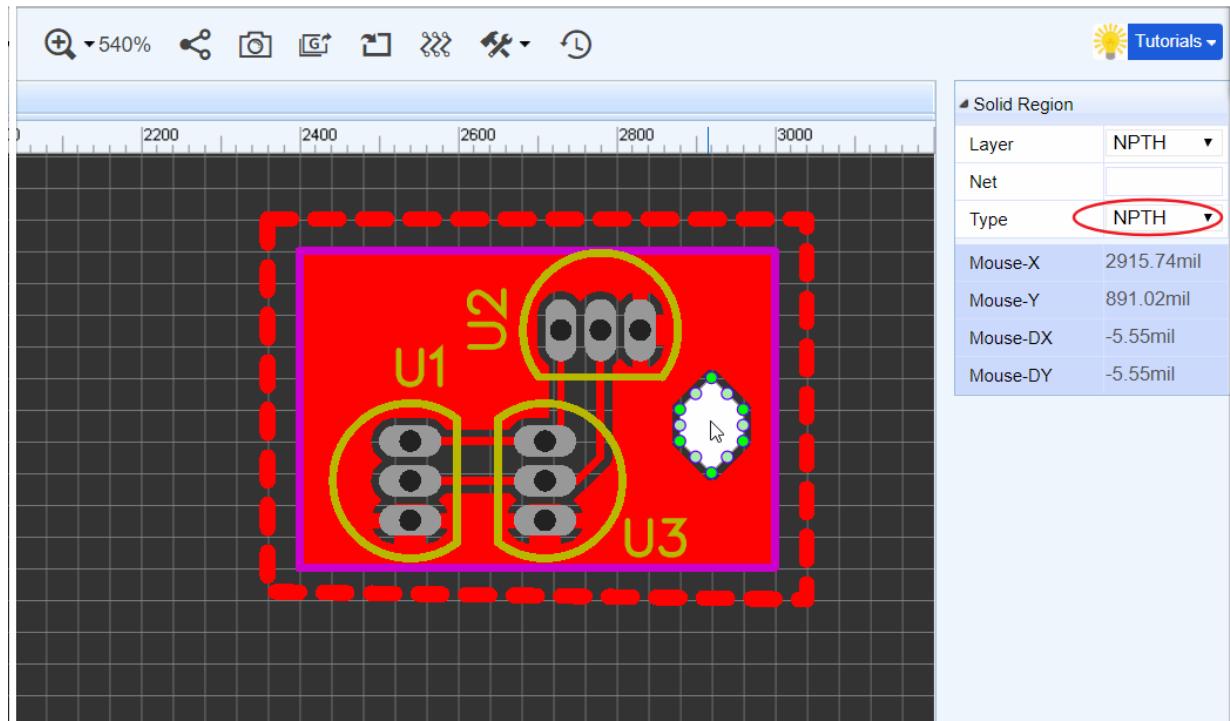
The Solid Region can also be used to create a cutout in a copper area.

If you have a copper area but need an area inside it to not be filled then you can draw a Solid Region and set the Type attribute (in the right hand Properties panel) to Cutout, then this area will be free of copper, as shown in the image below:



Lastly, by setting the Type attribute (in the right hand Properties panel) to NPTH(Non Plated Through Hole), Solid Region can be used to create a *Non Plated Through Hole* of an arbitrary shape.

When the Gerber files are generated, an area defined by a Solid Region set to a Type NPTH in the PCB editor will create an area defined to be a NPTH hole and you can see it in the PCB photo view as below:

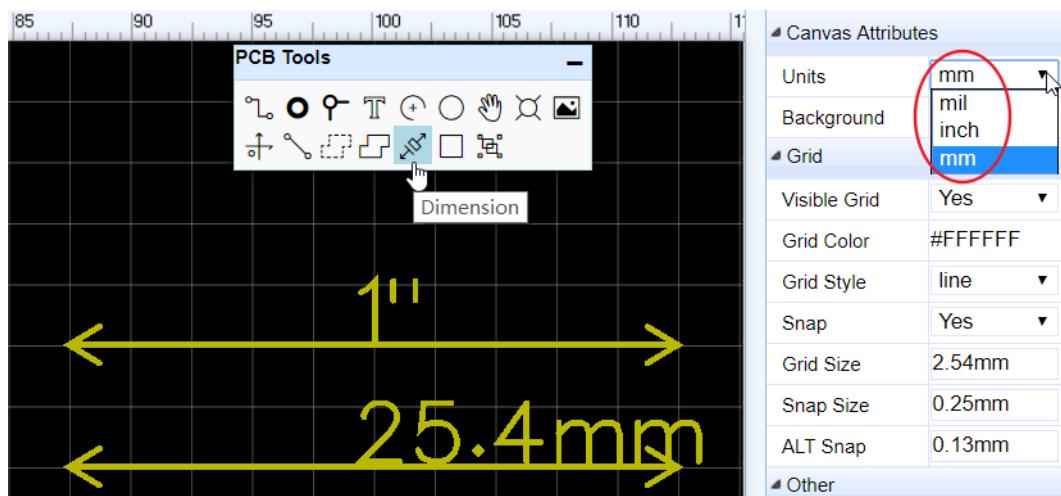


Measure/Dimension

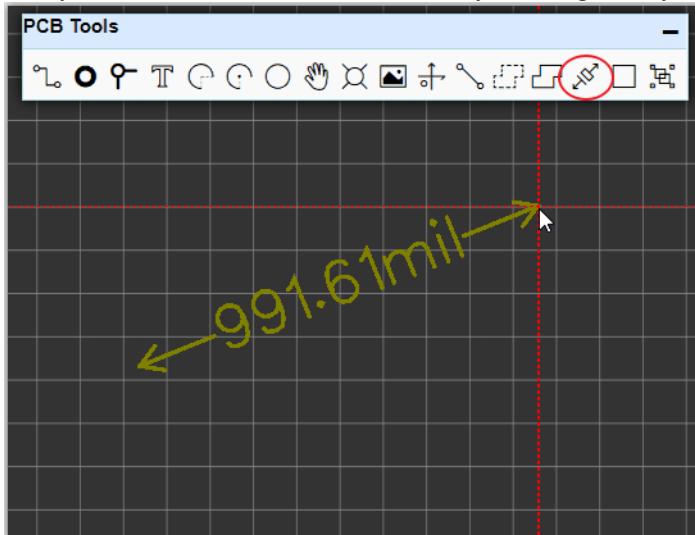
Making and adding measurements is useful in PCB design. EasyEDA provides two methods to do this.

1. Dimension tool in the PCB Tools palette:

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

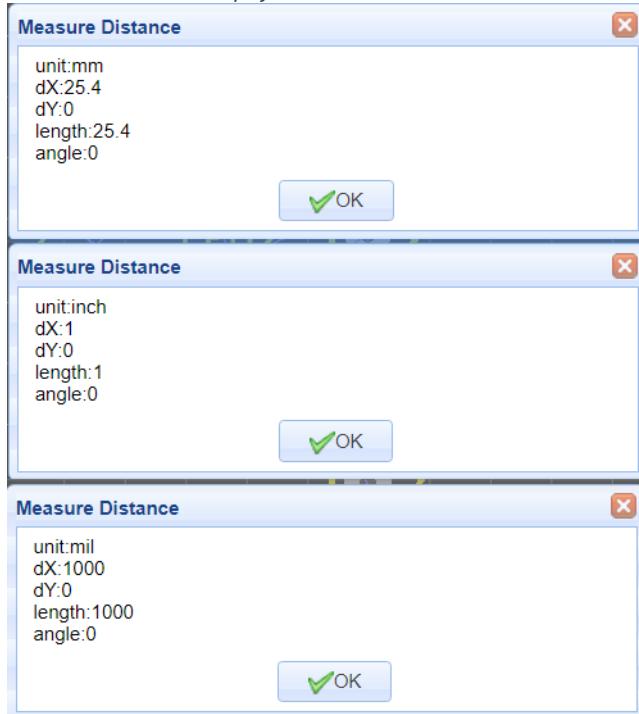


When you click one side of the dimension on the PCB, you can drag it for any directions or change its length.



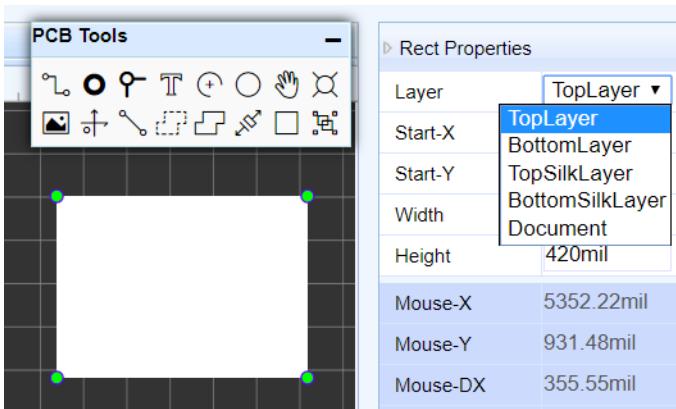
1. Measure a distance using M Hotkey: press M, Or Via: Super menu > Miscellaneous > Measure Distance, then click the two points which you would like to measure.

Note: This method will display the distance units which is the canvas' units.



Rect

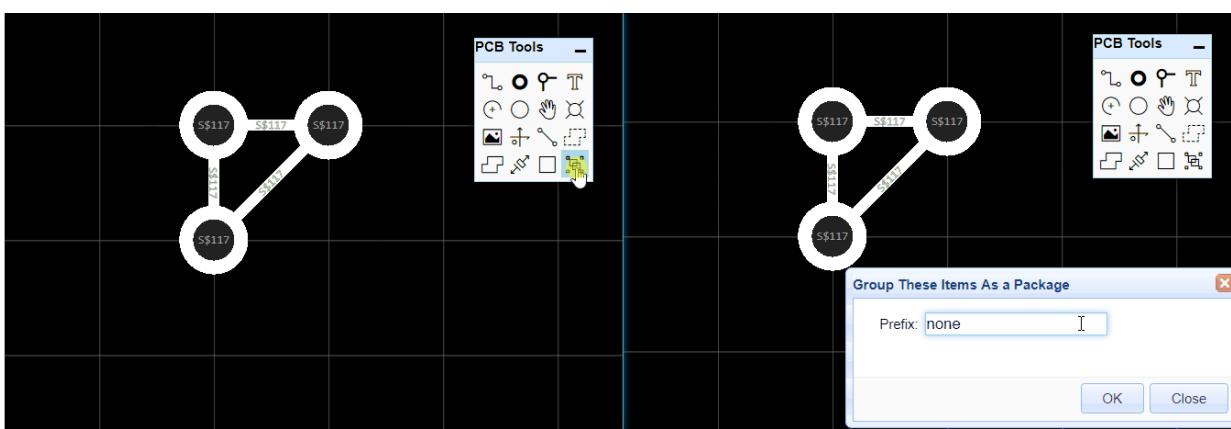
It looks like a Solid Region, but it can't be set Nets and you can't set the Layer as NTPH.



Group/Ungroup

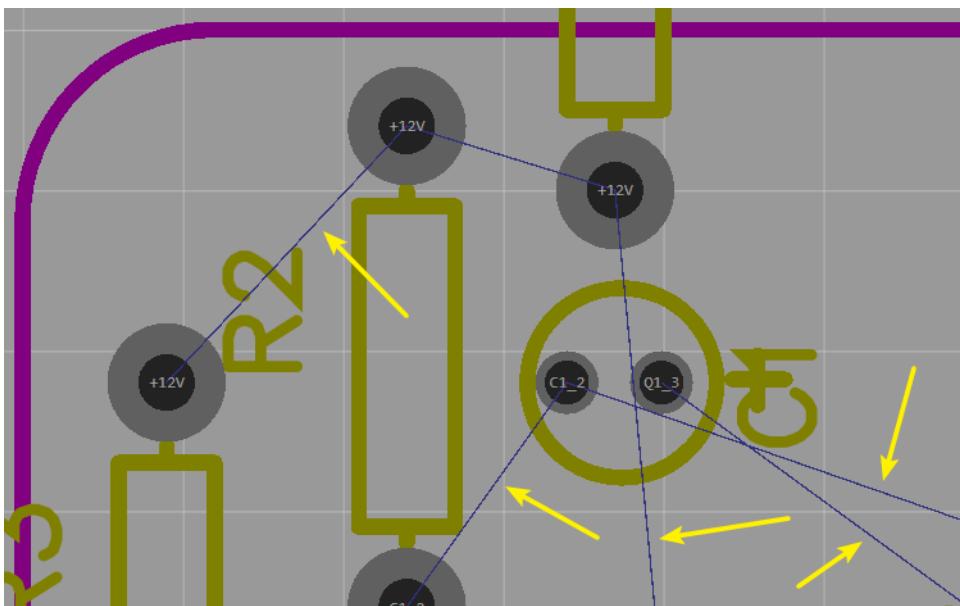
Just like Group/Ungroup in the Schematic Editor can be used to create a schematic lib symbol, you can use Group/Ungroup from the PCB Tools palette to create a PCB Lib footprint in the PCB editor.

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

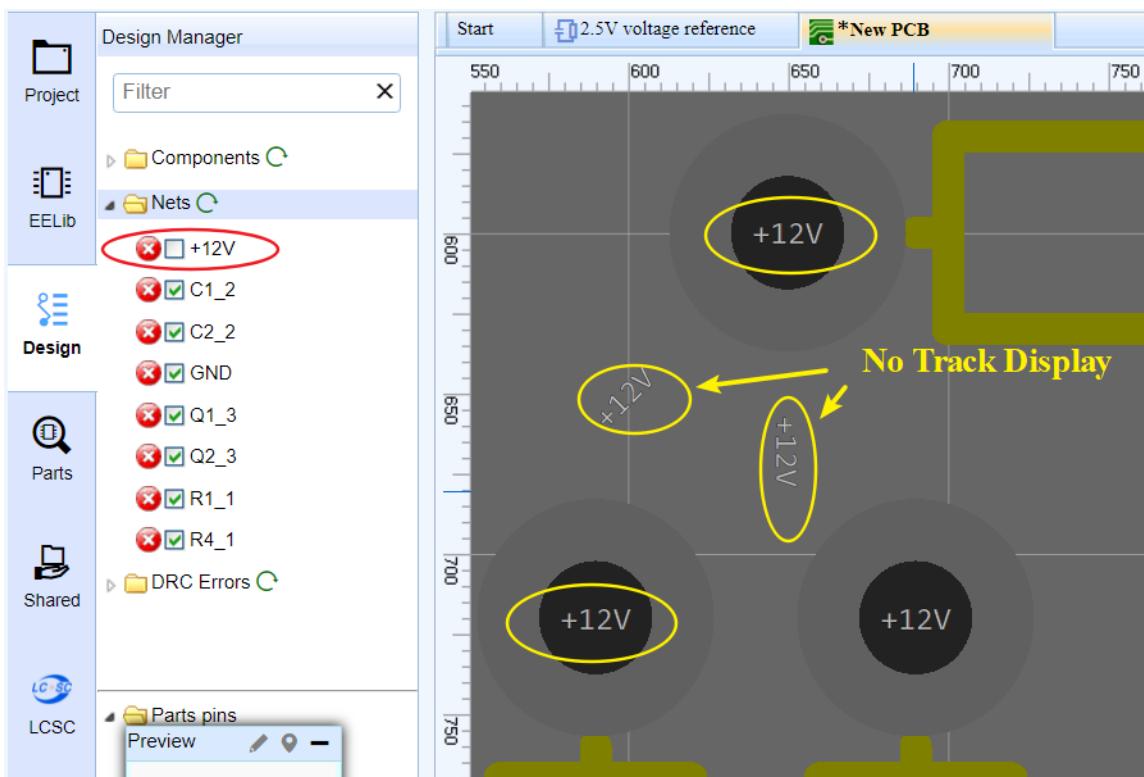


Ratline

When you lay the track in the PCB, Between PIN and PIN 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.

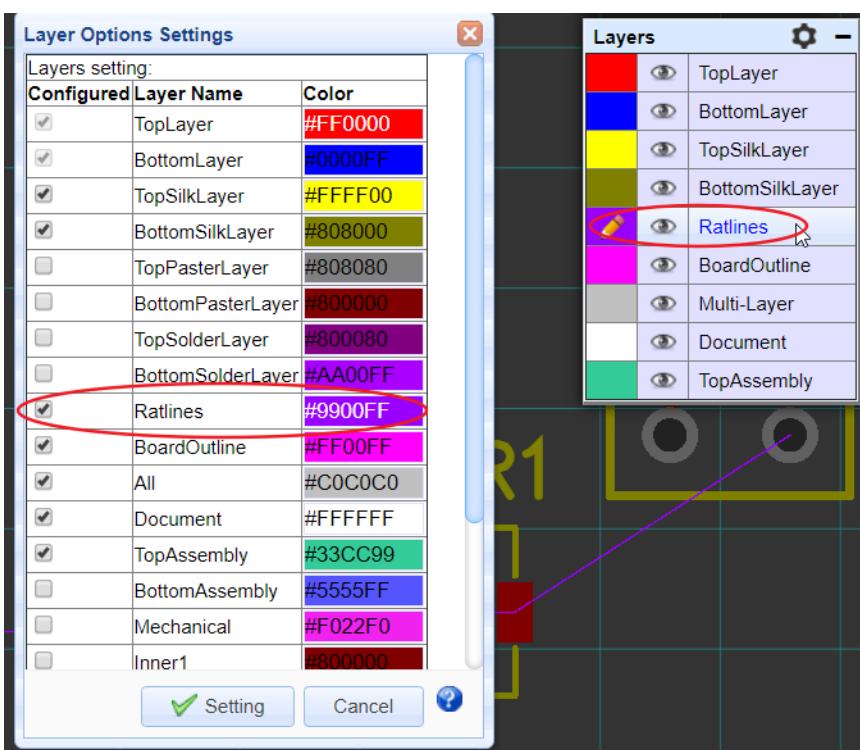


If you want some type of ratline to not show on the PCB editor, you can untick the net you like in the design manager, as below deselect +12v: If you still draw a track in +12v after deselecting, canvas will not display this track , but it will show a text with +12v as below.



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

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.

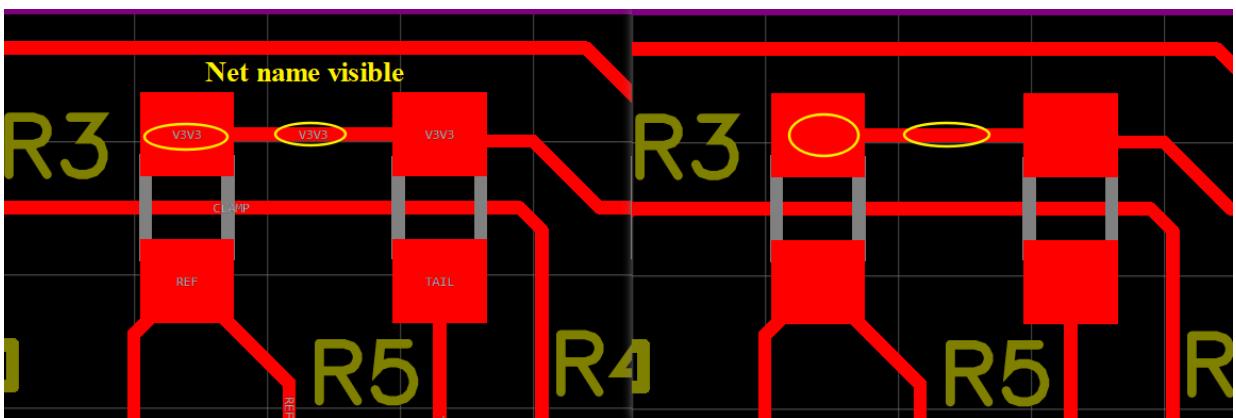


PCB Net

Net Name Visible

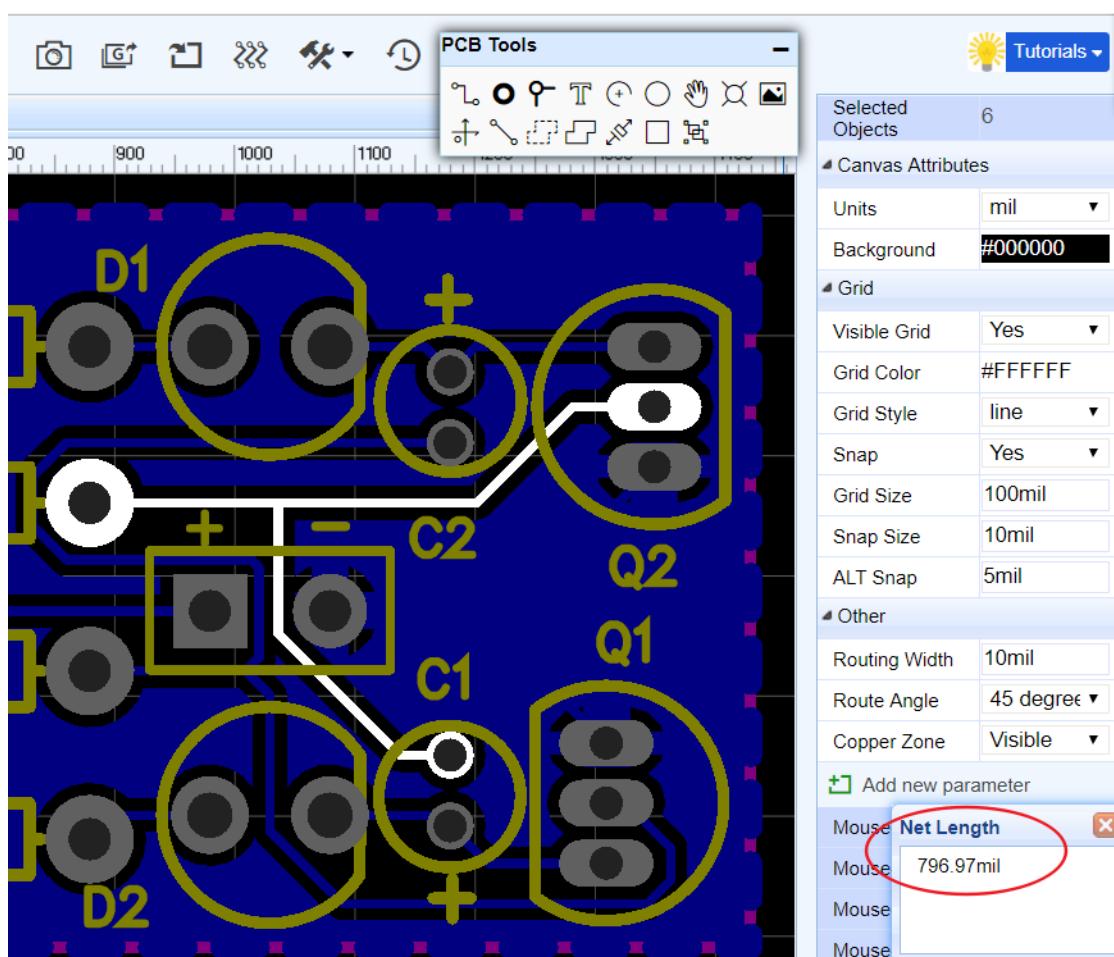
PCB editor can display net name in the track or Pads, if you don't need this feature, just need to turn it off via :

Super menu > View > PCB Net Visible, or press hotkey Q .



Net Length

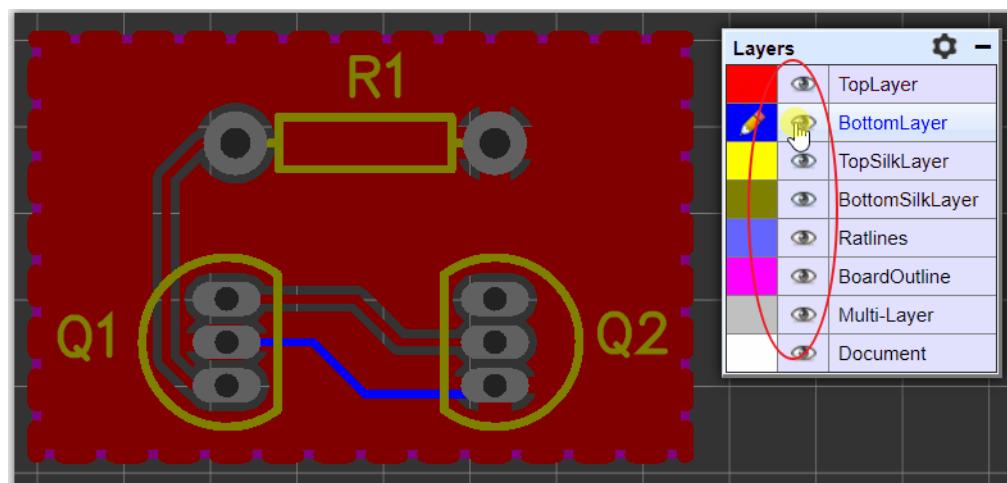
After selecting a track, and then pressing H key, EasyEDA will highlight the whole net and pop a message box to tell you the whole net's length. like in the image below



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 coloured rectangle representing the required layer. The pencil icon in the coloured rectangle indicates that this is the active layer.

Show/Hide layers: click on the eye icons to show/hide layers.



HotKeys for layer activation:

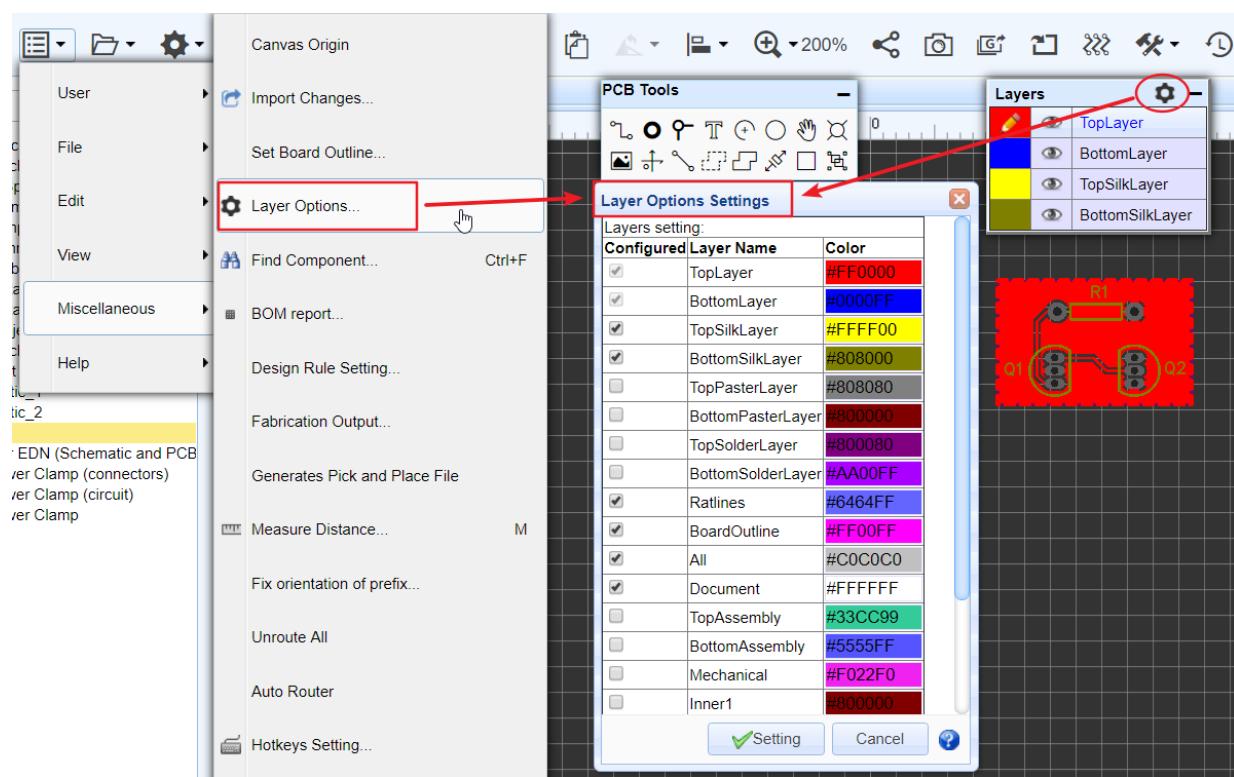
- **T:** Top Layer is active
- **B:** Bottom Layer
- **1:** Inner1 Layer
- **2:** Inner2 Layer
- **3:** Inner3 Layer
- **4:** Inner4 Layer

Layer Setting

Via Super menu > Miscellaneous > Layer Options..., Or Click Layers' gear icon.

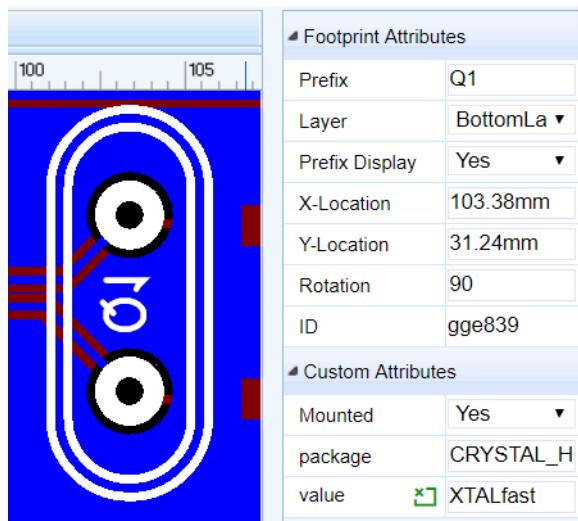
You can find the Layer Options Settings dialog.

In this dialog, you can change the layer's Color and configure which layers are shown in the Layers Tool. If you plan to design a PCB with more than 2 layers, then you must tick Inner1 and Inner2 for a 4 layer PCB plus Inner3 and Inner4 for a 6 layer PCB.



Footprint attributes

When selecting a Footprint, you can find its attributes at the right hand Properties panel.



Layer: You can set a footprint to be on the TopLayer or BottomLayer.

Note: The footprint mirrors when swapping layers.

X-Location and Y-Location: Moves the origin of the footprint to a precise position.

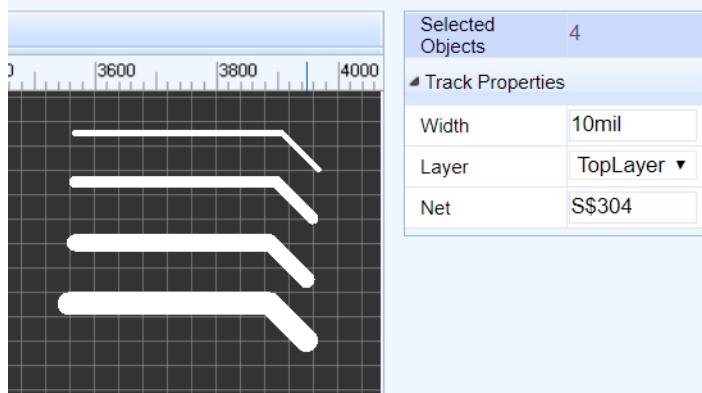
Rotation: Rotates the footprint about its origin over the range from 0o to any angle in 10 steps (visually of course multiples of 360o will appear identical).

ID: EasyEDA will assign a unique ID for each footprint automatically, you can't modify it.

Change Attributes in Batch on PCB Editor

Sometimes, we need to change some attributes of multiple objects together, such as the track width, hole size and font size.

Now, you can select them and do some changes. Taking the track for an example. If you select 3 tracks, now you can change their `Width`, `Layer`, `Net` together.



You could also use it with other items such as **Pad**, **Via**, and **TEXT**.

Layout A PCB Without Schematic

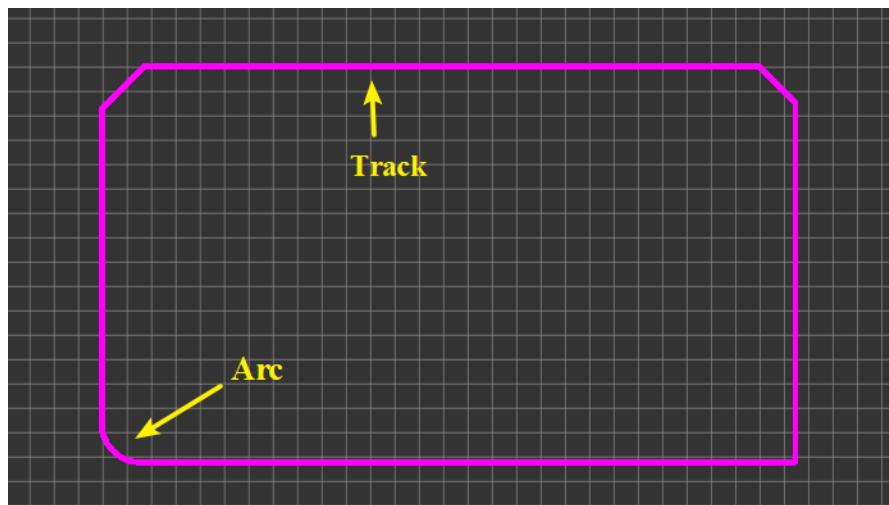
For some small PCB projects, maybe you don't need a schematic. EasyEDA allows you to lay the PCB directly from the PCB Editor.

Start a new PCB and you can add footprints directly from the PCB Libs from Left Navigation Panel **Parts** and then just track them.

For setting pad to pad connections, you can check the above section : [Connect Pad to Pad](#)

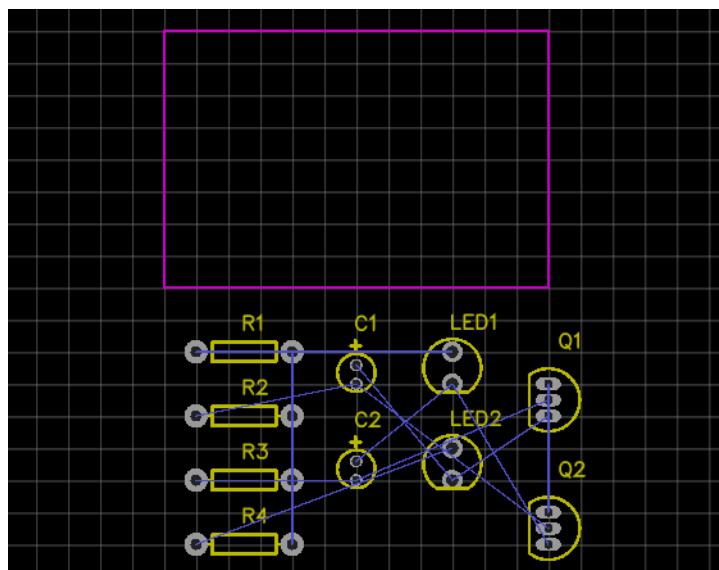
Board Outline

Before placing footprints we need to create a board outline. The board outline must be drawn on the **BoardOutLine** layer. So first, set **BoardOutLine** 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.

