

---

# Quick Start Guide To *ExpressPCB*

---

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 provides 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.

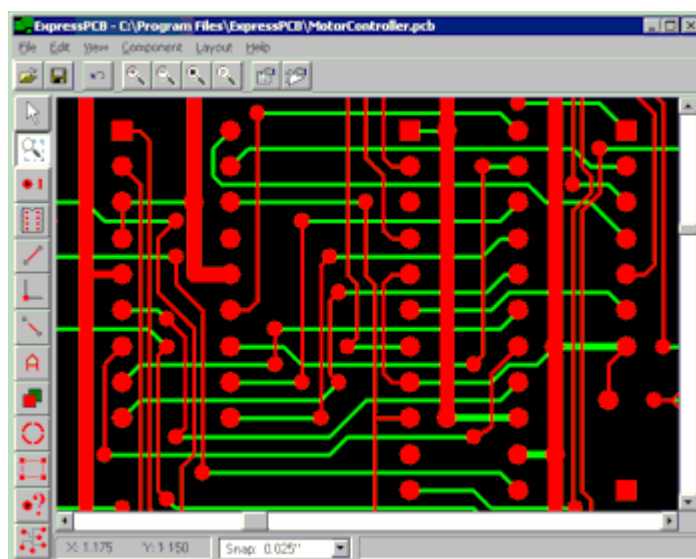
---

## Designing a PCB

*ExpressPCB* is a very easy to use Windows application for laying out printed circuit boards.

While not required, we suggest that you draw a schematic for your circuit using the *ExpressSCH* program. By linking your schematic, *ExpressPCB* will guide you by highlighting the pins that should be connected together with traces.











**Note:** *ExpressPCB* defaults to coordinates in inches. The exact position of the mouse is displayed on the status line.



---

## The Side Toolbar

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

	Select, drag or set properties of an item in the layout.		Place text in the layout.
	Zoom to view a selected area.		Place a rectangle.
	Place a pad (either a through hole or surface mount pad).		Place a circle or an arc.
	Place a component.		Insert an edge segment in the board perimeter.
	Place a trace.		Display information about a pad.



Insert a corner into a trace.



Highlight net connections.



Disconnect a trace.

At the bottom of the side toolbar are 4 additional buttons that toggle on and off features of the display:



Toggle display of silkscreen layer.



Toggle display of bottom copper layer



Toggle display of top copper layer.



Toggle Snap-to-grid.

## 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 the *Component Manager*.

These buttons are added on top as needed:



Move selected items to top layer.



Set rotation up.



Highlight first net.



Move selected items to bottom layer.



Set rotation left.



Highlight previous net.



Select top silkscreen layer.



Set rotation down.



Highlight next net.



Select top copper layer.



Set rotation right.



Highlight last net.



Select bottom copper layer.



Set text weight to normal.



Set text weight to bold.

## Beginning a New Layout

1. Begin a new layout by running *ExpressPCB*. If you are designing a four-layer board, select *Board properties* from the *Layout* menu and check the *4-Layer* option.
2. In the main window, the yellow rectangle defines the perimeter of the PC board. Set the size of your board by moving three of its four corners (the upper left corner is fixed at 0,0). Move the corners by dragging them with the mouse, or by double-clicking them and entering coordinates. Additional corners can be added to the perimeter to change its shape (see [Changing the Board Perimeter](#)).
3. Select the size of the *Default via*. In some cases as you place traces, *ExpressPCB* inserts via (plated-through holes) when a trace changes between the upper and lower layers. When inserted, these via pads are always visible. Set the *Default via* in the *Board properties* dialog box.
4. If you have drawn a schematic of your circuit using *ExpressSCH*, link the schematic file to your circuit board layout using the *Link schematic to PCB* command found under the *File* menu.
5. Finally, give your board a name by selecting *Save As* from the *File* menu.

## The Grid

The *Snap-to-grid* is an invisible grid that helps you align traces 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 .

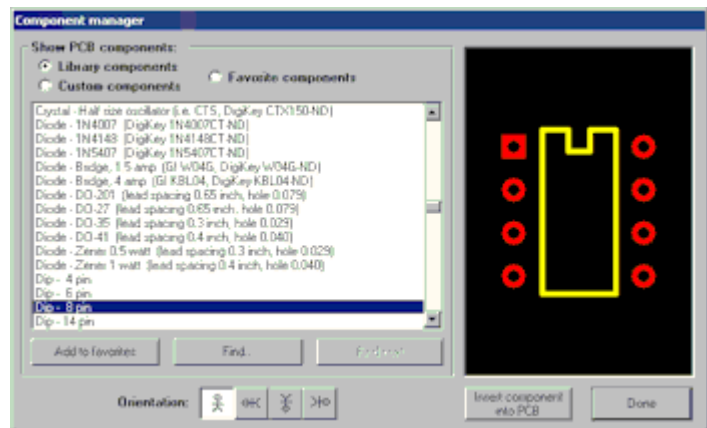
## Placing Components



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 PCB footprint 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 then select the component's orientation (rotated up, left, down, or right) by clicking one of these buttons: 
4. Press the *Insert component into PCB* button, then drag the component to the desired location.
5. Assign the component's *Part ID* (such as R1 or U2). To do this, select  then double-click on the component to display its *Component properties* dialog box.

