

## Table of Contents

[Overview](#)  
[Beginning a New Schematic](#)  
[Placing Items in the Schematic](#)  
[Placing Symbols and Ports](#)  
[Editing the Schematic](#)  
[Copying all or Part of a Schematic to Another File](#)  
[Making Custom Schematic Components](#)  
[Modifying Library Components](#)  
[Exporting Your Schematic Drawing](#)  
[Working with Sheets and Ports](#)  
[Checking the Schematic for Errors](#)  
[Linking the Schematic and PCB Files](#)  
[Creating a Bill of Materials](#)  
[Keyboard Shortcuts](#)  
[How to Get the Most Recent Version of ExpressSCH](#)  
[How to Contact Us](#)

<p><b>This document is formatted for easy printing.</b></p>
---

---

## Overview

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. After completing your PC board design, we provide a low cost, high quality and fast source for having your boards made. Here is how it works:

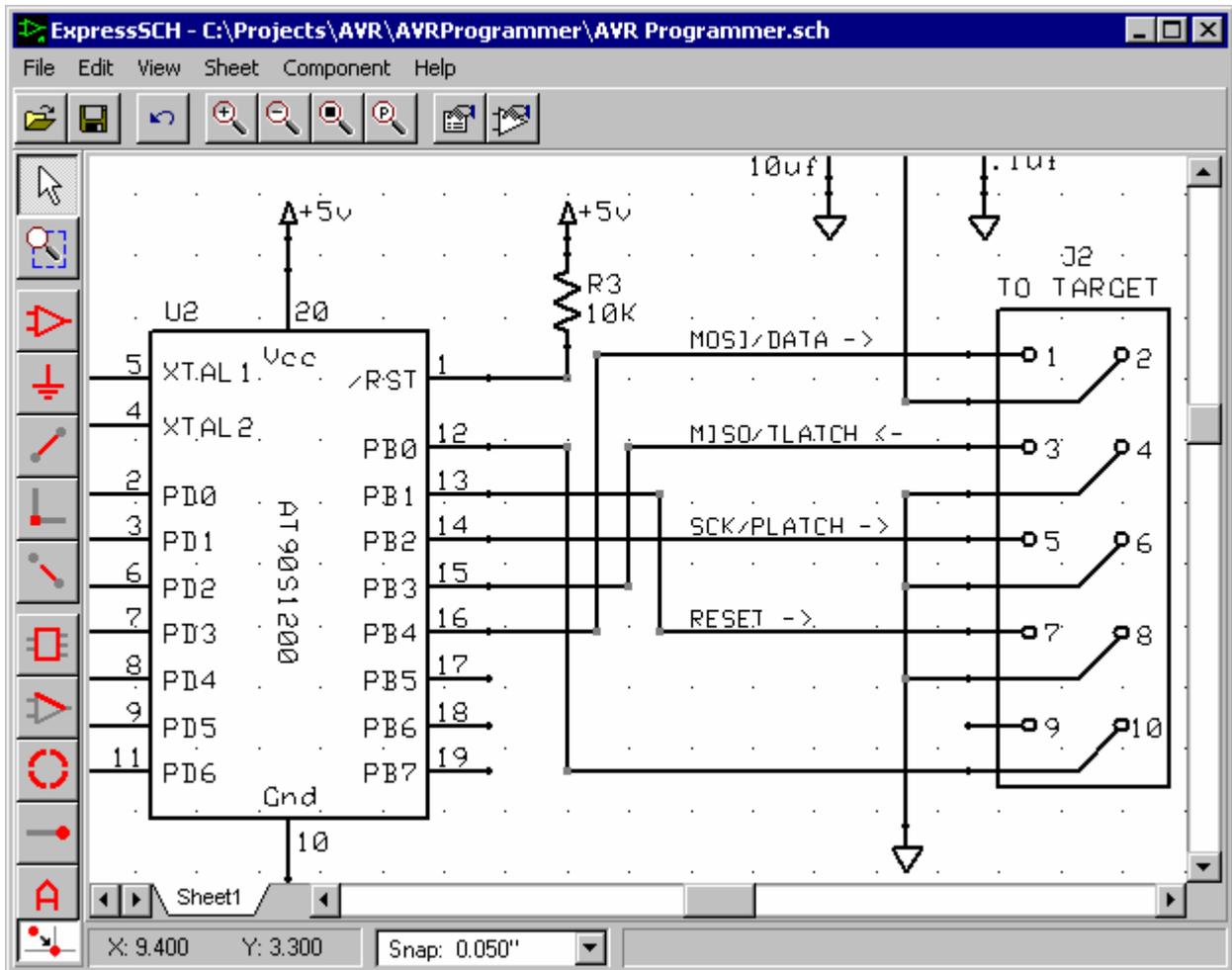
- 1.** We recommend that you begin your project by drawing a schematic using *ExpressSCH*. Drawing a schematic is not required, but it will save you time when designing your board and reduce the possibility of wiring errors.
- 2.** Next, use the *ExpressPCB* program to lay out your PC board. If you link your schematic to the *ExpressPCB* program, the program will guide you through the wiring process by highlighting how the components should be connected.
- 3.** When your layout is complete, the *ExpressPCB* program will tell you the exact cost to have your boards made.
- 4.** To order the boards, simply enter your name, address and billing information into *ExpressPCB* and press the *Send* button in 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.

