Creating new symbols

Circuit Wizard allows you to extend the component library by creating your own circuit symbol and PCB layout components. *Note that user-created symbols are for design only and will not simulate.*

Components are created by drawing shapes and turning these into specific component symbols.

💡 It is common to create both a new circuit symbol and a new PCB symbol for a component that you wish to include. To do this you would create the two types of symbol separately. The two symbols would be linked together so that the circuit symbol converts into the PCB symbol (see page 3).
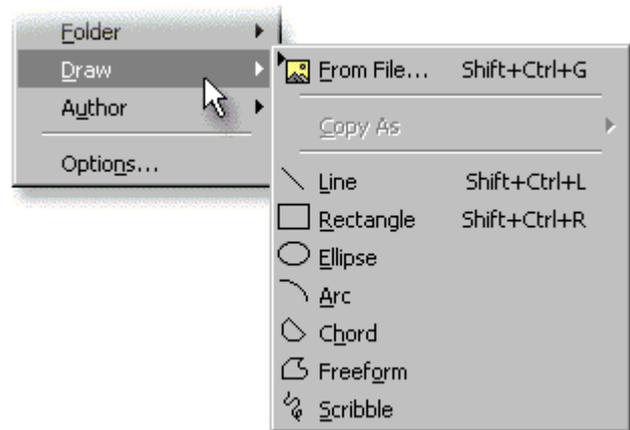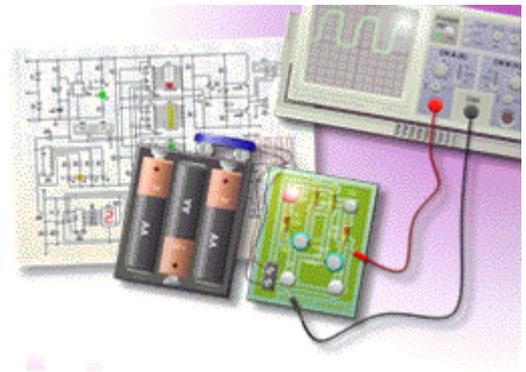
Drawing a symbol shape

Symbols are drawn using standard drawing tools (Line, Rectangle, Ellipse, Arc, Chord, Freeform and Scribble), available from the **Project** | **Draw** menu.

PCB components can also include copper pads for each component leg. You can add a copper pad by clicking on the Pad button or choosing **Pad** from the **Project** | **PCB Components** menu.

Layering is important when drawing new symbols. The shapes for circuit diagram symbols should be placed on the *Circuit Diagram* layer, whilst the shapes for PCB symbols should be placed on one of the PCB layers, such as the silk screen. For PCB symbols you can also add a real world view by placing shapes onto the *Real Components* layer.

You can move an object to a specific layer by first selecting that object and then choosing from the **Edit** | **Layer** menu.

Folder	▶	
Draw	▶	
Author	▶	
Options...		

|		
🖼 From File...	Shift+Ctrl+G
Copy As	▶
＼ Line	Shift+Ctrl+L
☐ Rectangle	Shift+Ctrl+R
◯ Ellipse	
⌒ Arc	
◖ Chord	
⌔ Freeform	
⸙ Scribble	

Adding symbol pins

Pins are used in symbols to specify the location of connections. Each pin is numbered and you should add a symbol pin for each pin, or component leg, on each new component.

You can add a pin by choosing **Insert Pin** from the **Edit** | **Symbol** | **Library** menu. When a circuit symbol is converted into a PCB component symbol, the pin numbers are used to determine how the connections are mapped (pin 1 on the circuit diagram symbol will be mapped to pin 1 on the PCB component symbol). You should align pins on a 0.1" grid to match the existing components.

Once placed, the symbol pin will appear as a small red cross. You can double-click on the pin to change its associated number.
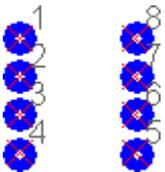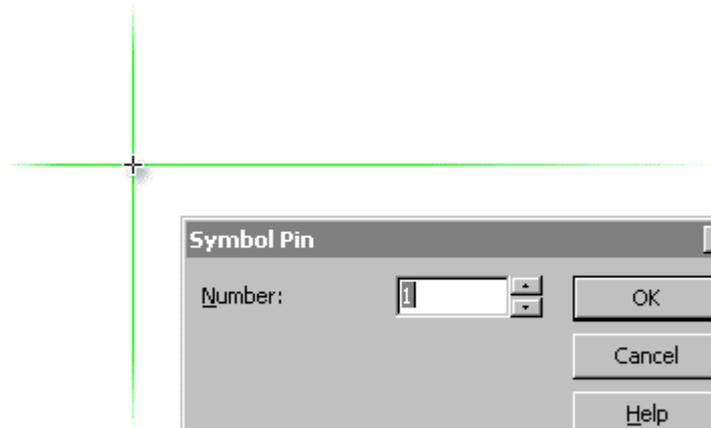
Pins on circuit diagram symbols are placed at the end of each terminal leg, however this will vary depending on the layout or shape of the symbol.