**Note:** Setting the *Part ID* is required if you link your schematic and PCB. In order to mate a schematic component with one on the board layout, the *Part IDs* must match.

## Placing Pads

Individual pads are used to build new components and as via. To insert a pad:

1. From the side toolbar, select  or press the **P** shortcut key.
2. Select a pad size from the drop down list box on the top toolbar. There are several pad types: round, square, surface mount and via. Vias are used to pass traces between layers. Regular pads differ from via pads in that *ExpressPCB* may eliminate a via if the traces connecting to it are all on the same layer.

**Tip:** Remember, when choosing a pad, it is better to pick a hole that is too big, rather than too small.

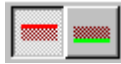
3. To place the pad, click on your layout at the desired location.

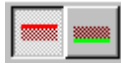
## Placing Traces

These are the steps to add traces to your layout. If you have drawn a schematic for your circuit, be sure to read the section [Linking the Schematic and PCB](#) before you begin.

1. From the side toolbar, select  or press the **T** shortcut key.

2. Select the trace width from the drop down list box on the top toolbar: A width of 0.010" is a good default for digital and analog signals. For power lines, use traces 0.05" or wider.



3. Select the layer by clicking  or by pressing the **L** shortcut key.
4. Move the mouse to the trace's first end point and click. Drag the trace to the second end point, then click again. Continue placing trace segments (use the **L** key to change layers) until you have reached the final end point.

**Tip:** Keep an eye on the statusbar when connecting traces to components. It will display the pin and part number to which the trace is connected.

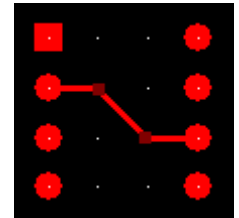
5. As you drag the trace, the **L** key changes layers, the **Del** key deletes the previous segment, the **+** and **-** keys zoom in and out, the **G** key toggles the *snap-to-grid* on and off. The **Spacebar** sets the trace, and the **Esc** key cancels it.
6. To complete the operation, press the **Spacebar** or click right.

---

## Working with Traces


Traces themselves cannot be moved. The path of a trace is determined by the straight line between its two connections. Therefore, to move a trace, you need to connect it to something different or to move what it is connected to.

Corners in traces allow them to bend. They are displayed as small square blocks at the ends of traces. A trace with two corners is shown here. In your final PC board layout, the corners will not be included.




Corners can be dragged, inserted or deleted to change the trace's path. To insert a corner, select



then click  to choose the layer on which the corner will be placed. Next click on the trace at the point where you want to insert the corner.



To disconnect a trace and reconnect it elsewhere, select  from the side toolbar. Next, click on the trace near the point you want to disconnect, then drag the trace to a new pin.

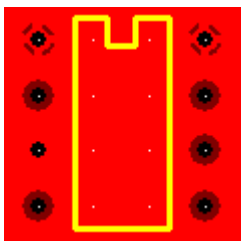
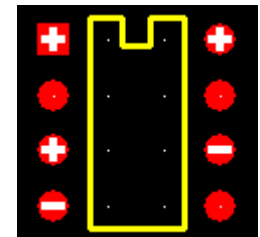
---

## Designing a Four-Layer PCB

Begin a four-layer design by selecting the *4-Layer* option in the *Board properties* dialog box. This adds two inner layers to your layout, a *Power* plane and a *Ground* plane.

Any through-hole pad in a layout can be connected to the *Power* or *Ground* internal planes by right clicking on the pad. In the popup menu select one of the *Power layer* or *Ground layer* options.

Connections to the *Power* plane are displayed with a **+** over the pad. *Ground* plane connections are marked with a **-** symbol.




To look at the actual inner layer planes, select *View Power layer* or *View Ground layer* in the *Options* dialog box. This example shows pins 1, 3, and 8 connected to the *Power* plane. The other pads are isolated.

You will notice that pins 1 and 8 are connected with *Thermal* pads, while pin 3 is connected with a *Solid* pad. Typically *Thermals* are used with pads connecting to a plane because they improve the solderability of the pad, reducing the chance of a "cold solder joint".

---

## Setting Properties of an Item

All the items in your layout have properties. For example, the properties of a pad include the pad type and its XY coordinates.

To view or change the properties of an item, select  then double click on the item.

---

## Copying, Deleting and Moving Items

*ExpressPCB* 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 from the *Edit* menu choose *Paste*.



Deleting items from your layout 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 *ExpressPCB* and paste. It is not possible to copy items between two programs running at the same time.

---

## Changing the Layer of Items

To move traces, rectangles and text between the top and bottom copper layers, select them with the mouse then click  to move the selection to the top layer; click  to move it to the bottom.

