Step 1 of 5: Introduction

This tutorial shows you how to create your own circuit symbol and PCB component by customizing the PCB Wizard library. You should have experience of PCB Wizard's drawing and editing tools.

Difficulty Level: **Advanced** (suitable for experienced users)

Getting started

In this tutorial you will create a new component (both as a circuit symbol and as a PCB layout symbol) and then add it to the library.
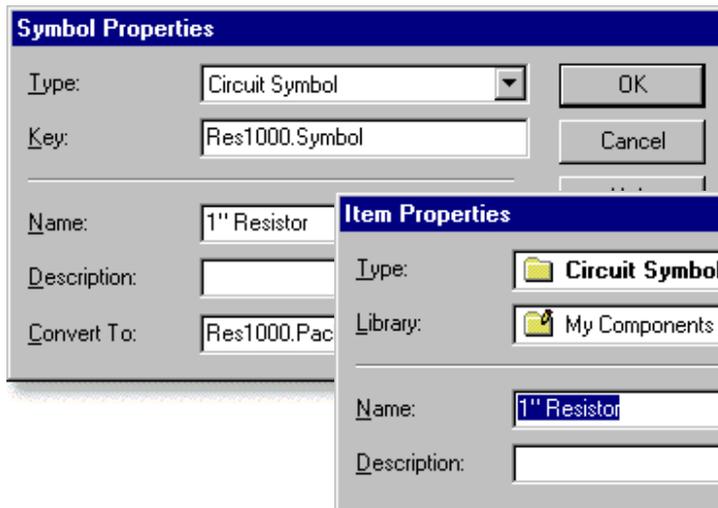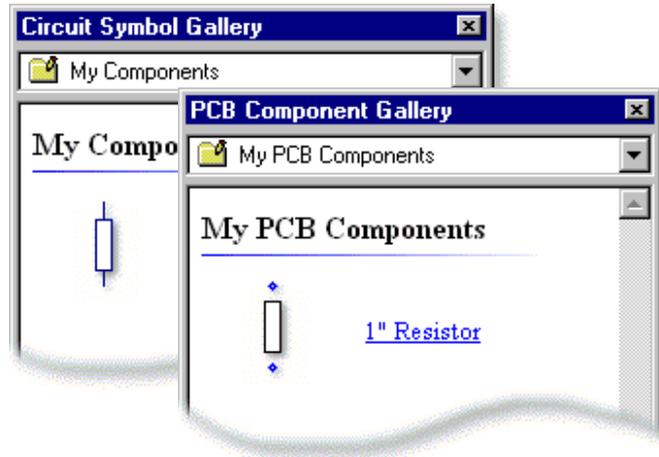
The component to be created is a large 1 inch resistor that does not currently exist the library.

Along the way, you will learn how to:

Create new circuit symbols
Create new PCB layout symbols
Add symbols to the library

Step 2 of 5: Creating the circuit symbol

The first step is to create the circuit symbol for the new component.

Using the drawing tools in the **Insert | Picture** menu, draw a resistor shape using a rectangle and two vertical lines.

Next, select all three of these objects and then choose **Circuit Diagram** from the **Edit | Layer** menu. This will place each object on the layer used for circuit diagram drawing.

With the shape drawn, you need to specify exactly where the connection points are on the symbol. This is done by placing pins on the symbol.

Choose **Insert Pin** from the **Edit | Symbol** menu and then click on the top of the upper-most vertical line. A window will appear allowing you to specify the pin number. Enter a pin number of 1 and then click on the **OK** button.

Add a second pin at the bottom of the bottom-most vertical line with the number 2. It should now look as shown opposite.

Step 1: Drawing the circuit symbol shape.

Step 2: Adding the pins.

The symbol is now ready to be created. Select all of the symbol objects (including the pins) and choose **Make Symbol** from the **Edit | Symbol** menu.

A window will appear in which you can specify a range of properties for the symbol. Complete the fields as shown on the left.

Type determines what category of symbol is created. Ensure that **Circuit Symbol** is selected.

Key is a unique identifier that will be used by the software to access the symbol. The Key field is normally based on the name of the symbol being created. In this example the value **Res1000.Symbol** is used (since it is a 1 inch, or 1000 mil, resistor circuit symbol).

Name and **Description** are used to describe the symbol being created. Enter the name **1" Resistor**.

Convert To specifies which PCB layout symbol should be used when this circuit symbol is converting into a PCB layout. This is done by entering the same value here as in the Key field of the appropriate PCB symbol. As this symbol is to be linked to an equivalent PCB symbol, you should enter a value of **Res1000.Package**.

Finally click **OK** to create the circuit symbol.

Step 3: Making the symbol.

💡 **Tip!** You need to remember which Key values you have used when creating your own circuit and PCB layout symbols.

Step 3 of 5: Creating the PCB layout symbol

With the circuit symbol created, you can now create the associated PCB layout symbol. This process follows much the same approach as for creating circuit symbols.

Draw two pads and then a rectangle (the rectangle tool can be found in the **Insert | Picture** menu) to represent the body of the resistor. The pads should be 1 inch apart (using a grid spacing of 0.1 inch will help with this).

Step 1: Drawing the PCB layout symbol shape.

Next, select the rectangle and choose **Silk Screen** from the **Edit | Layer** menu. This will place the package outline on the component-side silk screen layer.

*Note: If you want to include a real-world view of the component you should do it at this point. Using the drawing tools draw how the component would look (remembering to ensure that these drawn objects are placed on the **Real Components** layer).*

With the shape drawn, you need to specify exactly where the connection points are on the symbol. This is done by placing pins on the symbol.

