

DocType	Shortcut	Function
PCB	+	Increase Routing Width; Switch to the next signal layer
PCB	*	Cycle switch to the next signal layer
PCB	Delete	Delete selected object; Undo the track when routing
PCB	ALT--	Decrease Snap Size
PCB	ALT++	Increase Snap Size
PCB	CTRL+R	Depend on reference point for copy object repeatedly
PCB	CTRL+L	Open layer manager
PCB	CTRL+Q	Hide/show network text
PCB	SHIFT+M	Remove All Copper Area fill data
PCB	SHIFT+B	Rebuild All Copper Area
PCB	SHIFT+D	Move Object(s) by reference point
PCB	SHIFT+G	Display track length while routing
PCB	SHIFT+W	Show favorite track width while routing
PCB	SHIFT+R	Change routing conflict
PCB	SHIFT+S	Toggle layers which is not active
PCB	SHIFT+Double Click	Delete selected track segment
PCB	CTRL+SHIFT+V	Paste object(s) and keep the prefix, and hide the ratline layer
PCB	CTRL+SHIFT+SPACE	Change routing angle, same as hotkey L
PCB	CTRL+ALT+L	Enable all layers
Footprint	CTRL+SHIFT+ALT+D	Open custom data dialog

Basic Skills

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, and create a new schematic:

File > New > Schematic , and play!

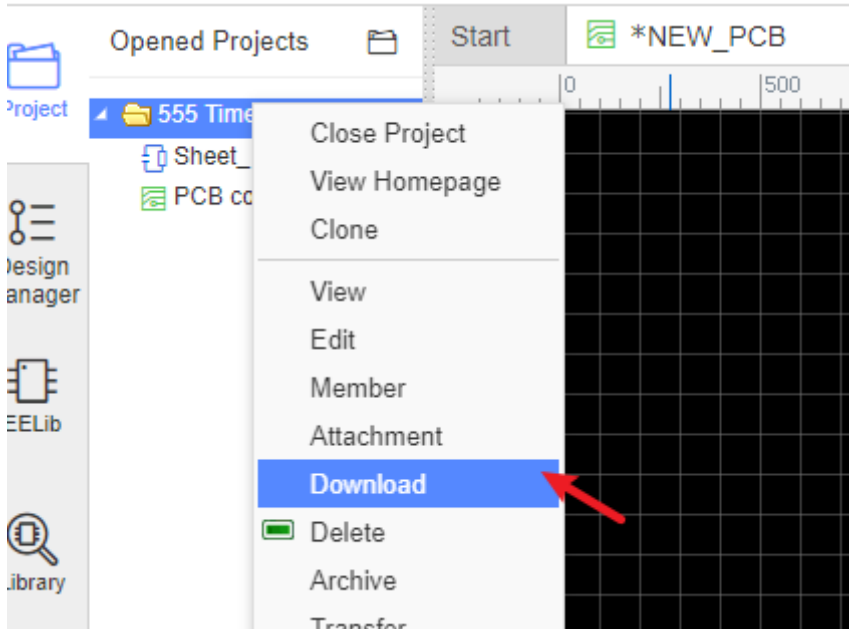
Saving Your Work Locally

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

You can right-click your project folder, and click "Download Project", or export your design as EasyEDA source file via "File > EasyEDA Source".

For more detail you look here: [Export EasyEDA Source](#) section.