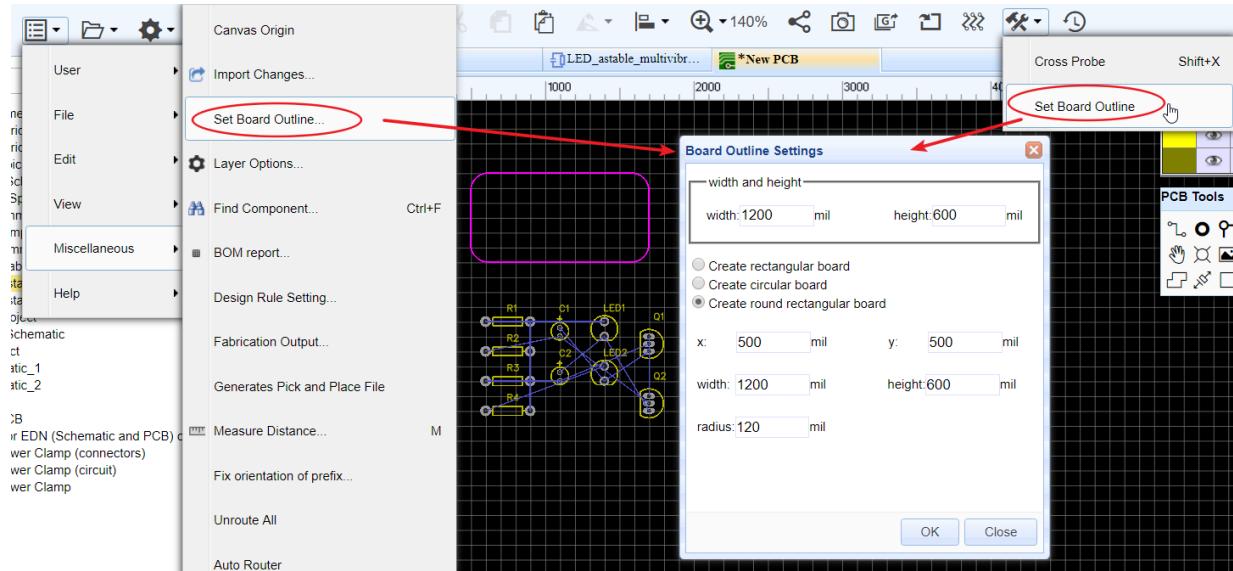


To create a simple rectangular board outline, this arc can be removed and the line X and Y end points edited - either directly in the Properties panel or by dragging the line ends - to close the rectangle.

Alternatively, an outline with more rounded corners can be created by copying the arc and rotating it in 90 degree steps to position it over the

desired right angle corners and then editing the line X and Y end points - either by dragging the line ends or directly in the Properties panel - to overlap the arc end points (also shown but not editable in the Properties panel).

And EasyEDA provides a **Board outline wizard**, so it is very easy to create a board outline. Via: **Super menu > Miscellaneous > 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.

Design Manager

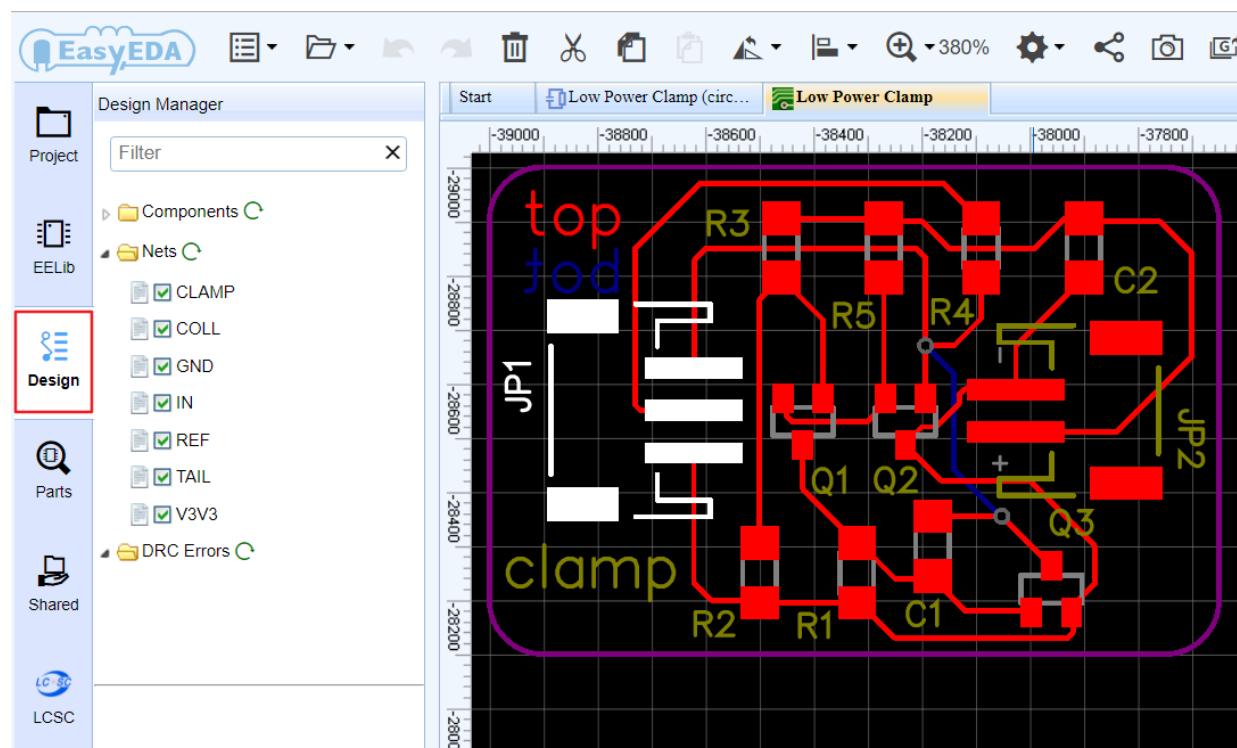
Just like Schematic's Design Manager, PCB's Design Manager can be found via:

Left Navigation panel > Design

or just press the **CTRL+D** hotkey to open the Design Manager dialog.

In this dialog, you can:

1. Click a component to highlight it.
2. Check/uncheck a component to show/hide it.
3. Filter to find a component or net.
4. Click a net to highlight the tracks/vias with the same net.
5. Check/uncheck the net to show/hide the net. For example, very often you may want to use this to hide a GND or supply net which has had a copper flood added to turn it into a plane and then show it again later.
6. Double click the net to remove all of the tracks and vias with the net name. If you want to reroute a net, this is the recommended method to use to un-route it first.



Import Changes

Before using "Convert to PCB", "Update PCB" in Schematic and "Import Changes" in PCB, please read [Essential Check Before Clicking "Convert to PCB" or "Update PCB" or "Import Changes"](#) section.

Sometimes, while working on a project, you need to make changes to the schematic and then update your board, to incorporate them.

It's easy to do this with EasyEDA.

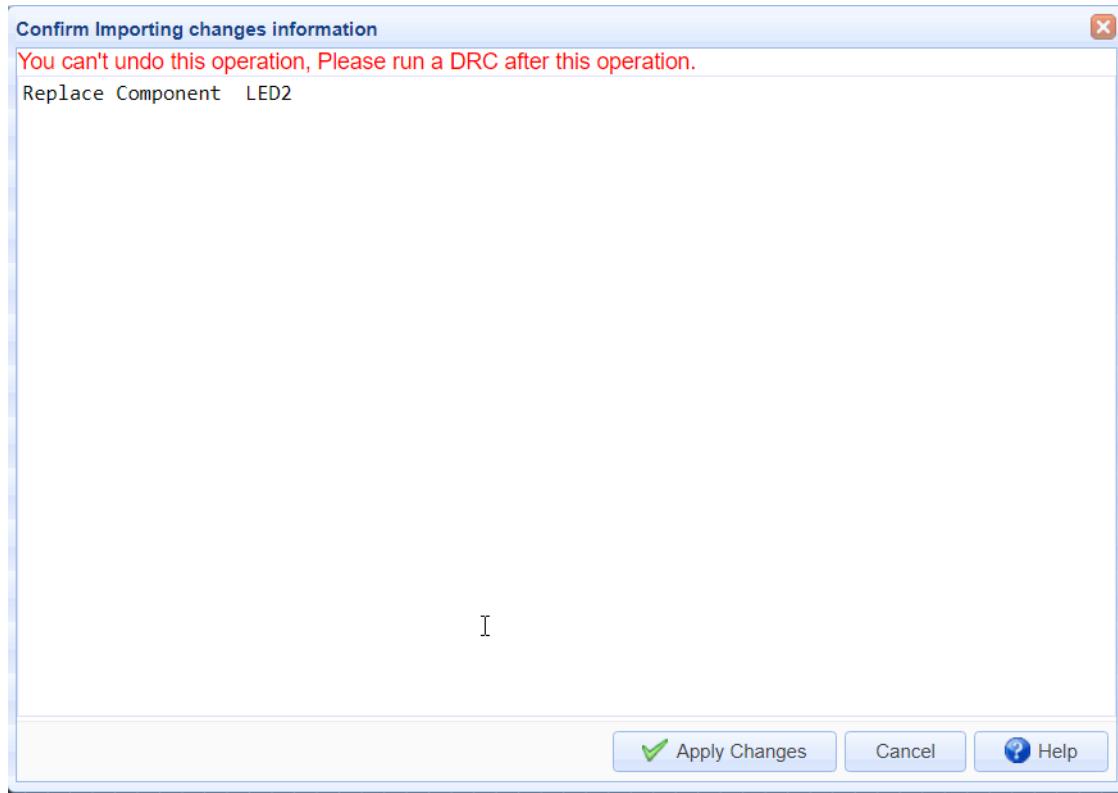
Go to the **PCB Editor**,

Super menu > Miscellaneous > Import Changes

Or click that button at the tool bar



You will get a Confirm Importing changes information dialog:

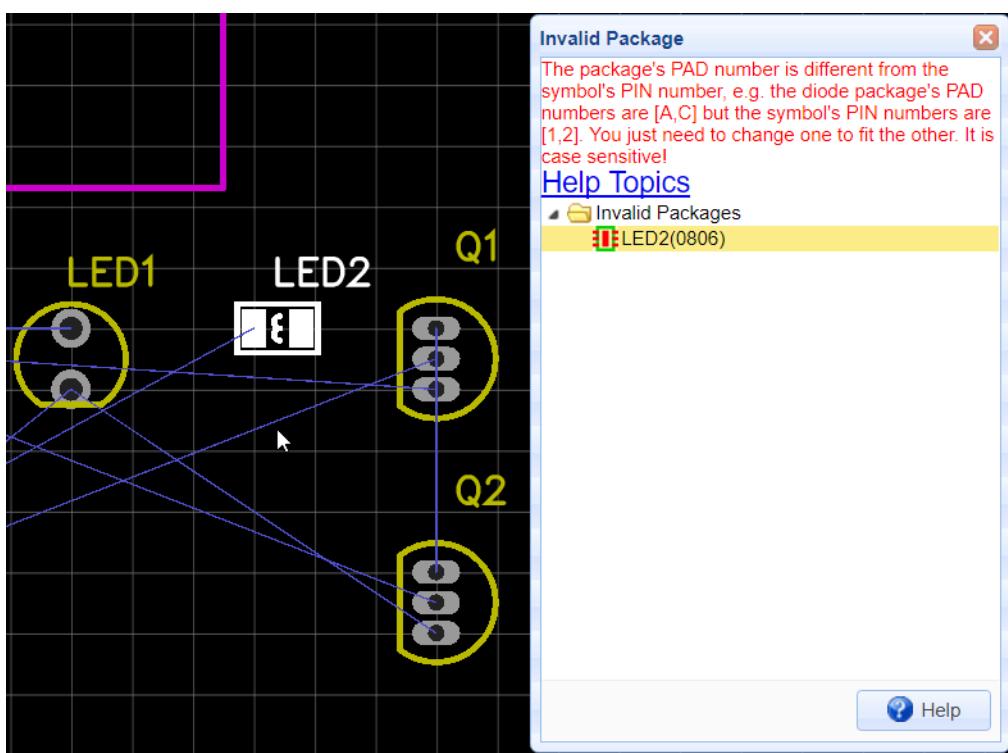


If you are happy with your changes, just click the Apply Change button.

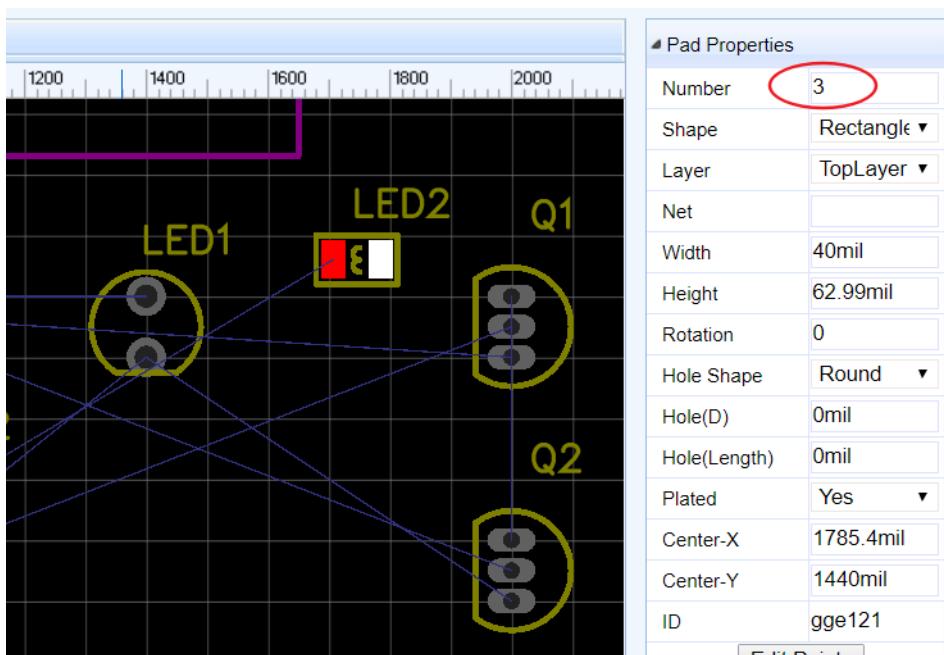
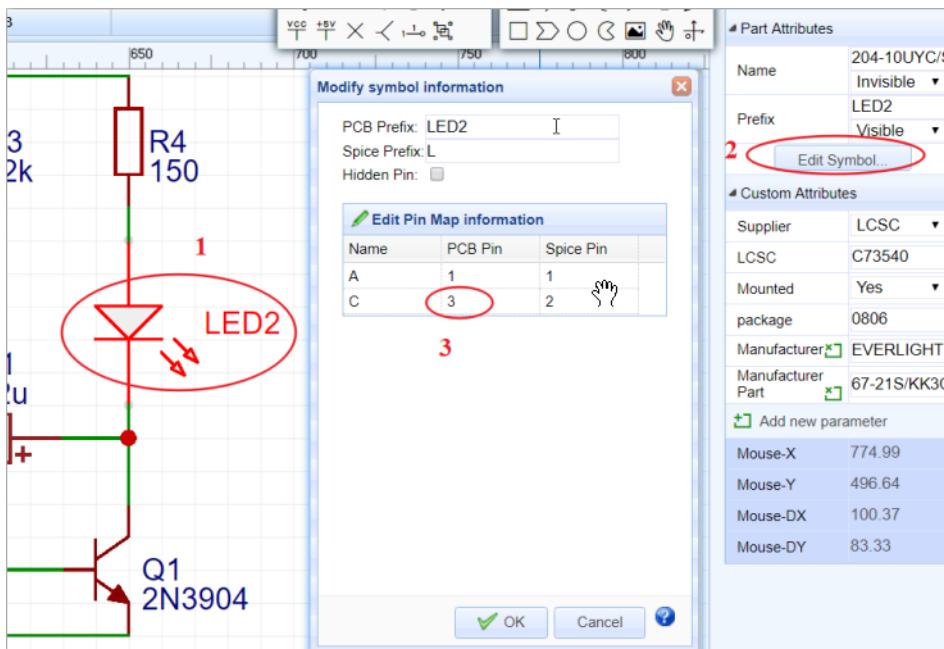
The changes will then be passed into the PCB layout and you can then adjust the tracking to suit.

Invalid Packages

Sometimes, when you try to convert a schematic to a PCB, you will get an error message dialog like below. Don't worry, it is easy to fix this problem.



From the error message, you will find that the symbol's PIN number is different from PAD number. What caused that? Check the image below,



From the image, we can get the PIN number in the schematic symbol is set as 3, but the PAD Number in the PCB Footprint is set as 2. Now that

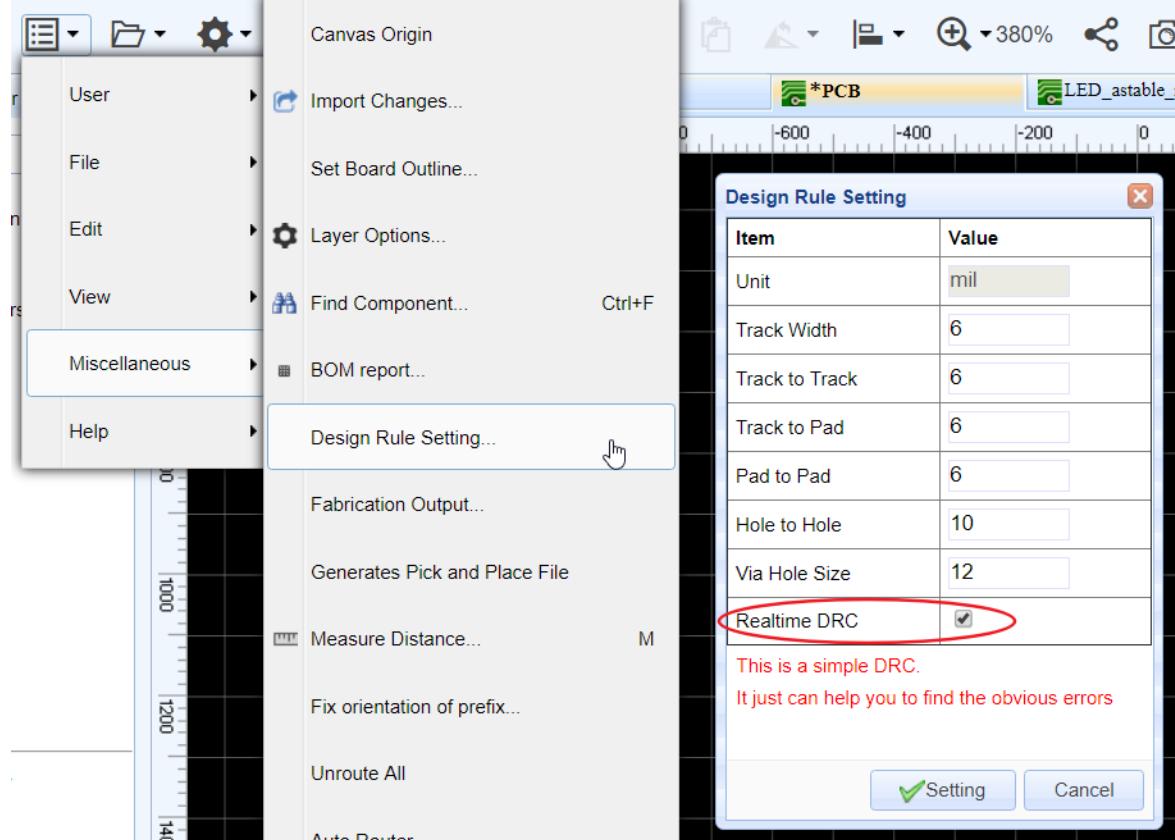
we've found the problem, how to fix this?

- Solution One: Change the schematic symbol. Using [PinMap function](#). Change the PCB PIN from 3 to 2. And save your schematic, and update PCB.
 - Solution Two: Modify the Footprint. Edit the Footprint, change the PAD from 2 to 3. And set this PAD net name to be the same as LED2 net name in the schematic.
- So, we should be aware that PIN number should be the same as Pad number.

Design Rule Check

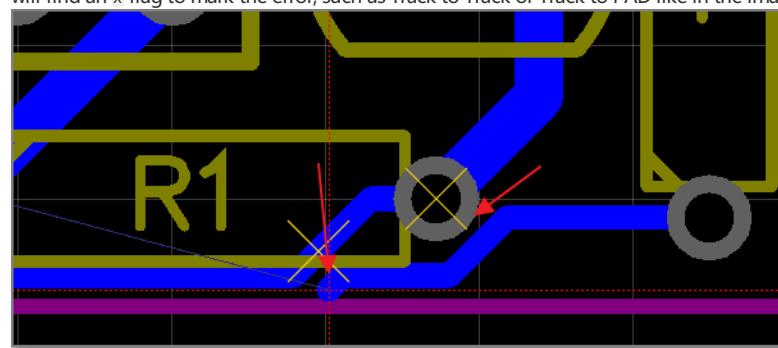
EasyEDA provides a powerful real time DRC(Design Rule Check) function.

Via at:Super menu > Miscellaneous > Design Rule Setting to open the DRC setting dialog:



Note: When you convert a schematic to PCB, the real time DRC is open. But in the old PCB, the real time DRC is closed. you can open it as in the image above.

This is a big feature of EasyEDA. It is hard to fix DRC errors after laying out the PCB. Now EasyEDA will let you know the error in routing. You will find an X flag to mark the error, such as Track to Track or Track to PAD like in the image below

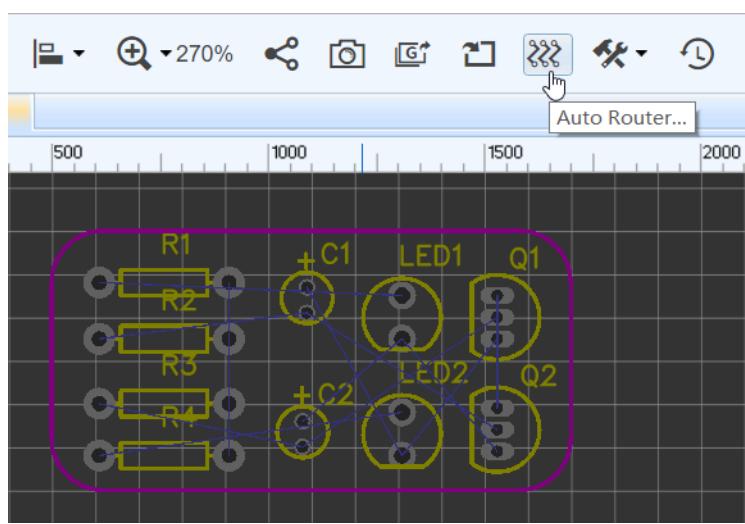


Auto Router

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.

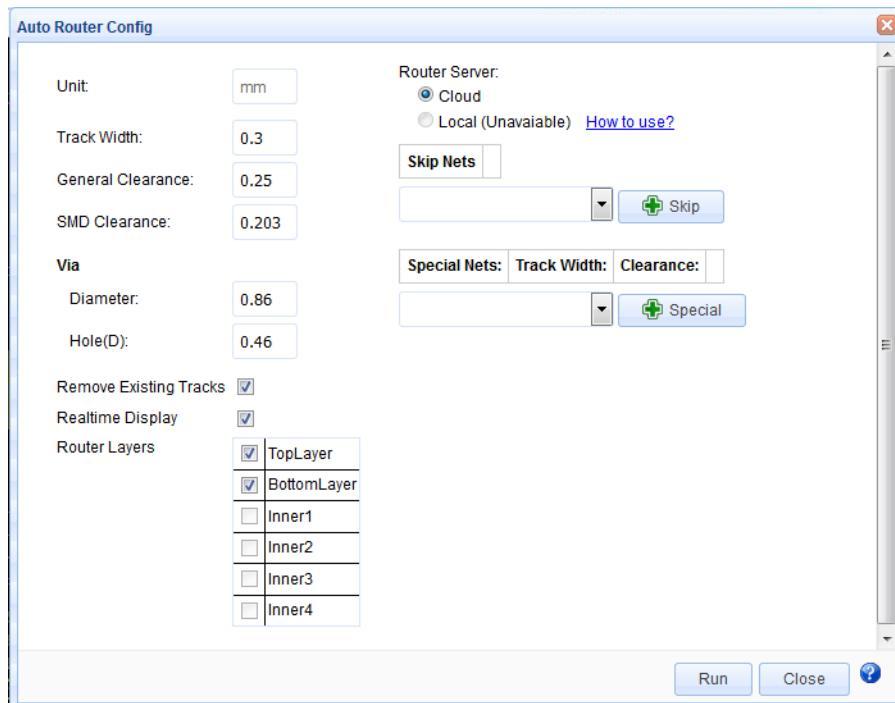
Steps:

- 1 Click the the auto router button from the toolbar or "Super Menu > Miscellaneous > Auto Router"



2 Config the auto router

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

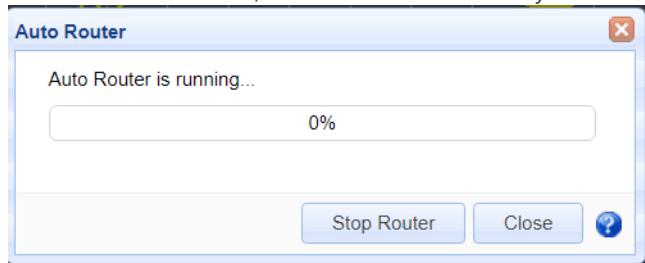


In the config dialog, you can set some rules to make the auto router result professional. These rule must equalize or more than DRC setting.

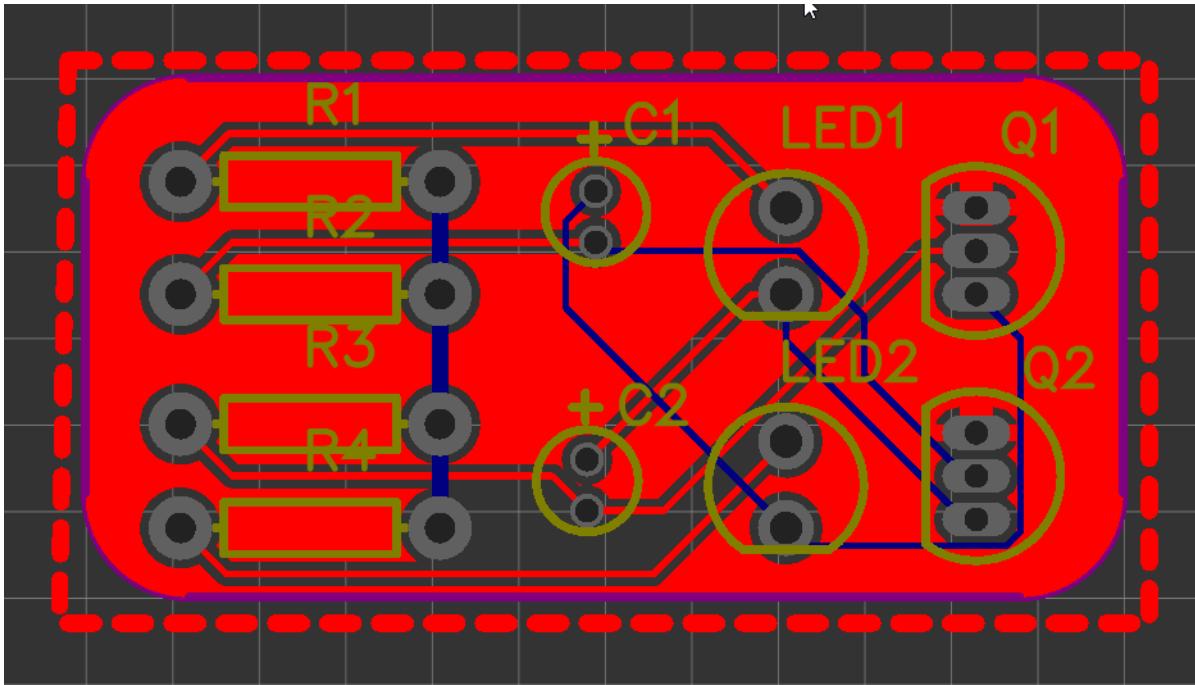
- Remove Existing Tracks:** If you want to reserve the routed track, you need to deselect it.
- Realtime Display:** when you select it, the real time routing status will show on.
- Router Layers:** If you want to route inner layer, you have to enable the inner layer first at [Layers Setting](#).
- Router 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.
- 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.
- Special Nets:** For the power supply track, you may want it to be bigger, so you can add some special rules.

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 like in the image below.

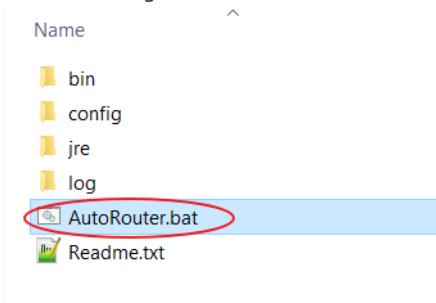


Local Auto Router

EasyEDA suggest that using local auto router rather than using the cloud server, because when many users using cloud server, the cloud auto router will fail.

The local auto router need to download and unzip it to the Non-System folder, this version only works on windows7(x64) or later. Download via: [EasyEDA Router.zip](#)

You need to configure the browser and execute the AutoRouter.bat first before click the **Auto Router** icon at editor.



Notice: Please use the latest Chrome or Firefox !!!

1)Chrome

If the local auto router is unavailable, you have to upgrade Chrome to version 60.0.3112.78 or later.

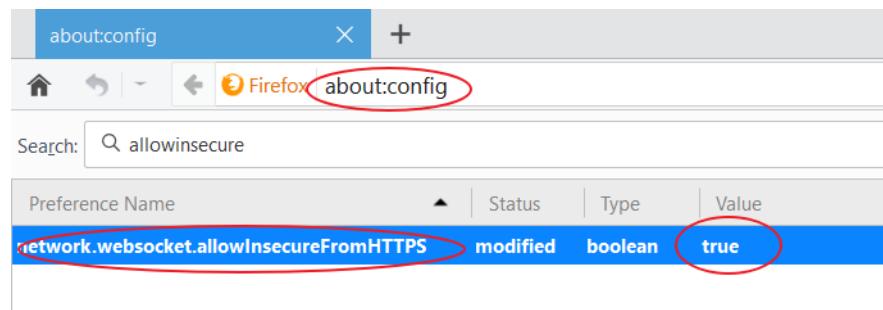
2)Firefox

Configure Firefox:

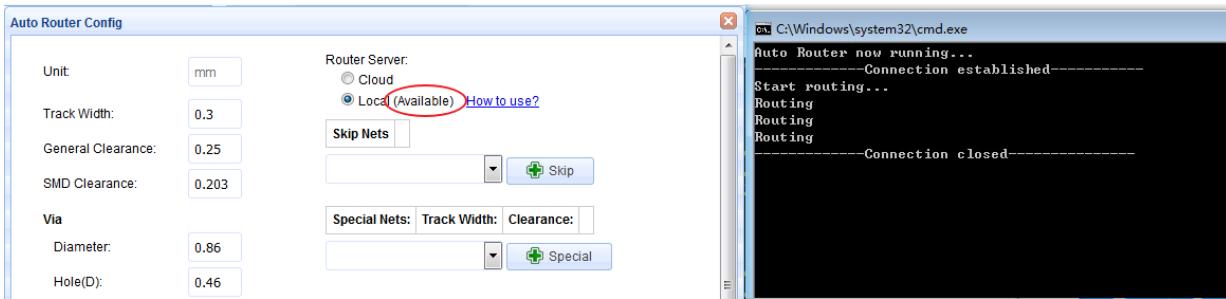
Type "about:config" into the address bar then press enter, and search "allowInsecure", the completely preference name as below :

```
network.websocket.allowInsecureFromHTTPS
```

double click it, its value will change as "true", re-open Firefox and try again.



If the local router server is available, the dialog will tell you. Click the **Run** button, the AutoRouter.bat dialog will show the process as below:



Sometimes, if you can't get it done, try the tips below.

1. Skip the GND nets, add copper area to GND net.
2. Use small tracks and small clearance, but make sure the value is more than 6mil.
3. Route some key tracks manually before auto routing.
4. Add more layers, 4 layers or 6 layers
5. Use local auto router rather than cloud server.
6. Tell the error detail to us.

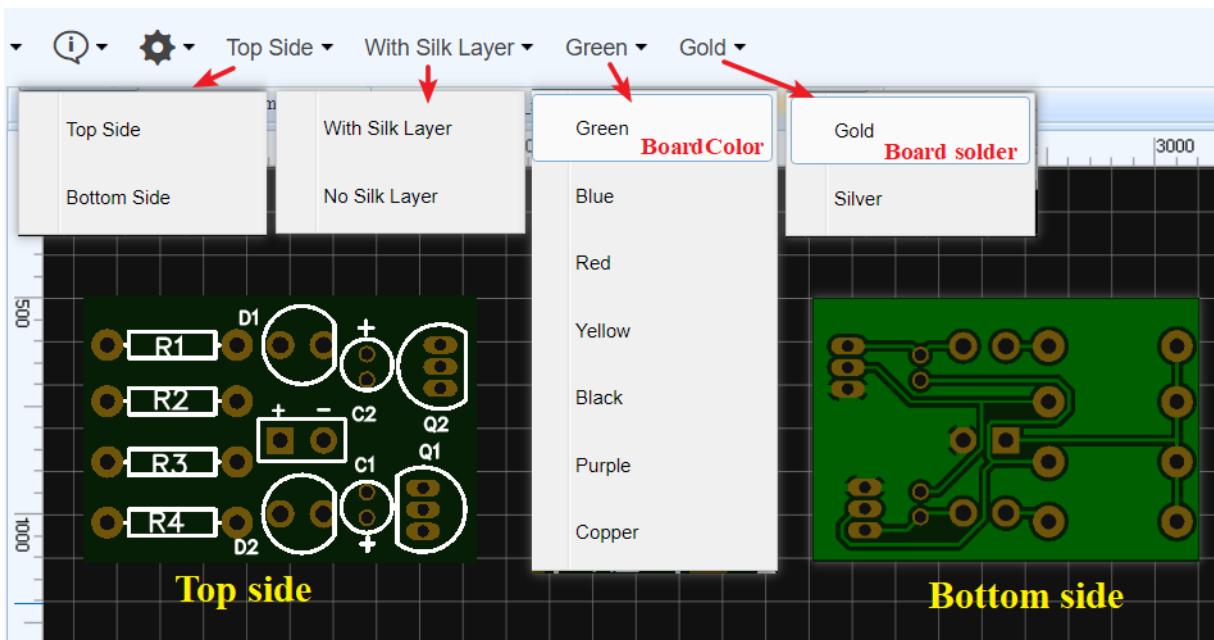
Some professional people don't like the auto router, because they think auto router is not professional, but you can use the auto router to check your placement. to check the density of your PCB.

Photo View

EasyEDA has no 3D View at present, but we provide a nice Photo View to help you to check the PCB. There is a `PhotoView` button on the PCB document toolbar, like in the image below. If you can't see this button, try to `reload` the PCB again.



After converting the PCB to Photo View, you can see the result as in the image below.

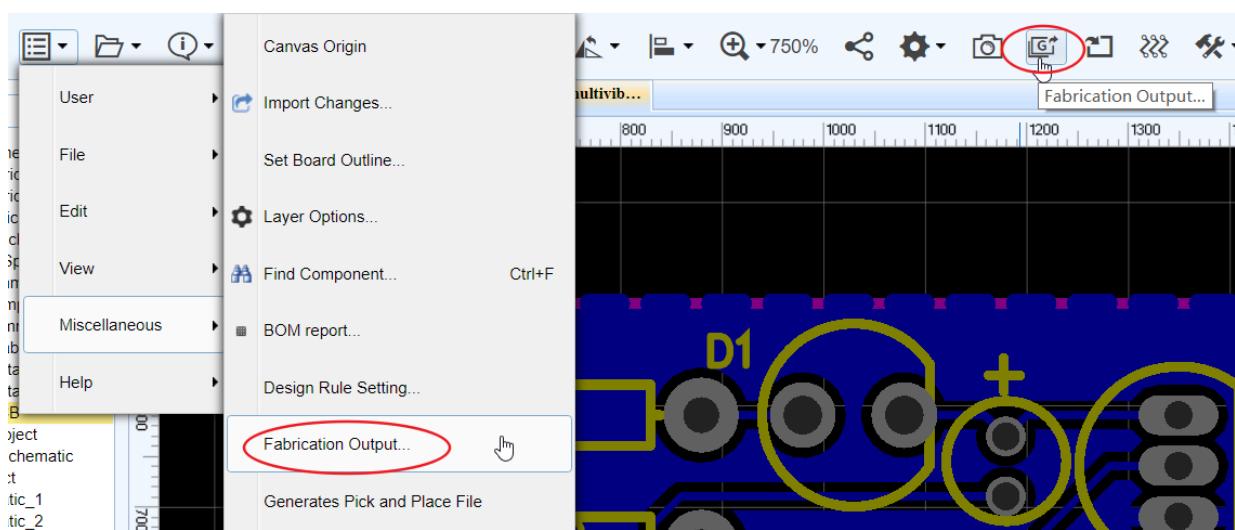


The photo view background default set as black and the right panel was hidden , you can popup up the right attribute panel and modify it.