Step 2: Adding the pins.

Choose **Insert Pin** from the **Edit | Symbol** menu and then click in the centre of the top-most pad. A window will appear allowing you to specify the pin number. Enter a pin number of 1 and then click on the **OK** button.

Add a second pin on top of the bottom-most pad with the number 2. It should now look as shown opposite.

The PCB layout symbol is now ready to be created. Select all of the symbol objects (including the pins) and choose **Make Symbol** from the **Edit | Symbol** menu.

A window will appear in which you can specify a range of properties for the symbol. Complete the fields as shown on the left.

Type determines what category of symbol is created. Ensure that PCB Component is selected.

Step 3: Making the symbol.

Key is a unique identifier that will be used by the software to access the symbol. The Key field should match the value entered into the **Convert To** field on the circuit symbol. For this symbol enter a value of **Res1000.Package**.

Name and **Description** are used to describe the symbol being created. Enter the name **1" Resistor**.

The **Convert To** option only applies to circuit symbols and is therefore disabled. You can ignore this field.

Finally click **OK** to create the PCB layout symbol.

Step 4 of 5: Adding the symbols to the library

Now that both symbols have been created, you need to add them to the library.
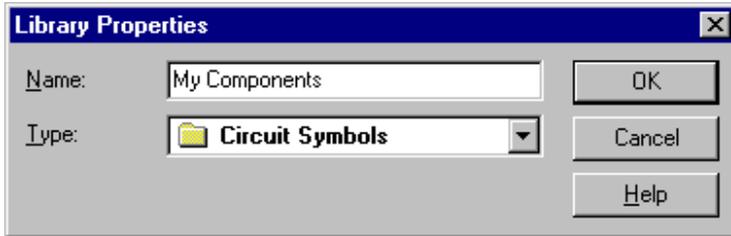
Starting with the circuit symbol, a new library needs to be created. Choose **New Library** from the **Tools | Library** menu.

In the window that appears enter **My Components** as the name, select **Circuit Symbols** as the Type and then click on the **OK** button.
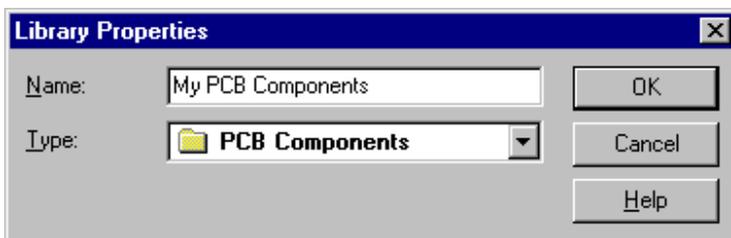
Next select the circuit symbol created earlier and choose **Add to Library** from the **Tools | Library** menu.

Another window will appear. Ensure that the new **My Components** library has been selected and then click on the **OK** button.

Once done you will see the symbol appear in the gallery.
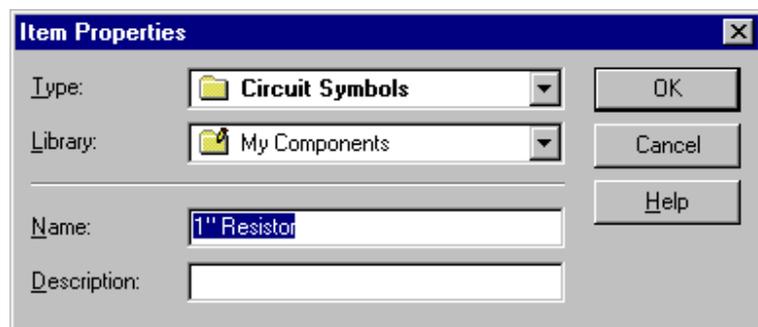
This process is then repeated for the PCB layout symbol.

A second library needs to be created for the PCB layout symbol. Choose **New Library** from the **Tools | Library** menu.

In the window that appears enter **My PCB Components** as the name, select **PCB Components** as the Type and then click on the **OK** button.

Next select the PCB layout symbol created earlier and choose **Add to Library** from the **Tools | Library** menu.

Another window will appear. Ensure that the new **My PCB Components** library has been selected and then click on the **OK** button.

Once done you will see the symbol appear in the gallery.

Step 5 of 5: Using the new symbols

Your new components are now ready to be used.

The circuit symbol appears in the **My Components** library within the **Circuit Symbol Gallery**.

You can drag it into your designs just as you would with any other circuit symbol. Once in your design you can wire it up to other components and then automatically generate a PCB layout.

When selecting **Design to Printed Circuit Board** from the **Tools** | **Convert** menu, the new circuit symbol will be automatically converted into the 1" resistor PCB layout symbol.
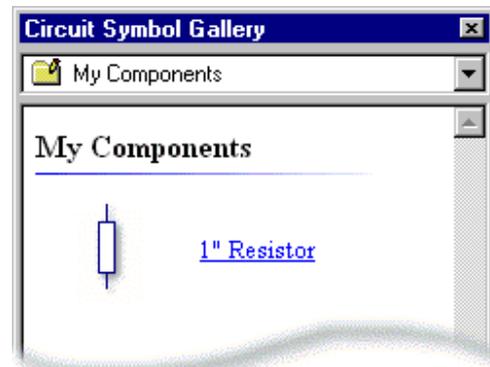
Alternatively if you are creating your PCB layouts manually, you can drag the new PCB layout directly into your design. It is available in the **My PCB Components** library within the **PCB Component Gallery**.