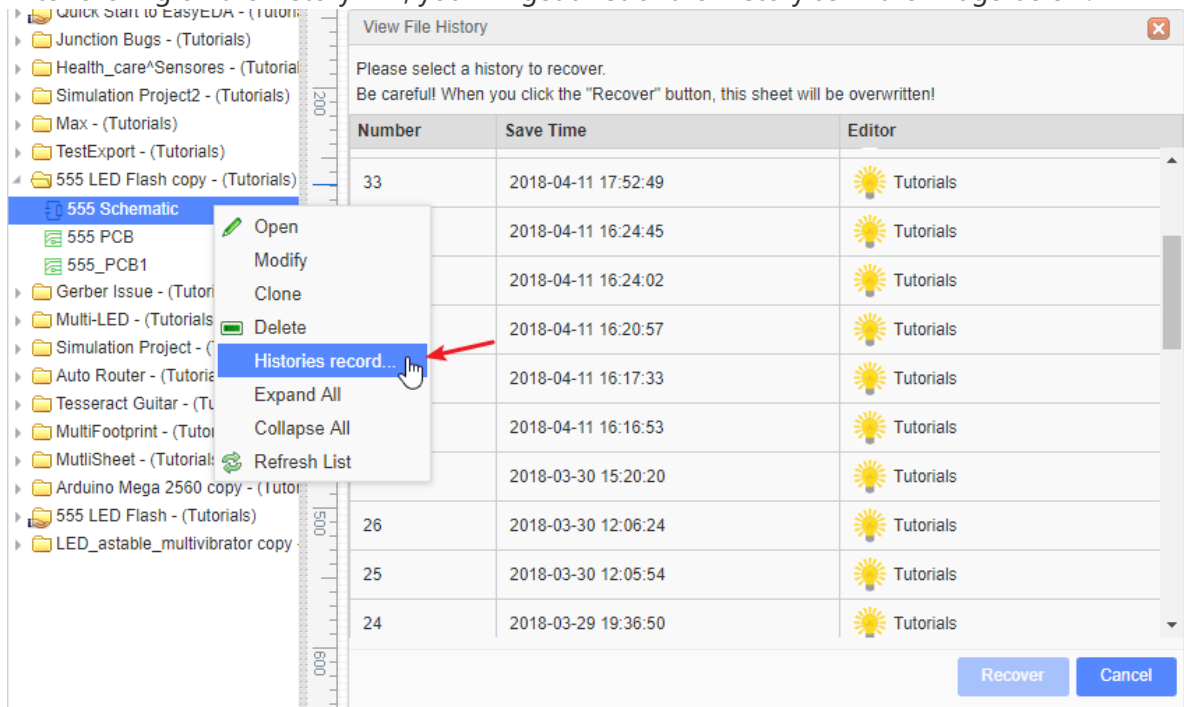
To download your project:



History Record

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

After clicking on the history link, you will get a list of the History as in the image below.



Click the History number to 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 the History entries are tagged with a number. An upcoming feature will allow you to add a custom tag.
2. Saving your files too frequently will create many History entries and it will become more difficult to find the exact one you want.