**Note:** See [Board Manufacturing Options](#) to learn about the different PC board manufacturing choices we offer.

---

## The Display

Take a few minutes to acquaint yourself with the *ExpressSCH* program's display. You will notice that there are two toolbars, one along the top and another along the left side. At the bottom of the display is a statusbar.



### The Statusbar

The statusbar shown along the bottom has three fields. The first field shows the XY position of the mouse. The units are in inches. The coordinate (0,0) represents the upper left corner of page.

In the middle of the statusbar is the *Snap list box*. Here the *snap-to-grid* feature can be turned on and its spacing set. When on, placing items such as components and wires will automatically align to the snap-grid.

The field on the right displays helpful information as you work. For example, the display might read: *Connecting to pin 11 of U4* when you attach a wire to that pin.

### The Sheet Tabs

Displayed at the bottom left corner of the main window is a tab for each page of your schematic. Initially there is one just tab named *Sheet1*. Simple schematics will often fit on a single page. However more complex designs can be divided into multiple sheets. Shown here are the tabs from a schematic divided into three pages:



To display a particular page of a your schematic, click on it's tab. Use the left and right arrows to scroll the tabs sideways should your schematic include more sheets than can be shown at

once.

To learn more about creating schematics with multiple pages, see: [Working with Sheets](#).

---

## The Side Toolbar

The toolbar along the left side of the main window is used to select between the different editing modes, such as *place component* and *place wire*. Each of these modes can be selected either with the mouse or a shortcut key.

You will notice that when you click on one of the side toolbar buttons, additional buttons are added to the top toolbar that are specific to the function selected.

These are the buttons along the side toolbar:

### 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.
-  Wire components together.
-  Insert a corner to bend an existing wire.
-  Disconnect a wire.

### 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.
-  Draw text.

At the bottom of the side toolbar is an additional button  that toggles the *Snap-to-grid* on and off.

---

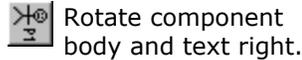
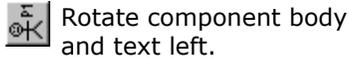
## The Top Toolbar

These are the buttons found on the top toolbar. The buttons shown depend on the mode selected with the side toolbar. Always displayed on the top toolbar are these:

-  Open a schematic file.
-  Zoom in.
-  Zoom to previous size.
-  Save the schematic file.
-  Zoom out.
-  Display *Options* dialog box.
-  Undo last operation.
-  Zoom to see all.
-  Display the *Component Manager*.

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).



---

## Units and the Coordinate System

By default, the *ExpressSCH* program displays coordinates in *Inches* with the origin (0, 0) located at the upper left corner of the page. Sometimes it can be very useful to move the origin or select metric units.

To change the display units, select either *Inches* or *Millimeters* in the *Options* dialog box. Then update the *Snap-to-grid* spacing also in the *Options* dialog with an appropriate value.

The (0, 0) origin can be positioned anywhere on the schematic. Feel free to move it as often as you like as it only affects the display of coordinates, it does not actually make changes to objects on the page.

To move the origin, from the *View* menu select the command *Set origin with mouse*. Next click on your schematic at the point where you want the origin to be located. If the *Snap-to-grid* is on, then the origin is positioned at a grid location. The origin can be restored to its original position using the menu command *Reset origin to upper left*.