Getting Fabrication Files

When you finish your PCB, you can output the Fabrication Files(gerber file) via :

Super menu > Miscellaneous > Fabrication Output , or by clicking the Fabrication Output button from the toolbar.



It will open a webpage to you, and you can download the gerber as a zipfile.

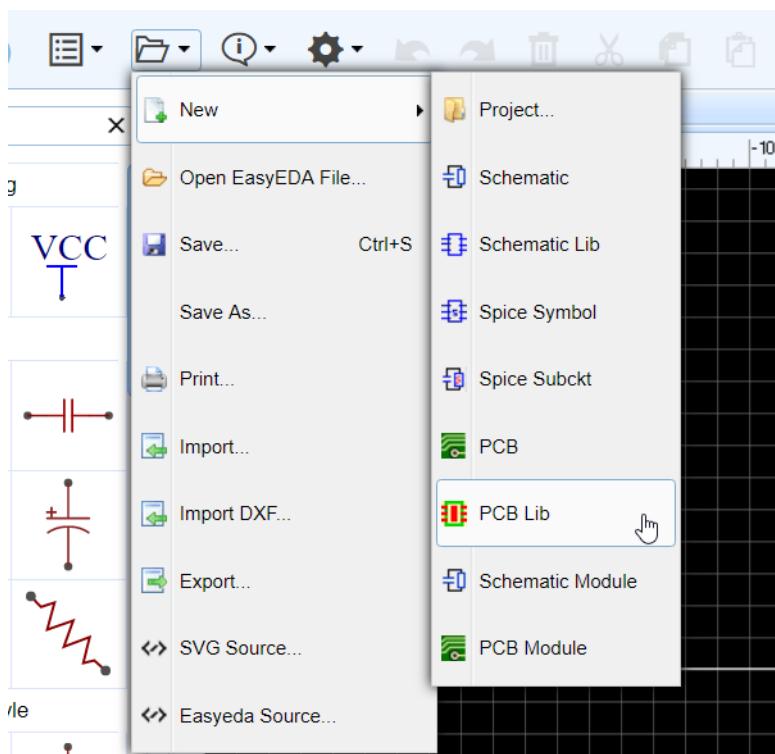
2.5V voltage reference.zip [Download Gerber Files](#)
[Gerber-viewer](#)

Creating The PCB Libs

There will be times when you will need a PCB footprint that is not already in the EasyEDA libraries.

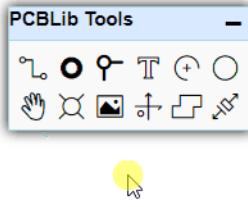
The process of creating your own PCB Libs is very similar to how you make symbols for your own Schematic Libs.

You can start a new PCB lib as shown below:



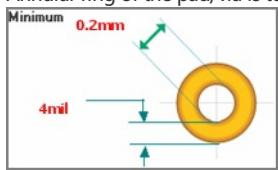
PCBLib Tools

PCBLib Tools almost are the same as PCB tools, just lacking some of the functions.



Others

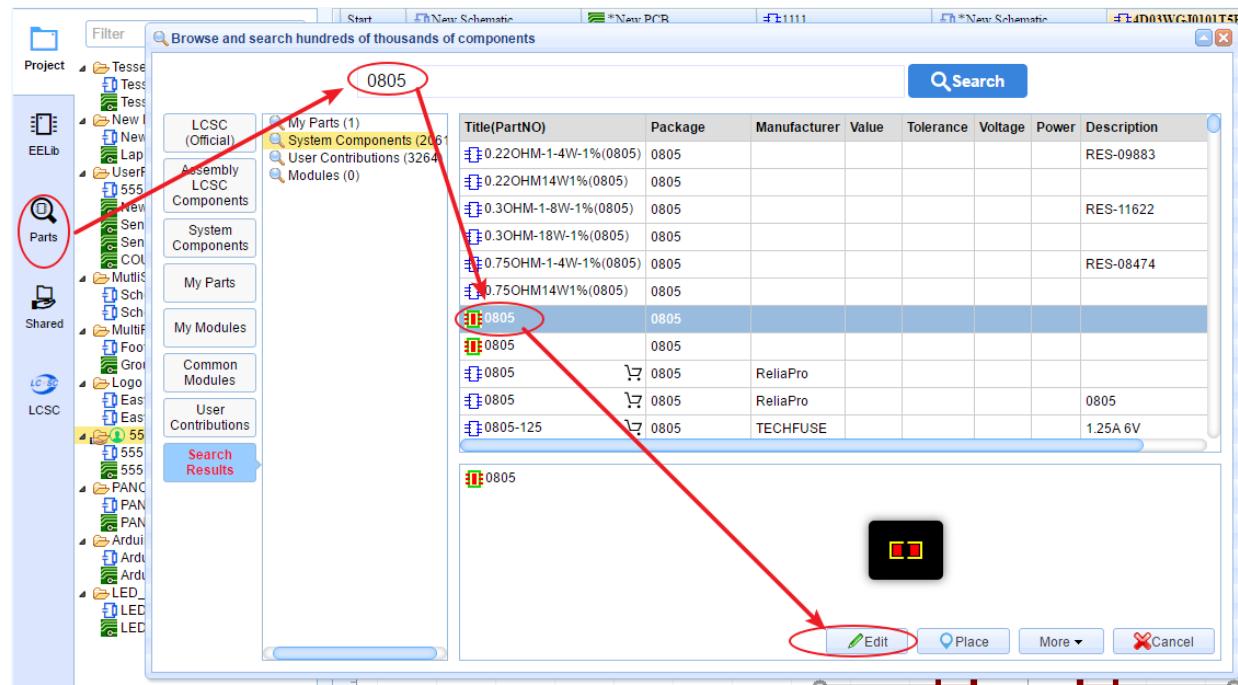
1. It is important to set the right Snap and Grid sizes to ensure that the pads on the finished footprint snap exactly to the grid and so connect the nets. For example, if you are creating a DIP package, set the Grid size to 100mil.
2. Keep all other shapes such as component outlines and any associated pin identification marks or text on the TopSilkLayer. EasyEDA will automatically take care
3. of the actual layer assignment when you place the footprint on the PCB.
4. **CTRL+S** to save your footprint designs and you will find them saved into the **Parts > My Parts > Packages** section of the left Navigation panel.
5. Annular ring of the pad/via is too small, keep the annular ring $\geq 4\text{mil}$. In this case, you can add a **Hole**



Edit PCBLibs

When you feel the PCB Libs(footprint) can not be satisfy for you, you can edit it.

Via "Parts" > "Search Part/My Parts/LCSC Parts/System Components/User Contributions" > Select Package > Edit



when you finish and save , it will be saved to your personal libraries "My Parts" and become your personal libraries.

Import

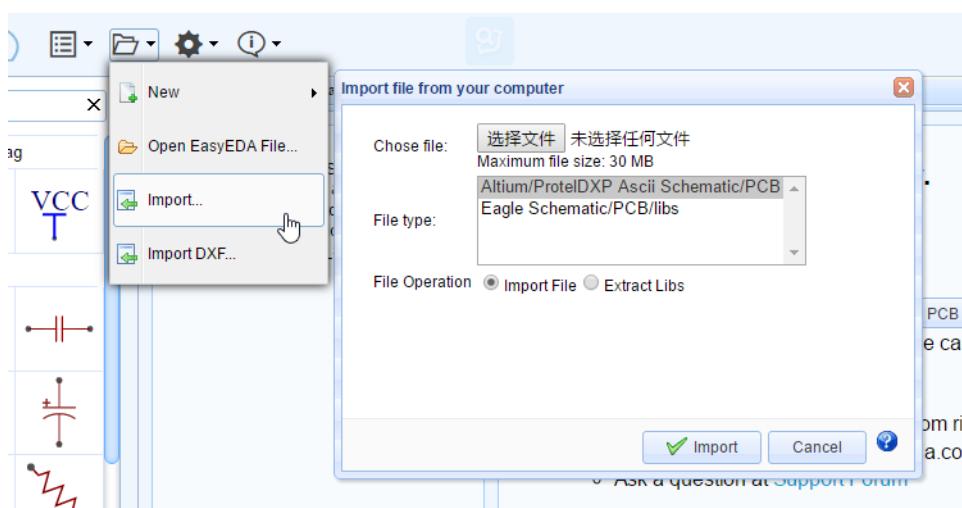
EasyEDA provides importing from:

- Altium/ProtelDXP Ascii Schematic/PCB
- Eagle Schematic/PCB/libs

You can find the import menu from the Document menu:

Document > Import...

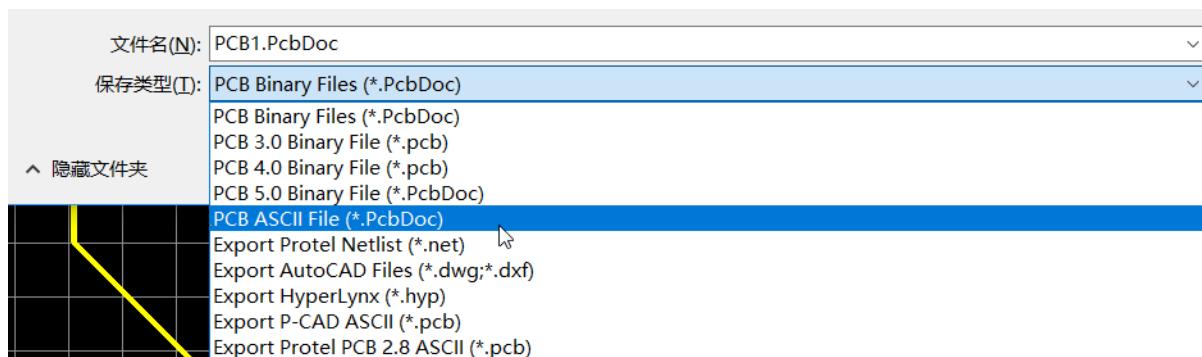
which opens the Import file from your computer dialog:



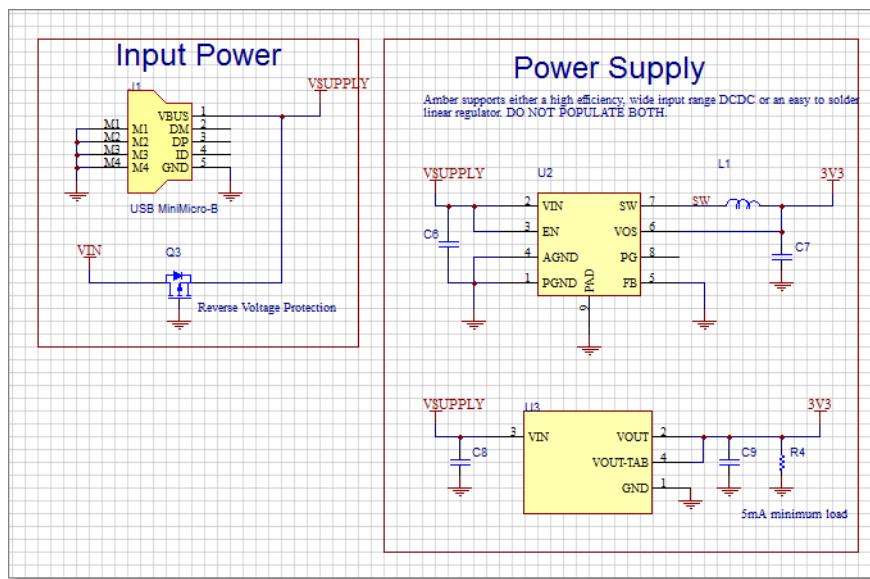
Please note that in File Operation: the Extract Libs option is only supported when importing Altium Designer and Eagle libraries.

Import Altium Designer

You can import Altium Designer's Schematic and PCB files into EasyEDA but only from **ASCII** files, so you need to save the designs as Ascii files like this.



EasyEDA offers an excellent experience in importing Altium Designer's Schematic and PCB: as you can see from the image below of a schematic imported from Altium Designer:



Altium Designer's Schematic and PCB libraries are not available as **ASCII** files, so how can you import them?

In the Import file from your computer dialog to the right of File Operation; tick the Extract Libs option and EasyEDA will extract all of the libs from the Schematic files or PCB Files. So, if you want to import Altium Designer's Libs, you can add them to your Altium Designer Schematic or PCB and then extract them again into your EasyEDA library.

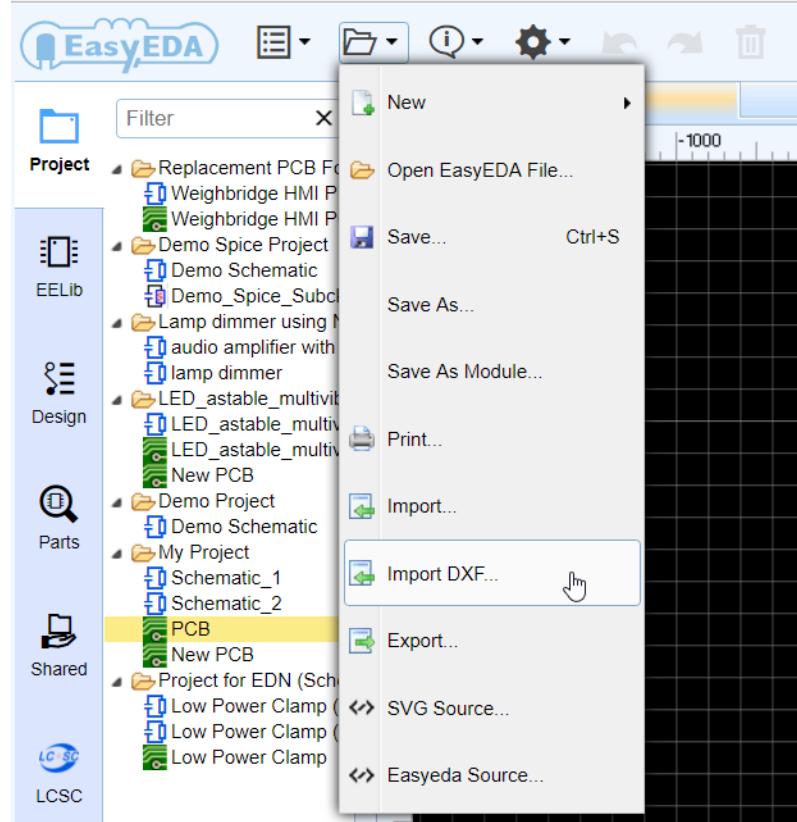
Import Eagle

Eagle Schematic/PCB/libs can be imported, but EasyEDA can only support version 6 and later (6+) because that was when Version 6 Eagle adopted an **ASCII XML** data structure as their native file format.

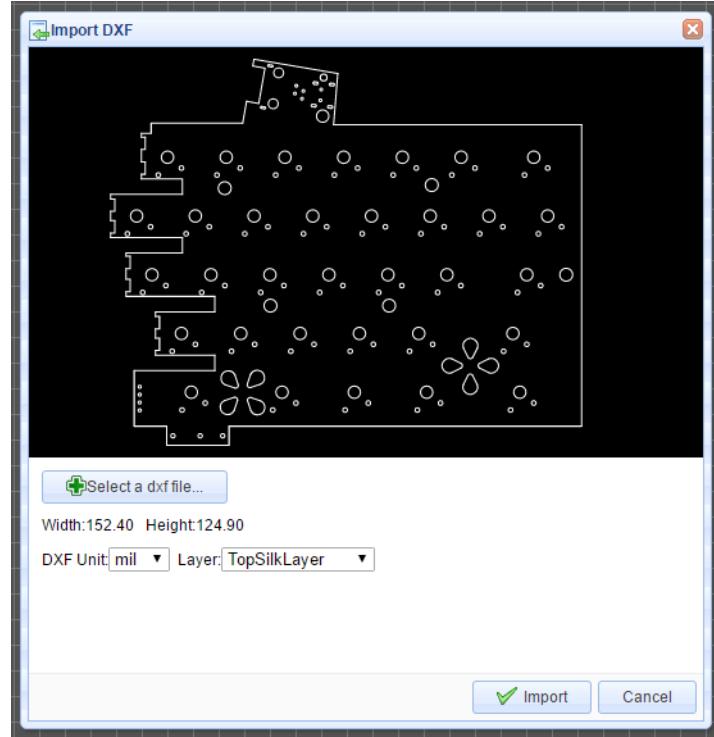
Import DXF File

How to create irregular board outlines or complex board outline in EasyEDA? This is sometimes needed when you are designing a PCB for an enclosure that may have a curved profile, or other unavoidable mechanical features for which one must design.

Find the import DXF menu under the file menu.

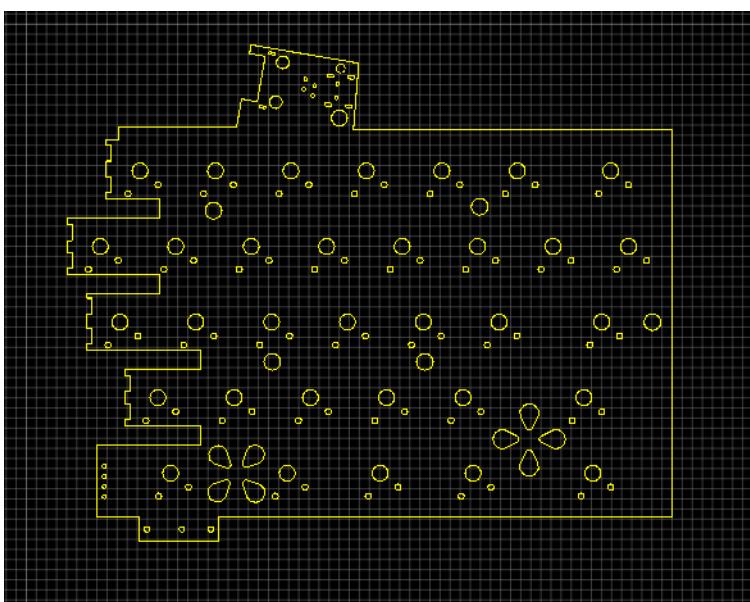


After selecting the *.DXF file, you will find a dialog like in the image below



EasyEDA provides two options, unit(mm or inch), and selection of the layer to which the shapes will be applied.

After clicking the import button, you will find them on your PCB canvas.



You can try this to import this example by yourself. [DXF example](#)

Please note:

1. The file must have a *.dxf filename extension
2. The circles will be converted to holes if you choose the layer as board outline.
3. There are some items which are not supported.

Export

For documentation and other purposes, you can export your Schematic and PCB designs for many items.

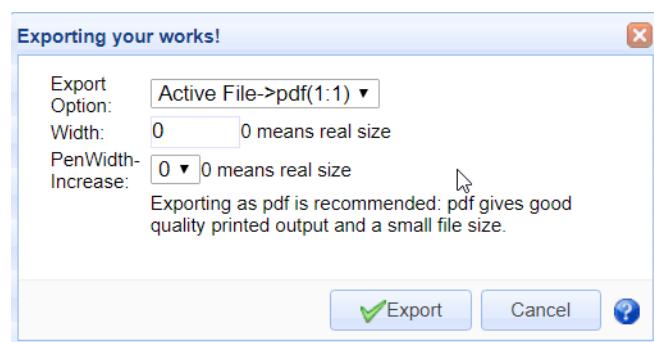
Exporting Schematics

Exporting Schematics In Documentation Formats

Using:

Document > Export...

will open this dialog:

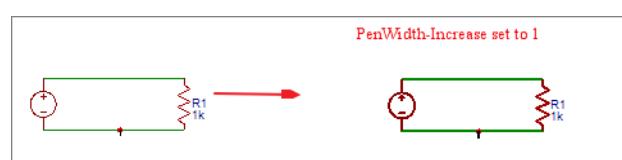


From here you can choose to export your design to SVG, image (.png) and PDF file format.

For all file formats:

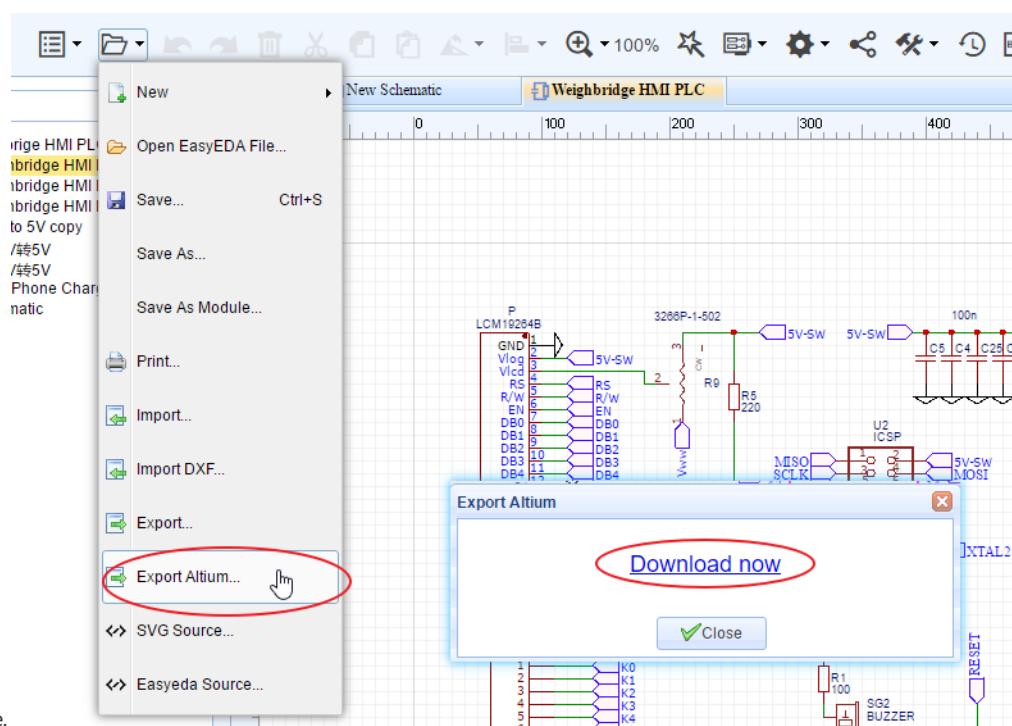
Width: This is images' width , 0 is a 1:1 export of your image, higher numbers scale your image , if you set number as 1024 , the width will be 1024 pixels of the export PNG .

PenWidth-Increase: 0 represents a default line width of 1 pixel; if you set this to 1, the line will be 2 pixels. This is illustrated in the image below.



Exporting Schematics In Altium Designer Format

EasyEDA support exporting the schematics in Altium Designer format. Via "**Documents > Export Altium...**", and click the "**Download now**"

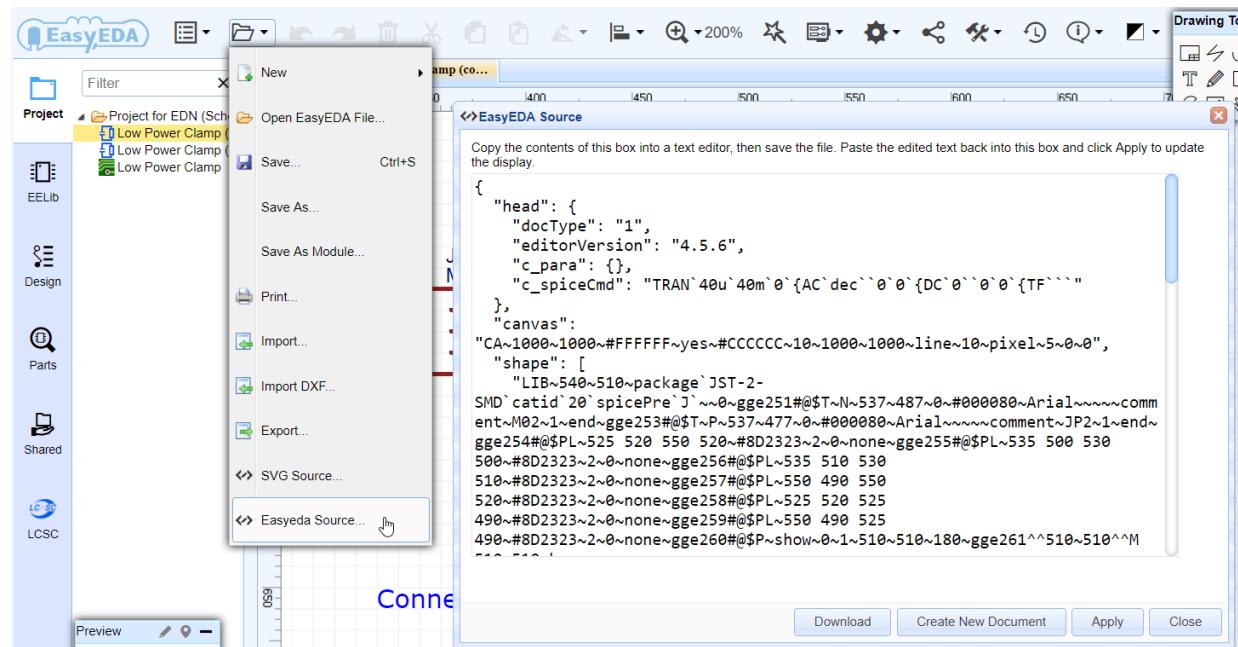


you will get a .schdoc file.

Download Schematics

You can download the schematic when it is opening, via:

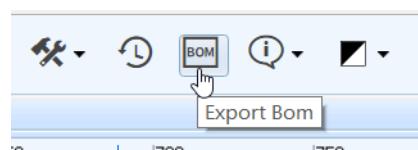
Document > EasyEDA Source..., click the download button, you will get a .json file.



Or **Project > Right Click > Download Project**, you will download a zip file with EasyEDA Source files for Schematics and PCBs.

Exporting BOM

You can **export** the Bill of Materials (BOM) for the active schematic (Document) and PCB or for the active project (i.e. the BOM for all the sheets in the project) as shown below , click the top toolbar BOM icon:



After clicking the BOM export option, the dialog below will open.

In this dialog , you can assign LCSC part's order code for your components.

Export BOM								
ID	Value	LCSC Part #	Supplier	Price(USD)	Quantity	Package	Components	Manufacturer Part Manufacturer
13	DS1034-09MUNSI44	C75752	LCSC	\$0.2976	1	DSUB9-2	J1	DS1034-09MUNSI44 CONNFLY
14	RJ11	C45827	LCSC		1	6P4C	RJ1	RJ11 LCSC
15	RJ45	C36373	LCSC		1	RJ45-3.68	RJ2	RJ45 LCSC
16	Audio-PJ001	C3792	LCSC	\$0.0304	2	AUDIO-PJ001	J2,J4	Audio-PJ001 LCSC
17	USB-A-2	C2345	LCSC	\$0.0315	1	USB-A-2	USB1	USB-A-2 LCSC
18	SWITCH-6x6x5_TH	C69330	LCSC	\$0.0161	1	SWITCH-6x6x5_TH	SW1	SWITCH-6x6x5_TH LCSC
19	VDG-02HG-R	C3661	LCSC	\$0.1691	1	VDG-02HG-R	DIP1	VDG-02HG-R LCSC
20	SRD-03VDC-SL-C	C24585	LCSC	\$0.3281	1	RELAY-SL-SRD	RELAY1	SRD-03VDC-SL-C LCSC
21	1N4148	C14516	LCSC	\$0.0063	1	DO-35	D1	1N4148 ST
22	204-10UYC/S530-A3	C73643	Assign	\$0.0433	1	LED-3MM/2.54	LED1	204-10UYC/S530-A3 EVERLIGHT
23	MBR0520LT1G	C23848	LCSC	\$0.0399	1	SOD-123	D2	MBR0520LT1G ON
24	PESD5V0S1BA	C19224	LCSC	\$0.0465	1	SOD-323	D3	PESD5V0S1BA NXP
25	2W10	C3064	LCSC	\$0.0731	1	BRIDGE-WOB	D4	2W10 LCSC
26	2N3904	C2081	LCSC	\$0.02	1	TO-92(TO-92-3)	Q1	2N3904 CJ

Export BOM at LCSC

Cancel



After clicking on the assign icon , the components and packages search dialog will pop up, and you can choose which component you want to assign.

Browse and search hundreds of thousands of components

Search components and modules

LCSC (Official)

Resistor

- General Resistor(SMD)
- General Resistor(TH)
- Precision Potentiometer
- Photo Resistor(TH)
- Array Resistor(TH)
- Array Resistor(SMD)
- Varistor

Capacitor

- Ceramic Capacitor(SMD)
- General Capacitor(SMD)
- Electrolytic Capacitor
- Tantalum Capacitor
- Tantalum Capacitor
- Monolithic Capacitor
- Array Capacitor(SMD)
- CBB Capacitor
- High Voltage Capacitor

Inductor

- General Inductor(TH)
- General Inductor(SMD)
- Radial Inductor(TH)
- RJ45 Transformer
- Ferrite Bead(SMD)
- Power Inductor(SMD)
- HF inductor

Diode

- General Diode
- Rectifier Bridge

Search results:

Title(PartNO)	Package	Manufacturer	Value	Tolerance	Power	Description
MFR03SF1003A10	AXIAL-1.0	UniOhm	100KΩ	±1%	3W	100KΩ (1003) ±1%
MFR03SF1000A10	AXIAL-1.0	UniOhm	100Ω	±1%	3W	100Ω (1000) ±1%
MFR03SF1000A10	AXIAL-1.0	UniOhm	100Ω	±1%	3W	100Ω (1000) ±1% 25ppm
MFR03SF100JA10	AXIAL-1.0	UniOhm	10Ω	±1%	3W	10Ω (10R0) ±1%
MFR03SF1001A10	AXIAL-1.0	UniOhm	1KΩ	±1%	3W	1KΩ (1001) ±1%
MFR03SF2003A10	AXIAL-1.0	UniOhm	200KΩ	±1%	3W	200KΩ (2003) ±1%
MFR03SJ0200A10	AXIAL-1.0	UniOhm	20Ω	±5%	3W	20Ω(200)±5%
MFR03SF2203A10	AXIAL-1.0	UniOhm	220KΩ	±1%	3W	220KΩ (2203) ±1%
MFR03SF2320A10	AXIAL-1.0	UniOhm	232Ω	±1%	3W	232Ω (2320) ±1% 25PPM
MFR03SF200KA10	AXIAL-1.0	UniOhm	2Ω	±1%	3W	2Ω(2R0)±1%
MFR03SF3000A10	AXIAL-1.0	UniOhm	300Ω	±1%	3W	300Ω(3000)±1%
MFR03SE3301A10	AXIAL-1.0	UniOhm	33Ω	±1%	3W	33Ω /33R0) ±1%

\$0.0318

BUY

0 In Stock Out of Stock

Minimum: 10

Distributor: LCSC

Assign

Cancel

When you click "Export BOM at LCSC", we will help you to list all the components of your BOM, If you want to buy the components form LCSC, and you just need to put them to the cart and check out.

Export BOM								
ID	Value	LCSC Part #	Supplier	Price(USD)	Quantity	Package	Components	Manufacturer Part Manufacturer
13	DS1034-09MUNSI44	C75752	LCSC	\$0.2976	1	DSUB9-2	J1	DS1034-09MUNSI44 CONNFLY
14	RJ11	C45827	LCSC		1	6P4C	RJ1	RJ11 LCSC
15	RJ45	C36373	LCSC		1	RJ45-3.68	RJ2	RJ45 LCSC
16	Audio-PJ001	C3792	LCSC	\$0.0304	2	AUDIO-PJ001	J2,J4	Audio-PJ001 LCSC
17	USB-A-2	C2345	LCSC	\$0.0315	1	USB-A-2	USB1	USB-A-2 LCSC
18	SWITCH-6x6x5_TH	C69330	LCSC	\$0.0161	1	SWITCH-6x6x5_TH	SW1	SWITCH-6x6x5_TH LCSC
19	VDG-02HG-R	C3661	LCSC	\$0.1691	1	VDG-02HG-R	DIP1	VDG-02HG-R LCSC
20	SRD-03VDC-SL-C	C24585	LCSC	\$0.3281	1	RELAY-SL-SRD	RELAY1	SRD-03VDC-SL-C LCSC
21	1N4148	C14516	LCSC	\$0.0063	1	DO-35	D1	1N4148 ST
22	204-10UYC/S530-A3	C73643	Assign	\$0.0433	1	LED-3MM/2.54	LED1	204-10UYC/S530-A3 EVERLIGHT
23	MBR0520LT1G	C23848	LCSC	\$0.0399	1	SOD-123	D2	MBR0520LT1G ON
24	PESD5V0S1BA	C19224	LCSC	\$0.0465	1	SOD-323	D3	PESD5V0S1BA NXP
25	2W10	C3064	LCSC	\$0.0731	1	BRIDGE-WOB	D4	2W10 LCSC
26	2N3904	C2081	LCSC	\$0.02	1	TO-92(TO-92-3)	Q1	2N3904 CJ

Export BOM at LCSC

Cancel



102	<ul style="list-style-type: none"> value: AT91SAM9260B-QU package: PQFP-208_28x28X05P Manufacturer Part: AT91SAM9260B-QU Supplier: LCSC More		AT91SAM9260B-QU Package: PQFP-208_28x28X05P LCSC Part #: C22665 Mfr.Part #: AT91SAM9260B-QU Mfr: ATMEL	1+ \$ 7.8352 10+ \$ 6.6463 30+ \$ 6.4096 100+ \$ 6.1728	1		136 in stock
103	<ul style="list-style-type: none"> value: ATGM336H-5N-3X package: 9.7X10.1MM Manufacturer Part: ATGM336H-5N-3X Supplier: LCSC More		ATGM336H-5N-3X Package: 9.7X10.1mm LCSC Part #: C90770 Mfr.Part #: ATGM336H-5N-3X Mfr: ZHONGKEWEI	1+ \$ 4.5290 10+ \$ 4.1806 30+ \$ 4.0064 100+ \$ 3.8322	1		90 in stock
104	<ul style="list-style-type: none"> value: 3A/250V package: 3.6X10 Manufacturer Part: 3A/250V Supplier: LCSC More		3A 250V Slow break Package: 3.6x10 LCSC Part #: C30449 Mfr.Part #: Glass tube fuse Slow break 3A/250V Mfr: ReliaPro	10+ \$ 0.0512 100+ \$ 0.0391 300+ \$ 0.0369 1000+ \$ 0.0348	10		3189 in stock
105	<ul style="list-style-type: none"> value: 5S15A 250V package: 5*20MM-15A Manufacturer Part: 5S15A 250V Supplier: LCSC More		Slow break 5S15A 250V Package: 5*20mm 15AWithout lead LCSC Part #: C48473 Mfr.Part #: Slow Break 5S15A 250V Mfr: ReliaPro	5+ \$ 0.0626 50+ \$ 0.0470 150+ \$ 0.0440 500+ \$ 0.0409	5		114 in stock

And Click the "BOM" button to download the BOM file. You can open it in any text editor or spreadsheet.