---

## Linking the Schematic and PCB

Running the traces from component to component is simplified if the schematic and PCB are linked. Embedded in *ExpressSCH* schematic files are wiring lists (called NetLists). When linked, the *ExpressPCB* program highlights which pins need to be wired together.


Before linking your schematic, check it for errors. Run the command *Check schematic for netlist errors* from the *File* menu of *ExpressSCH*. If errors are reported, correct them, then save the schematic file. For more information, see: [ExpressSCH: Checking the Schematic for Errors](#)

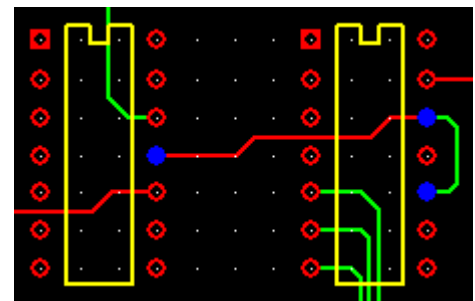
Next, for every part referenced in your schematic, you need to insert the component footprint for that part in your PCB design. Also, the *Part ID* values for each footprint must be set. To learn how to insert the footprints and set the IDs, see: [Placing Components](#)

**Note:** The *Part IDs* are used to match a schematic component with its PCB footprints when highlighting a net. (i.e. a *Part ID* of R12 in the schematic will be matched with a PCB footprint that also has the *Part ID* set to R12).



Now link your schematic and PCB files. From the *File* menu of the *ExpressPCB* program, select the command *Link schematic to PCB*.

Once linked, *ExpressPCB* can show you which pins should be connected together by highlighting them in blue.

To highlight a "Net", select  from the side toolbar, then click on one of the pads of a component. This highlights the other pads that should be wired to the pad you've selected. If the schematic has nothing connected to that pin, then no pads are highlighted.



You can step through all of the "Nets", from first to last, to visually inspect your PCB for missing

connections. This is done by selecting  and then clicking on  to navigate through all of the nets:

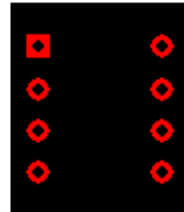
**Tip:** The ExpressPCB program reloads the linked schematic file every time the *.PCB* file is opened. If you run ExpressSCH and ExpressPCB at the same time, you can update the link by first saving the schematic within ExpressSCH, then selecting *Refresh link to schematic* from the *File* menu in ExpressPCB.


## Making Custom PCB Components

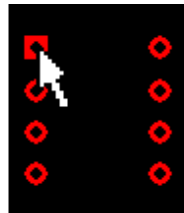
ExpressPCB includes many components that you can use to create your PCB. However if you need a footprint not found in our library, you can easily build your own. Here is how:




1. First add the pads by selecting  and choosing a pad type on the top toolbar. Carefully position the pads with the correct spacing. It may be helpful to change the *Snap-to-grid* spacing in the *Options* dialog.

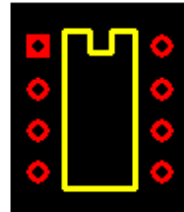
**Tip:** When choosing a pad, it is better to pick a hole that is too big, rather than too small.




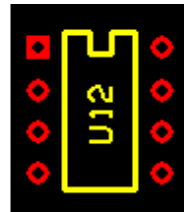
2. Assign each pad a *Pin number*. This must be done if you link your PCB to its schematic. Assign pin numbers by selecting  and then double clicking on each pad to display its *Pad Properties* dialog box. In the *Pin number* field, enter the pin number.



3. Draw the component outline for the new part on the silkscreen layer. Draw straight lines by selecting  or circles and arcs using . Be sure to select the silkscreen layer before drawing the outline by clicking . The recommended line width for drawing component outlines is 0.012".



4. Group everything together to combine it into a single component as follows. Select all of the objects in your new part by clicking  and then dragging the mouse to enclose them. From the *Component* menu, choose the command *Group to make PCB component*. Next, double click the new component to assign its *Part ID* (i.e. U12).



5. Save the 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:** ExpressPCB stores every component in its own file. These files have the *.P* extension and are saved in the directories:

- Program Files\ExpressPCB\PCBComponents\_Library
- Program Files\ExpressPCB\PCBComponents\_Custom

## Modifying Library Components

Sometimes it is easiest to build a new component by starting with one from our library, for example to change the pad sizes. This is done by first inserting the library component into your layout.

Next select it with the mouse then choose the *Ungroup PCB component* command from the *Component* menu. Now select only the pads and double click on one. In the *Pads Properties* dialog, change the pad size. Finally, regroup the component back together as described above to create the new part. Your changes will NOT affect the original part in the ExpressPCB library.

---

## Having PC Boards Made

When you have completed your layout, the next step is to have the boards made. Before ordering, you can determine the cost using the *Compute Board Cost* command from the *Layout* menu.

To order your boards, connect your computer to the Internet then select the *Order Boards Via The Internet* command from the *Layout* menu. Fill in the forms that follow and press the SEND button. Your order is sent to *ExpressPCB* for manufacturing.

For more information, see the online help section [Having PC Boards Made](#).

---

## 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)