**Note:** Internally the *ExpressSCH* program saves the coordinates of objects in inches with a resolution of 0.001". Values entered in millimeters are converted to the nearest thousandth of an inch.

---

## The Grid

Displayed in the main window is a grid of small black dots. You will find this grid helpful when aligning components and wires. The grid is only a display tool and will not appear on the schematic when you print it.

As you zoom out to view a large area of the page, you will notice that the grid is not always displayed.

The grid is configurable and can be modified in the *Options* dialog box. You can display or hide the grid, and also set the spacing between grid lines. The spacing is typically set to 0.2 inches.

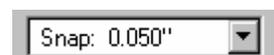
The *Snap-to-grid* is an invisible grid that is also configured in the *Options* dialog box. It is very helpful when placing items to ensure that they are automatically aligned. If the *Snap-to-grid* feature is turned on, you will notice that as you drag components and wires with the mouse, they always gravitate to a grid boundary. Typically the *Snap-to-grid* spacing is set to 0.05 inches (one fourth of the *Grid* spacing).

The *Snap-to-grid* feature is most easily toggled on or off by pressing the **G** key or by clicking



at the bottom of the side toolbar.

**Tip:** Quick changes to the *Snap-to-grid* spacing can be made from the list box on the statusbar shown here:

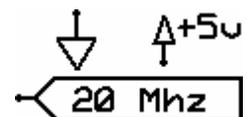


---

## Components and Symbols

It is important to understand the difference between components and symbols in *ExpressSCH*. *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. Symbols are also used to make connections from one page of a schematic to another. Here is how they work: Each symbol is assigned a *NetName* (such as Gnd). All

symbols having the same *NetName* on your schematic are electrically connected to each other. For example, all component-pins that are wired to a ground symbol (which has the *NetName* Gnd) are connected. Read [Working with Sheets and Ports](#) to learn more about *Port* symbols.

---

### Example Schematic Files

When learning to use our software, it is often helpful to first review a few example schematics. In the directory where *ExpressSCH* is installed, you will find several example files having an *.SCH* extension. Load these files into the *ExpressSCH* program using the *Open* command from the *File* menu. Typically they are located in the directory:

C:\Program Files\ExpressPCB

---

## Beginning a New Schematic

1. Begin a new schematic by running *ExpressSCH*. You can launch *ExpressSCH* from your desktop by clicking on the  icon.
2. Select *New* from the *File* menu and then choose the desired page size, typically *Letter – 8.5" x 11"* is used. 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 your printer can handle, the printed schematic is compressed to fit the printer.
- 3 Document your project by filling in the title block located on the lower right corner of the page.

Company Name		
Schematic Name		
Designer's name	Rev 1.0	Page # or name
	11/19/2002	

Double click on the text in each field to set it. Adjust the position of each line of text by dragging it with the mouse. It is often helpful to turn off the *Snap-to-grid* by pressing the **G** key before dragging these fields.

4. You may want to divide your schematic into two or more sheets. Each sheet is printed on a separate page. To learn more about creating schematics with multiple pages, see: [Working with Sheets](#).
5. Finally, give your schematic a filename by selecting *Save As* from the *File* menu.

---

## Scrolling and Zooming

### Scrolling the Main Window

In addition to scrolling using the scrollbars, you can also scroll with the mouse and the arrow keys. While dragging an item such as a pad or a trace with the left button down, moving the mouse outside the main window causes it to scroll. You can also scroll the using the keyboard by pressing the *Up*, *Down*, *Right* or *Left* arrow keys.

---

### Zooming In and Out

There are four buttons along the top toolbar that are used to zoom the main window in and out:

-  Zoom in. (+ shortcut key)
-  Zoom out. (- shortcut key)
-  Zoom out to fit the entire schematic. (**F** shortcut key)
-  Zoom to the previous display.

You can also zoom into a specific region of the schematic by choosing  and then dragging the mouse. Simply enclose the area you wish to expand and center.

 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 view side to side and up and down.

## Placing Items in the Schematic

### Placing Components - Using the Toolbars

*Components* represent parts that you will add to your circuit such as resistors and ICs. There are a couple of ways to place them into your schematic. Here is how to insert a component using the toolbars:

1. From the side toolbar, select  or press the **C** shortcut key.
2. Select the component type from the drop down list box on the top toolbar as seen here:

There are three categories in the list:

- Favorite components - parts that you have book-marked.
- Custom components - parts that you have built.
- Library components - parts that are included with the program.