	A	B	C	D	E	F	G	H	I	J
1	id	value	quantity	package	components	Manufacturer Part	Manufacturer	Supplier	LCSC	price
2	1	150	2	AXIAL-0.3	R1,R4	25121WJ020KT4F	UniOhm	LCSC	C45278	\$0.02
3	2	22k	2	AXIAL-0.3	R2,R3	25121WF300LT4F	UniOhm	LCSC	C16074	\$0.03
4	3	22u	2	CAP-D3.0XF1.5	C1,C2	1812B225K500NT	FH	LCSC	C28503	\$0.28
5	4	204-10UYC/S53I	2	LED-3MM/2.54	LED1,LED2	67-21S/KK3C-H2727QAR3LED EVERLIGHT	LCSC	C73540		\$0.04
6	5	2N3904	2	TO-92(TO-92-3)	Q1,Q2	MURA220T3G	ON	LCSC	C37995	\$0.17
7										

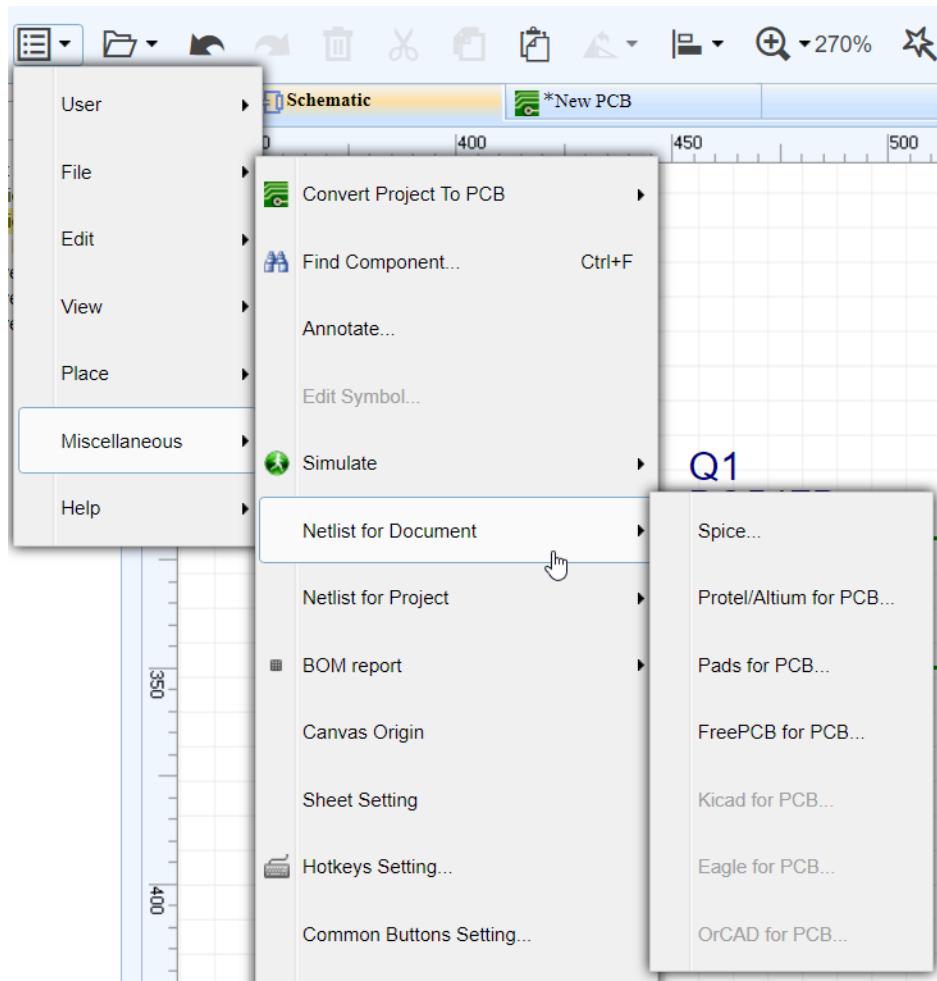
Exporting Netlist

EasyEDA can export the netlist for the active schematic (Document) and/or for the whole active project:

Super menu > Miscellaneous > Netlist for Document or Netlist for Project

EasyEDA can export a netlist in a variety of formats:

- Spice:** this is a Spice3f5 compatible netlist generated by the simulation engine of EasyEDA, [Ngspice](#). It is not normally used as the basis for a PCB layout.
- KiCad:** a PCB netlist in a format that can be imported straight into Pcbnew, the PCB layout tool part of the free, open source cross-platform EDA suite.
- Altium Designer:** a PCB netlist in a format that can be imported straight into Altium Designer and it's predecessor, Protel.
- Pads:** a PCB netlist in a format that can be imported straight into Pads PCB layout tools.
- FreePCB:** a PCB netlist in a format that can be imported straight into FreePCB, a free, open source PCB editor for Windows.



Exporting PCB Designs

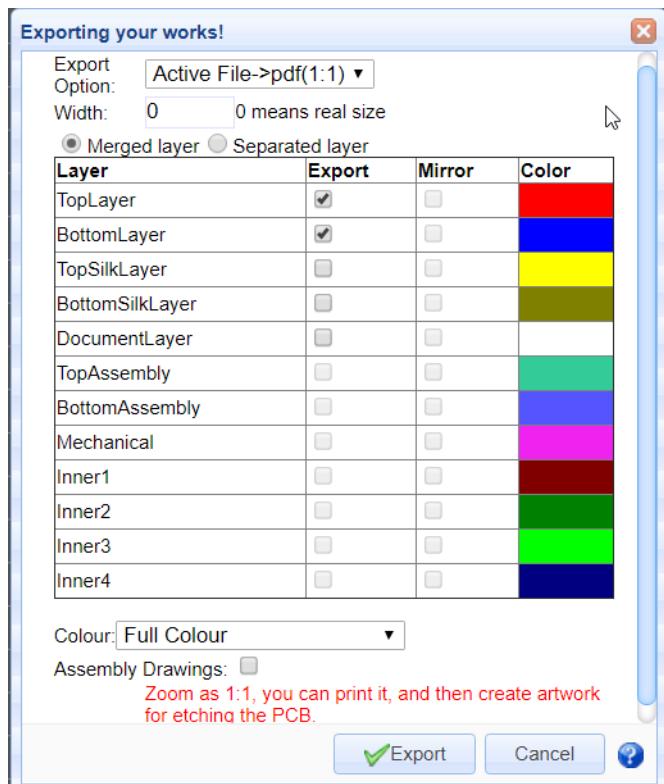
Exporting PCB In Documentation Formats

Exporting a PCB design or footprints from EasyEDA is very similar to exporting a Schematic or a Symbol.

Using:

Document > Export...

you can open this dialog:



You can select to export in PDF, drawing (.PNG) or SVG format.

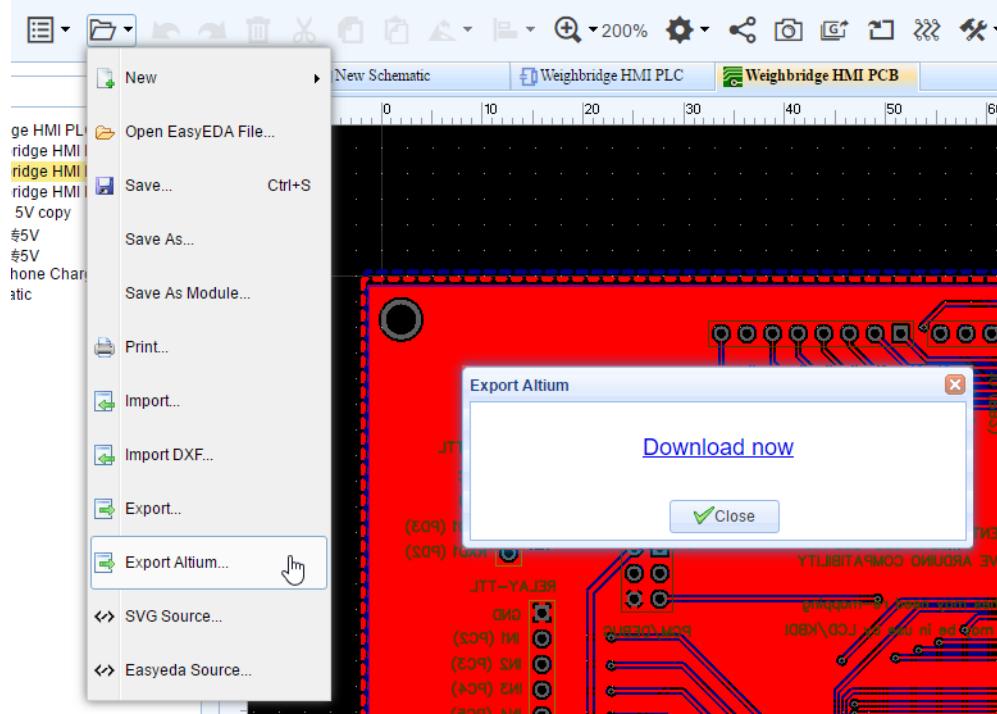
Note: If you want to print the PCB 1:1 with the paper, you need to choose to export PDF(1:1).

You can select to print individual layers or selected layers merged into a single file.

It is also possible to mirror selected layers for example to show bottom layers in easily readable orientation.

Exporting PCB In Altium Designer Format

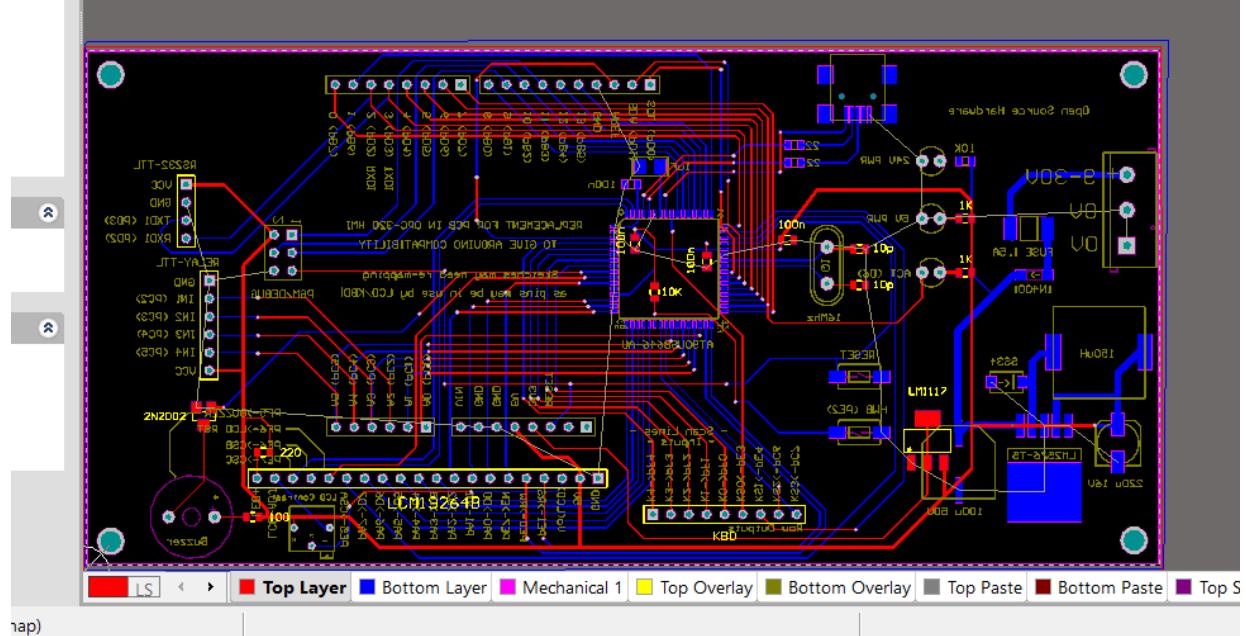
EasyEDA support exporting the PCB in Altium Designer format. Via "Documents > Export Altium...".



When open the exported PCB file at Altium Designer, there will open a dialog of DXP Import Wizard, don't worry, just cancel it to continue.

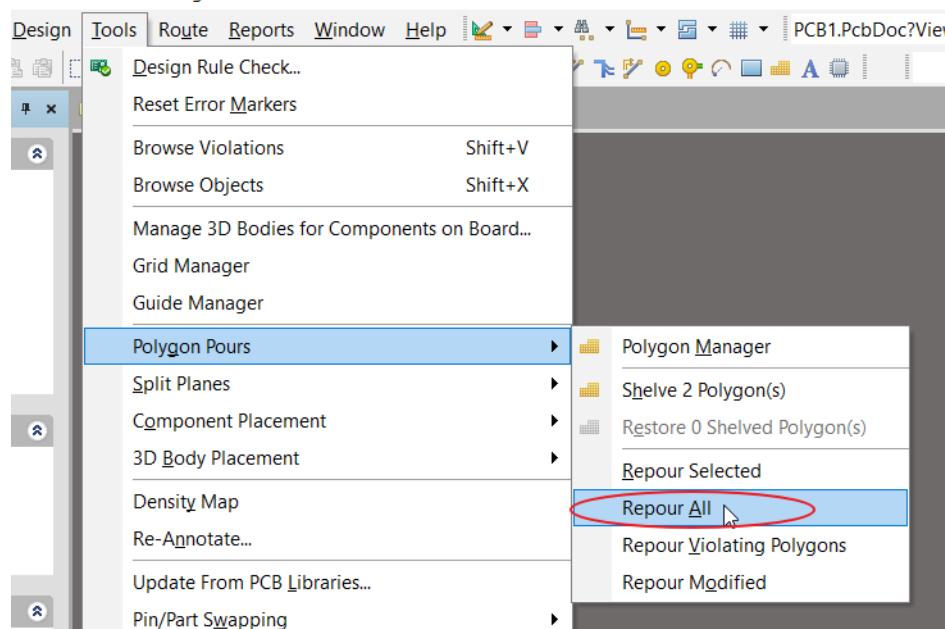


And then, you will see the PCB file, which looks like without copper area as below:

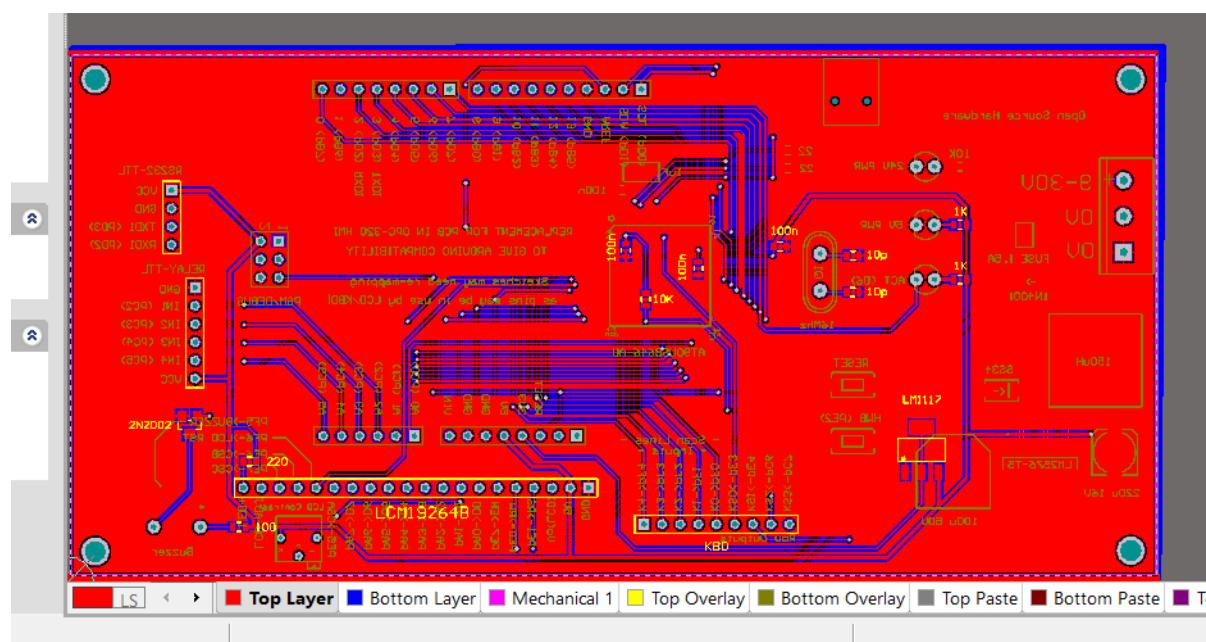


At now, you need to repour all polygons at Altium Designer. Via: **Tools > Polygon Pours > Repour All**:

See Documents. Not signed in.



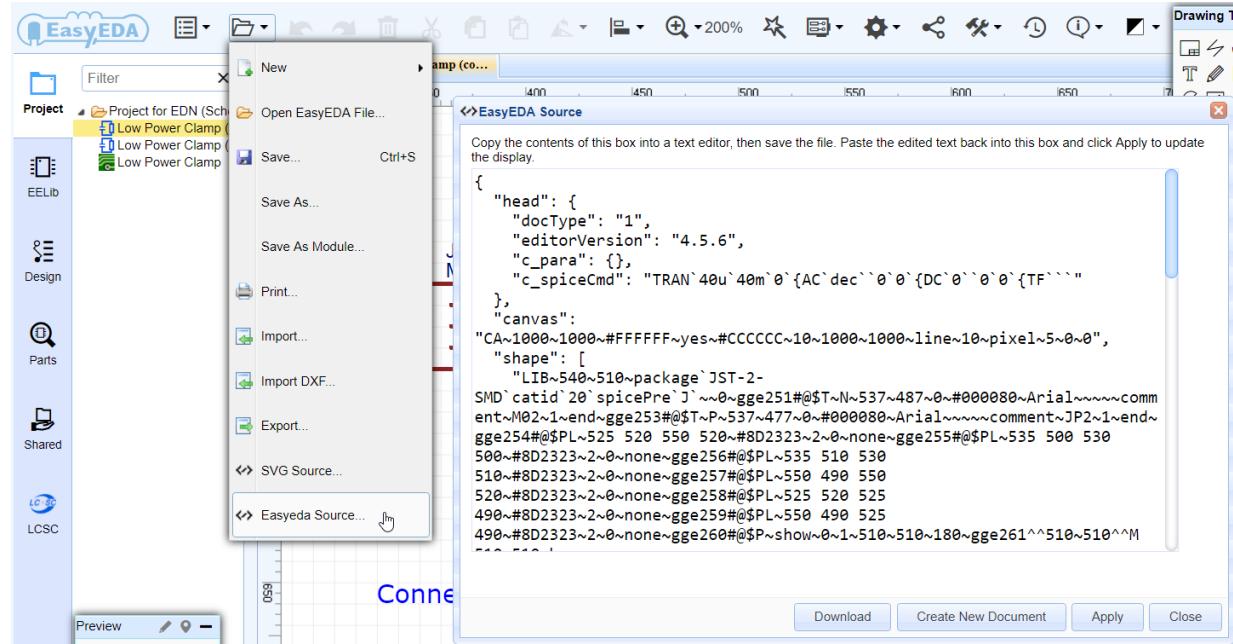
And the last, save it.



Download PCB

You can download the PCB when it is opening, via:

Document > EasyEDA Source..., click the download button, you will get a json file.

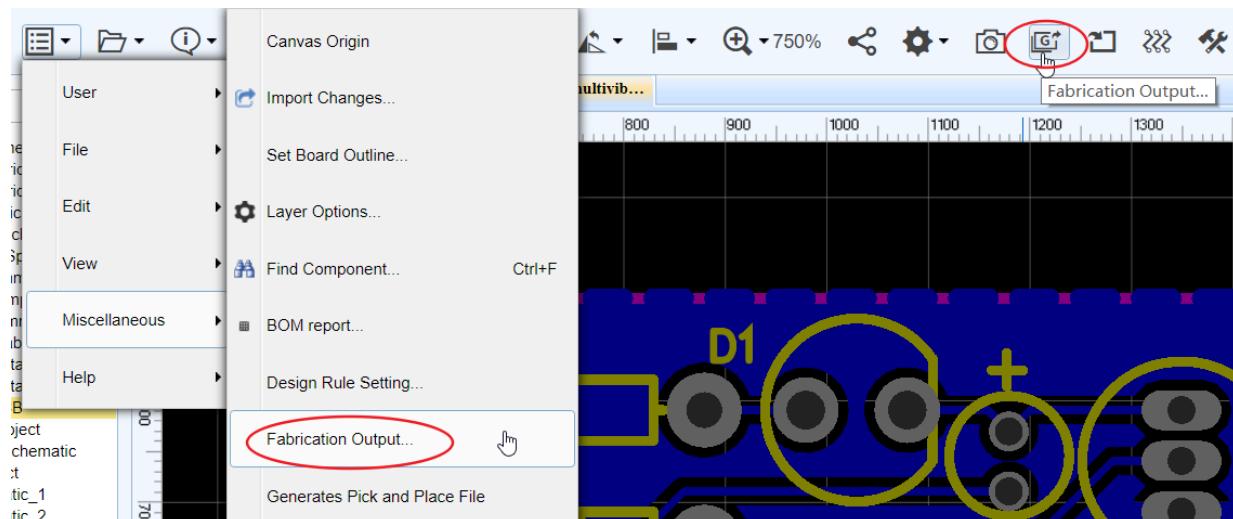


Or **Project > Right Click > Download Project**, you will download a zip file with EasyEDA Source files for Schematics and PCBs.

Exporting Fabrication Files

When you finish your PCB, you can output the Fabrication Files(gerber file) via :

Super menu > Miscellaneous > Fabrication Output , or by clicking the Fabrication Output button from the toolbar.



It will open a webpage to you, and you can download the gerber as a zipfile.

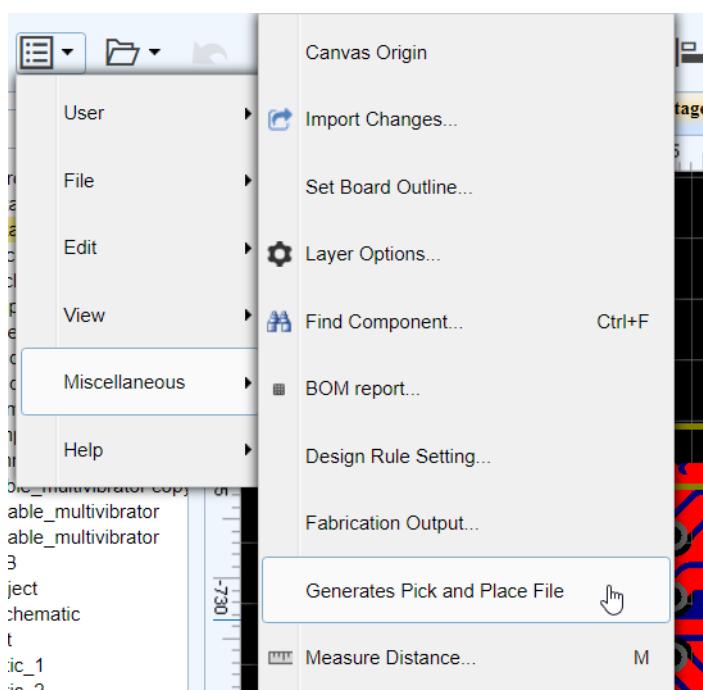


Notice: Before order your PCB, Please read [Essential Check Before Placing a PCB Order](#) section!!

Exporting Generates Pick And Place File

In PCB editor, if you want to generate Pick And Place as a CSV file, you can via:

Super Menu > Miscellaneous > Generates Pick And Place File



When you open the exported CSV file, you can see:

	A	B	C	D	E	F	G	H	I	J
1	Designator	Footprint	Mid X	Mid Y	Ref X	Ref Y	Pad X	Pad Y	TB	Rotation
2	Q1	SOT23	580mil	430mil	580mil	430mil	617mil	473mil	T	180
3	Q2	SOT23	770mil	430mil	770mil	430mil	807mil	473mil	T	180
4	Q3	SOT23	1040mil	120mil	1040mil	120mil	1003mil	77mil	T	0
5	R1	1206	680mil	150mil	680mil	150mil	680mil	95mil	T	90
6	R2	1206	500mil	150mil	500mil	150mil	500mil	95mil	T	90
7	R3	1206	540mil	750mil	540mil	750mil	540mil	695mil	T	90
8	R4	1206	910mil	750mil	910mil	750mil	910mil	695mil	T	90
9	R5	1206	730mil	750mil	730mil	750mil	730mil	695mil	T	90
10	C1	1206	820mil	200mil	820mil	200mil	820mil	255mil	T	270
11	C2	1206	1100mil	750mil	1100mil	750mil	1100mil	805mil	T	270
12	JP2	JST-2-SMD	1076.5mil	450mil	1120mil	450mil	974mil	489mil	T	270
13	JP1	JST-3-SMD	275.5mil	450.5mil	190mil	450mil	378mil	372mil	T	90

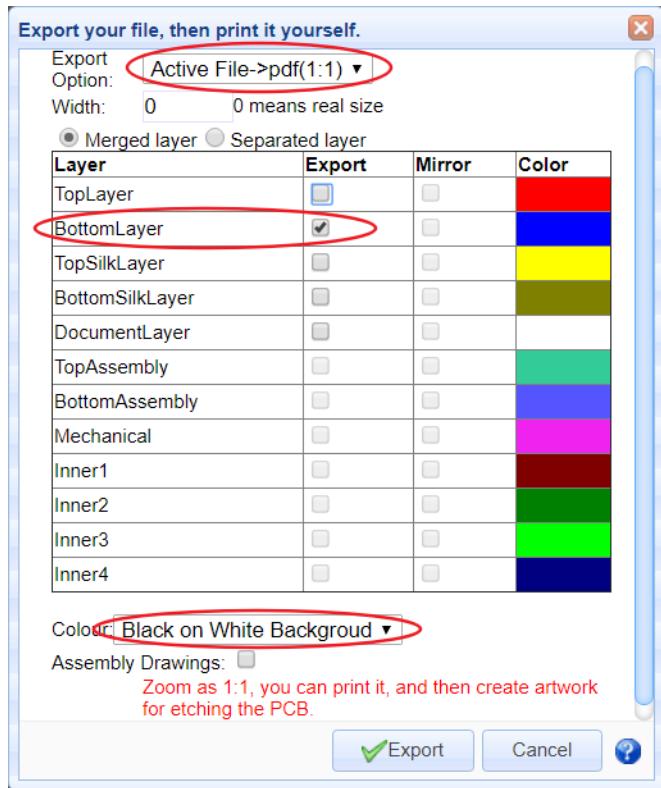
Exporting Print For Etching

If you don't want to order your PCBs from EasyEDA then maybe - for single and double sided PCB designs - you might like to try like using some home made PCB tech:

<http://hackaday.com/2012/12/10/10-ways-to-etch-pcb-at-home/>

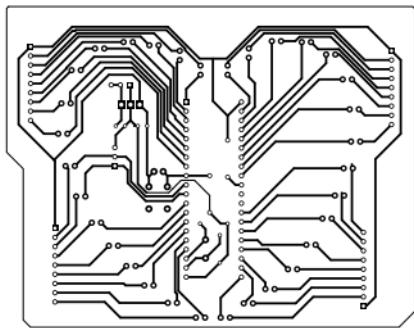
So here's how you can print your PCB layer by layer and then etch it onto a PCB.

Step 1) Export it to PDF, Using: **Document > Export...**, or **Document > Print...**

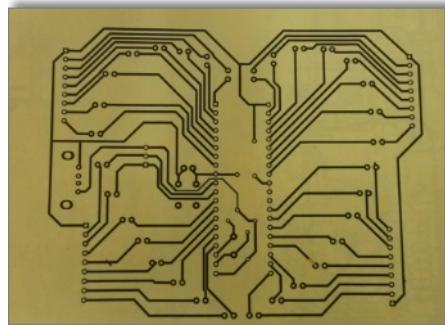


Note: Make sure the Colour is Black on White Background.

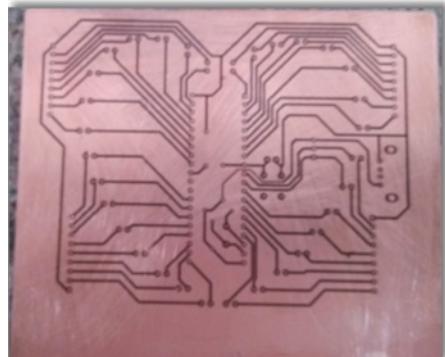
Step 2) Open the pdf file in a viewer



Step 3) Print it to paper

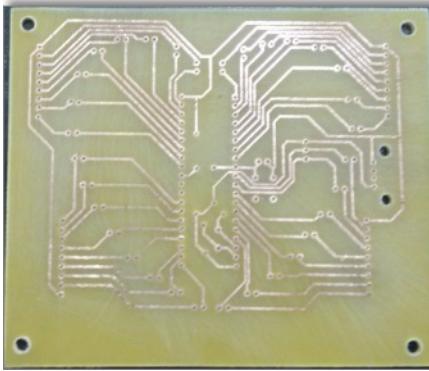


Step 4) Copy it to the copper



Step 5) Etch it.

Step 6) Drill it ...



Step 7) Get your soldering iron out!



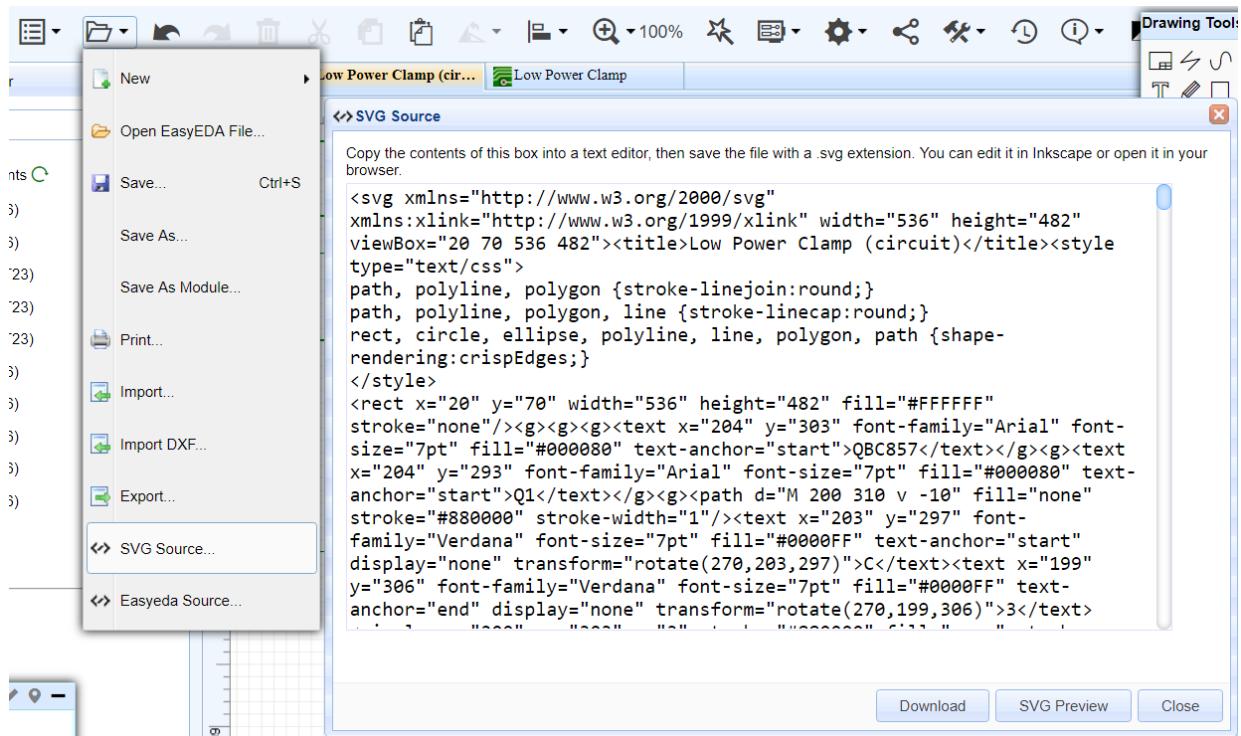
Exporting SVG Source

You can create an SVG sourcefile via:

Document > SVG source...

then copy the contents of this box into a text editor and save the file with a .svg extension. You can edit it in [Inkscape](#) or open it in your browser.

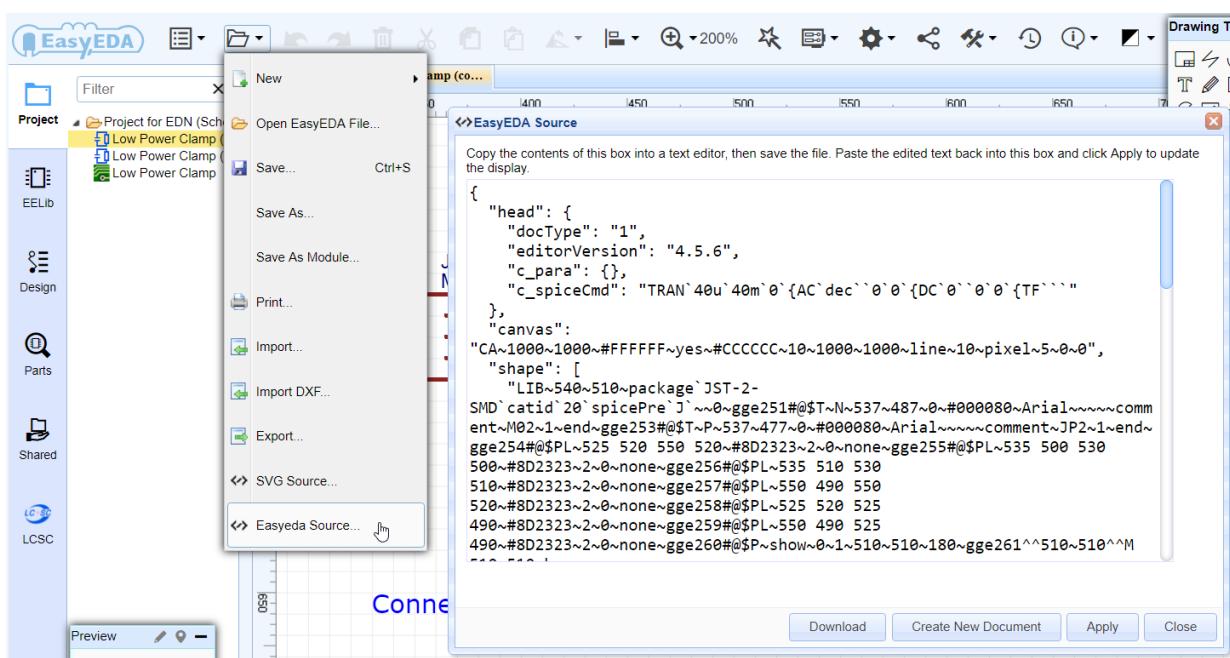
This solution doesn't need an internet connect so if you open EasyEDA offline, you can use it.



Exporting EasyEDA Source

You can create an EasyEDA source file via:

Document > EasyEDA Source...



Or Project > Right Click > Download Project, you will download a zip file with EasyEDA Source files for Schematics and PCBs.

EasyEDA Source is a **JSON** file which can be read by many other programs. Please see:

<http://en.wikipedia.org/wiki/JSON>

for more information.

The open EasyEDA Source file allows you to work on files at a text level which enables some powerful ways to manipulate schematic and spice files and symbols as well as PCB files and footprints.

Click on the **Download** button or copy the contents of this EasyEDA source into any text editor, then save the file. You can paste the text back into this box and click **Apply** to update the display. If you have made no changes to the text then the canvas will show your file exactly as if it was saved and reopened from the EasyEDA server.

This is a good way to share/backup your works. Your file doesn't need to be saved to EasyEDA's server. It can be highly compressed in any readily available format such as such as zip or 7z. It can be emailed to anyone who can then open it in EasyEDA without worrying if they have the same libraries as you.

EasyEDA team will provide more details of the EasyEDA Source soon to show how you can edit and even create drawings, schematics, symbols, footprints and PCB layouts in EasyEDA Source. It is also possible to copy and edit symbols straight out of a Schematic and save them as new Schematic Lib or Spice Symbols and even to create a new Spice Subckt from a Schematic.

Sharing

Sharing your work with others is a big feature of web based EDA tools and EasyEDA is no exception in offering you some nice features.

Share to Public

Did you create a really cool project with EasyEDA? Show it off and be super helpful to other EasyEDA users, you just need to set your projects to public, so others can explore your circuits.