With PCB component symbols, a pin is normally placed on top of each pad. On the left you can see how a simple 8-pin dual-in-line (DIL) component might be created as a PCB symbol, with 8 pins placed in sequential order.

Pins for circuit diagram symbols should be aligned on a 0.1" grid so as to match the existing, built-in component symbols.
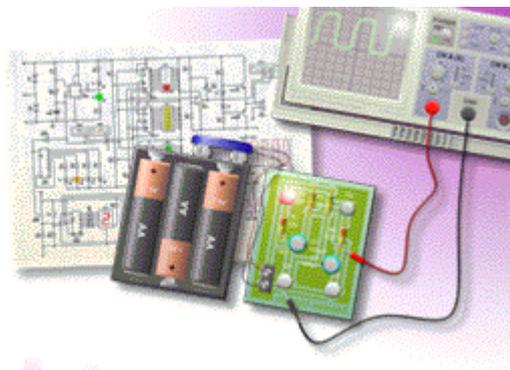
Editing symbol properties

With the shape of a new symbol drawn, you can turn it into a symbol by selecting all of the objects that will make up the symbol and then choosing **Make Symbol** from the **Edit** | **Symbol** | **Library** menu.

A window will appear allowing you to enter details for the new symbol.

Type determines the category of symbol that is created. It is fixed for existing symbols.

Key is a unique value that identifies the symbol and is normally based on the name of the symbol being created.

It is good practice to also add '.Symbol' or '.Package' to the end of the Key for circuit symbol and PCB symbols respectively.
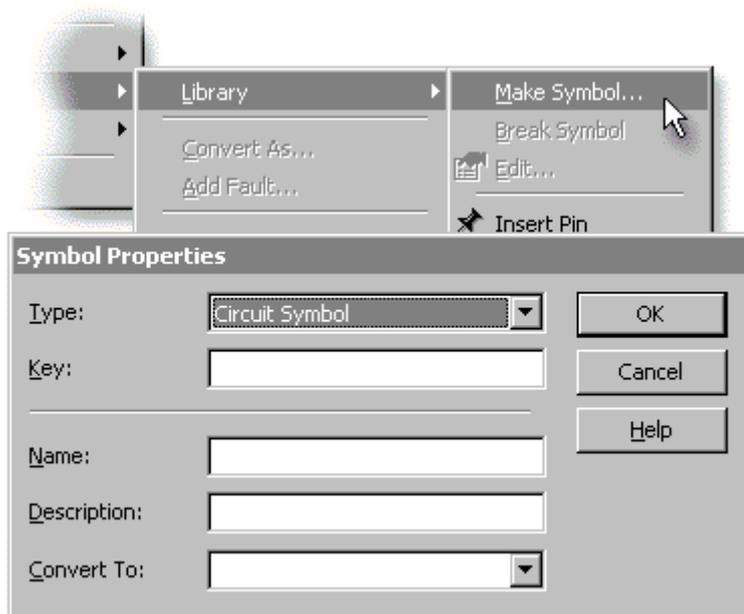
Examples of the Key field include:

Res1000.Symbol
for a 1 inch resistor circuit symbol

DIL8.Package
for an 8-pin dual-in-line PCB symbol

Key is also used to reference component information when a Bill of Materials report is generated.

**Name** and **Description** allow you to describe the symbol.

Convert To is only available for circuit symbols and determines which PCB symbol is used whenever this symbol is converted into a PCB component. This should be the same as the Key field of the required PCB component. Note that you can only link to your own user-created PCB symbols.

Once the properties have been entered, click on **OK** to create the symbol.

You can change an existing symbol by choosing **Break Symbol** from the **Edit** | **Symbol** | **Library** menu. This will turn an existing user-created symbol back into its constituent parts.

Edit, from the same menu, allows you to change the properties of an existing user-created symbol.
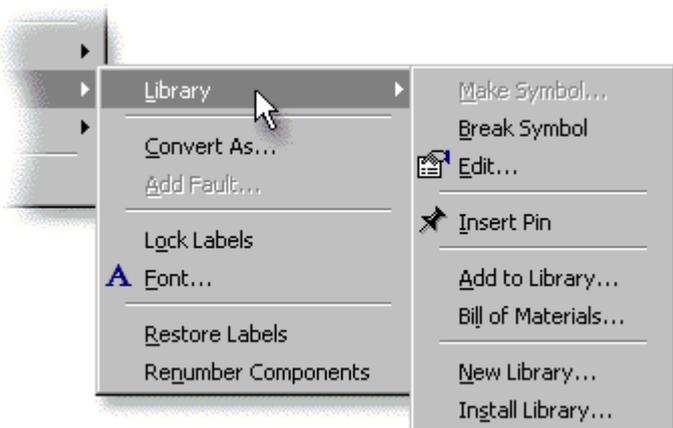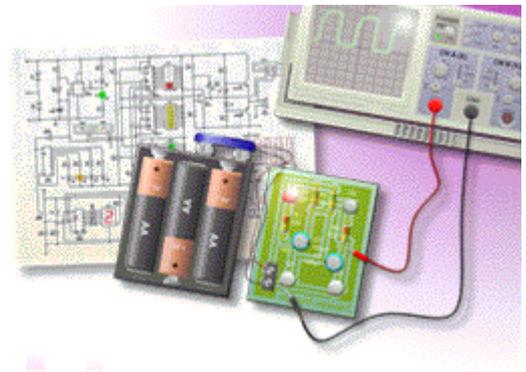

Adding symbols to a library

To add a symbol to a library requires a user library to have been created. Symbols can only be added to user libraries—user libraries allow you to modify their contents, unlike the standard fixed libraries which normally make up the Gallery.