3. Select how the component and its text will be rotated on the page by clicking one of:

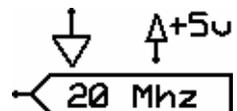


4. Move the mouse into the main window, press the left mouse button down and drag the component to the desired location.
5. While dragging the component, the **R** shortcut key can be used to rotate the component, the **+** and **-** keys zoom in and out, and the **G** key toggles *snap-to-grid* on and off.
6. After placing the component, assign it a *Part ID* (such as R1 or U1) and a *Part name* (such as 74LS00 or 3.3K). Do this by selecting  then double click on the component to display the *Component properties* dialog box.

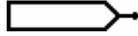
 Often it is easiest to place additional copies of a part using *Copy and Paste*.

### Placing Symbols and Ports - Using the Toolbars

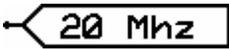
*Symbols* such as these are inserted into your schematic just like components. However symbols are different from components in that they don't have a physical part associated with them.



Symbols are used to make electrical connections to common buses (such as *Gnd* or *+5v*) without having to draw wires.

 User defined *Port* symbols allow connections to be made from one page of a schematic to another. Here is how they work: All *Ports* are assigned a *NetName* (such as *20 Mhz* ). Every *Port* on your schematic with the same *NetName* is electrically connected together. For example, every component pin that is wired to a port with the *NetName* *20 Mhz* is connected. To learn more about *Port* symbols, read: [Working with Sheets](#).

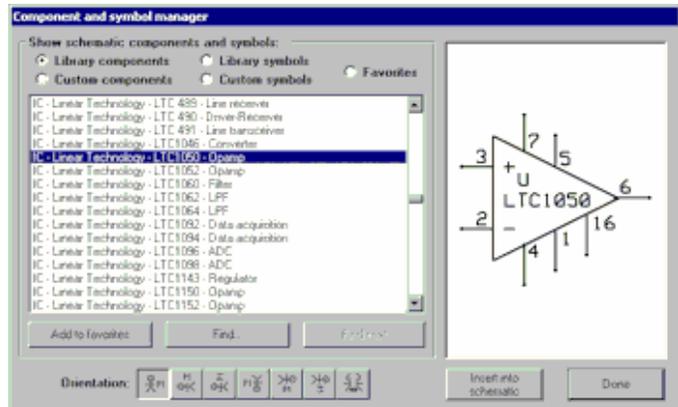
To insert a symbol such as *Ground* into your schematic or to insert a user defined *Port*, do the following:

1. First select the  button from the side toolbar.
2. From the top toolbar, choose the symbol in the listbox as seen here: 
3. Move the mouse into the main window, press the left mouse button down and drag the symbol to the desired location.
4. After placing a *port* symbol, assign it a *NetName*. Do this by double clicking on it to display the *Symbol properties* dialog box. The port symbol shown here has been assigned the *NetName*: *20 Mhz* 

## Placing Components and Symbols - Using the Component Manager

An alternate way to place *Components* and *Symbols* in your schematic is to use the *Component Manager*.

The advantages of using the *Component Manager* are that it displays a bigger window of part names; it also shows the parts in a preview window and you can search for parts by name.



To place a component or symbol:

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
  - Library symbols - Symbols (such as *Gnd* or *+5v*) that are included
  - Custom symbols - Symbols that you have drawn
  - Favorite components - Components or symbols that you have book-marked
3. From the list box, choose the item to insert. 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 with the mouse.
6. Double click on the item just inserted to display its properties dialog box. Here you will assign the component's *Part ID* (such as R1 or U1) and *Part name* (such as 74LS00 or 3.3K). The properties dialog is also used to set the *NetName* after placing a *Port* symbol.

**Note:** Setting the *Part ID* is required if you choose to link your schematic and PCB files. In order for the *ExpressPCB* program to mate a component from your schematic with the part in your board layout, the *Part IDs* must match.

 The *Find* button in the *Components Manager* makes it easy to search for a part in the long list of component names.