All projects in EasyEDA are set to private by default, your private project can not be shared with anyone.
i.e. to make it public, you should create a new project or right click and edit your existing project to be a Public project:

Create New Project :

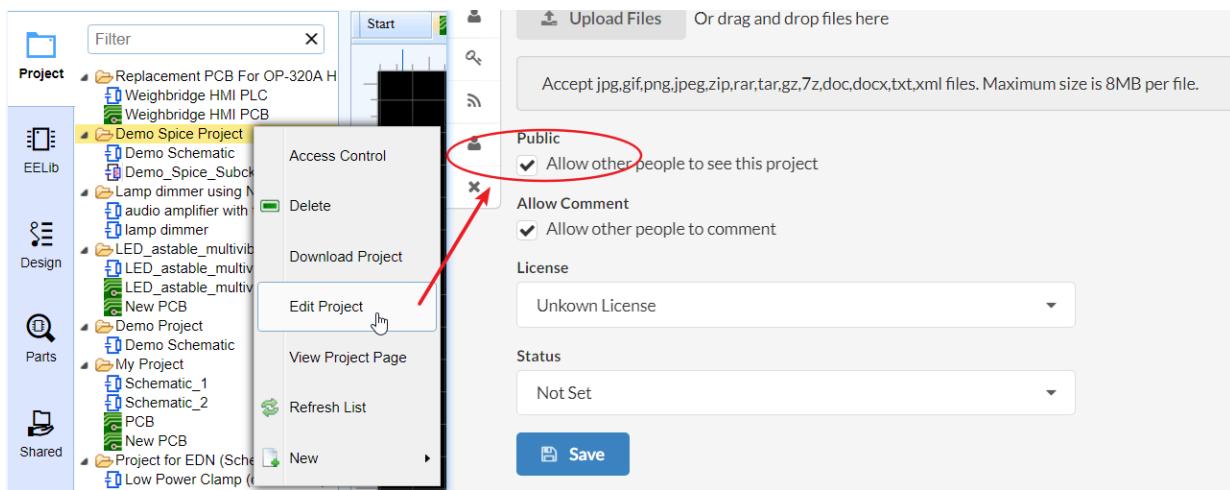
New Project

Title:	New Project
Visibility:	<input checked="" type="radio"/> Public (Only you can modify this project. Anyone can see it.) <input type="radio"/> Private (Only you can see and modify this project)
Description:	This a Public project

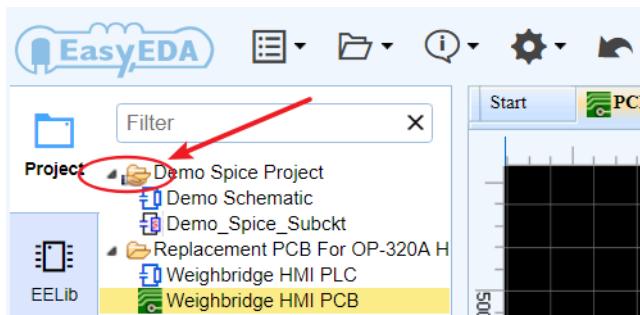
OK Cancel

Edit Existing Project:

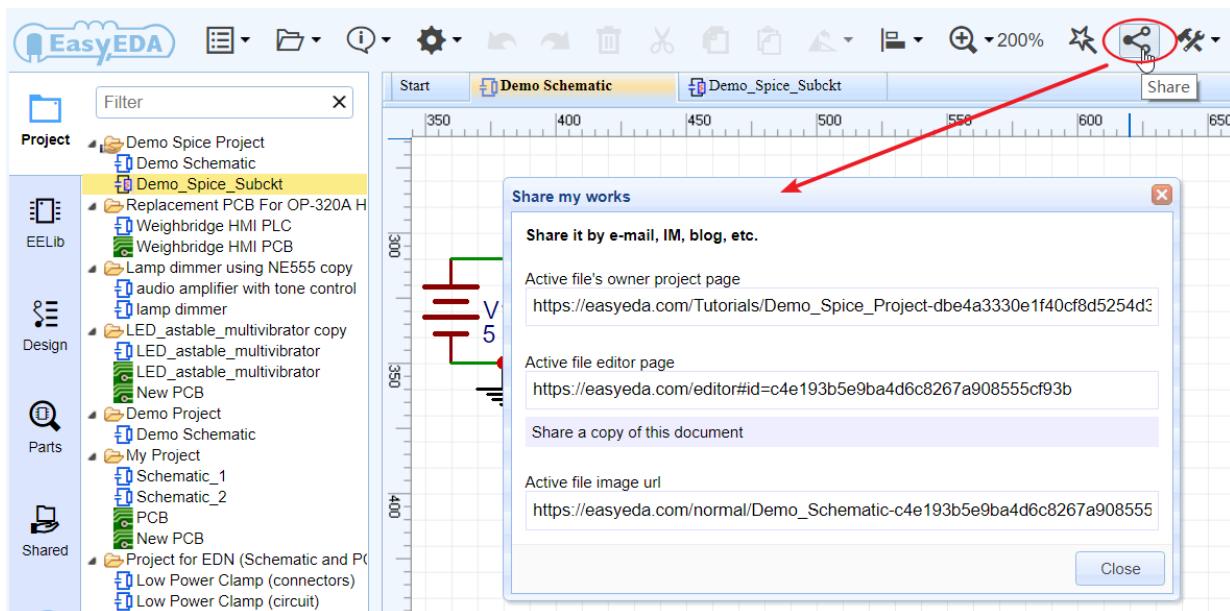
When you click **Edit Project**, it will open a website to allow you to set your project to be public.



After setting the project as public, you will see that the Project folder Icon is now shown as a hand holding the folder.



If you then open one of the documents in this share folder, you can then click the Share icon on the toolbar to open the Share my works dialog.



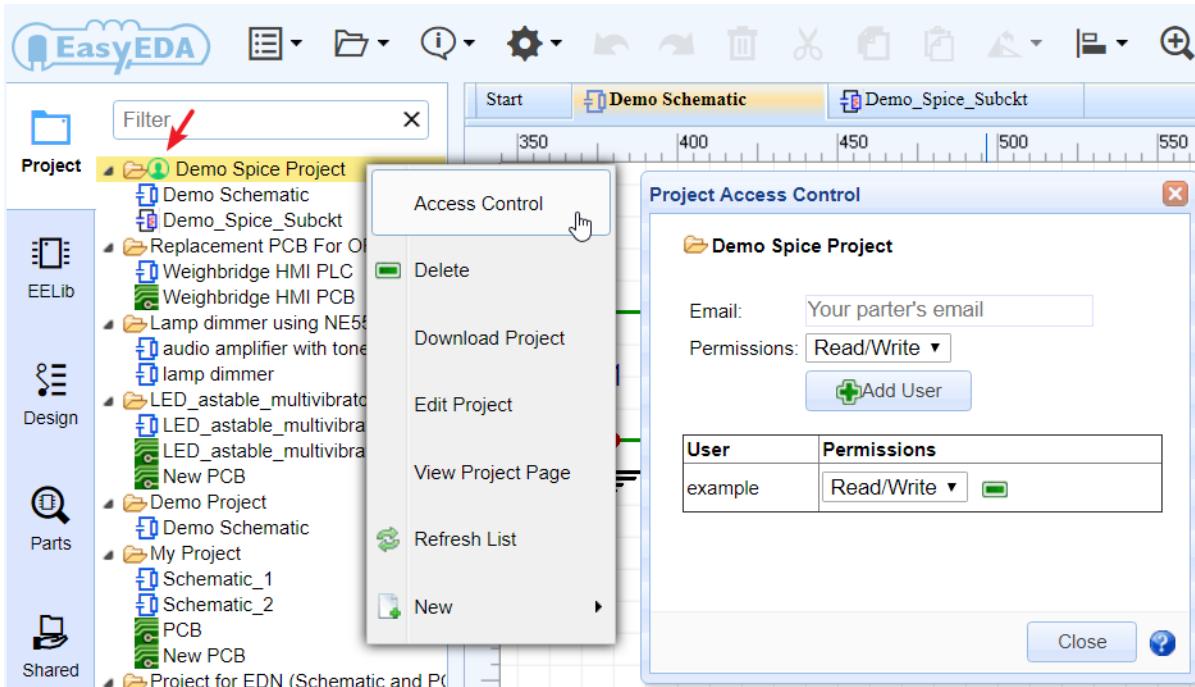
Access Control

How about sharing with selected people?

Can you share a private project with your partner? Can your partner modify your designs?

Yes, you can use **Access control** to do this.

Right click the project and you will see the Access Control on the context menu; clicking on it will open the Access Control dialog. After adding a user, a user icon will show up beside the project folder icon as below.

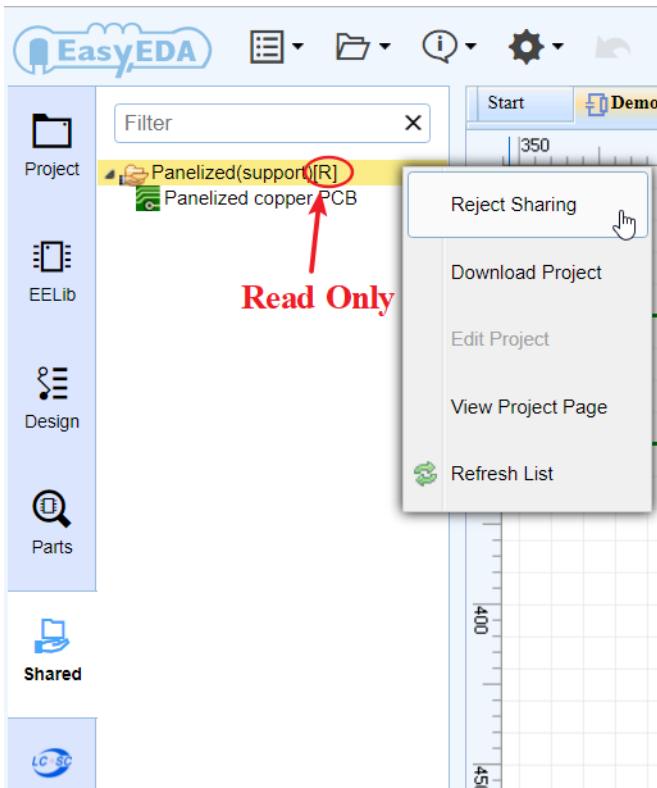


So if you want to share a project with someone,

1. You just need to know their E-mail address which they have used to create an account with EasyEDA
2. You can share your project as **read only** or **read/write**.

After setting up **Access Control** and Permissions, your partner will find your project in the **Shared** section from the left **Navigation Panel** when they login.

If your partner doesn't wish to accept the shared project, they can reject it by right clicking on the project in the Shared with Me section and then clicking on Reject Sharing:



And you also can check projects that your partner has shared with you in the account dashboard:

ID	Project	Create Time
1	Panelized	1 year ago

Essential Check

Introduction

By following - and constantly checking against - a set of procedures, it is possible to avoid just about all of the common mistakes and omissions that can significantly delay or even stop a schematic being successfully converted to a PCB and then that PCB being successfully updated from the schematic as a design progresses.

It can also significantly reduce the likelihood of a PCB being made that subsequently is found to not work correctly due to mistakes made during the creation of the original schematic (Schematic Capture).

After spending hours on Schematic Capture, it is very frustrating to be presented with error messages about prefix conflicts, missing or invalid packages when first attempting to pass a schematic through to the PCB Editor by clicking on the **Convert Project to PCB...** button or, after making changes to a schematic, similar error messages or having components that disappear from the PCB when attempting to update an existing PCB using the **Update PCB...** button in the Schematic Editor or the **Import Changes...** button in the PCB Editor.

These issues can be avoided by running through a series of checks for the first time each new Part (i.e. the first instance) is placed into the schematic.

There are several other issues that arise from mistakes in and omissions from the schematic that people encounter only after they are part way through a PCB design or - worse still - only when they receive their PCBs in the post.

Almost all of these other issues can be avoided by running through a further series of checks (i) during Schematic Capture, (ii) once Schematic Capture is complete but before first attempting to convert the schematic into a PCB and (iii) when updating the PCB as work progresses.

This document pulls together all the essential procedures to follow and things to check in the schematic before clicking on the **Convert Project to PCB...**, **Update PCB...** or **Import Changes...** buttons.

Things to understand before using this document.

Before using the document it is important that the following points are clearly understood:

What constitutes a Part in a schematic and a PCB?

A Part is any element of the circuit that is to be mounted on the PCB plus any element which is ultimately intended to be mounted on or form an integral part of the PCB such as heat sinks, PCB mounting holes, mounting holes for PCB mounted potentiometers and switches (for example where a PCB is used as a front panel or as a self-contained test jig), test points, wire links or jumpers, fuse holders and even image based elements such as high voltage warnings and logos.

Note that fuses that are fitted into PCB mounted fuse holders are best dealt with in a schematic by showing the fuse using a fuse symbol in the schematic but assigning to that fuse symbol the BoM information - including the package - that is for the required fuse holder. The required fuse ratings, type and supplier information can then be included in the BoM using Add new parameter function.

Other socketed devices can be treated in the same way.

What is the relationship between Parts, Schematic Symbols and PCB Packages?

Any Part must have a Schematic Symbol to represent it in the schematic (a.k.a. Schematic Lib) and that Schematic Symbol must have a PCB Package (a.k.a. PCB Lib) assigned to it either when the symbol is created or after placing the first instance of it into the schematic.

The associated PCB Package must exist in the library.

- Ensuring that every Part has a Schematic Symbol with a PCB Package associated with it **and** that that PCB Package actually exists in the

Parts (SHIFT+F) library will avoid a **Missing package** error being issued on Conversion or Update to PCB.

It is possible that a component may comprise more than one device in a package, for example logic gates. Some symbols represent both devices in a single symbol but quite often a separate symbol is used to represent each of the devices. This may mean that some of the pin numbers and/or names on the symbols representing each of the two devices may be different although both may have the same power and ground pin numbers and/or names.

High pin count devices such as processors and FPGAs may be split into several symbols representing different sections or ports. It is important to ensure that pin numbers and names are unique across all the symbols.

It is possible that a component may be available in different packages. For example the LM358-N dual operational amplifier is available in several different packages. The pin numbering and/or naming of the symbol may be different depending on which package the component is supplied in.

It is easy in EasyEDA to change the pin numbering and/or naming for a Schematic Symbol (using the *I* Hotkey) or a PCB Package so it may be tempting to think of an LM358-N as the same part in a different package and just put down a symbol, edit the package assigned to it and then hack the pin numbers and names about until they match the PCB package.

However, an LM358-N in a SOIC-8 package has a different pinout, a different part number and has to be physically ordered as a different part from an LM358-N in a DSBGA-8 package.

- They are therefore two *different Parts*.

When thought of like this it should be clear that there should be one Schematic Symbol (or pair if each device has a separate symbol) and a matching PCB Package for an LM358-N in a SOIC-8 package and another Schematic Symbol (or pair if each device has a separate symbol) and a matching PCB Package for an LM358-N in a DSBGA-8 package.

- Ensuring that the pin numbers/names of the Schematic Symbol for a Part are correct, unique and match those of the PCB package associated *with that particular part* will avoid the generation of the **Invalid package** error being issued on Conversion or Update to PCB.

Why do Parts in a PCB disappear when the PCB is updated from the schematic?

It is important to understand that any Part that is supposed to form part of or be mounted on the PCB must have a corresponding Schematic Symbol in the schematic.

If it *does* then as soon as the PCB is created, the PCB Package for that Part, even such a seemingly abstract item as a mounting hole, warning sign or a logo, will be pulled into the PCB layout without having to be added to the PCB later by hand.

If it *does not* then not only will the PCB package for that Part not be pulled into the PCB layout as it is created but when it is added to the PCB later by hand and the PCB is then updated to bring in changes made to the original schematic, *that PCB package will be deleted*.

Such elements can be added to the schematic later and then imported into the PCB but if they do not exist in the schematic at the time the PCB is updated from that schematic then they will always be deleted and will therefore have to be added back to the PCB by hand.

What is the relationship between User Contributions and the other Parts categories (LCSC (Official), Assembly LCSC Components, System Components, My Parts, My Modules and Common Modules)?

Any Schematic Symbol or PCB Package chosen from the **User Contributions** category MUST be added to your local library by doing:

Parts (or SHIFT+F) > Search for and select the part then > More > Add Favorite

- Failure to do this will result in the package not being found and a **Missing package** error being issued on Conversion or Update to PCB.

Packages in the other library sections will be found automatically.

Note however that although Schematic Symbols and PCB packages created by a user within a Team will automatically appear in that user's My Parts library, once that user swaps to another Team, those parts will no longer appear in their My Parts library but will only be available via the **Add Favorite** option from the User Contributions library.

Procedures and Checklist

- Verify that the Schematic has been drawn to show clearly how the components in the circuit are connected together;

The schematic must help the reader understand signal and power flow in the circuit with inputs on the left, outputs on the right, positive supplies at the top, negative supplies at the bottom and netlabels used to clarify connections and reduce congestion.

Components may be grouped by function and boxes may be drawn around them.

Decoupling components may be drawn adjacent to the devices they are associated with or symbols with dedicated sub-parts for power pins can be used to reduce congestion;

- Use the Design Manager (left-click the **Design** button in the left hand panel) to check that all components are present in the schematic and that all nets have been assigned reasonable mnemonic names and have at least two connection.

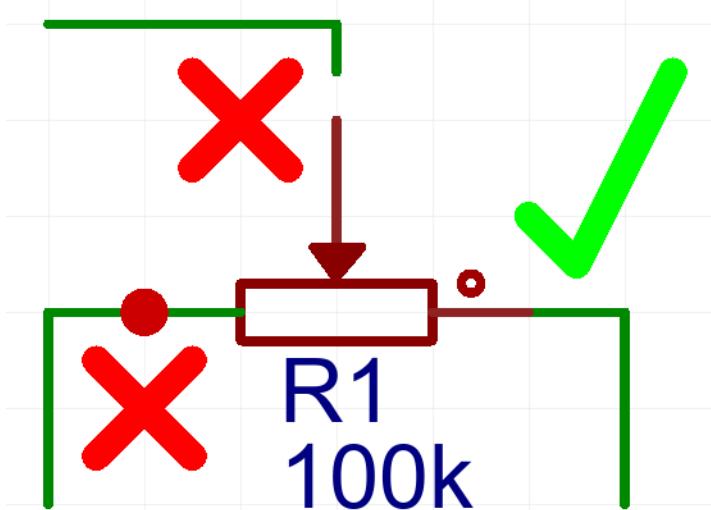
Naming nets instead of relying on the EasyEDA auto-generated alphanumeric names makes signal tracing and debugging the final PCB much easier but care must be taken to ensure that names are correct and that there are no unintended duplicate names or accidental increments in numbered nets;

- Verify that there are no duplicate prefixes in the schematic:

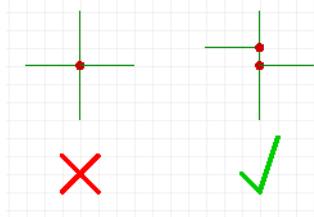
https://easyeda.com/forum/topic/How_to_resolve_quotPrefix_Conflictquot_error_-gpca8642

Remember to check across all sheets of a multi-sheet schematic;

- Check that the schematic is drawn correctly.
In particular, check that no nets have been accidentally cross connected, that wires have join dots where they are intended to be joined, that they are properly connected to component pins and that nets joined by netlabels are correctly named and that there are no unintended duplicate net names.



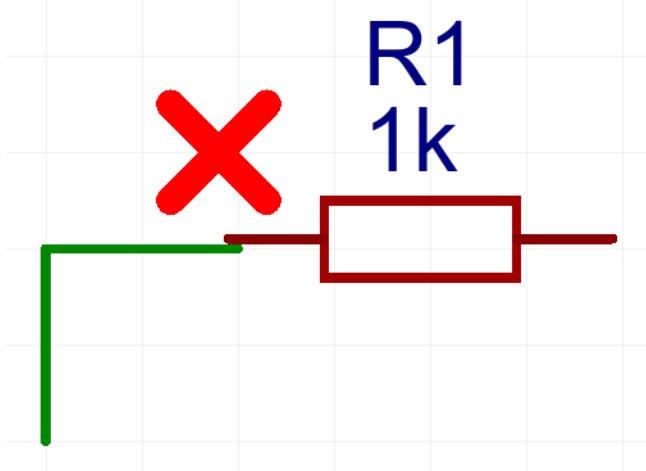
- Verify that junctions of 4 or more wires are drawn to show staggered junctions to avoid confusion with wires that cross but are not joined at the crossing point.



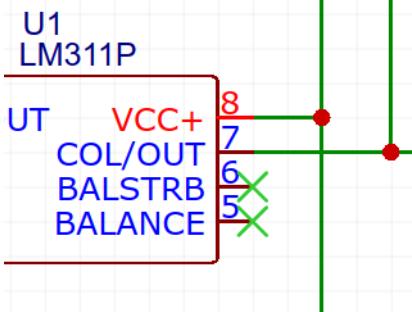
- Check that all parts and nets have been placed with the Canvas Attribute **Snap = Yes**:

Selected Objects	0
▲ Canvas Attributes	
Background	#FFFFFF
Visible Grid	Yes
Grid Color	#CCCCCC
Grid Style	line
Grid Size	10
Snap	Yes
Snap Size	10
ALT Snap	5

and that no parts have been placed off grid so that although they may appear to be connected on close inspection it can be seen that they are not:



- Check that pins have been terminated (pulled up, down, left open etc.) as specified in manufacturers' datasheets.
 - Check that all unconnected pins have **No Connect** symbols attached directly to them. Unconnected pins without No Connect symbols attached directly to them will show up as alphanumeric net names in the Design Manager but will not highlight when clicked on in the Design Manager.
- No Connect symbols must be attached directly to component pins. There should be no wire between the No Connect symbol and the pin.



Note that the No Connect symbol changed in V4.8.5 of EasyEDA from a red cross to a green one to make the highlighted state of a selected symbol clear;

- Check that the device ratings are suitable for the circuit in which they are to be used. For example, capacitor, diode, transistor, connector and switch voltage ratings, transistor, resistor and zener diode power dissipations, inductor, diode (including LED), transistor, connector and switch current ratings.

Although these parameters should have been checked at the time of specifying the components as an essential part of the circuit design stage prior to or during Schematic Capture, there is plenty of scope for them to have gone astray during the part selection, placement and editing steps of Schematic Capture.

An undetected mistake now can result in the wrong size part being chosen. For example a larger diameter or even a taller electrolytic capacitor may be needed. Whilst this is easy to correct in the PCB design stage, at best this may waste time in having to redesign part of the PCB. At worst the mistake may not be discovered before the PCB design is completed and sent for manufacturing.

- Check that diode (including LED) and bipolar transistor base-emitter junction reverse breakdown voltage ratings and that input differential and common mode voltage ratings of operational amplifiers and comparators are not exceeded during any state of operation of the circuit including power up and power down.

Consider adding diode or MOSFET reverse supply protection especially for battery powered circuits.

An example of MOSFET reverse protection is described in:

<https://easyeda.com/example/UberclampSchematicPCBandBoM-r4YgysK2k>

Pay special attention to this in operational amplifier or comparator devices that exhibit `output phase reversal` under some input conditions. For more information about this see:

<http://www.analog.com/media/en/training-seminars/tutorials/MT-036.pdf>

For example, the TL081 exhibits this behaviour but it is not documented in more recent versions of the datasheet. See Applications Hints on page 5 of this earlier version:

<http://www.physics.ucc.ie/fpetersweb/FrankWeb/courses/PY2108/spec%20sheets/TL081%20OpAmp.pdf>

Consider adding diode or MOSFET reverse supply protection especially for battery powered circuits.

An example of MOSFET reverse protection is described in:

<https://easyeda.com/example/UberclampSchematicPCBandBoM-r4YgysK2k>

- Check that LED currents are supplied through series current limiting resistors or from constant current sources.

For background on this please see:

<https://easyeda.com/andyfierman/LEDsmusthaveseriesresistors-OoGYgCK2k>

- Check that signal connectors have sufficient ground pins to maintain signal integrity by minimising signal return path impedances (i.e. ground loop area). This is especially important in designs with high speed signals through the connectors but can also be important for lower speed signalling with long wire interconnects and/or fast edge speeds.
- Check that power connectors have sufficient ground and power pins pins to maintain power integrity by minimising power and ground return path impedances.
- Verify that device power supply decoupling complies with manufacturers' recommendations.

Where possible, check datasheets, applications notes and schematics and PCBs for Reference Designs or Evaluation Boards.

For some background on the importance of adequate decoupling please see:

https://easyeda.com/andyfierman/Power_supply_decoupling_and_why_it_matters_-451e18a0d36b4f208394b2a2ff7642c9

- Verify that a Schematic Symbol and an associated PCB Package has been created for every Part needed to construct the complete PCB.

Remember to include Schematic Symbols and an associated PCB Packages for things like heat sinks, PCB mounting holes, mounting holes for PCB mounted potentiometers and switches (for example where a PCB is used as a front panel or as a self-contained test jig), test points, wire links or jumpers, fuse holders and even image based elements such as high voltage warnings and logos;

Look in:

Parts (or SHIFT+F) > MyParts > Schematic Lib > Favorite Schematic Lib

and:

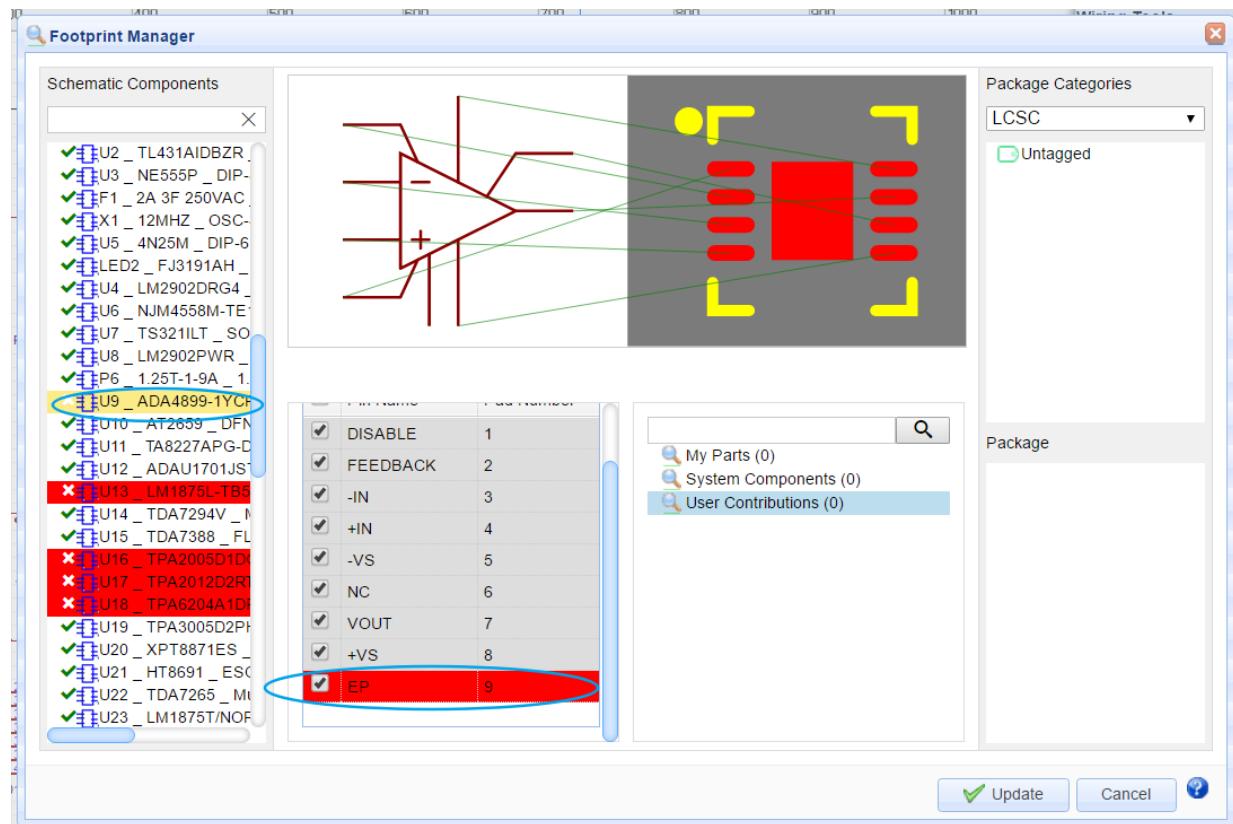
Parts (or SHIFT+F) > MyParts > Package > Favorite Package

and verify that every Schematic Symbol and associated PCB Package chosen from the **User Contributions** category has been added to your local library;

- Verify that the pin numbers/names of the Schematic Symbol(s) for every Part are correct, unique and match those of the PCB package associated *with that particular part*;
- Verify that the pin order (pin mapping) of the PCB Package associated with every part is correct.

This task is simplified using the EasyEDA Footprint Manager:

<https://easyeda.com/Doc/Tutorial/Schematic.htm#Footprint-Manager>



Remember that in EasyEDA, the PCB Footprint is viewed looking down onto the component side of the board. This view is assumed to be with all components mounted on the Top Layer. Packages can subsequently be placed on the top or bottom layers as required.

- Verify that all necessary information about the specific components (and any suitable alternatives) that are to be used in the circuit and which are ultimately intended to be mounted on or form an integral part of the PCB, from which a Bill of Materials (BoM) can be generated has been added to the Schematic Symbols.

For more information about this please see:

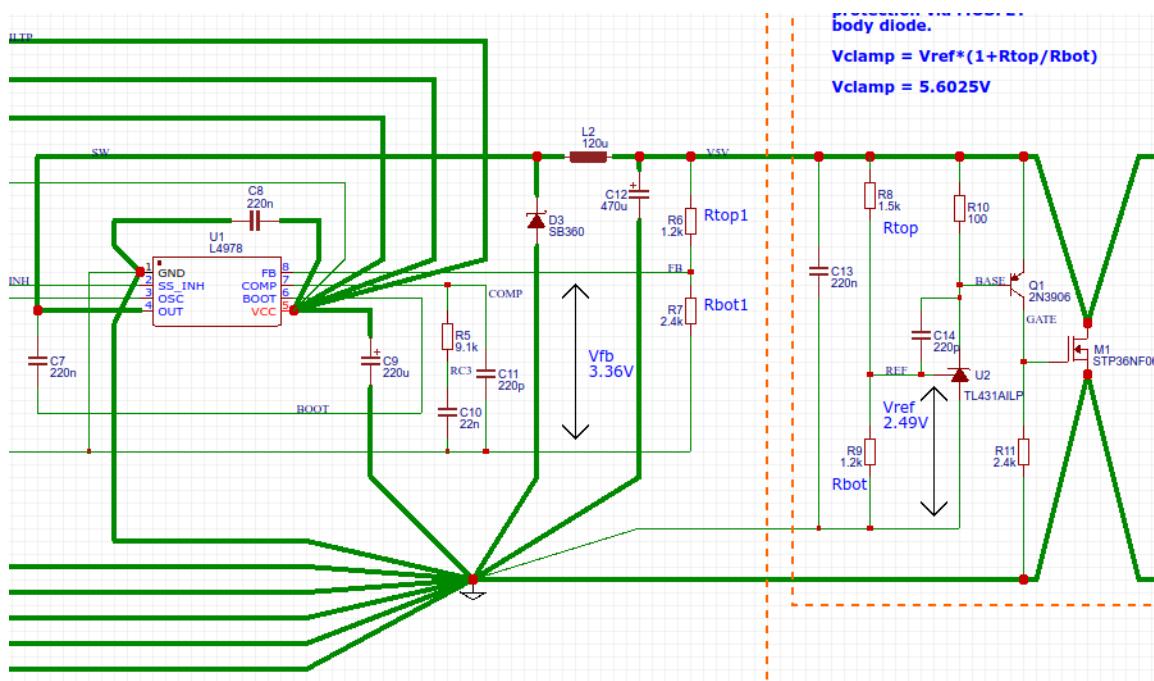
https://easyeda.com/forum/topic/How_to_add_extra_information_to_the_Bill_of_Materials_BOM-Hp9rJCUcu

- Verify that any necessary information relating to the physical placement of components and layout of copper traces and areas has been annotated in the schematic.

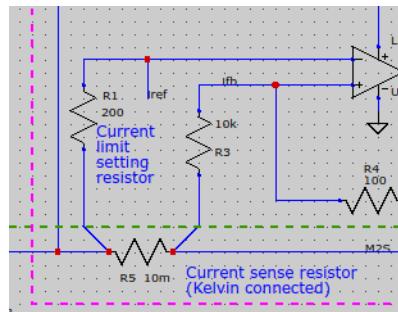
Including text (or even diagrammatic information in the schematic about component positioning and orientation, clearances around heatsinks for airflow or copper areas for heat sinking, current and voltage ratings of traces, trace length matching, controlled impedance transmission lines and differential pairing can all help in the following stages of PCB design.

Nets that are carrying high currents may be drawn using thicker wires (Stroke width).

Nets can be drawn converging at star points to help illustrate where this type of PCB layout is required on the PCB:



Kelvin connections to current sense resistors can be drawn in a similar way:



Nets can be colour coded but beware using red because it can be very hard to see when such nets are highlighted.

- Generate - and check - the Bill of Materials (BoM) information and check the availability of components.

Whilst it is easy to change parts in the PCB design stage, at best this may waste time in having to redesign part of the PCB. At worst the unavailability of a part may not be discovered before the PCB design is completed and sent for manufacturing.

- Use the Design Manager (left-click the **Design** button in the left hand panel) to check everything again!

FAQ

Please spend a few minutes reading this FAQ, it will save you lots of time getting started with EasyEDA.

Hot Question

How to find the list of hotkeys.

<https://easyeda.com/Doc/Tutorial/Introduction.htm#Hotkeys>

Where are my files?

Your files are stored on EasyEDA servers, so you can access them anywhere and share them with your partners.

Why does EasyEDA focus on Cloud based EDA?

EasyEDA is built for people who like to work anywhere, who like to build projects together with other team members, who like to share their projects, who like something that operates like a github for hardware design. The only way to meet these needs is to build a Cloud version EDA.

How can I work if there is no internet?

Although most of the time there are ways to access the internet easily and cheaply there may be times when, for whatever the reason, internet access is simply not possible. For times like this, EasyEDA is working to provide a desktop client soon.

Does EasyEDA have a desktop version?

At present, no but EasyEDA is developing and testing a desktop version to be introduced soon.

A Windows version will be available at the end of this year. Mac and Linux versions will be available early next year.

Which Browser is best for EasyEDA?

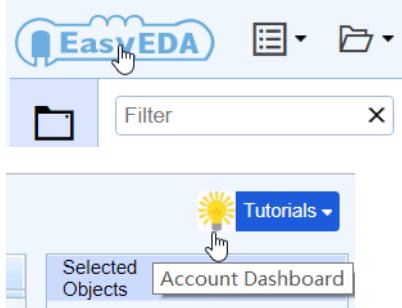
Chrome. Firefox and Safari are OK too. If you are restricted to using other browsers, it would be better to download the EasyEDA desktop client when it becomes available (see above).

How to go to your dashboard.

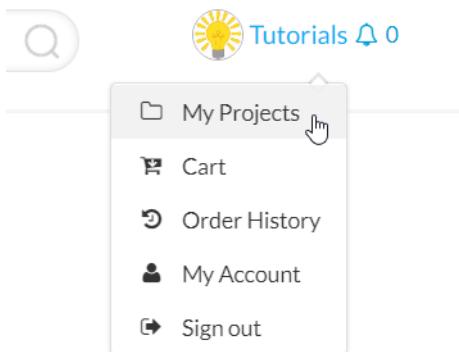
In the [Dashboard](#), you can check all your Projects, Modules and Components and Favorites, projects others have shared with you, forum posts and orders.

There are two ways to arrive there.

1. From the Editor, you can click on the EasyEDA logo or user logo:



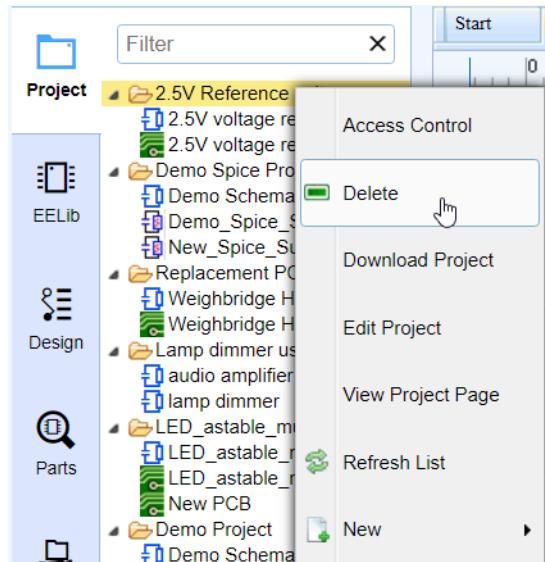
2. From the homepage, you can click My Projects:



Projects and Files

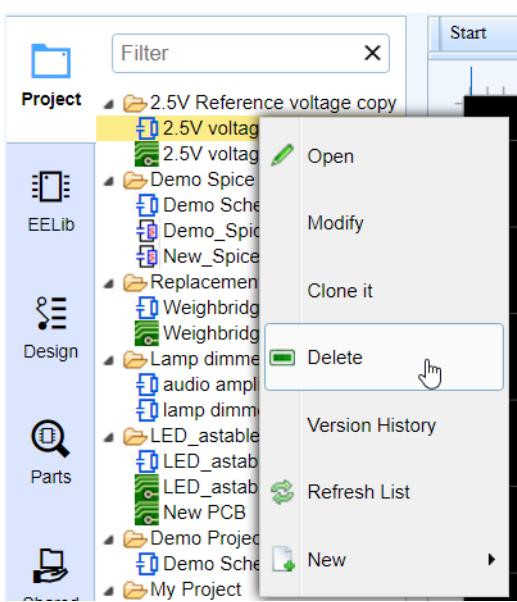
How to delete a project.