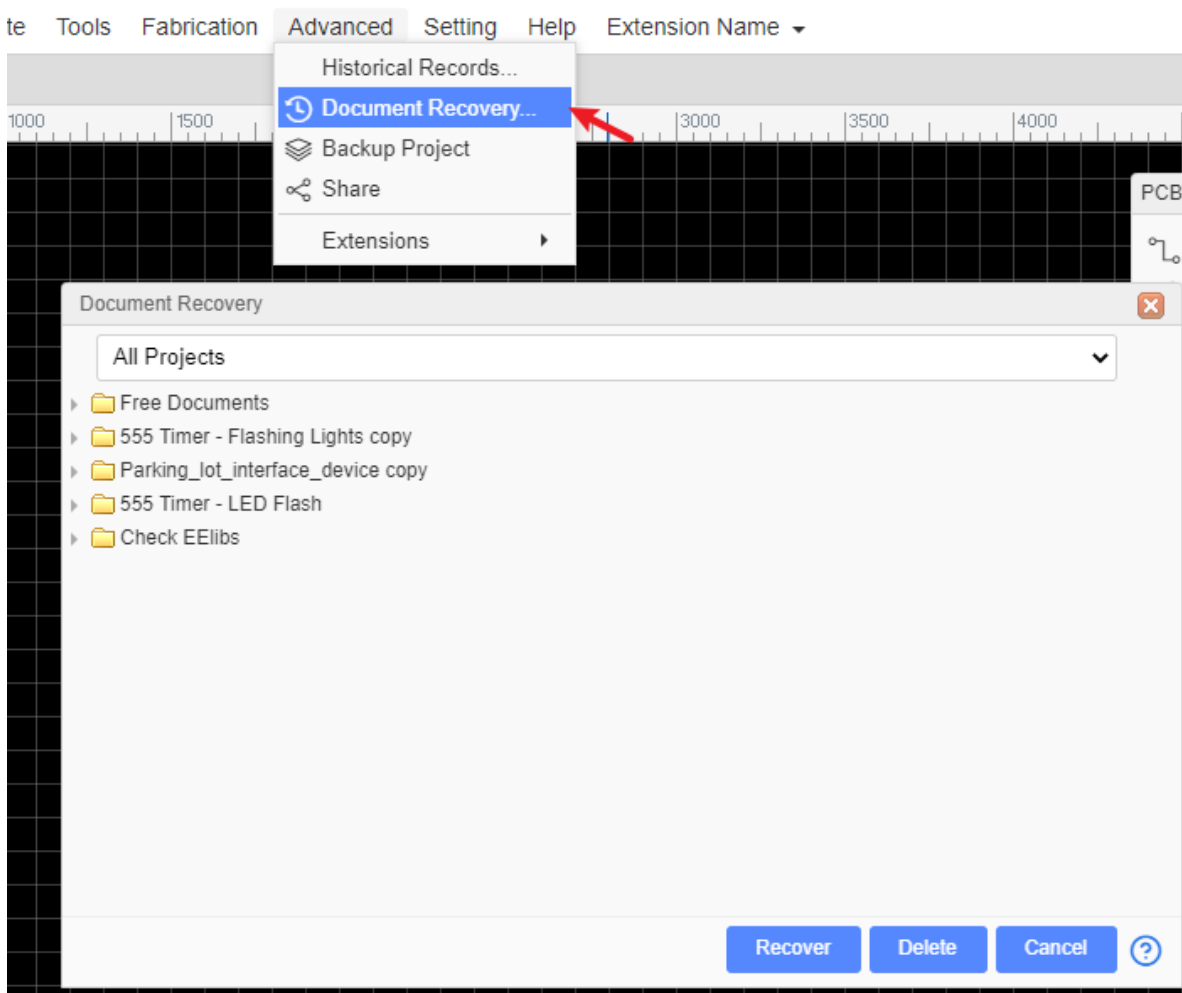
Document Recovery

No operating system, software or network is perfect, so sometimes things 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, crash recovery is built into EasyEDA.

On the top menu, click **Menu - Advanced - Document Recovery** as below:



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

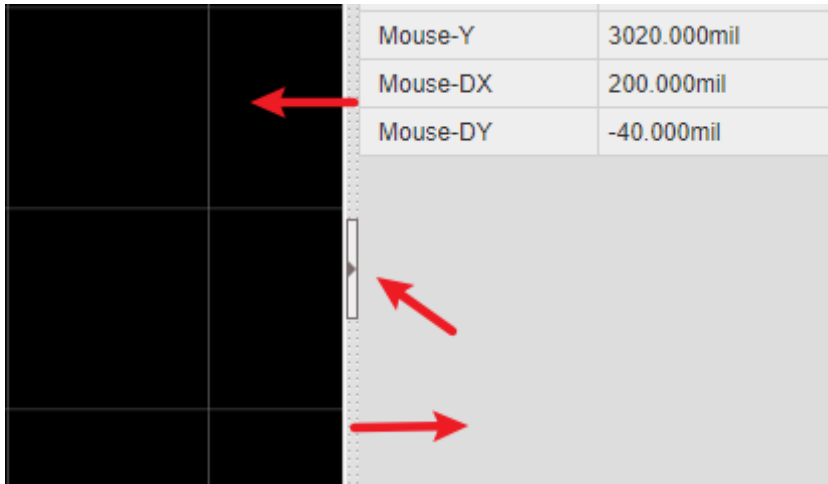
Please note:

- *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.*
- *If you cleaned your browser's cache, the recovery files will disappear.*

- If you made a mistake deleting a file and removing the cache, you might be able to find your document in the recycle bin: <https://easyeda.com/account/user/recycles/personal>.