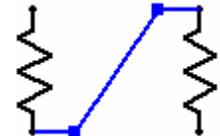
### Wiring Components Together



1. From the side toolbar, select  or press the **W** shortcut key.
2. Move the mouse to the wire's first end point and click left. Typically you will begin by starting on the pin of a component. Drag the wire to the second end point, then click left again. Continue placing wire segments until you have reached the final end point.
3. As you drag the wire, 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 wire, and the **Esc** key cancels it.
4. To complete the operation, press the **Spacebar** or click right.

### Wires with Corners

Corners allow wires to bend. They are displayed as small square blocks at the ends of wire segments. A wire with two corners is shown here. Corners in a wire can be dragged, inserted or deleted to change the wire's path. When you print your schematic, the corners will not be visible.



### Inserting Corners in Wires

1. Corners can be inserted in an existing wire to change its route. To insert a corner, select  or press the **I** shortcut key.
2. Click on the wire where you want to add the new corner.
3. Drag the corner to the desired location.

### Placing Text

Most of the text that appears on a schematic, such as component pin descriptions, pin numbers and Part IDs are built into the component and not separately added to the schematic. These fields are set by double-clicking on the item and then filling in the value in the items properties dialog box.

Additional text, such as comments, notes and signal descriptions can be inserted into your schematic by using the side toolbar. Here is how:

1. To place text, from the side toolbar, select  or press the **A** shortcut key.
2. Select the height of the text from the drop down list box.
3. Enter your text in the text box along the right hand side of the toolbar as shown in this example: 
4. Move the mouse into the main window, press the left mouse button and drag the text to the desired location.
5. As you drag the text, the **L** key toggles the layer, the **R** key rotates the text, the **+** and **-** keys zoom in and out, and the **G** key switches the *snap-to-grid* on and off.

---

## Editing the Schematic

### Setting Properties of an Item

Each of the items in your schematic has specific properties. For example, the properties of a component include its *Part name* (i.e. 74LS00 or 3.3K) and its *Part ID* (i.e. U1 or R2). To view

or change the properties of an item, select  from the side toolbar, then double click on the item.

---

### Moving Items in the Schematic

There are three ways most items can be moved. Typically, items are moved by selecting and dragging them. They can also 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.

---

### Moving Wires

Wires themselves cannot be moved. The position of a wire is determined by the straight line between the two endpoints of what it is connected to. Therefore, to move a wire, you need to either connect it to something different, or move what it is connected to.

Often the solution to moving a wire is to drag its corners. Several corners can be selected at once and then dragged together. In some cases, you will want to insert additional corners to form new bends.

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

---

### Deleting Items From the Schematic

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

**Tip:** Keep in mind that almost every operation can be undone with the *Undo* command in the *Edit* menu.

---

### Copying Items

*ExpressSCH* uses standard *Edit* commands for *Copy*, *Cut* and *Paste*. To copy, first select the item or several items of interest. Next, choose *Copy* from the *Edit* menu. Then choose *Paste*. The new copy will appear selected in the middle of the window.

---

## Copying all or Part of a Schematic to Another File

It is not possible to copy sections of a schematic between two *ExpressSCH* programs running at the same time. However, you can easily copy all or part of your schematic to another file as follows: 1) Using the mouse, select the region of your diagram that you would like to copy. 2) From the *Edit* menu, choose *Copy*. 3) From the *File* menu, open the .SCH file into which you want to insert this region into. 4) Select the *Paste* command from the *Edit* menu to copy the circuit into this new file.

---

## Rotating Items 90 Degrees

A section of a schematic can be easily rotated in 90-degree increments. Do this by first selecting the region of interest. Then from the *Edit* menu choose *Rotate Selection 90°*. Repeat the command to rotate 180 or 270 degrees.

Individual components in the schematic are rotated with the commands found in the *Component* menu. The menu command *Rotate Body 90°* rotates the outline of the component, but leaves its text (such as the pin numbers and pin descriptions) upright. The command *Rotate Body & Text 90°* rotates both the component's outline and its text. A component can be changed from right facing to left facing using the command *Flip right to left*.

---

## Right Click Menu

Many editing functions can be executed from *Right Click Popup* menus. Do so by clicking on an item in your schematic with the *Right* mouse button. A popup menu appears with several commands listed, such as *Copy*, *Paste*, *Rotate*, *Flip*, and *Set Properties*.