You can create a new user library by choosing **New Library** from the **Edit** | **Symbol** | **Library** menu.

A window will appear allowing you to choose a name for the new library.

In addition, you can decide in which part of the Gallery the new library will reside. This will dictate which type of symbols can be stored in the library. Click on the **Type** list and select the required library type.

Clicking on **OK** will create the new library.
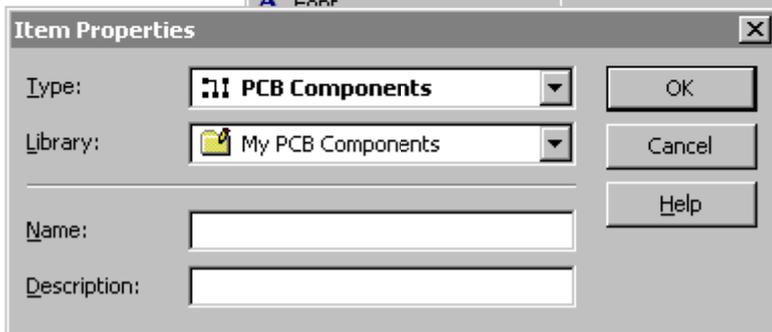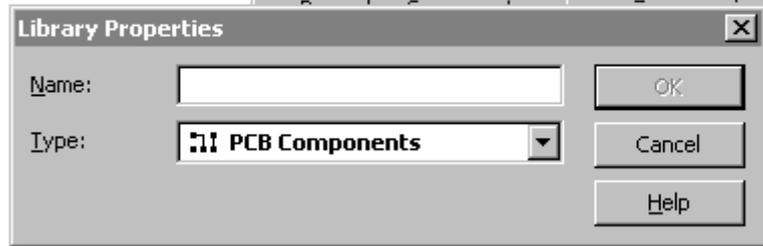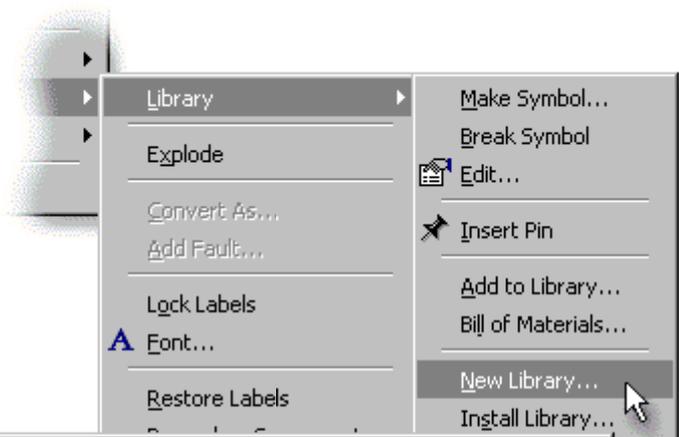
With a new user library created, you can add a symbol to the library by first selecting your new symbol and then choosing **Add to Library** from the **Edit** | **Symbol** | **Library** menu.
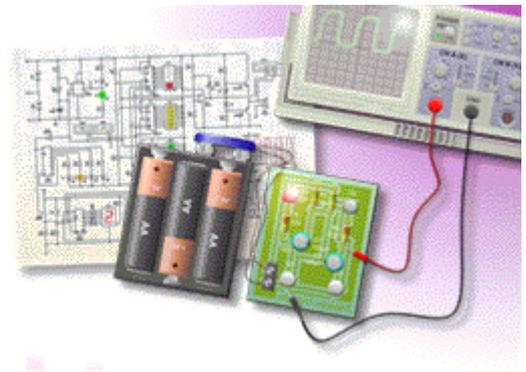
From the window that appears you can decide in which library the symbol will be placed.

A name and description can also be provided.

Click on the **OK** button to add the symbol to the library.

Copyright 2005-2012 New Wave Concepts Limited. All rights reserved. **www.new-wave-concepts.com**

Manipulating library items

Clicking the right mouse button over an item in a Gallery window and opening the **Item** menu allows you to control that item. *Note that only items in user libraries can be modified.*

Insert allows you to add that item to your document.

Edit opens that library item in a new document window for editing. You should click on the Save button once you have finished editing.

Copy allows you to create a duplicate of the selected library item. The duplicate can be placed in any existing user library.

Update will overwrite the contents of the selected library item with the current selection.

Delete removes the selected library item.

Hide removes the item from view, but leaves the symbol in the library. This allows you to tailor user libraries to different users.

Properties allows you to adjust the name and description of the selected library item.