Resizing the canvas area

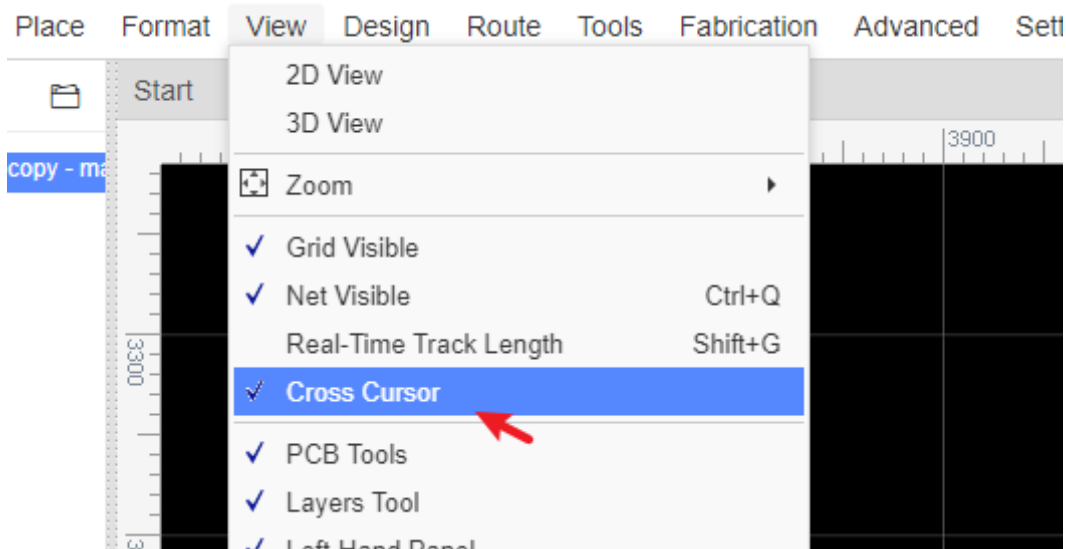
Hovering the mouse cursor over the areas indicated by the three green ellipses will bring up the blue sidebar toggle lines. Clicking on them will toggle the visibility of their associated right and left areas to expand the canvas area. The vertical lines can be dragged horizontally to resize the panels.



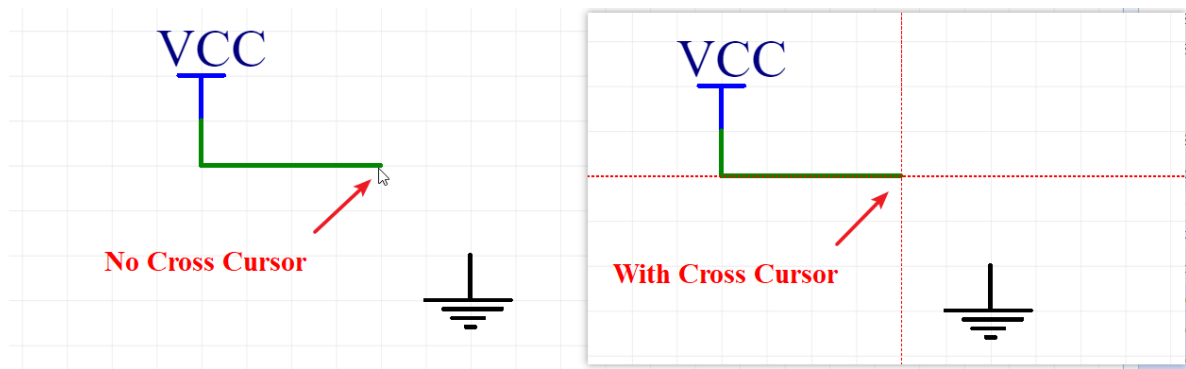
Cursor Style

Some users do not like the cross cursor, so you can change it to an arrow cursor like in the image below.