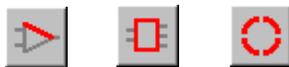
---

## Other Commands

### Making Custom Schematic Components

*ExpressSCH* includes hundreds of components and symbols you can use in your schematics. If you need something not included in our library, you can easily build your own. Here is how:

1. Begin by drawing the body of the component right in your schematic. Draw lines, rectangles and arcs by selecting these toolbar buttons:

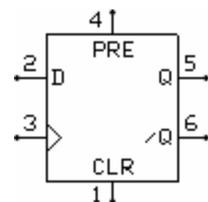
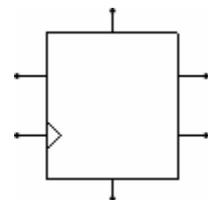
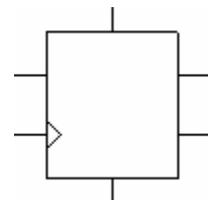


**Note:** Wires & corners should never be used when building components.

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

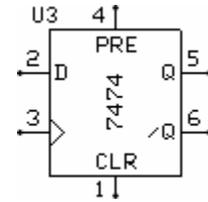
component by selecting  from the side toolbar and clicking at each endpoint. Pins are shown as small circles on the screen, but will not be printed on the schematic. We recommend setting the *snap-to-grid* to 0.050" when positioning the pins.

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.

**Note:** *ExpressSCH* stores each custom component in its own file (having a .S extension). You may want to backup your components, or copy them to other computers where *ExpressSCH* is installed. To locate the directory where these files are saved, open the *Options* dialog box, then select the *Misc* tab.

**FYI:** Find more information about designing custom schematic symbols online at:  
<http://www.expresspcb.com/ExpressPCBBin/ExpressSCHComponentGuide.pdf>

---

### Modifying Library Components

In many cases, it is faster to build a new component by starting with a close fit from our library. This is easily done by first inserting the library component into your 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.

---

### Exporting Your Schematic Drawing

You can export your schematic drawing so that it can be inserted into document or placed on the web. Use the *Export schematic image* command in the *File* menu to save an image of the schematic. The file can be saved as either a .BMP file, or as a scalable .EMF metafile.

An image can also be generated using the *Edit* menu command *Copy schematic image to clipboard*. The image can then be pasted into a Microsoft Word or other document type. To make an image of your schematic for the web, you can paste it into a graphics package, and then convert it to a .GIF file type.

---

### Working with Sheets and Ports

Complex 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 easily 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 of your schematic, simply click on one of the tabs. The left and right arrows are used to scroll the tabs sideways should your schematic include so many sheets that not all the tabs can be shown.

**Tip:** The order that the tabs appear can be changed using the *Move sheet tab left* and *Move sheet tab right* commands in the *Sheet* menu.

When you divide a circuit into multiple sheets, you will need a way to make electrical connections between them. With *ExpressSCH*, 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*. The use of these two port symbols is equivalent to running a wire between them.

Our library includes many port symbols to choose from. Some point to the left and some to the right and there are several sizes to accommodate long or short NetNames. To learn how to insert port symbols in your schematic, read [Placing Symbols and Ports](#).

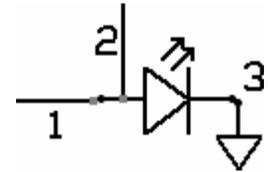
---

### Checking the Schematic 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 spot. 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.

---

### Linking the Schematic and PCB Files

After completing your schematic, the next step is to design the printed circuit board for your project. This process can be greatly simplified by linking the schematic and PCB files. Embedded in *ExpressSCH* schematic files are wiring lists (called NetLists) that describe which pins should be connected together. By loading this information into the *ExpressPCB* program, it can guide you when you place the traces by showing you which pins need to be wired together.

Before linking, be sure to check the schematic for errors. Run the *Check schematic for netlist errors* command from the *File* menu of *ExpressSCH*. If any errors are reported, correct them and then save the schematic by choosing *Save* from the same menu.

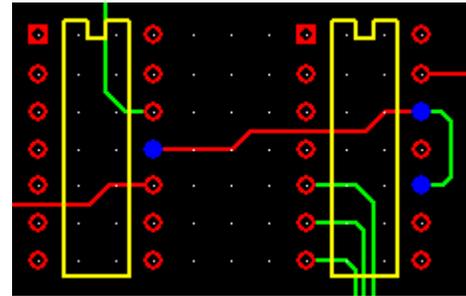
To link your schematic and circuit board, run *ExpressPCB* and load your *.PCB* file. At this point, you should have already manually inserted the footprints for each component referenced in the schematic into your PCB design. Also, the PCB footprints should have their *Part ID* values set. The *Part IDs* are used to match the schematic components to their PCB footprints (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).

