

---

## Quick Start Guide To ExpressSCH

---

There are two parts to ExpressPCB, our CAD software and our board manufacturing service. Our CAD software includes *ExpressSCH* for drawing schematics and *ExpressPCB* for designing circuit boards. We also provide a low cost, high quality and fast source for having your boards made. Here is how it works:

**This document is formatted for easy printing.**

1. We recommend that you begin by drawing a schematic using *ExpressSCH*.
2. Next, use the *ExpressPCB* program to layout your PC board. If you link your schematic to *ExpressPCB*, it will guide you through the wiring process.
3. When your layout is complete, determine the exact cost to have boards made with the *Compute Board Cost* command.
4. To order the boards, enter your name, address and billing information into *ExpressPCB* and press the *Send* button within the *Order Boards Via The Internet* dialog box.
5. In a few business days (typically 2 or 3) an overnight courier will deliver your PC boards.

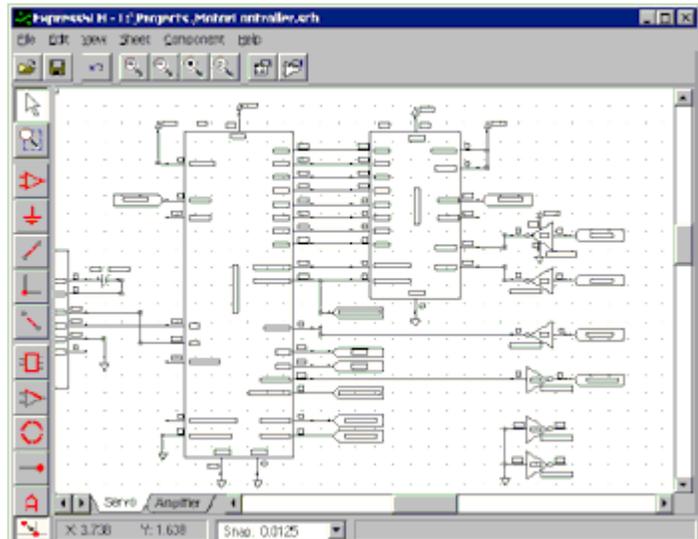
---

## Drawing a Schematic

*ExpressSCH* is a very easy to use Windows application for drawing schematics.

While not required, we suggest that you draw a schematic for your circuit before designing the PC board. By linking the schematic to the PCB, you will save time designing and be less likely to make mistakes.

You will also discover that the user interface for *ExpressSCH* and *ExpressPCB* is so similar that after spending the few minutes to learn one, the other will already be familiar.



---

## The Side Toolbar

The toolbar along the left side of the *ExpressSCH* main window is used to select editing modes, such as *place component* and *place wire*. These are the side toolbar buttons:

### Tools for drawing the schematic

-  Select, drag or set properties of an item in the schematic.
-  Zoom to view a selected area.
-  Place a component in your schematic.
-  Place a symbol (such as GND or +5v) in your schematic.

### Tools for creating custom components

-  Draw a rectangle as part of a new component.
-  Draw a line as part of a new component.
-  Draw a circle or an arc as part of a new component.
-  Place a pin on a new component.



Wire components together.



Draw text.



Insert a corner to bend an existing wire.



Disconnect a wire.

## The Top Toolbar

When you click on a side toolbar button, additional buttons are added to the top toolbar, specific to the function selected. Always displayed on top are these:



Open a schematic file.



Zoom in.



Zoom to previous size.



Save schematic file.



Zoom out.



Display *Options* dialog box.



Undo last operation.



Zoom to see all.



Display *Component Manager* dialog box.

When the *Place Component* button is selected on the side toolbar, these buttons are added on top. They are used to set the component's rotation before it is inserted into the schematic.



Place component body and text up.



Rotate component body down (text up).



Flip the component right to left.



Rotate component body left (text remains up).



Rotate component body right (text up).



Rotate component body and text left.



Rotate component body and text right.

## Zooming and Panning

The easiest way to move around your schematic is with the scroll wheel on the mouse. Turning the wheel zooms in and out. Pressing the wheel and dragging the mouse pans.

## The Grid

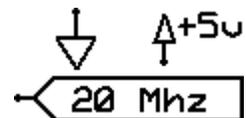
The *Snap-to-grid* is an invisible grid that helps you align wires and components. With the *Snap-to-grid* on, you will notice that as you place objects, they gravitate to a grid boundary. The *Snap-to-grid* spacing is set from the *Options* dialog box or from the *Snap* listbox located on the

lower statusbar. To toggle the *Snap-to-grid* on or off, press the **G** key or click .