Select it and right click to open a context menu, like the image below.



How to delete a schematic or PCB.

Select it and right click to open a context menu, like the image below.



How to share a project with others.

1. Make your project public.

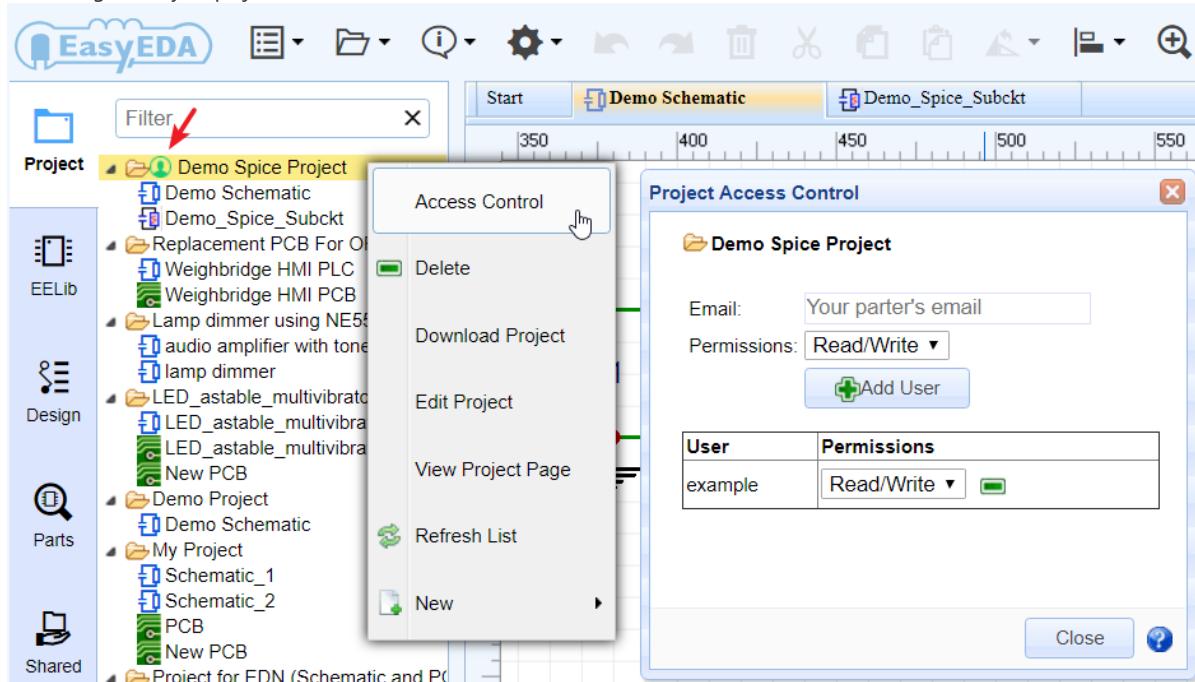
Open <https://easymaker.com/projects/mylists>, then click the red 'No entry' icon where indicated by the arrows. This icon will change to a green 'Tick' icon to show that the project is now public.

Project	Create Time	Docs	Views	Like	Star	Show in Editor	Comment	Share	Edit
2.5V Reference voltage copy	3 days ago	2	0	0	0	✓	✓	✗	✗
Replacement PCB For OP-320A HMI which gives partial Arduino compatibility copy	6 days ago	2	0	0	0	✓	✓	✗	✗

2. To share a project privately with only selected collaborators via:

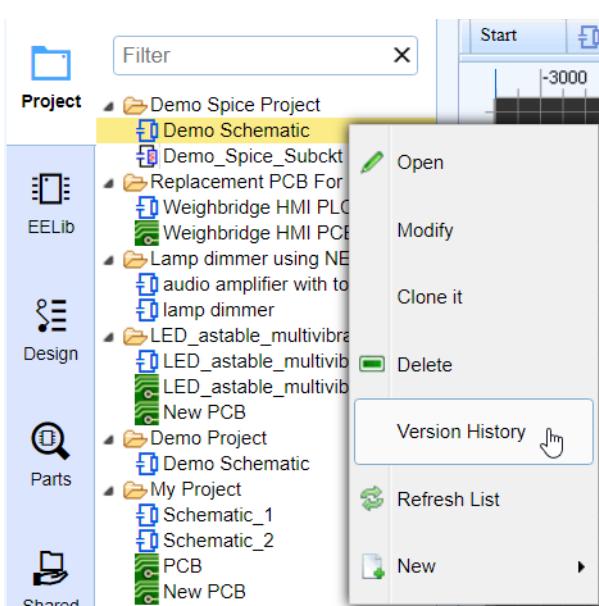
[Access Control](#)

You can right click your project and select the access control menu:



How to find the version history of schematics and PCBs.

The version history of your EasyEDA schematics and PCBs can be accessed by right-clicking on the file you wish to query to open the context menu as shown in the image below:



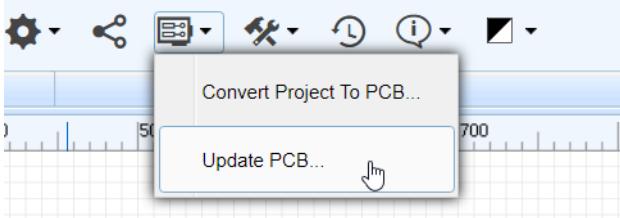
Then click on the version number that you wish to view.

Note: saving a previous version will restore that version to being the current version of the file.

Schematic

If I update the schematic, how do I then update the PCB?

The initial conversion of a schematic to PCB is done from within the Schematic Editor using the **Convert Project to PCB...** button as illustrated in the toolbar below but a new **Update PCB** button has been added so that modifications to the schematic can immediately be passed forward to update a selected PCB without having the PCB editor window already open.

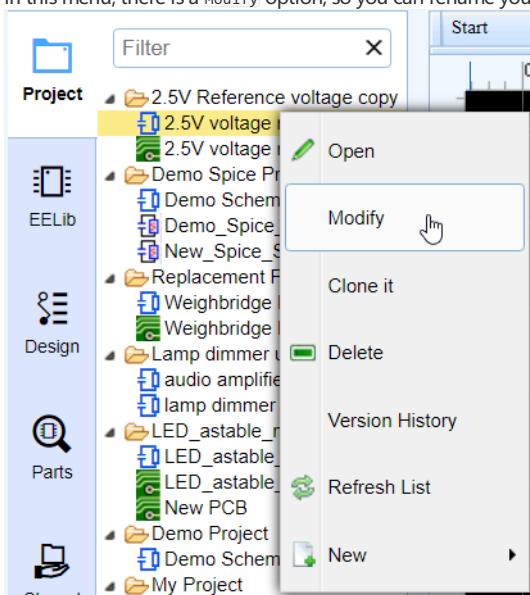


Alternatively, you can import changes from the schematic from within the PCB Editor:

<https://easyeda.com/Doc/Tutorial/PCB.htm#Import-Changes>

How to rename a Sheet/Document or modify description.

In this menu, there is a **Modify** option, so you can rename your files.



How to find components

The component search function has been significantly improved to make finding part symbols and footprints quicker and easier. Press **SHIFT+F** or click on the **Parts** icon on the left navigation panel:

In the new components dialog, it is easy to select the right components via tags and you can set tags for your own components.

How to add sub parts to a schematic.

You can add sub parts to a schematic one by one but please note that the sub parts prefix must be in the form of U1.1 U1.2 etc, and not U1.A U1.B.

Title(PartNO)	Package	Description
New Schematic Lib2		
SIP4	HDR1X4	
CY7C1049CV33	TSOP II - 44	
ADTL082ARZ	SOIC8	
ADTL082ARZ.1		
ADTL082ARZ.2		
TEENSY-3.1-ALL-PINS-AND-PADS-VIN_TEENSY_3.1_ALLPINS-VIN	TEENSY_3.1_ALLPINS	
NotGate	DIP	
IRFP460	-TO-247AC	
SOLDERJUMPERNC_SJ_2S		
ATMEGA256RFR2QFN_QFN-64		
ADTL082ARZ.1		

What is the unit of the schematic sheet?

The basic unit of the schematic sheet is the pixel. 1 pixel is about 10mil (0.001 inch) but please note that this use of the pixels as a unit in a schematic is just for reference.

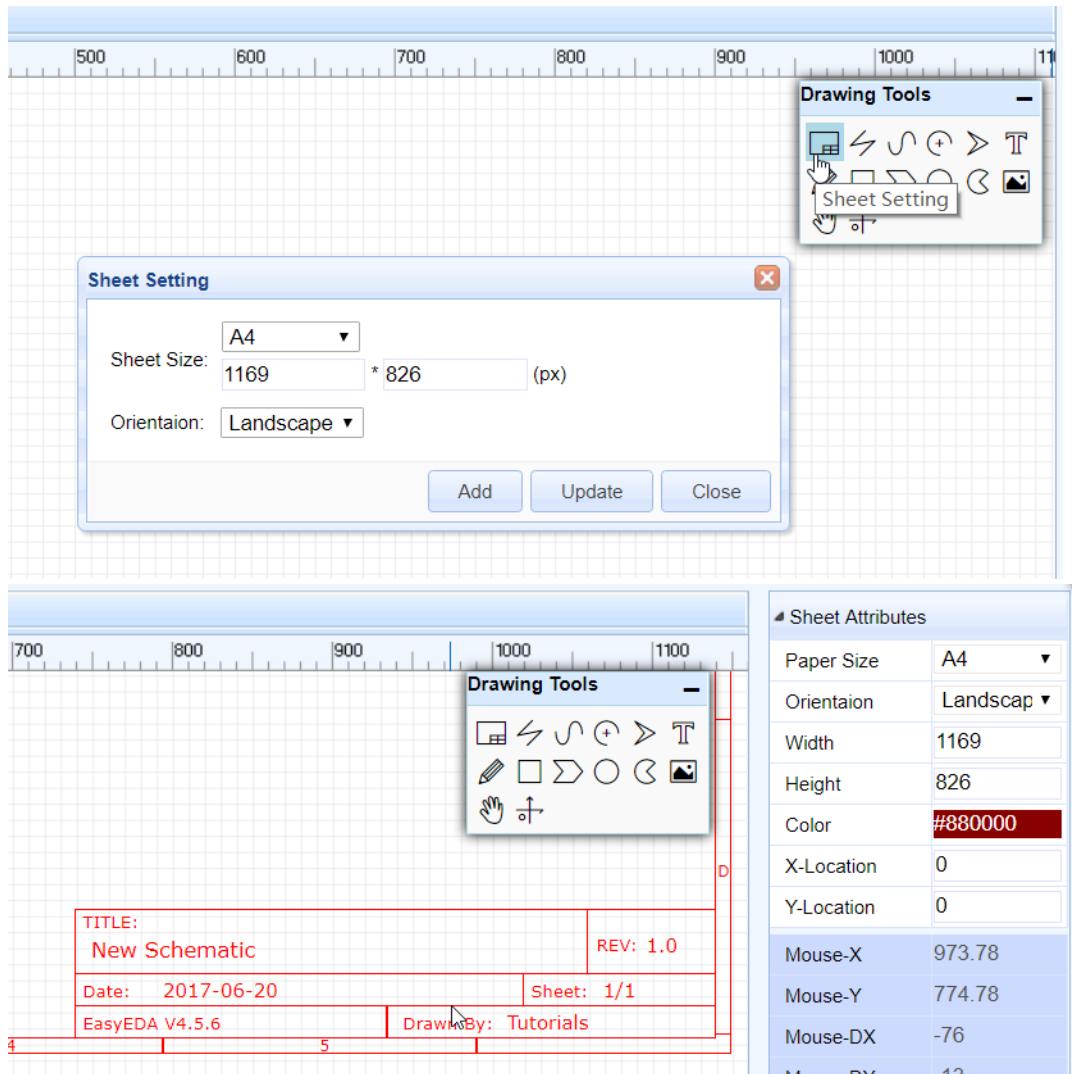
For a complex project, I want to split the schematic over several sheets. Does EasyEDA support hierarchy?

Please check out this link <https://easyeda.com/Doc/Tutorial/Schematic.htm#Hierarchy>

How to change the sheet size and modify the design information.

To change the sheet size, move the mouse anywhere over the lower right area of the drawing border or frame until the whole border highlights red and then right-click on it. Paper size and orientation can then be changed in **Sheet Attributes** in the right hand panel.

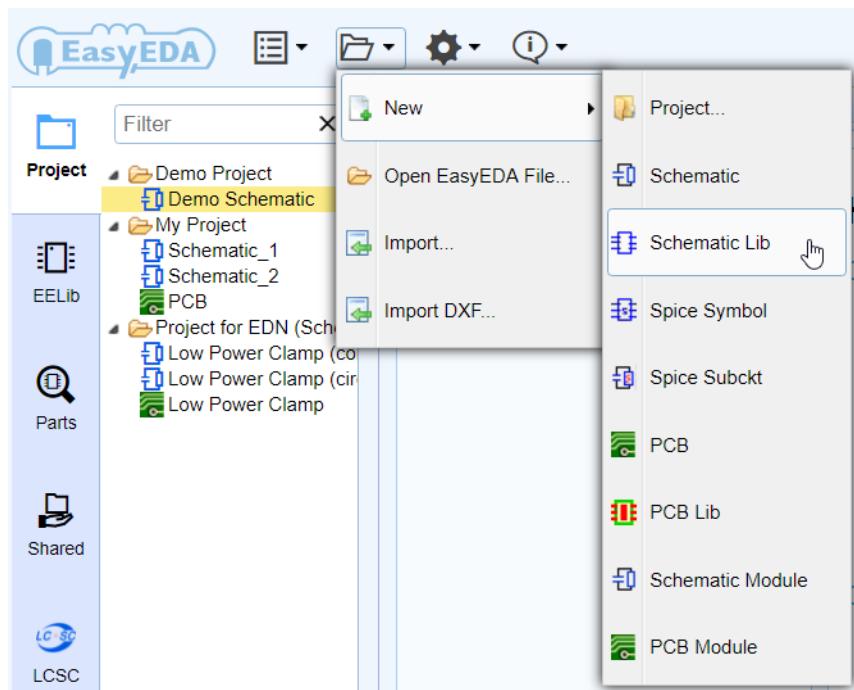
To modify the design information, left-click on the relevant blue text in the lower right area of the drawing border or frame to change it in **Text Attributes** in the right hand panel. Double left-clicking the blue text will allow you to type new information directly into the field.



Schematic library symbol

How to create a schematic library symbol.

Document > New > Schematic Lib



How to tag my schematic library symbol.

Browse and search hundreds of thousands of components

Search components and modules

LCSC Parts - save 40%

System Components

My Parts

My Modules

Common Modules

User Contributions

Search Results

Schematic Lib

Untagged Favorite Schematic Lib

Package

EasyEDA Package KiCad Tinkerforge Footprints Untagged Favorite Package

Title(PartNO)	Package	Description
3455	SOP8	Demo schematic lib

3455

Modify
Delete
Clone
Add Sub Part
Add Favorite
Cancel

How to create sub parts for multi-part components.

In My Parts, Right click the part then select Add Sub Part from the menu that opens:

Title(PartNO)	Package	Description
74HCT04		74HCT04
74HCT04.1		
74HCT04.2		
74HCT04.3		
74HCT04.4		
74HCT04.5		

Modify
Delete
Clone
Add Sub Part
Add Favorite

How to change the Package for a component.

<https://easyeda.com/Doc/Tutorial/Schematic.htm#Update-Package>

PCB

How to change the Units of PCB from mil to mm or inch.

There is an option for that in PCB canvas attributes:

Canvas Attributes

Units mm
mil
inch
mm

Background #FFFFFF

Grid Yes

Grid Color #FFFFFF

Grid Style line

Snap Yes

Grid Size 2.54mm

Snap Size 0.25mm

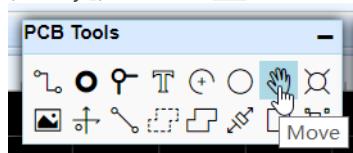
ALT Snap 0.13mm

Other

How to pick and move the components on the PCB canvas quickly.

Before routing the PCB, the components need to be positioned in suitable places on the PCB. In the PCB Editor, it can sometimes be quite

difficult to select components by clicking on the silkscreen outline or the pads. To select and move them more easily, please use drag mode (Hot Key D) or click the Move icon in the PCB Tools toolbar:



Can I create a PCB without creating schematic?

Yes but for any but the simplest PCBs, please see:

https://easyeda.com/forum/topic/The_best_way_to_design_a_PCB_in_EasyEDA-ThR3pwqlC

How to add more fonts for PCB.

You can refer to [Text](#) of PCB section.

How to insert an Image/Logo to PCB.

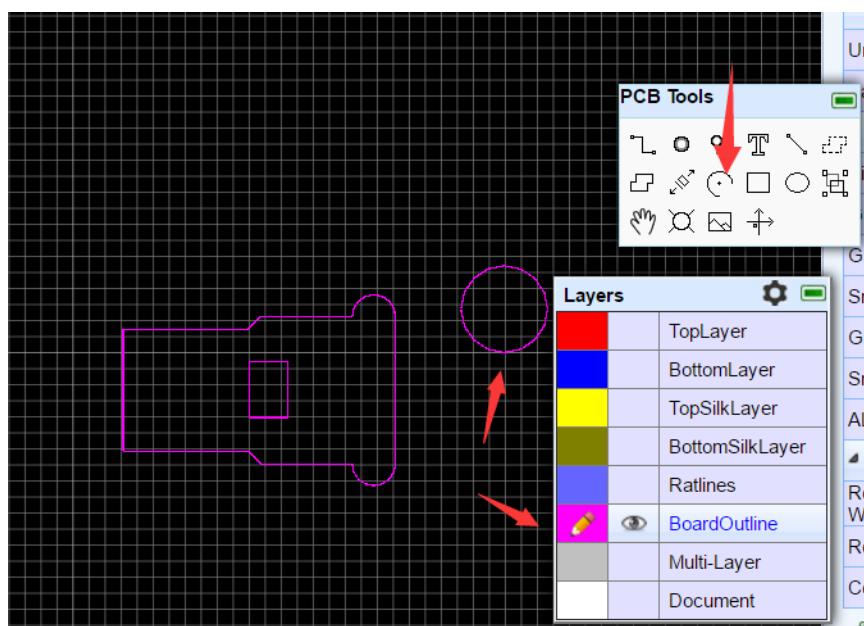
You can refer to [Image](#) of PCB section.

How to insert a DXF board outline.

You can refer to [Import DXF File](#) of Import section.

How to create non rectangular pcb outline such as round?

You can import a DXF file for the board outline. For a round board outline, you can use an arc to do that, you just need to change to the board outline layer, then draw 1 arc like in the image below (need to adjust a bit later), you can use lines and arcs to create complex board outlines.



How to add a slot and cut out.

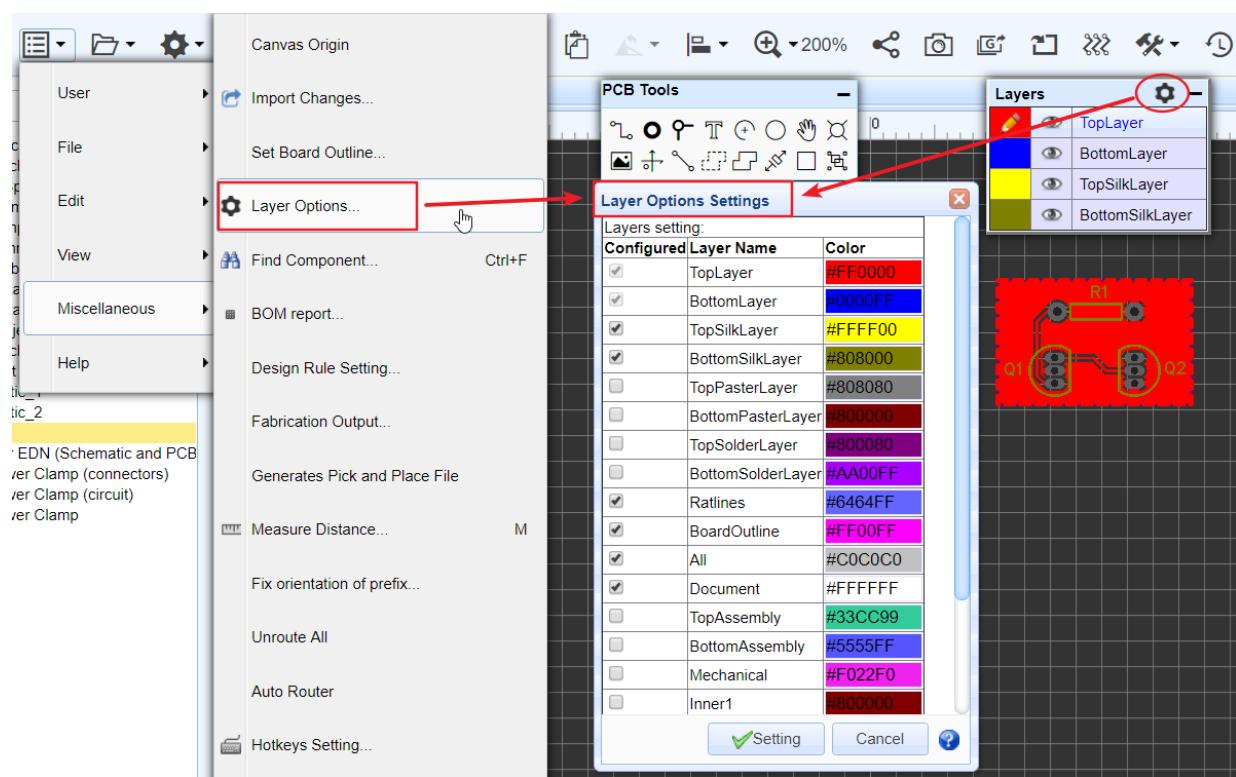
<https://easyeda.com/Doc/Tutorial/PCB.htm#Pad> and <https://easyeda.com/Doc/Tutorial/PCB.htm#Solid-Region>

How to measure dimensions on a PCB.

<https://easyeda.com/Doc/Tutorial/PCB.htm#Measure-Dimension>

How to add more layers.

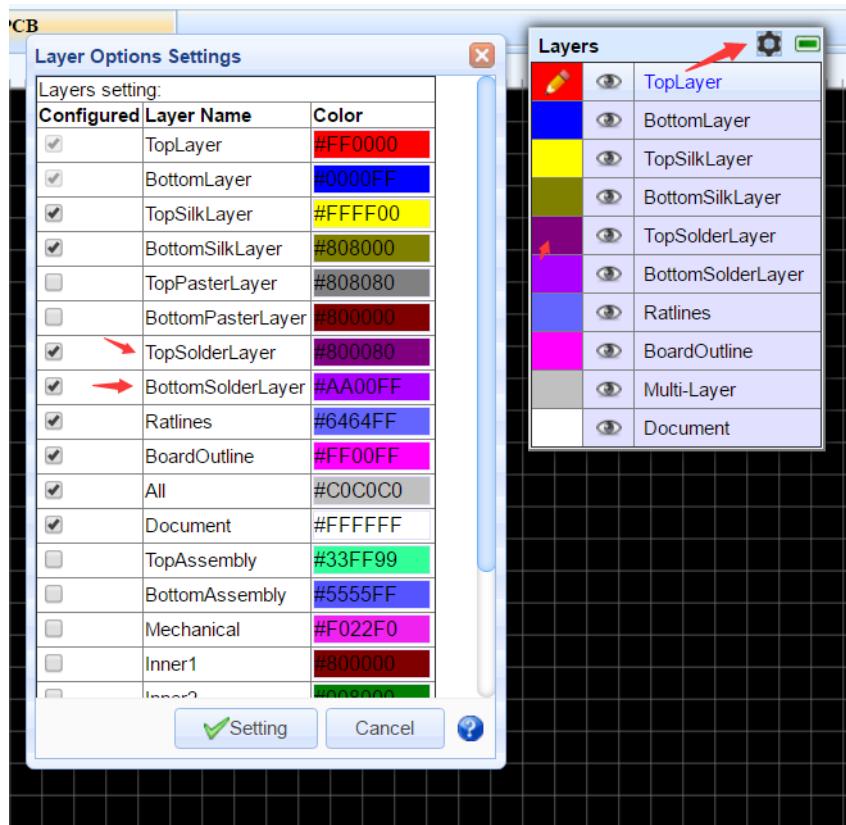
Click the layer options button, then tick the extra layers in the dialog that opens.



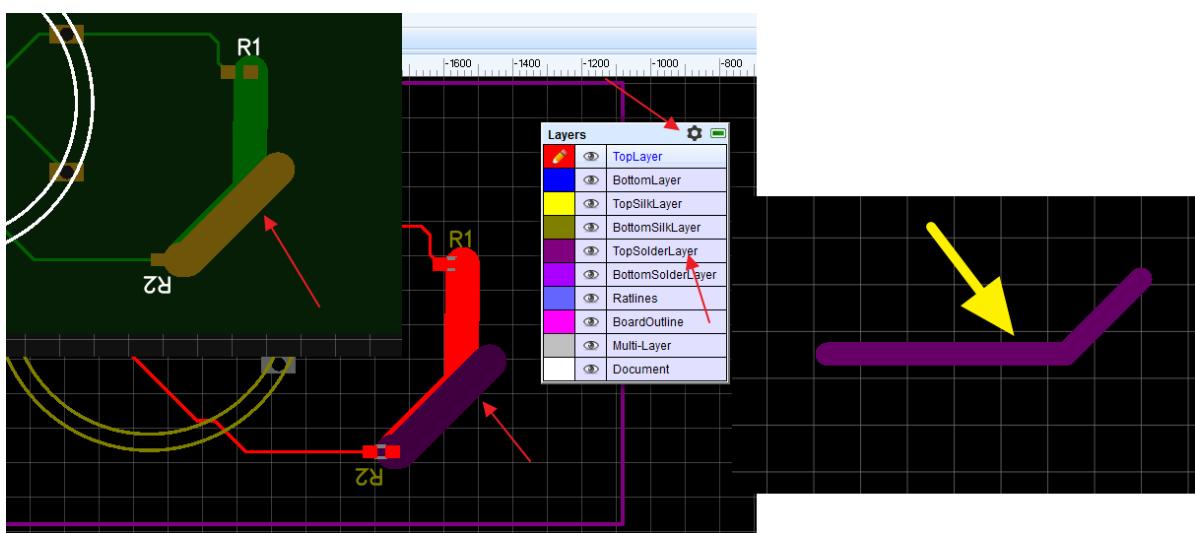
How to add solder mask.

It is possible to get boards with the copper exposed so that you can apply a layer of solder over those tracks to further increase their current carrying capacity. In this case, you need to add solder mask over a copper (copper area, track, solid region). EasyEDA will add solder mask for pads automatically. Sometimes however, you may need to add an aperture in the solder mask to expose an area of copper.

1. First, add a top or bottom solder mask layer, as required.



2. Next, draw a region in the solder mask layer over a copper item as illustrated in the image below. This in effect draws an aperture in the solder mask so that the copper item inside the region, in this case the track, will not be covered by the green film of solder mask.



A common mistake is to just draw a solder mask, without a copper area, like the track pointed to by the yellow arrow. That is incorrect and does not produce the desired result.

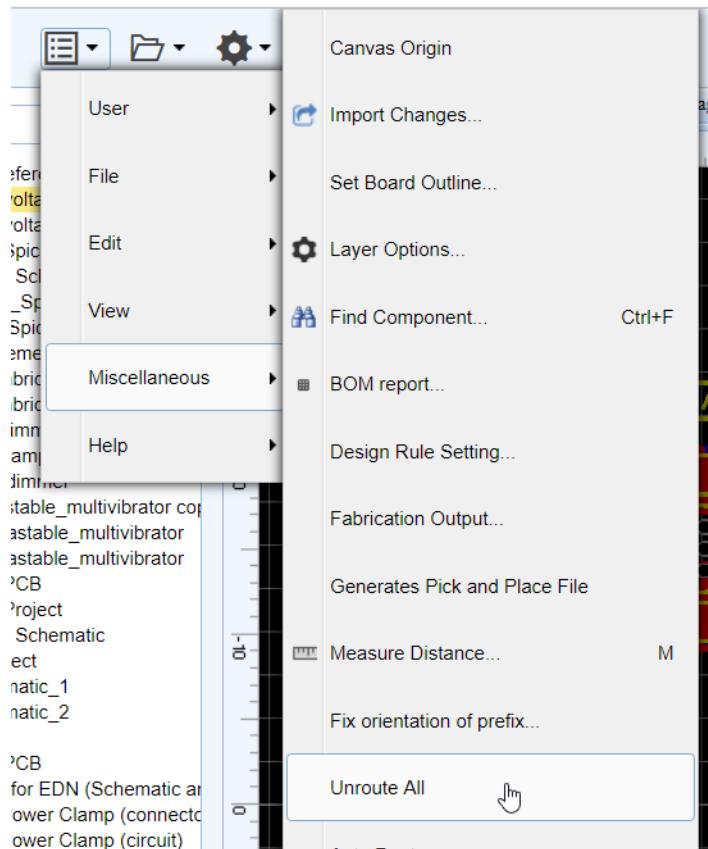
How do I set the dimensions of my PCB in the layout?

PCB's dimension/size depends on the board outline, you can create your board outline, please refer to [Board Outline](#) of the PCB section.

My PCB is complex, how can I be sure that I have routed all of the tracks?

Please refer to [Design Manager](#) of PCB section.

I need to start my layout again, how can I remove all of the tracks?

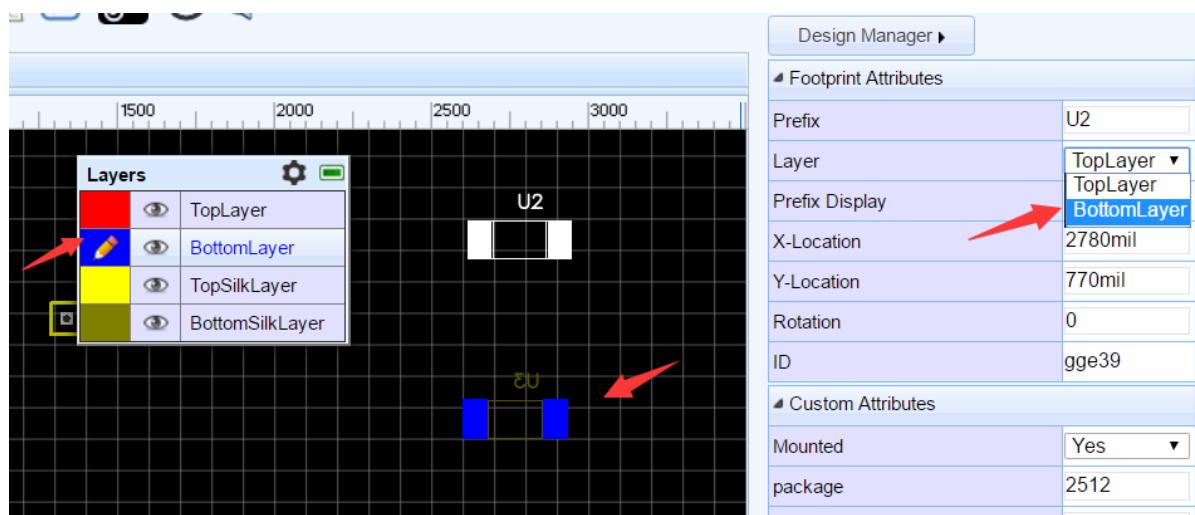


Unroute All

How to put a component on the bottom layer?

There are two ways to do this.

1. If your active layer is the bottom layer, then every component you place will be placed on the bottom layer automatically.
2. You can place a component then select it and change its layer attribute to `Bottom_layer` in the right hand panel.

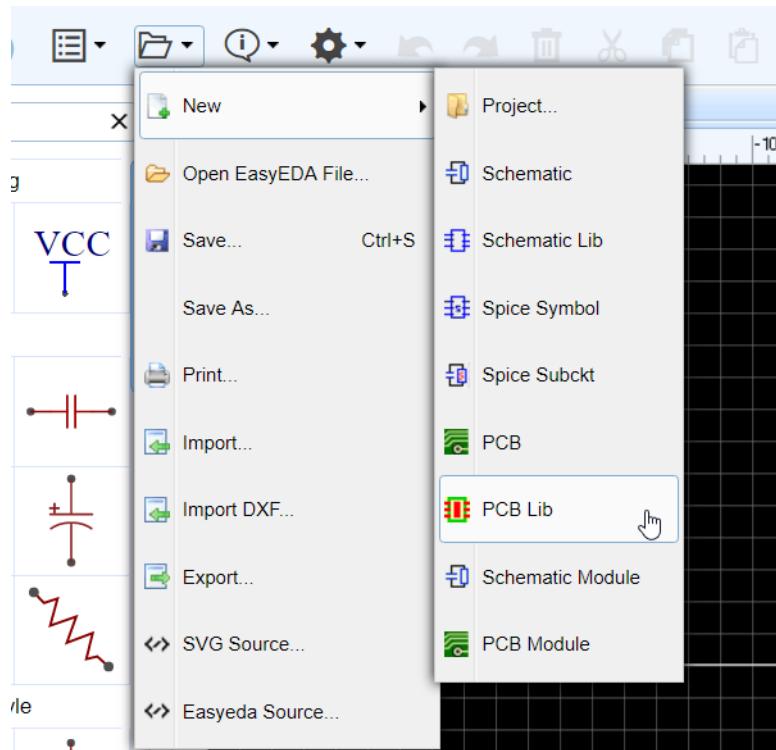


I can't convert schematic to PCB. Why is this?

1. You have not set the right packages for your components.
2. <https://easyeda.com/Doc/Tutorial/Schematic.htm#Prefix-Conflict-Error>
3. <https://easyeda.com/Doc/Tutorial/PCB.htm#Invalid-Packages>

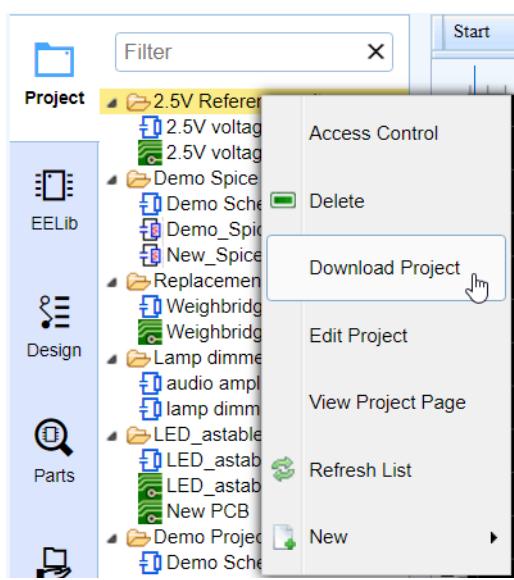
PCB package.

How to create a PCB package/library.