Via: Top Menu - View - Cross Cursor



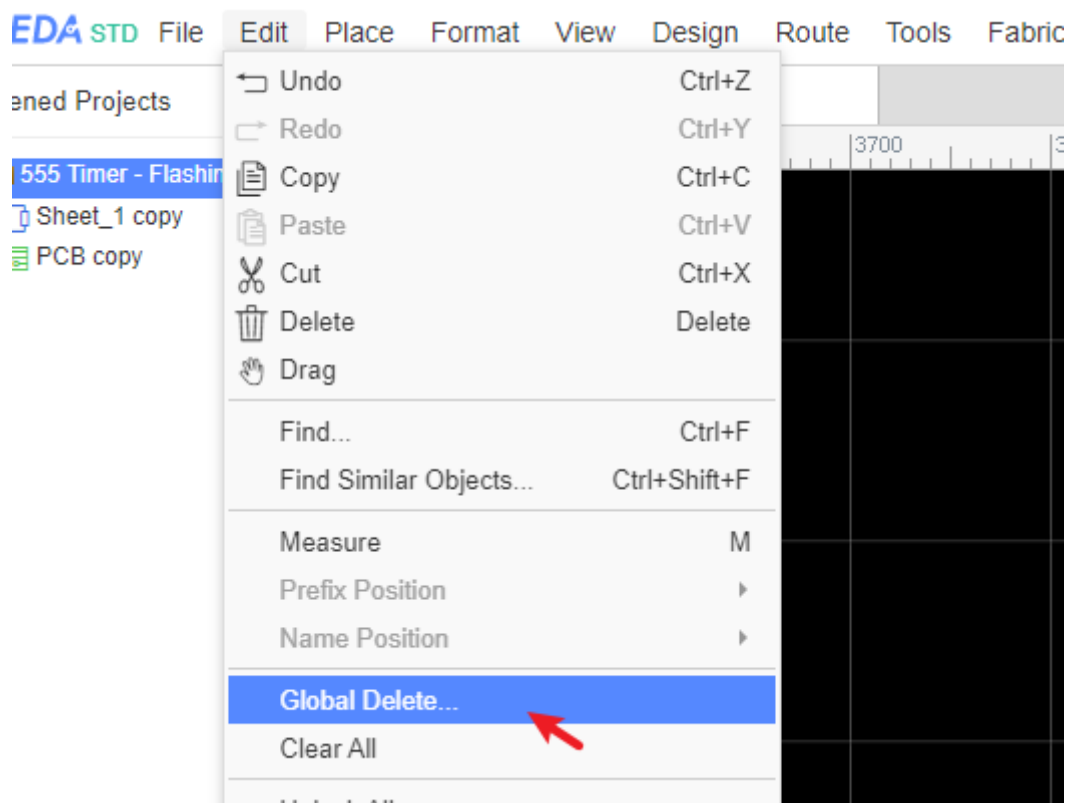
The difference between these options is shown below:



Clear and delete

If you think your schematic or PCB looks terrible, and you want to redraw everything, you can do this:

- **Top Menu > Edit > Global Delete.**



- Delete this schematic and create a new one.
- Click one object or CTRL+A, press delete key to remove all objects.

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 the text;
- The exact left click functionality depends on what item is being selected and in what Canvas the item exists (Schematic or PCB).

Right clicking

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

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

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

ESC key

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

Select more shapes

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

Zoom in and Zoom out

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

Please note:

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

Double clicks

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