## Components and Symbols

You build a schematic by first placing components and symbols on the page and then wiring them together. In *ExpressSCH*, components and symbols are different. *Components* represent parts that you add to your circuit such as resistors and ICs.

*Symbols*, such as these, are inserted into your schematic just like components. However they differ from components in that they don't have a physical part associated with them.



Symbols are used to make electrical connections to common buses (such as ground) without having to draw wires. They are also used to make connections from one page of a schematic to another. Here is how they work: All symbols are assigned a NetName (such as GND). Every symbol placed on a schematic with the same NetName is electrically connected together. For example, every component pin that is wired to a ground symbol (which has the NetName GND) is connected. Read

[Working with Sheets and Ports](#) to learn more about *Port* symbols.

## Beginning a New Schematic

1. Begin a new schematic by running *ExpressSCH*.
2. Select *New* from the *File* menu and then choose the desired page size, typically *Letter – 8.5" x 11"*. It is best to set the page size to the same dimensions as the paper in your printer. By doing this, your schematic will be printed one-to-one. If you create a schematic with a larger page size than in your printer, the printed schematic is compressed to fit the printer.
3. Fill in the text in the title block located in the lower right corner of the page. Double click on "Company Name", "Schematic Name" and "Designer's Name" to set these fields.
4. Finally, give your schematic a filename by selecting *Save As* from the *File* menu.

## Placing Components

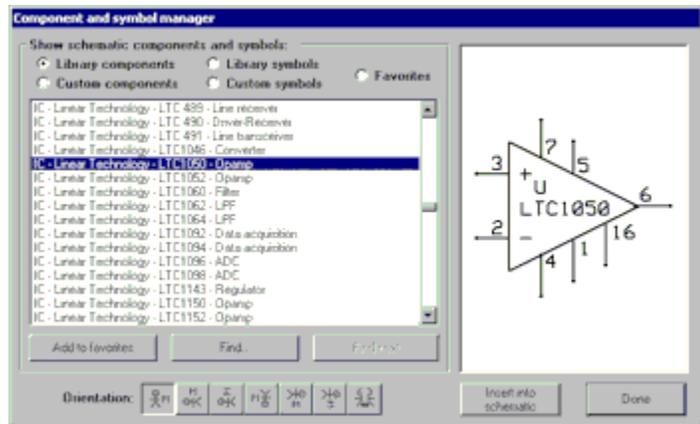
Components represent parts that you will add to your circuit such as resistors and ICs.

The easiest way to place components in your board layout is to use the *Component Manager*.

**Tip:** Use the *Find* button to search for a component by its name.

To place a component:

1. Click the  button located on the top toolbar to display the *Component Manager*.



2. Select one of these categories:
  - Library components - Components that are included with the program
  - Custom components - Components that you have drawn
  - Favorite components - Components or symbols that you have book-marked
3. From the list box, choose the item to insert. Note that the components list is organized by the prefixes:
  - Connector - connectors
  - IC - integrated circuits
  - Misc - batteries, buzzers and motors
  - Passive - resistors and capacitors
  - Semiconductors - transistors and diodes
4. Select how the component and its accompanying text are rotated by clicking one of:



5. Press the *Insert into schematic* button, then drag the component to the desired location.
6. Double click on the item just inserted to display its properties dialog box. Here you will assign the component its *Part ID* (such as R1 or U1) and *Part name* (such as 74LS00 or 3.3K).

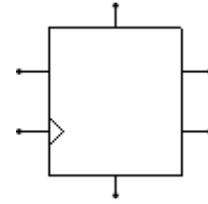
## Placing Symbols and Ports - Using the Toolbars



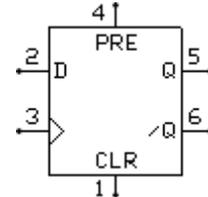
components.

2. Next, place pins at the locations where wires will attach to the finished

component by selecting  from the side toolbar and clicking at each endpoint. Pins are shown as small circles on the screen, but will not be printed on the schematic. We recommend setting the *snap-to-grid* to 0.050" when positioning the pins.

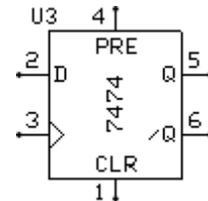


3. After placing the pins, double-click on each to assign its pin number and pin name (i.e. CLR). Then drag the pin number and name text to the desired locations. We recommend selecting a *text height* of 0.060" and setting the *snap-to-grid* to 0.025" when positioning the text.