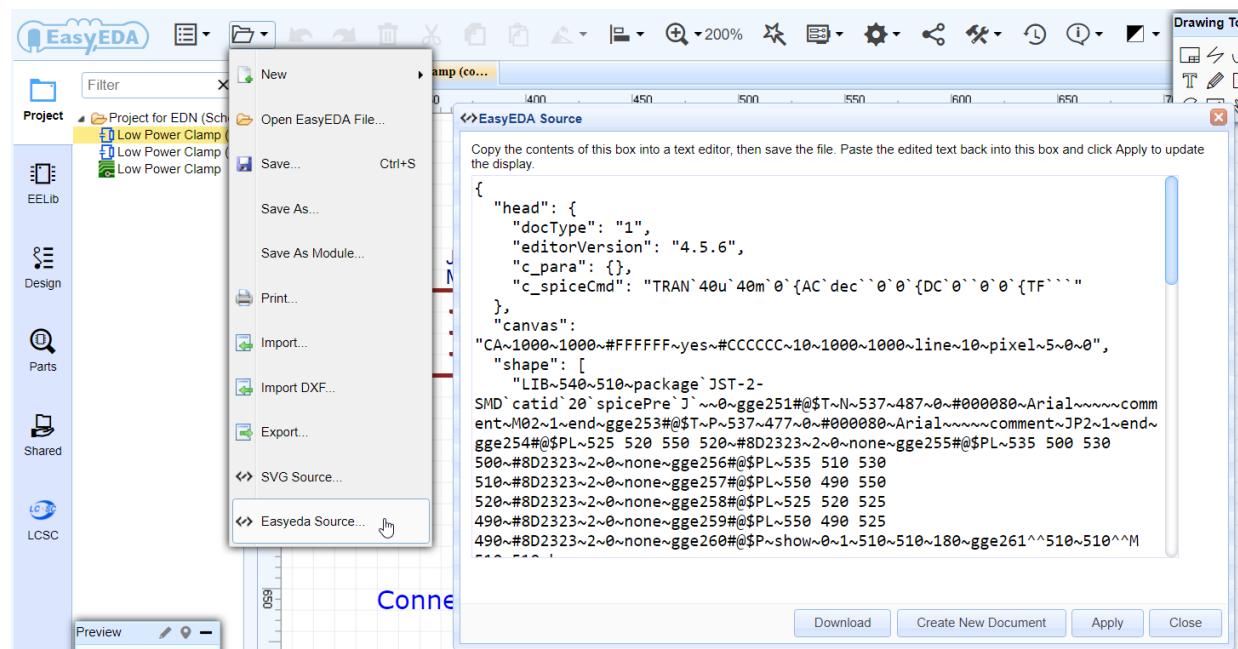
Keep in Mind

1. There is no need to back up your schematics and PCBs manually. After the first save of any file, EasyEDA will back up all saved files automatically under the [Version History](#). If you want to back up your files locally, you can download a copy of the whole project or of individual files in a project in EasyEDA Source (JSON) format:



and;

Document > EasyEDA Source > Download



2. If you need help, you can contact us or ask via our [Support Forum](#); we will respond ASAP.

Most Common Errors on EasyEDA.

1. Manually creating backup schematics into the same project. When a project is converted to PCB, EasyEDA will merge all of the schematics under the same project into a single PCB. If there are multiple copies of the same schematic in a project then this will create errors such as duplicate part prefixes. Especially if you are new to EasyEDA, just keep one copy of each unique schematic in any one project.
2. Saving schematic and PCB into different projects. Unless you are absolutely sure that you will not need to update (Synchronise) your PCB from changes made to your schematic then please keep the schematics and PCB under the same project.
3. Bad packages. Schematic symbols must have the appropriate footprints assigned to them, these footprints must exist in the library and - for any footprint that you have not created yourself - you must have clicked on the **Favorite** option in the component search window to add it to your **Favorite Parts** list in the left hand Navigation panel.

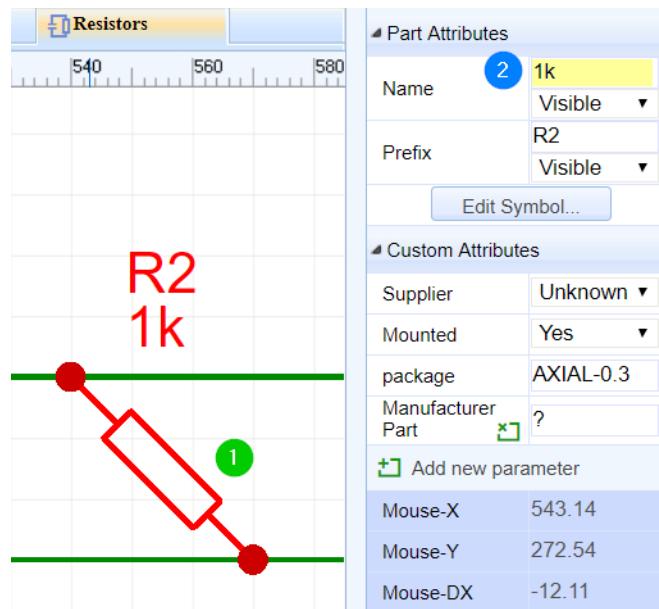
4. Using the polyline from the Drawing Tools Palette to connect symbol pins. To connect components together, you must use Wires from the Wiring Tools Palette.

Spice Simulation FAQ

EasyEDA's main target is schematic and PCB, not simulation. EasyEDA only support simple schematics simulation.

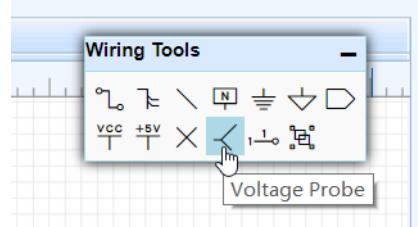
How to set the resistance of a resistor

You can use the name attribute. Just set the name or double click the value text.



Where Can I find the Probe?

Voltage probe



Why I can't simulate my schematic

EasyEDA only has very few simulation models, EasyEDA is powered by <http://ngspice.sourceforge.net/> please check Ngspice to know what can be simulated.

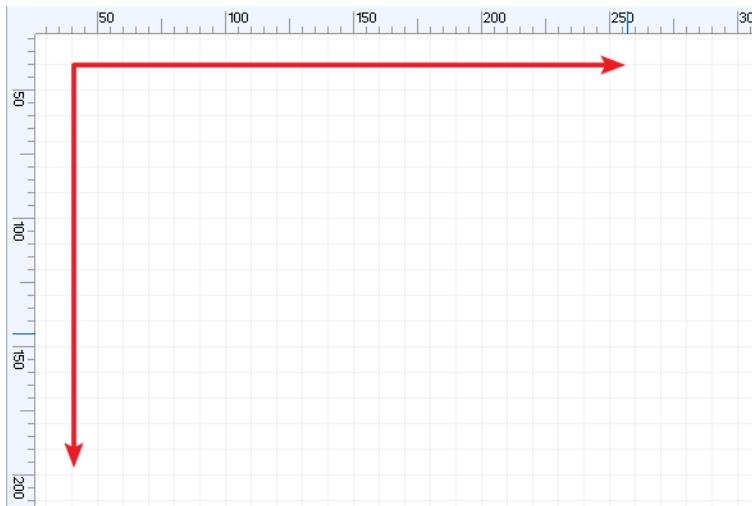
Others.

Does EasyEDA canvas use the Cartesian coordinate system?

Yes and no.

It uses X and Y coordinates where the horizontal X coordinate is positive to the right of the origin and negative to the left but the vertical Y coordinate is positive **below** the origin and negative above it.

Actually, we think our coordinate system is not very good but it is hard to change.



PCB Order

After laying out your PCB, you probably want to order some **PCBs**. We have made it easy for you to save time and money by using our **awesome service** to order **low cost, high quality** PCBs *directly* from EasyEDA. More importantly, if you are not satisfied with the quality of our PCBs, EasyEDA will refund your money in full.

Although EasyEDA makes it easy to order PCBs for your projects and offers an exceptionally low PCB Manufacturing fee, you are free to download the Gerber files and order your PCBs from any other vendor. However, if you like EasyEDA, please give us a chance to fab. the PCB for you. We think you won't be disappointed.

PCB Quality

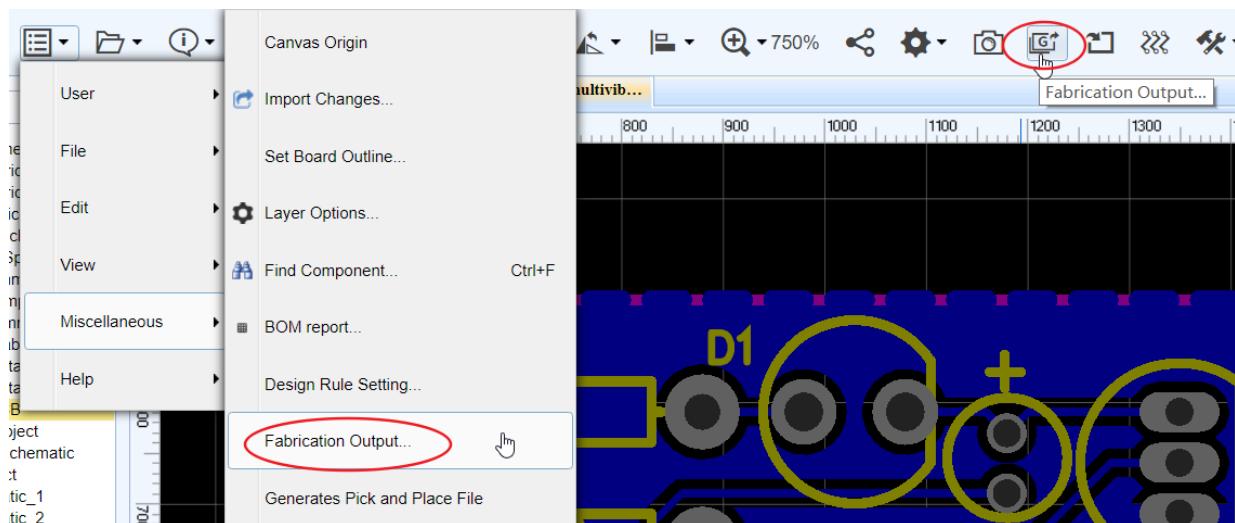
As engineers, we have spent more than 6 years building EasyEDA. As artisans, we believe that if you like using our EDA tools, then you will like our PCBs.

EasyEDA's PCBs are in a group buy model and all PCBs are given 100% E-test. This allows us to provide you with good quality, tested PCBs at a great price. We have shipped thousands to our users, all of whom like our PCBs.

No matter how good we tell you our service is, maybe you still have doubts. The easy way to remove those doubts is to try it out because - as we have said above - if you are not satisfied with the quality of our PCBs, we will refund your money in full. For details of this offer, please check the [Warranty and Return](#).

Order Button

To order PCBs from us, just click the **Fabrication Output...** button in the PCB Editor window, as shown in the image below, and you will be redirected to an order page. In that page you can place an order quickly and easily. At the same time, at the click of a button, you can check the Gerber and drill files in our Online Gerber Viewer and then download your files. Obviously, we hope that you will support EasyEDA by ordering your PCBs from us but you are welcome to download the Gerber and drill files and send them to your favorite PCB house.



Essential Check Before Placing a PCB Order

A simple mistake can make a batch of PCBs useless so before submitting an order for PCBs, there are a lot of things to check.

The list below is a good starting point for the essentials but it by no means exhaustive!

- 1) Check that all nets and netnames are as intended. For example nets can be accidentally connected together while drawing and editing a

schematic and also by inadvertently assigning duplicated netlabels;

2) Check that the appropriate package has been assigned to each and every schematic symbol. Don't forget that different transistors, capacitors and even resistors may be in different packages in different locations in a schematic. Also check that components have been rated correctly as this may affect their package size (don't forget their height too!);

3) Check that the pin designations of the schematic symbol and the PCB package are the same. The pins on the schematic symbol for a bipolar transistor may be labelled B, C, and E but if the corresponding pins on the PCB footprint are labelled as 1, 2, and 3 then EasyEDA will flag this as an error when an attempt is made to convert the schematic into a PCB. It is simple to correct or - better - avoid this: change the labelling on the corresponding pins of the PCB footprint to B, C and E **or** change the labelling on the corresponding pins of the schematic symbol to 1, 2 and 3.

To change the pin labelling in the schematic to match the PCB footprint: select the part, press the **i** key then edit the **Names** in the **Edit Pin Map Information** section of the **Modify symbol information** dialogue that opens and click **OK** when finished.

To change the pin labelling in the PCB to match the schematic symbol: select each pin of the relevant part, then edit the pin **Names** in the right hand **Properties** panel.

Note: do not confuse the schematic symbol pin numbering with the spice pins numbering. For more about this see:

Schematic symbols: prefixes and pin numbers in:

https://docs.google.com/document/u/1/d/1OWZVVFRAe_2NW3WratpkA_SGuHa5AcRow5ZRfvcoVTU/pub#h.pkwqa1

4) Check that the pin designations of the PCB footprint chosen for each and every device actually matches the pinout of the device that will be soldered to it. It is very easy to assign a SOT23 package to a BC846 bipolar transistor where the pin order is:

Pin 1 = Base

Pin 2 = Emitter

Pin 3 = Collector

and then to forget that the pin order for a MMBF5485 junction FET going round the same SOT23 package in the same order is:

Pin 1 = Drain

Pin 2 = Source

Pin 3 = Gate

To change the pin labelling in the schematic to match the PCB footprint: select the part, press the **i** key then edit the order of the **PCB Pin** information in the **Edit Pin Map Information** section of the **Modify symbol information** dialogue that opens and click **OK** when finished.

To change the pin labelling in the PCB to match the schematic symbol: select each pin of the relevant part, then edit the pin **Numbers** in the right hand **Properties** panel.

5) Check that all necessary Bill of Materials (BoM) information is present and correct. Correct it and add more if required, making sure that fields such as **Description** are consistently labelled so that they form a coherent column structure in the BoM;

6) Check that all PCB footprints are correct for the intended devices (yes: whether they have come from the library or you have created them yourself: check them thoroughly);

7) Check that silkscreen markings such as the polarity markings for electrolytic capacitors and diodes are the right way round. Even if the pin names, numbering and sequence around the package are correct, it can all go wrong if the footprint markings show the device in the wrong orientation.

9) Check that devices have been placed on the correct side of the board;

10) Refresh and check all the Components and Nets in the schematic Design Manager tab in the right hand panel;

11) Refresh and check all the Components, Nets and DRC Errors in the PCB Design Manager tab in the right hand panel;

12) Check connector and on-board pot and switch orientations;

13) Check the dimensions and locations of mounting holes and any components that have to line up with respect to these mounting holes or to apertures in an enclosure;

14) Check that the order of the top and bottom (and any inner layers) is correct;

15) Check that a Board Outline exists, is closed and that it is shaped and dimensioned correctly and is on the correct (Board Outline) layer;

16) Check that silkscreen markings do not overlap pads;

17) Check that all required silkscreen markings are present, in the correct locations on the correct layers, are within the recommended dimensions, are legible and spelled correctly;

18) Check that any additional information such as notes about the PCB stackup etc, are present and on the correct (Documentation) layer;

19) Check that no board outline, silkscreen or documentation layer information has accidentally been placed on any copper layers;

20) Having completed the layout, check that the assembled component heights do not foul any enclosure (At the time of writing (160922) this is not something that can be achieved directly in EasyEDA as there is no 3D viewer yet available so this must be checked by other means);

21) If copper areas are used with heat shunt spokes enabled, check that any tracks that join pins that are also joined by copper areas run within the area of a spoke. If they do not - for example a 45 degree diagonal track coming out of a pad with 90 degree heat shunt spokes - the track forms an extra spoke which increases the heat shunting to a pad and so may make soldering more difficult;

22) Check that all copper areas assigned to a net are joined by reasonable widths of copper, i.e. they are not just joined by thin slivers of copper;

23) Check that any tracks that require special routing considerations such as Kelvin connections to low value current sense resistors or increased clearances for high voltage traces have been correctly implemented;

24) Check clearances of copper on all layers and components on both sides to the edges of the board;

25) Check that no traces have been set to hidden in the Design Manager Nets list;

26) Use the Design Manager (the **Design** button in the left hand panel) to check that all components are present in both the schematic and the PCB and that all nets have at least two connection;

27) Use the Design Manager (the **Design** button in the left hand panel) to check, investigate and correct all DRC errors. A completed PCB design should have no DRC errors;

28) Check that the Gerbers to be generated are from the correct version of the PCB layout. It may have changed as a result of the above checks so always regenerate the Gerbers from the latest version unless there is a specific reason to use them from an earlier version. Putting a version number on the PCB helps but is not perfect as this has to be updated manually anyway;

29) Download and check items (6) to (24) in the Gerbers in either the EasyEDA Gerber Viewer:

<https://gerber-viewer.easyeda.com/>

or using a 3rd party Gerber viewer such as the free and open source gerbv:

<http://gerbv.sf.net/>
<http://flatcam.org/>
<http://kicad-pcb.org/>
<http://www.gerber-viewer.com/>

30) Check the order options such as number of boards, copper finish, silkscreen colour, solder mask colour, panellisation, any solder paste mask requirement and so on;

31) Lastly, check that the order is being placed with the correct delivery option. The default delivery method is by express courier. This is the most expensive option but avoids the mistake of ordering boards that are needed urgently with a slow delivery method.

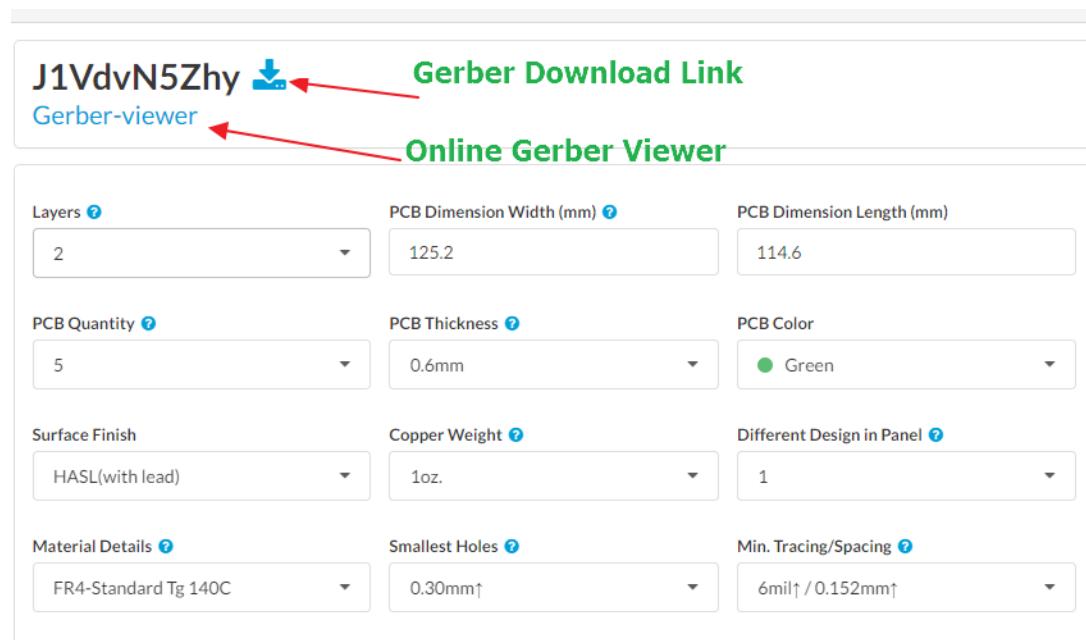
PCB Order from EasyEDA Editor

When you click the `Fabrication Output...` button your order is coming from within the EasyEDA editor environment so you don't need to input information about **Layers**, **Width** and **Height**; EasyEDA fills this information in for you.

On the order form page you will find a real time price. Most of the time this price is the final cost however if, for example, you change the Layers to 4 or 6, you will find the price field changes to **Quote**. If that happens, don't worry: just click the `Save to Cart` button and we will email a quote for the final price to you ASAP.

Note:

- When you add your PCB to the cart, EasyEDA saves a copy of Gerber files at that time.
- If you then change your PCB back in the PCB editor, **EasyEDA does not synchronize your Gerber files to the updated PCB design**.
- The only way is to delete the earlier version of the PCB from your Cart and then add the updated design as a new item.



The screenshot shows the EasyEDA Order Form interface. At the top, there are two prominent links: "Gerber Download Link" with a blue download icon and "Online Gerber Viewer" with a blue camera icon. Below these, there are several input fields for PCB parameters:

Layers	PCB Dimension Width (mm)	PCB Dimension Length (mm)
2	125.2	114.6
PCB Quantity	PCB Thickness	PCB Color
5	0.6mm	Green
Surface Finish	Copper Weight	Different Design in Panel
HASL(with lead)	1oz.	1
Material Details	Smallest Holes	Min. Tracing/Spacing
FR4-Standard Tg 140C	0.30mm†	6mil† / 0.152mm†

PCB Order from Order Link

If you would like to upload your own Gerber files from a third party PCB tool such as Eagle, Pads, or Altium Design, just click on this link <https://easyeda.com/order> to order. This page will let you upload your own Gerber Files.

Drag and Drop

your gerber file here

Or

Add Files

Zip or Rar, 4MB Max

Layers <small>?</small>	PCB Dimension Width (mm) <small>?</small>	PCB Dimension Length (mm)
2	100	100
PCB Quantity <small>?</small>	PCB Thickness <small>?</small>	PCB Color
5	0.6mm	● Green
Surface Finish	Copper Weight <small>?</small>	Different Design in Panel <small>?</small>
HASL(with lead)	1oz.	1
Material Details <small>?</small>	Smallest Holes <small>?</small>	Min. Tracing/Spacing <small>?</small>
FR4-Standard Tg 140C	0.30mm↑	6mil↑ / 0.152mm↑

PCB Capabilities

Number of Copper Layers*	1-16
PCB Material	FR-4, FR4-Tg, FR4-High Tg
Available Solder Mask Colors	Green, Red, Yellow, Blue, White, Black
Silk Screen Colors	White, Black (For White Solder Mask only)
Minimum Quantity	5
Minimum dimensions*	0.4cm x 0.4cm
Maximum dimensions*	100cm x 100cm

- If your PCB requires more than the default maximum of 6 layers (up to a maximum of 16) or larger dimensions, then please contact us before placing your order
- If your PCB dimensions are bigger than 45cm * 45cm, it may add some additional cost

Manufacturing Specifications:

Item	Specs	
	Unit: mm	Unit: mil
Available Board Thickness	0.4, 0.6 (except 4 layer), 0.8, 1.0, 1.2, 1.6, 2.0	15.7, 23.6 (except 4 layer), 31.5, 39.4, 47.2, 63.0, 78.7
Thickness Tolerance	(t >= 1.0) ± 10%	(t >= 39.4) ± 10%
Thickness Tolerance	(t < 1.0) ± 0.1%	(t < 39.4) ± 0.1%
Insulation Layer Thickness	0.075 - 5.0	2.95 - 196.85
Minimum trace width	0.089	3.5
Minimum inner trace width	0.127	5
Minimum trace/vias/pads space	0.102	4
Minimum inner trace/vias/pads space	0.102	4
Minimum silkscreen width	0.1524	6
Minimum silkscreen text size	0.8128	32
Outer Layer Copper Thickness	> 0.03	> 1.18

Drilled Hole Diameter (Mechanical)	0.3 - 6.35	11.81 - 250.00
Drilled Hole Diameter (Laser)	0.2 - 0.3	7.87 - 11.81
Diameter Tolerance (Mechanical)	±0.08	± 3.148
Solder Mask Bridges	0.1	3.94
Circuit to edge	≥0.3	≥11.8
Slot	≥0.6	≥23.6
Slot Tolerance(Mechanical)	±0.15	±6
Aspect Ratio	8:1	
Solder Mask Type	Photosensitive ink	

If you have any special PCB requirements, please contact us before placing your order.

Price

All Prices stated are FOB Shenzhen. This does not include transportation costs which shall be borne by the customer.

Manufacturing Price

Price is dependent on many factors, such as the quantity of PCBs you order, PCB Color, Surface Finish, PCB Thickness, PCB Dimensions, Hole size etc.

EasyEDA uses a group buy business model and we are sure it will be hard to find a better PCB supplier than EasyEDA offering the same price and quality.

EasyEDA needs 2~4 days to manufacture the PCBs after you submit payment.

Shipping Costs

Method	note	Price	Service
Air Mail	Delivery Time: 8-35 days. Most of our users receive their PCBs in two weeks.	From \$6	Usually http://www.singpost.com/
Express	Delivery Time: 3-7 days	From \$24	Usually delivered by http://www.DHL.com/

Note: The shipping cost is estimated. EasyEDA will always try to find the best shipping option. If you are in some [Remote Areas](#), we will ask you to pay for more or change to some other express service such as Fedex, UPS. Sometimes, we will use [Hongkongpost](#) for delivery by Airmail.

File Name

If your Gerber file names are good, this will save us a lot of time in checking your design. There are many different PCB design software packages so there are many variations of Gerber file names and filename extensions.

Gerber Type

File Type	
.TXT, .DRL, .dri, .drd	Drill holes
.GML, .GKO, .GML, outline EDGE_CUTS.GBR	Outline
.GTP, F.PASTE.GBR	Solder Paste Top
.GBP_B_PASTE.GBR	Solder Paste Bottom
.GTS, .stc, .smt, F.MASK.GBR	Solder Mask Top
.GBS, .sts, .smb_B.MASK.GBR	Solder Mask Bottom
.GTO, .sst, F.MASK.GBR	Silkscreen Top
.GBO, .ssb, B.SILKS.GBR	Silkscreen Bottom
.GTL, .cmp, .top, F.CU.GBR	Top Layer
.GBL, .sol, .bot, B.CU.GBR	Bottom Layer
Inner layer in 4 layers PCB:	
.GL2	Layer 2
.GL3	Layer 3

If you don't know how to map your files, don't worry about changing the file names and please contact support for help.

We encourage you to use our free online [gerber viewer](#) to check your gerber files before placing an order.

E-Test

All PCBs undergo a 100% AOI (Automated Optical Inspection) to make sure that all tracks and pads are connected. In addition to this the PCBs can be tested by a flying probe to make sure that all vias are connected, because this is not visible by the AOI. Single layer PCBs do not require this test because there are no vias but boards with 2 layers and above will always be 100% tested with a flying probe.

Payment

We accept the PayPal, Credit Card and Wire Transfers.

PayPal and Credit Card

We use Paypal as our payment; it is safe and easy. If you don't have a Paypal account, you can still use Paypal to pay with a debit or credit card.

The screenshot shows a payment interface with a sidebar labeled "summary". The sidebar displays the amount £18.60 and the total £48.60 GBP. The main area is titled "Choose a way to pay" and offers two options: "Pay with my PayPal account" and "Pay with a debit or credit card". A red arrow points to the "Pay with a debit or credit card" option. Below it, a sub-section says "(Optional) Join PayPal for faster future checkout". The payment form includes fields for Country (set to United States), Card number, Payment types (listing VISA, MasterCard, Discover, and American Express), Expiration date (mm/yy), and CSC (Card Security Code). There is also a link "What is this?".

Wire Transfers

Wire Transfers can only be used on orders with a grand total (subtotal plus all additions and deductions but excluding shipping fees) of at least \$600. For orders > \$2000, payment by Wire Transfer is preferred. In this circumstance, 3.5% extra discount will be applied for the grand total (subtotal plus all additions and deductions but excluding shipping fees). Wire Transfer payments usually take 3-5 business days to clear. We will not ship your order until your payment is verified by our bank. Please send a copy of the Wire Transfer receipt to our customer service because although it is not sufficient to release an order, it will help us to push the delivery date.

Customs, Duties and Taxes

You should expect to pay any amount charged by the government in your respective country. This includes but is not limited to: duties, taxes and any extra fees charged by the courier company. We will not be held responsible for any extra charges once the original package has been shipped. If the customer refuses to pay these extra charges, the return shipping and any additional fees will be taken out of the cost of the order, with any remaining funds being refunded to the customer. Customs are quite different in each country. Please include information about particular Customs requirements as necessary, while you are placing your order: we will support you as much as possible.

Warranty and Return

For your first order for a PCB laid out in **EasyEDA**, we have the top return policy on the planet! If you don't like them, just send an email to

support@easyeda.com, no reason needed. We will provide your full money back - including product + shipping costs - in one working day.

For subsequent orders, because you now know the quality level of our PCBs, if you are not satisfied with a product you bought from us for whatever reason, you just need to email us some pictures of the product and explain why you are not happy with it. We will then refund the full money of the product. Shipping fees will only be refunded if the return is a result of a shipping error on our part.

PCB Parameter Description

PCB Dimension

EasyEDA supports the design of PCBs up to 50cm * 50cm however we suggest our customers try to limit the design to a PCB size of no larger than 45cm * 45cm. We may have to charge more for PCB sizes greater than this because they are harder to fabricate and need a bigger box with more packaging to protect them in shipping.

- If the size is smaller than 2cm, you need to panelize them to big size, or we can't help you to do the E-test.

PCB Quantity

EasyEDA uses a group buy model, so the price is very low, but the minimum number of PCBs should be 5 pcs. So for example, if you need 2 pcs, you need to order 5 pcs and similarly, if you need 7 pcs you need to order 10 pcs.

PCB Thickness

EasyEDA PCB provides 0.4mm, 0.6mm, 0.8mm, 1.0mm, 1.2mm, 1.6mm, 2.0mm thickness to choose from. If you need a 2.5mm or 3mm thickness board, please ask us for a quote via email. Please note that not all thicknesses will be available for all numbers of layers; for example no PCB house can fabricate a 32 layer PCB with a 0.4mm thickness!

PCB Stack up

2 Layer PCB stackup 1Oz

2 Layers 1.6mm (0.062 inch) Green for 1 Oz

Layer stack up	Layer	Thickness (mm)
	Top Solder Mask	0.01
	Top Layer	0.035
	Core	1.5
	Bottom Layer	0.035
	Bottom Solder Mask	0.01

2 Layer PCB stackup 2Oz

2 Layers 1.6mm (0.062 inch) Green for 2 Oz

Layer stack up	Layer	Thickness (mm)
	Top Solder Mask	0.01
	Top Layer	0.070
	Core	1.5
	Bottom Layer	0.070
	Bottom Solder Mask	0.01

4 Layer PCB stackup 1Oz

4 Layers 1.6mm (0.062 inch) Green for 1 Oz

Layer stack up	Layer	Thickness (mm)
	Top Solder Mask	0.01
	Top Layer	0.035
	Prepreg	0.18
	Inner 1	0.017
	Core	1.12
	Inner 2	0.017
	Prepreg	0.18
	Bottom Layer	0.035
	Bottom Solder Mask	0.01

4 Layer PCB stackup 2Oz

4 Layers 1.6mm (0.062 inch) Green for 2 Oz

Layer stack up	Layer	Thickness (mm)
	Top Solder Mask	0.01
	Top Layer	0.07
	Prepreg	0.18
	Inner 1	0.017
	Core	1.12
	Inner 2	0.017
	Prepreg	0.18
	Bottom Layer	0.07
	Bottom Solder Mask	0.01

6 Layer PCB stackup 1Oz

6 Layers 1.6mm (0.062 inch) Green for 1 Oz

Layer stack up	Layer	Thickness (mm)
	Top Solder Mask	0.01
	Top Layer	0.035
	Prepreg	0.1
	Inner 1	0.017
	Core	0.57s
	Inner 2	0.017
	Prepreg	0.1
	Inner 3	0.017
	Core	0.57
	Inner 4	0.017
	Prepreg	0.1
	Bottom Layer	0.035
	Bottom Solder Mask	0.01

For other numbers of layers or for a different layer stack up, please email us before placing your order.

Copper Weight

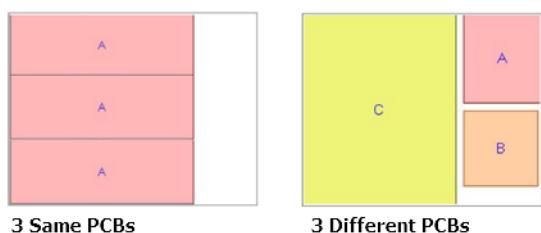
You can select 1oz, 2oz in our order page.

1. For 1Oz, the track width and clearance can be 3mil.
2. For 2Oz, please make sure the clearance is bigger than 8mil.

Different Design in Panel

Some customers would like to merge more than 1 PCB in the same Gerber. We know you want to save money but this may make it **hard to cut the board outline** and more importantly this will take a lot more time to pick up and package the PCB. Although by doing this, you just have the one order, this complicates the fabrication of the panel and separation of the individual PCBs, so we will usually charge more for this. Similarly, using holes or slots as break off sections between boards are treated the same way as putting more than one design on a panel, each with its' own board outline.

Note: This additional charge only applies if the PCBs on a panel are different. Boards such as in the left hand image below will not incur an additional charge because they are easy to pick up but boards such as in the right hand image would incur an additional charge.

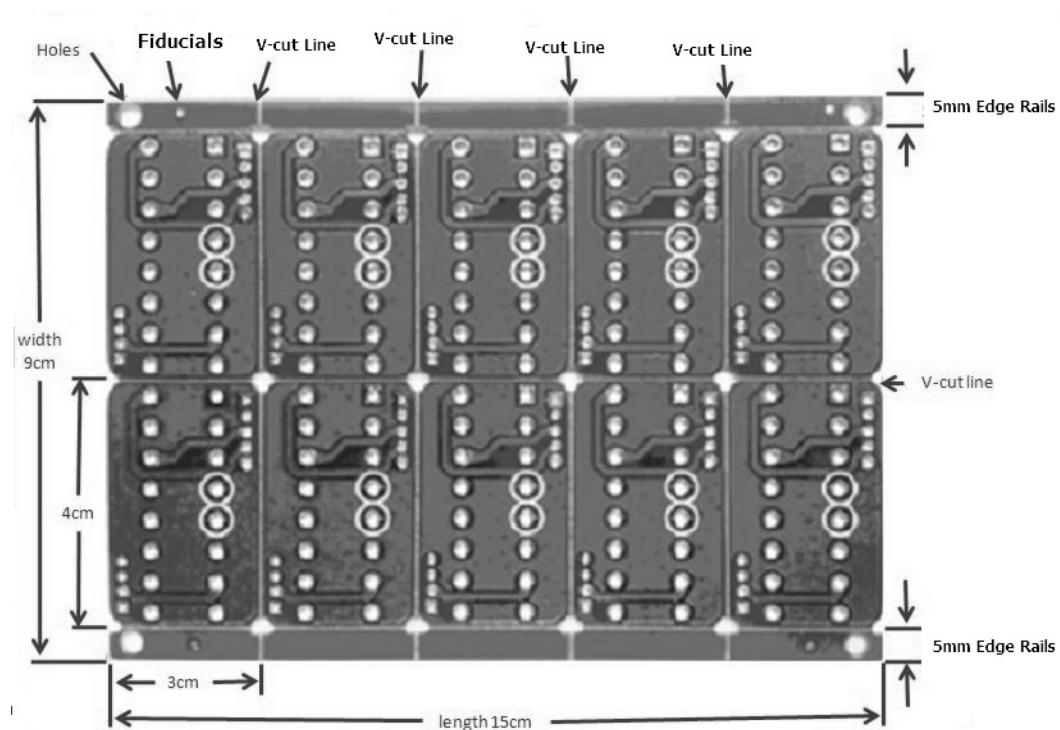


If you are just in the prototype phase and want to save money, then you can use the following trick. Place your different designs all inside one overall board outline and use lines drawn in the silk layer to mark out the separate the PCBs. Then, when you receive the PCB, carefully cut them apart yourself (we recommend you do this before you assemble the PCBs!). Like the PCB shown below, the yellow lines are drawn in the silk layers showing how you can merge 3 different PCBs in one gerber without incurring any extra costs.