Link the two files by selecting *Link schematic to PCB* from the *ExpressPCB* program's *File* menu. Here, enter the name of your schematic file.

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.



**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*.

---

## Creating a Bill of Materials

A useful document to accompany your schematic is a *Bill of Materials*. A bill of materials is a parts list that includes all of the components referenced in your schematic. This list can be automatically generated using the *Copy bill of materials to clipboard* command found in the *Edit* menu. This command copies the parts list for your schematic onto the clipboard. It can then be pasted in a text file, Word document or Excel spreadsheet.

---

## Keyboard Shortcuts

*ExpressSCH* includes many shortcut keys to help make drawing your schematic fast and easy. In some cases, a key will only be active while you are performing a specific operation.

<b>A</b>	Select the <i>place text</i> function from the side toolbar.
<b>C</b>	Select the <i>place component</i> function from the side toolbar.
<b>D</b>	Select the <i>disconnect wire</i> function from the side toolbar.
<b>F</b>	<i>Zoom to fit</i> the entire schematic.
<b>G</b>	Toggle <i>snap-to-grid</i> on and off.
<b>I</b>	Select the <i>insert corner</i> function from the side toolbar.
<b>R</b>	Change the <i>rotation</i> while dragging an item.
<b>S</b>	Select the <i>selection</i> function (shown as an arrow) from the side toolbar.
<b>W</b>	Select the <i>place wire</i> function from the side toolbar.
<b>Z</b>	Select the <i>zoom into a selected area</i> function from the side toolbar.
<b>+</b>	<i>Zoom in</i> .
<b>-</b>	<i>Zoom out</i> .
<b>Backspace</b>	When placing wires, delete the previous wire segment.
<b>Del</b>	<i>Delete</i> the selected items from the schematic.
<b>Esc</b>	<i>Cancel</i> the current operation.
<b>Return</b>	When placing wires, end the operation and set the new wires.
<b>Spacebar</b>	When placing wires, end the operation and set the new wires.
<b>Ctrl C</b>	<i>Copy</i> the selected items into the clipboard.
<b>Ctrl F</b>	Search the schematic for a component's PartID.

<b>F3</b>	Continue searching the schematic for a component's PartID.
<b>Ctrl N</b>	Create a <i>new file</i> .
<b>Ctrl O</b>	<i>Open</i> an existing file.
<b>Ctrl S</b>	<i>Save</i> the current file.
<b>Ctrl V</b>	<i>Paste</i> from the clipboard into the schematic.
<b>Ctrl X</b>	<i>Cut</i> the selected items from the schematic into the clipboard.
<b>Ctrl Z</b>	<i>Undo</i> the last operation.
<b>Left</b>	<i>Scroll</i> main window left.
<b>Right</b>	<i>Scroll</i> main window right.
<b>Up</b>	<i>Scroll</i> main window up.
<b>Down</b>	<i>Scroll</i> main window down.
<b>Ctrl-left</b>	<i>Move</i> selected items one pixel left.
<b>Ctrl-right</b>	<i>Move</i> selected items one pixel right.
<b>Ctrl-up</b>	<i>Move</i> selected items one pixel up.
<b>Ctrl-down</b>	<i>Move</i> selected items one pixel down.

---

## How to Get the Most Recent Version of *ExpressSCH*

The *ExpressSCH* software is periodically revised and improved. We recommend that you use the most recent version of *ExpressSCH* as updates become available.

You can check if you have the most recent program by comparing the version number you have (shown using the *About* command under the *Help* menu) with the version number found on the download page of our web site [www.expresspcb.com](http://www.expresspcb.com).

To receive the most recent version, visit the download page of [www.expresspcb.com](http://www.expresspcb.com). From here you can download the *ExpressPCBSetup* file free of charge. Included in the setup program are both the *ExpressSCH* and *ExpressPCB* applications. Run the setup program and install the new version on your PC in the same directory used by the older program. Any existing files that you have created will automatically be compatible with the new program.

---

## 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)

Our web site is a good source of information about *ExpressSCH*. You can also download the latest version of our software from our home page. Please visit [www.expresspcb.com](http://www.expresspcb.com)