4. Group everything together to combine it into a single component as

follows. Select your new part by clicking  and then drag the mouse to enclose it. From the *Component* menu, choose the command *Group to make component*. In the *Component properties* dialog box, enter the *Part name* (i.e. 7474 or 3.3K) and assign it a *Part ID* (i.e. U1 or R12).



5. Save your new component to add it to the *Custom components* list. Do this by selecting the part with the mouse and then choosing *Save custom component* from the *Component* menu.

**FYI:** ExpressSCH stores every component and symbol listed in the Component Manager in its own file. These files have the *.S* extension and are saved in the directories:

- Program Files\ExpressPCB\SchComponents\_Library
- Program Files\ExpressPCB\SchComponents\_Custom
- Program Files\ExpressPCB\SchSymbols\_Library
- Program Files\ExpressPCB\SchSymbols\_Custom

**Tip:** In many cases, it is fastest to build a new component by starting with a close fit from our library. This is done by first inserting the library component into a schematic. Next select it with the mouse and ungroup it using the *Ungroup component* command from the *Component* menu. Now make the desired changes as described above and finally regroup it back together to create the new part. Your changes will NOT affect the original part in the ExpressSCH library.

---

## Copying, Deleting and Moving Items

ExpressSCH uses standard *Edit* commands for *Copy*, *Cut* and *Paste*. To copy an item or several items, first select them. From the *Edit* menu choose *Copy*, then choose *Paste*.

Deleting items from your schematic is as easy as selecting them and pressing the **Del** key.

There are three ways most items can be moved. Typically, items are moved by selecting and dragging them. They also can be moved with the arrow keys by selecting them and then pressing *Ctrl-right*, *Ctrl-left*, *Ctrl-up* or *Ctrl-down*. Alternately, an item can be moved by changing the coordinates set in its properties dialog box.

**Tip:** To copy a group of items from one layout file to another, select and copy the items of interest into the clipboard. Load the second file into ExpressSCH and paste. It is not possible to copy items between two programs running at the same time.

---

## Working with Sheets and Ports

Schematics that have too many components to fit on a single page can be divided into two or more sheets. When you print your schematic, each sheet will be printed on one page. New schematics always start with a single sheet named *Sheet1*. Additional sheets are added using the *Add new sheet* command from the *Sheet* menu. To assign a sheet a meaningful name, select *Rename this sheet* from the same menu.

At the bottom left corner of *ExpressSCH* is a tab for each sheet in your schematic. Shown here are the tabs from a schematic divided into three sheets: *CPU*, *Power Supply* and *I/O*.



To switch between the sheets, click on one of the tabs. The left and right arrows are used to scroll the tabs sideways should there be so many that not all can be shown.

When a circuit is divided into multiple sheets, a method is needed to make electrical connections between them. This is done using *Port* symbols.



In this example, the output of the gate on the left is wired to a port symbol with the NetName *Beep*. The input to the circuit on the right (which can be on the same or a different sheet) is also wired to a port with the NetName *Beep*. This is the equivalent of running a wire between them.

Our library includes many port symbols to choose from. To learn how to insert port symbols in your schematic, read [Placing Symbols and Ports](#).

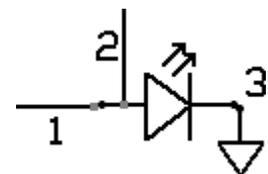
---

## Checking for Errors

After completing your schematic, you should check it for errors before beginning your PC board layout. Run the command *Check schematic for netlist errors* from the *File* menu. This will check for problems such as components that have not been assigned a *PartID*, multiple components having the same *PartID*, and missing pin numbers.

This command also scans for wires that are not properly connected and tries to correct them.

None of the wires leading to the LED seen here are connected. Wire 1 has not been dragged close enough to the LED's pin (shown as a dot). Though wire 2 ends on the LED, it does not attach to the right place. It needs to stop on the LED's pin. The gray *corner markers* at the ends of wires 1 and 2 indicate that they have not merged to the LED. When a wire is properly connected to a component's pin, the gray *corner marker* will disappear. The *ground* symbol (3) is also not connected. The pin on the *ground* symbol is too far away from the pin on the LED to make a connection.



**Note:** We strongly recommend that you visually inspect your schematic for errors before designing your PCB. The *Check schematic for netlist errors* command only identifies a few types of mistakes that you might have made. You should also make sure that all of your wires are properly connected to their components and other wires, because this command cannot fix all missed connections.

---

## How to Contact Us

Please feel free to contact us if you have questions, comments or suggestions. We can be reached by email at: [support@expresspcb.com](mailto:support@expresspcb.com)