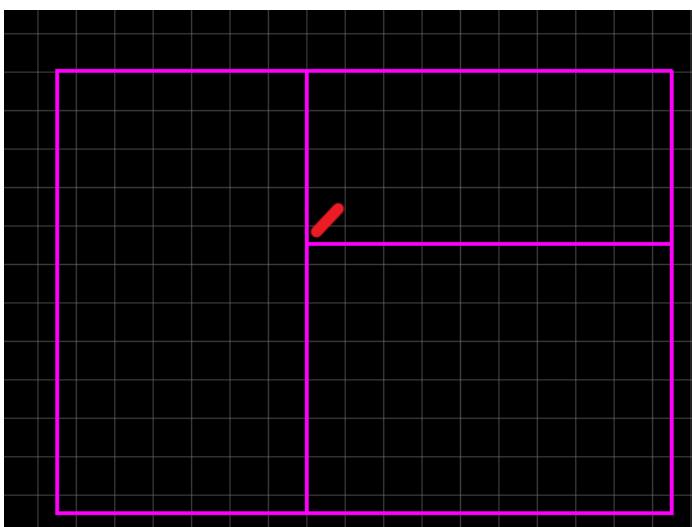


V-cut/V-Groove

This will help you to build a PCB Array to save time by making it faster to solder the PCB in SMT.



1. The V-cut line should be the same as the outline of the Sub-PCB. That is to say, there is zero space between the sub-boards.
2. The PCB panel needs to be larger than $8 * 8\text{cm}$.
3. The V-cut line must cross the whole panel or else the factory can't add a V-groove on the PCB because the milling cutter may destroy any sub-PCBs on the V-cut line. They also cannot stop part way across the panel. The V-cut lines shown below are not acceptable:



Material Details

EasyEDA supports FR4-Standard Tg 140C, FR4-Tg 150C, FR4-High Tg 170C. The FR4 TG's lead time may be more than 6 days. 90% of EasyEDA orders use FR4-Standard. For more information about this, please check the [FR4 Material pdf](#)

Smallest Holes Diameter

0.3mm for mechanical drill, 0.2mm for the laser drill. To save money, please use a minimum drill diameter of 0.3mm.

Ring

The width of the ring around vias or pads should be wider than 6mil/0.15mm.



Min. Tracing/Spacing

We support down to 4mil but to save money, please use 6mil.



Grid size

Make sure the Grid filled size is bigger than 8mil/8mil (track/space), if less than that, we will change it to 8mil/8mil .



Impedance Control

We support 5% and 10% precision. Please add enough information about your impedance control requirements to help us to fabricate your PCB.

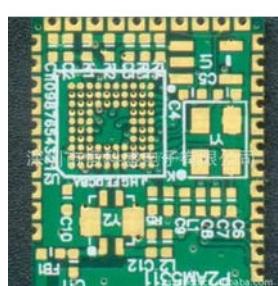
Gold Fingers

If you wish to built PCBs that plug directly into edge connectors, such as memory cards, please choose **Gold Fingers** as shown in the image below:



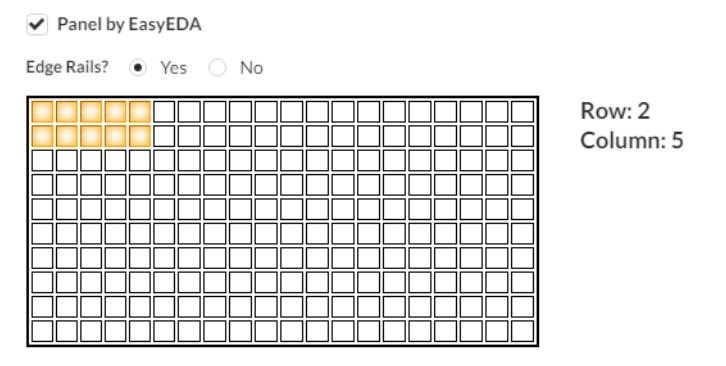
Half-cut/Castellated Holes

If you need to build some PCBs as shown in the image below, please choose **Half-cut** holes.



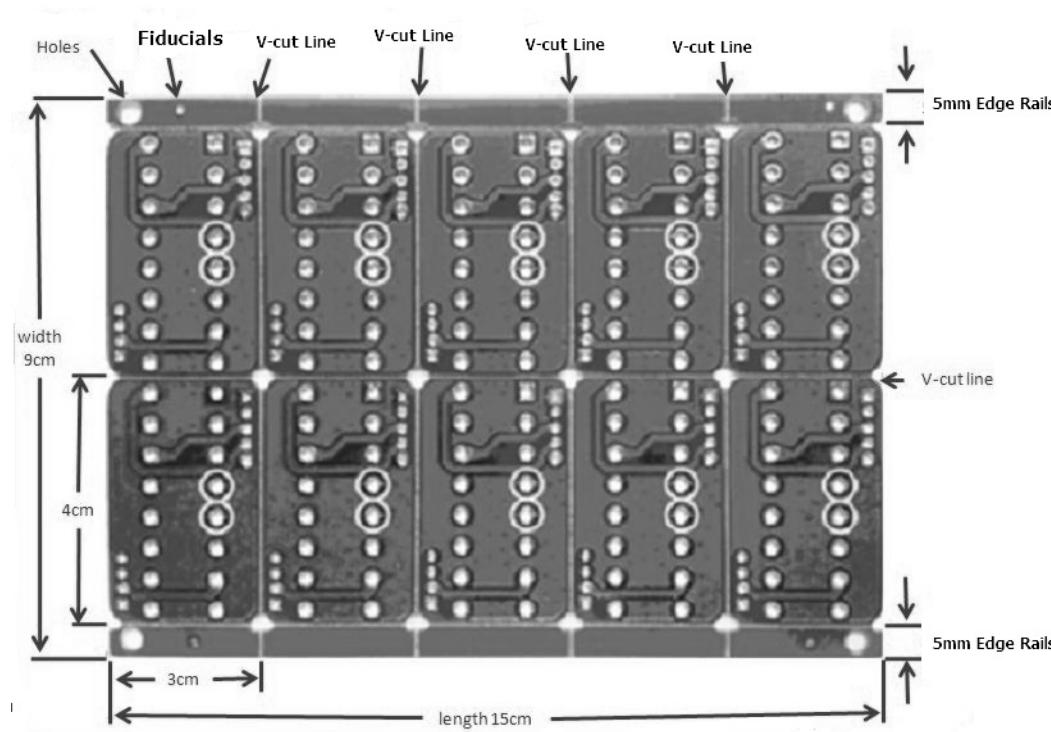
Panel by EasyEDA

When your Gerber is for just one design and you need EasyEDA to help you to duplicate many copies onto one panel, you can use this option. You can drag across the rectangles to select then click on the panel to set how many rows and columns as shown in the image below:



For this image, we will merge 10 small PCBs to 1 big panel. If you order 5 pcs, then we will send 5 big panels to you, each with 10 PCBs on it so you will end up with 50 small PCBs.

If you select the **Edge Rails**, we will add a 5mm board edge as shown in the image below: This is 2 rows and 5 cols panel PCB.

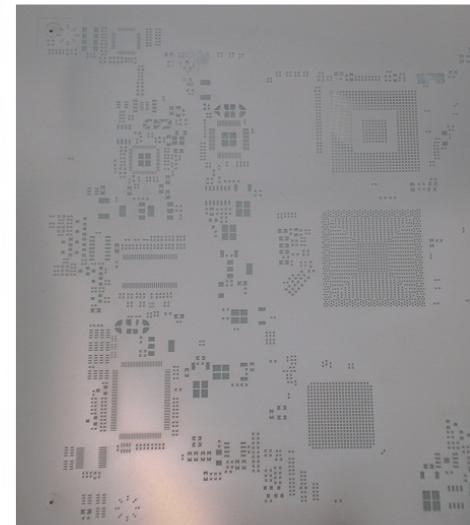
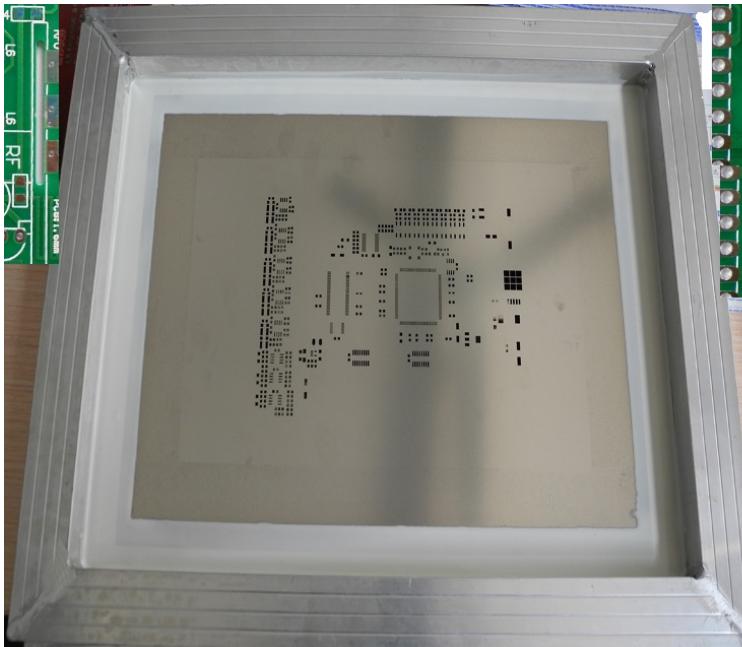


Note:

1. Make sure the single PCB size is bigger than 2cm x 2cm, or we will charge \$20 for the v-cut cost. Small PCB is hard to v-cut.
2. Make sure the board outline is simple, for complex board outlines, you need to panelize by yourself. Or you can pay us \$15 to do that, we will send the panelized gerber to you to confirm.

Stencil Parameter Description

A Stencil can help you to solder the PCB quickly. For efficient and reliable SMT assembly a Stencil is a must. EasyEDA can provide the option of NON-FRAMEWORK (or frameless) and FRAMEWORK stencils. The right hand image below shows a frameless stencil. Frameless stencils are cheaper and lower weight(0.2Kg) so they can help to reduce the shipping cost.



Order FAQ

How to check the Shipping Cost?

We can ship PCBs to any country. The shipping cost depends on the weight of the boards, country, and shipping method. Before you pay, you can see the shipping cost and select the shipping method. So you just need to add it to your cart and then you will see some options like in the image below:



How to order lots of PCB together?

EasyEDA allows you to order many different PCBs together, just add the PCB to cart one by one , at last to pay for together.

How to remove the customer ID on the PCB?

EasyEDA uses a group buy model to save the cost of production, for picking up your PCB easier, we need to add a very small string to your PCB, the string might be under some IC, and if you solder the PCB, the string will be hidden. If you don't like it, there are two ways.

1. Email us, we will fabricate your PCB in another way, but the cost will be higher.
2. When you order, you can use <https://easyeda.com/Doc/Tutorial/PCBOrderFAQ#Panel-by-EasyEDA> and keep the edge, we will add the string at the dirty edge, so there is no string on your product PCB.

EasyEDA API Plug

Before reading this capture, please check [Open EasyEDA File Format](#) first.

Why Need API

After route the PCB, you found out that you need to enlarge all tracks size a bit little, How? After route the PCB, you found out that all Vias' hole size is too small, How to fix this? How to create a board outline using code? EasyEDA API will let you control your designs in an easy way.

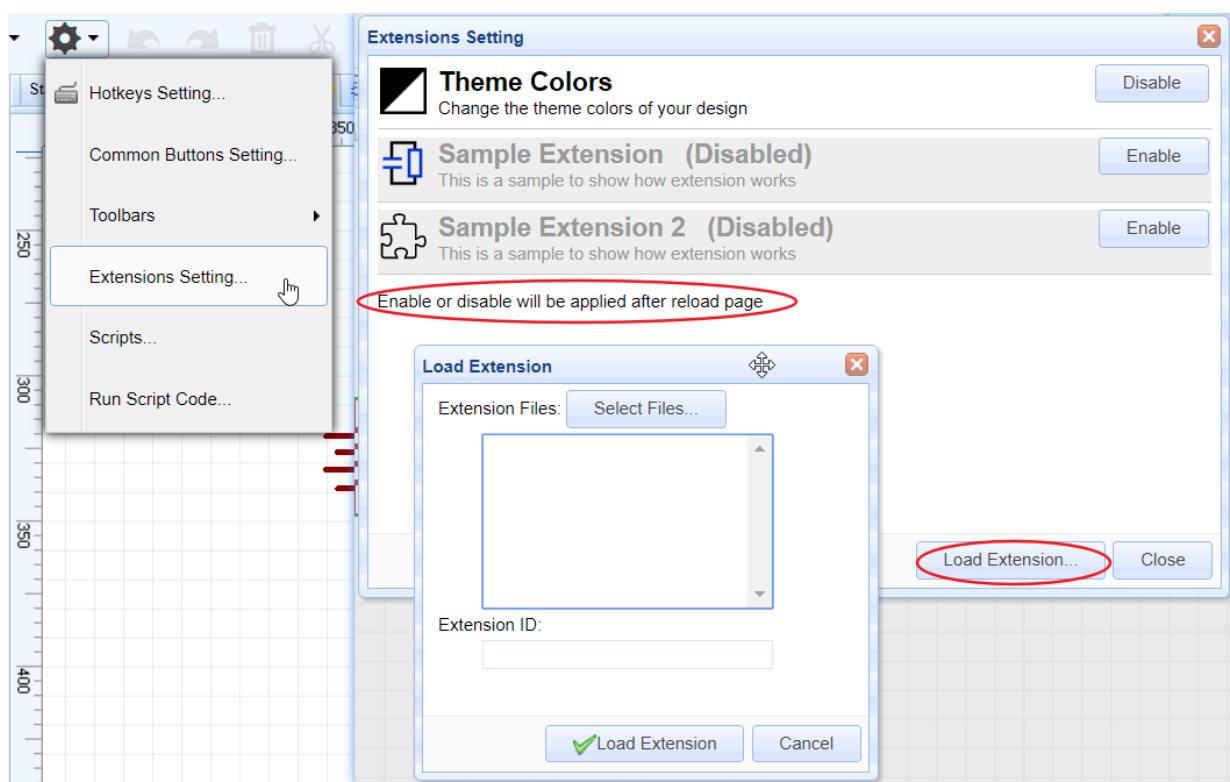
How to use API

How to find the plug entrance

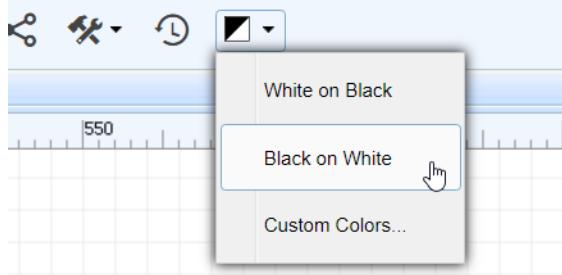
You can click **Config Icon > Extensions Setting** on the top toolbar image as below.

Extensions Setting

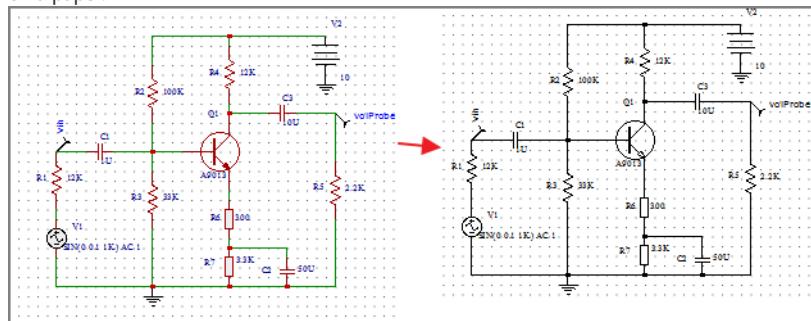
You can enable or disable the default extensions, after enable, please **reload** the EasyEDA editor. We will give you a file about how to create an extensions soon.



If you enable the **Theme Colors** Extension, you will find a button on the tool bar like bellow image:



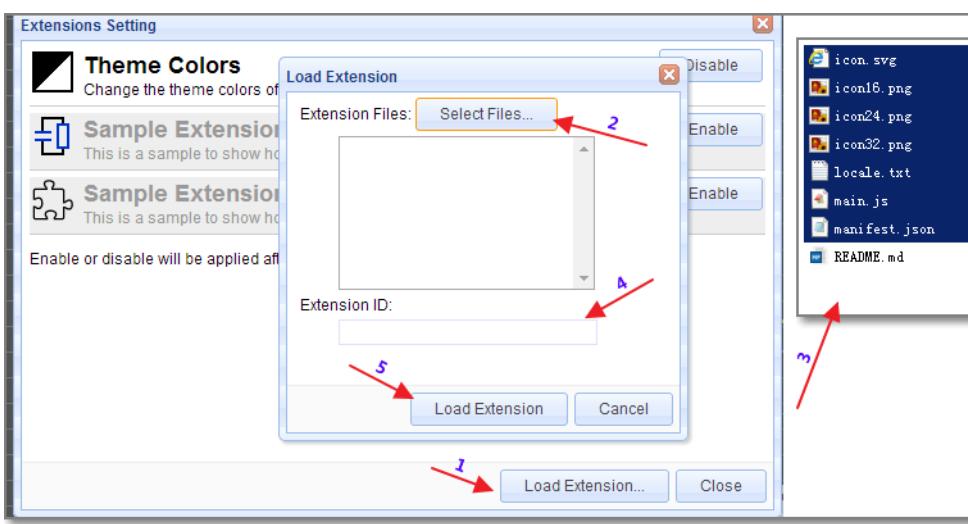
If you click the **Black On White**, you will find your schematic changes like bellow image, this is useful when you would like to print your design on a paper.



You can check our **github** codes of this API via <https://github.com/dillonHe/EasyEDA-Documents/tree/master/API/example/theme>, check the **manifest.json** and **main.js** out, you will find out how to create an extension.

How to install an extension

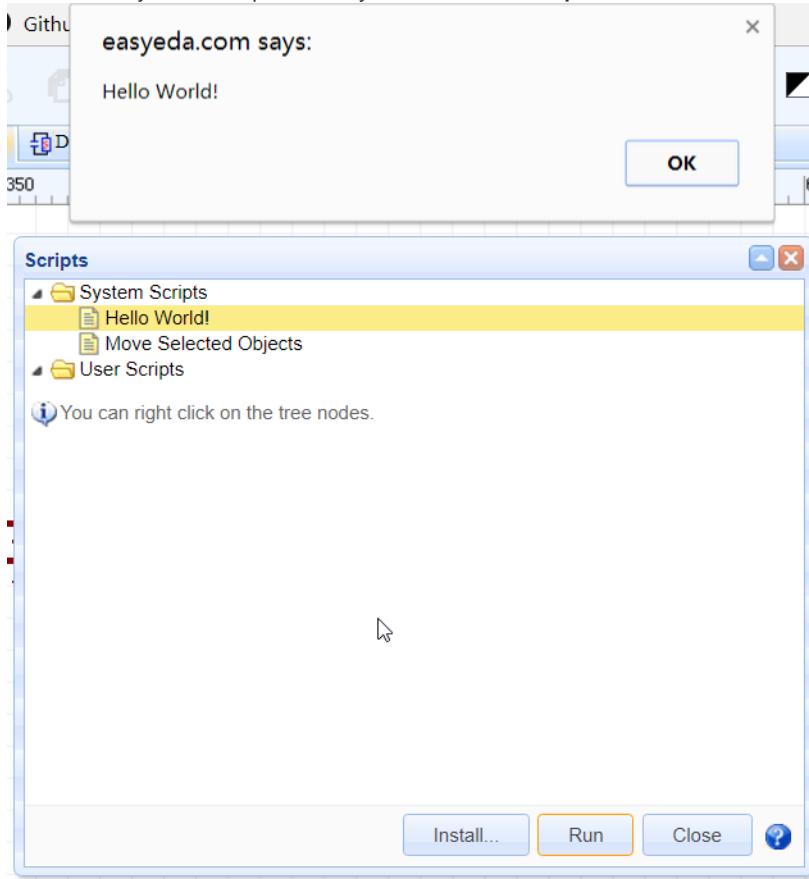
1. Click the Load Extension button
2. Click the select file button
3. Select All the files.
4. Type a name
5. Click the load button.
6. Close EasyEDA editor and open it again.



Scripts

If you just need some simple functions, you don't need to create an extension. You just need to create a single Javascript file and keep it in this list.

1. You can select the `Hello World`, then click the `Run` button, the response as below image.
2. You can select some items, then try `Move Selected Objects`.
3. You can install your own scripts, then they will show on **User Scripts**.



Run Script code

In some case, you just need to run the function one time, such as create a user define board outline in codes, changing the Track width, change the hole size etc. You can use this way.

Script

Script Content: [Load from js file...](#) (UTF-8 encoded js file)

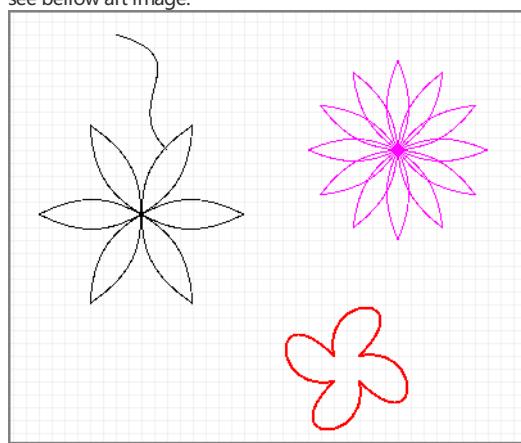
```
testInsertShape();

function testInsertShape() {
    api('insertShape', [
        {
            shapeTypeName: "path",
            fillColor: "none",
            pathString: "M520 500 C480 460 550 430 480 410",
            strokeColor: "#000000",
            strokeStyle: 0,
            strokeWidth: "1"
        }
    ]);
    api('insertShape', [
        {
            shapeTypeName: "path",
            fillColor: "none",
            pathString: shapeLotusFlower(500,550,3,80,40),
            strokeColor: "#000000",
            strokeStyle: 0,
            strokeWidth: "1"
        }
    ]);
}
```

Run Close

example 1 Art

You can open an empty schematic and copy [this example javascript codes](#) to the text box to run a test. After clicking the Run button, you will see bellow art image.

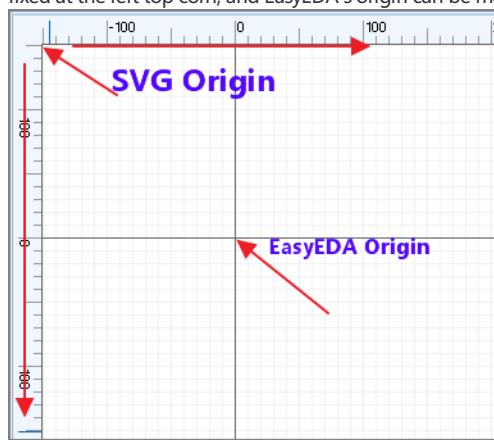


example 2 Change track width and via hole size

You can open a PCB and copy [this example javascript codes](#) to the text box to run a test. After that, All tracks will be 10mil.

EasyEDA Coordinate System

EasyEDA's editor is based **SVG**, **SVG viewport**, (Coordinates increase **left-to-right** and **top-to-bottom**, the same as EasyEDA). But SVG's origin is fixed at the left top corn, and EasyEDA's origin can be modified at the any place.



Be careful this, they are different from **Cartesian coordinate system**

Unit

There are two kinds of unit in our editor, **SVG Canvas unit** and **real world EasyEDA unit**. **SVG Canvas unit** is **Pixel**. The **real world EasyEDA unit** in schematic is also **Pixel**, but in PCB, there are **mm**, **mil** and **inch**. We use bellow map to convert Canvas to real world. - 1 pixel = 10 mil - 1 pixel = 0.254mm - 1 pixel = 0.01inch There are API for these convert.

```
//mm2pixel: convert 10mm to pixel
var result = api('unitConvert', {type:'mm2pixel',value:10});
```

```
//mil2pixel: convert 10mil to pixel
var result = api('unitConvert', {type:'mil2pixel', value:10});
```

There are other convert method, such as `inch2pixel`, `pixel2mm`, `pixel2mil` and `pixel2inch`.

All EasyEDA's value is based pixel, if you can keep in mind that 1 pixel equal 10mil or 0.254 mm, you don't need to use any convert function. For example, if you want to change a Track to 20mil, so you just need to use 2.

API List

Get EasyEDA Source

1. get EasyEDA JSON objects, type is `json`, you can check [PCB Json object](#) out to know more.

```
var result = api('getSource', {type:'json'});
```

2. get [EasyEDA compress string](#), EasyEDA save this string to our database, it is a bit little hard to read and understand, but it is small in size. EasyEDA save this string to our database.

```
var result = api('getSource', {type:'compress'});
```

3. Get SVG string

```
var result = api('getSource', {type:'svg'});
```

Check the [Get EasyEDA source example codes](#).

Apply Source

After you can use your codes to hack EasyEDA's source, then you need to apply the source to EasyEDA's editor. You can

1. Apply as compress string

```
//will open a new editor and convert compressStr to EasyEDA file.
api('applySource', {source:'compressStr', createNew: true});
```

2. Apply as Json object

```
//will modify the active file and convert json object to EasyEDA file.
api('applySource', {source: json, createNew: !true});
```

Check the [Apply Source example codes](#).

Get Shape

If you want to get an EasyEDA json object by `id`, you can try to use bellow code.

```
var obj = api('getShape', {id:'gge13'})
```

Delete Shapes

Removing shapes by follow code

Update Shape

If you want to modify an EasyEDA object, you can use this API.

```
//Change the net to GND and the shape to ELLIPSE
api('updateShape', {
  "shapeType": "PAD",
  "jsonCache": {
    "gId": "gge5",
    "net": "GND"
  },
  "shape": "ELLIPSE"
});
```

`shapeType` and `gId` are must provided.

1. Schematic

```
shapeType, schlib, rect, polyline, polygon, wire, bus, image, circle, ellipse, line, path, arc, annotation, junction, netlabel, busentry, arrowhead, noconnectflag, pin, netflag
```

2. PCB

```
shapeType, FOOTPRINT, TRACK, COPPERAREA, SOLIDREGION, RECT, CIRCLE, TEXT, ARC, DIMENSION, PAD, VIA, HOLE
```

Create Shape

If you want to create EasyEDA shape by codes, you can try. We will provide more information about this API soon, now we just provide examples. You will find out how to do.

```
/** with shortUrl
 * @example
 * api('createShape', {shapeType:'schlib', shortUrl:'nxlVIGgQ0', from:'system', title:'556_DIL14', x:400, y:300});
 * api('createShape', {shapeType:'FOOTPRINT', shortUrl:'Rrkew060i', from:'system', title:'ARDUINO_PRO_MINI', x:400, y:300});
```

```

/*
/** with jsonCache object
 * @example
* api('createShape', {
*   "shapeType": "PAD",
*   "jsonCache": {
*     "gId": "gge5",
*     "layerid": "11",
*     "shape": "ELLIPSE",
*     "x": 382,
*     "y": 208,
*     "net": "",
*     "width": 6,
*     "height": 6,
*     "number": "1",
*     "holeR": 1.8,
*     "pointArr": [],
*     "rotation": "0"
*   }
* });
*
* @example
* api('createShape', {
*   "shapeType": "polygon",
*   "stroke": "#000000",
*   "stroke-width": "1",
*   "stroke-style": "dashed",
*   "fill": "none",
*   "points": [
*     {"x": 390, "y": 580},
*     {"x": 450, "y": 450},
*     {"x": 520, "y": 580},
*     {"x": 610, "y": 490}
*   ]
* });
*
* @example
* api('createShape', {
*   "shapeType": "arrowhead",
*   "x": 300,
*   "y": 300,
*   "color": "#339933",
*   "size": "3",
*   "rotation": 0
* });
*
* @example
* var ts = ["no_connect_flag", "arrowhead", "busentry", "netLabel_GNd", "netLabel_GnD", "netLabel_gnD", "netLabel_Bar", "netLabel_bar", "netLabel_bar_bar"];
* for(var i=0;i<ts.length;i++){
*   api('createShape', {
*     "shapeType": ts[i],
*     "x": 300 + i%5*50,
*     "y": 300 + (i/5|0)*50
*   });
* }
*/
/** with cached or pre-defined libs
 * @example
* api('createShape', {"shapeType": "pcplib", from:'GeneralPackages', title:'C0402', x:400, y:300});
* @example
* api('createShape', {"shapeType": "schlib", from:'EasyEDALibs', title:'HDR2X2', x:400, y:300});
*
*/
*/
* @example 4
* api('createShape', {
*   "shapeType": "schlib",
*   "gId": "gge6",
*   "head": {},
*   "itemOrder": [],
*   "annotation": {
*     "gge8": api('createShape', 'annotation', {}),
*     "gge9": api('createShape', 'annotation', {})
*   },
*   "pin": {
*     "gge11": api('createShape', 'pin', {}),
*     "gge14": api('createShape', 'pin', {})
*   },
*   "polyline": {
*     "gge10": api('createShape', 'polyline', {}),
*     "gge12": api('createShape', 'polyline', {})
*   }
* });
*
* @example 5
* api('createShape', {
*   "shapeType": "schlib",
*   "gId": "gge6",
*   "head": {},
*   "children": [

```

```

        *     api('createShape', 'polyline', {}),
        *     api('createShape', 'polyline', {}),
        *     api('createShape', 'pin', {}),
        *     api('createShape', 'pin', {}),
        *     api('createShape', 'annotation', {}),
        *     api('createShape', 'annotation', {})
    *   ]
* });
*
* @example 6
* api('createShape', {
*   "shapeType": "schlib",
*   "gId": "gge6",
*   "head": {},
*   "children": api('createShape', [
*     ['polyline', {}],
*     ['polyline', {}],
*     ['pin', {}],
*     ['pin', {}],
*     ['annotation', {}],
*     ['annotation', {}]
*   ])
* });
*/

```

UI

If you want to create an extension, not just a run one time script, maybe need toolbar button. You can check the [example](#) before you read.

Create Toolbar Button

```

//@example create a button
api('createToolbarButton', {
  icon:'extensions/theme/icon.svg',
  title:'Theme Colors...',
  fordoctype:'sch,schlib',
  cmd:"extension-theme-setting"
});

* @example toolbar with menu
* api('createToolbarButton', {
*   icon:'extensions/theme/icon.svg',
*   title:'Theme Colors...',
*   fordoctype:'sch,schlib',
*   "menu" : [
*     {"text":"White on Black", "cmd":"extension-theme-WhiteOnBlack"},
*     {"text":"Black on White", "cmd":"extension-theme-BlackOnWhite"},
*     {"text":"Custom Colors...", "cmd":"extension-theme-setting"}
*   ]
* });

```

Create Extension Menu

```

/**
* @example
* api('createExtensionMenu', [
*   {
*     "text":"Theme Colors...",
*     "fordoctype": "sch,schlib",
*     "cmd": "extension-theme-white"
*   }
* ]);
*/

```

Create Dialog

check the [example](#)

Command List

Clone

```

// clone gge2 gge3 and return their new ids.
var newIds = api('clone', {ids:["gge2","gge3"]})

```

Delete

```

api('delete', {ids:["gge2","gge3"]});

```

Rotate

```

// rotate ids to 90 degree
api('rotate', {ids:["gge2","gge3"],degree:90});

```

Rotate Left

```
//anticlockwise  
api('rotate_left', {ids:["gge2","gge3"]});
```

Rotate Right

```
//clockwise  
api('rotate_right', {ids:["gge2","gge3"]});
```

Flip

```
api('fliph', {ids:["gge2","gge3"]});
```

Flipv

```
api('flipv', {ids:["gge2","gge3"]});
```

Align Left

```
api('align_left', {ids:["gge2","gge3"]});
```

Align Right

```
api('align_right', {ids:["gge2","gge3"]});
```

Align Top

```
api('align_top', {ids:["gge2","gge3"]});
```

Align Bottom

```
api('align_bottom', {ids:["gge2","gge3"]});
```

Selection

Change or get selection states of EasyEDA objects in editor.

Select

```
// gge2 and gge3 will be marked as selected.  
api('select', {ids:["gge2", "gge3"]});
```

Select None

```
//no objects will be selected.  
api('selectNone');
```

Get Selected Ids

```
var ids = api('getSelectedIds');
```

Move

You can use [Update Shape](#) to change the shapes position, but the Move method is better in this case.

Move Objects

Move shapes in relative coordinates, like move the shapes in arrow keys.

```
//Move gge2 and gge3 from left to right in 20pixel or 200mil step  
//from top to bottom in 20pixel or 200mil step.  
api('moveObjs', {objs:[{gId:"gge2"},{gId:"gge3"}], addX: 20, addY: 20});  
  
//Move gge2 and gge3 from right to left in 20pixel or 200mil step  
api('moveObjs', {objs:["gge2","gge3"], addX:-20});  
  
//Move selected objects from left to right in 20pixel or 200mil step  
api('moveObjs', {addX:20});
```

Move Objects To

How to move a VIA or junction to position `{x:'10mil', y:'10mil'}`? Move shapes to absolute coordinates.

```
//Move gge2 and gge3 to Canvas postion 20,20, the real coordinates are dedpend the origin.  
api('moveObjsTo', {objs:[{gId:"gge2"},{gId:"gge3"}], x:20, y:20});  
  
//move gge2 and gge3 to 10mm, 10mm coordinates
```

```

api('moveObjTo', {objs:["gge2","gge3"], x: api('coordConvert', {type:'real2canvas',x: '10mm'}), y: api('coordConvert', {type:'real2car
//Move selected objects to Canvas postion 20,20, the real coordinates are dedpend the origin.
api('moveObjTo', {x:20, y:20});

```

It is very easy to understand to move a PAD, VIA, Junction to absolution coordinates. But what are the effects of moving TRACK, FOOTPRINT, netlabel to some where. Just try to play the codes, you will find out the regular pattern.

SetOriginXY

EasyEDA's canvas origin is 0,0, you can't change it. But the real coordinates can be mapped to any where.

```

//set the real origin point to canvas x = 400, y = 300. X,Y is pixel all the time.
var result = api('setOriginXY', {x:400,y:300});

```

Coordinate Convert

You can use mm or mil or inch as units, but when you apply the Parameters to SVG graph, you must use coordinate convert.

```

//convert the canvas x 400 to real position, the value is depent your units and origin point.
var result = api('coordConvert', {type:'canvas2real',x:400})

//the default units is your canvas units, but you can add a units like 300mm.
//if your PCB's units is mil, then you will get the canvas coordinate 400mil,300mm.
var result = api('coordConvert', {type:'real2canvas',x:400,y:'300mm'});

```

If you set the origin to **0,0**. It is very easy to map the coordinate in your mind, you don't need to use API to convert. the canvas coordinate **100,100** equal the real coordinate **1000mil, 1000mil** or **1inch, 1inch** or **393.7mm, 393.7mm**

Value Convert

How to set the pad's hole size to 20mm? How to set the Track width to 20mil?

```

//the default units is your canvas units, but you can add a units like mm, mil, inch, even pixel.
var result = api('valConvert', {type:'real2canvas',val:400});
result = api('valConvert', {type:'real2canvas',val:'400mm'})

//convert the 400 pixel to real value, the value is depent your units , if the unit is mil, the result should be 4000
//result = api('valConvert', {type:'canvas2real',val:400})

```

If you can keep in mind 1pixel in canvas equal 10mil, so you don't need this API, you can do it in raw way. For example, If you want to update the track size to 20mil, you can do.

```

api('updateShape', {
    "shapeType": "TRACK",
    "jsonCache": {
        "gId": "gge5",
        "strokeWidth": 2
    }
});

```

Or

```

api('updateShape', {
    "shapeType": "TRACK",
    "jsonCache": {
        "gId": "gge5",
        "strokeWidth": api('valConvert', {type:'real2canvas',val:'20mil'})
    }
});

```

Get SVG Arc Path

[SVG Arc path Parameter](#) is very complex, We provide a API to convert human read ARC parameter to SVG path.

```

var result = api('getSvgArcPathByCRA', {cx:0, cy:0, rx:90, ry:90, startAngle:0.1, endAngle:0.7, sweepFlag:1});

```

result should be M89.55037487502231 8.985007498214534A90 90 0 0 1 68.83579685560396 57.97959185139219

Examples

Check [Github example](#)

Enjoy it, if you have any questions, do let us know.