

PCB Artist Layout Software

Library Creation

Jung-Chun Lu

Nov-15-2012

Abstract

The public software, PCB Artist, serves the prototype of the board layout before fabrication. Library creation, including making new Schematic Symbols, PCB symbols and components, becomes a significant step in beginning PCB design. The software requires the well-established component created in the library so that the actual components are solderable on the board.

Table of Contents

Introduction	3
Objective	3
Symbol Libraries	3
Creating Schematic Symbols	4
To open the library manager	4
To open the symbol manager	4
Adding the symbol detail	5
Save the symbol	6
Creating Footprints	8
To open the symbol editor	8
The footprint	8
Using the footprint wizard	9
Creating Component	12
To open the component editor	12
To map pins to pads	13
Recommendations	15

Introduction

The printed circuit board is used to sustain and connect electronic components using conductive pathways, or the signal traces etched from copper sheets on the non-conductive substrate. PCB initially must be designed, laid out, and fabricated. As the time elapsed, the whole process becomes faster and cheaper and potentially make on the vast quantity of production. The PCB Artist is the open source software for design and layout supplying with a component parts library. The program is extremely useful to home-developing hardware and large scale manufacturers. Hence, many electronic devises make the use of PCB layout as standards.

Objective

The PCB Artist provides built-in libraries; those can be used, modified and added by a user's desires. A unique set of libraries can be matched to new and existing library items together. To ensure the dimensions are used to the specific component, the datasheet of package is essential. Once one understands the basic concepts of PCB Artist libraries, library creation is straightforward. The libraries consist of three individual libraries: a component library made up with elements from Schematic Symbol and PCB Symbol (Footprints) library. In this application note, the library creation of INA333 is provided as an example so that the library can be stored for repeating usage of the given project.

Symbol Libraries

Two categories of symbols libraries exist: the Schematic Symbols, with the extension .ssl, and the PCB Symbols, with the extension .psl. Unlimited symbols may be kept in a given library and the symbols may be used multiple times.

Creating Schematic Symbols

To open the library manager

From the **File** toolbar, select the **Libraries** icon (shortcut **Ctrl + L**).



Figure 1

To open the symbol manager

From the Library Manager dialog, select the **Schematic Symbol** tab.

We will create a new library for each schematic and PCB symbol. A user, of course, can add a new symbol with existing library. The Symbol Editor takes a set of the default design, taken from the Technology file, for example, Units, Grids, Styles and Colors. While being familiar with PCB Artist, a user may customize the files. For now, we will leave default set using it as a standard.

Clicking the **New Lib** tab will create a new library into which to save the new symbol. For the **File** name, type in the name **ECG_Design**. A new library will be ready to use by clicking **Save**.

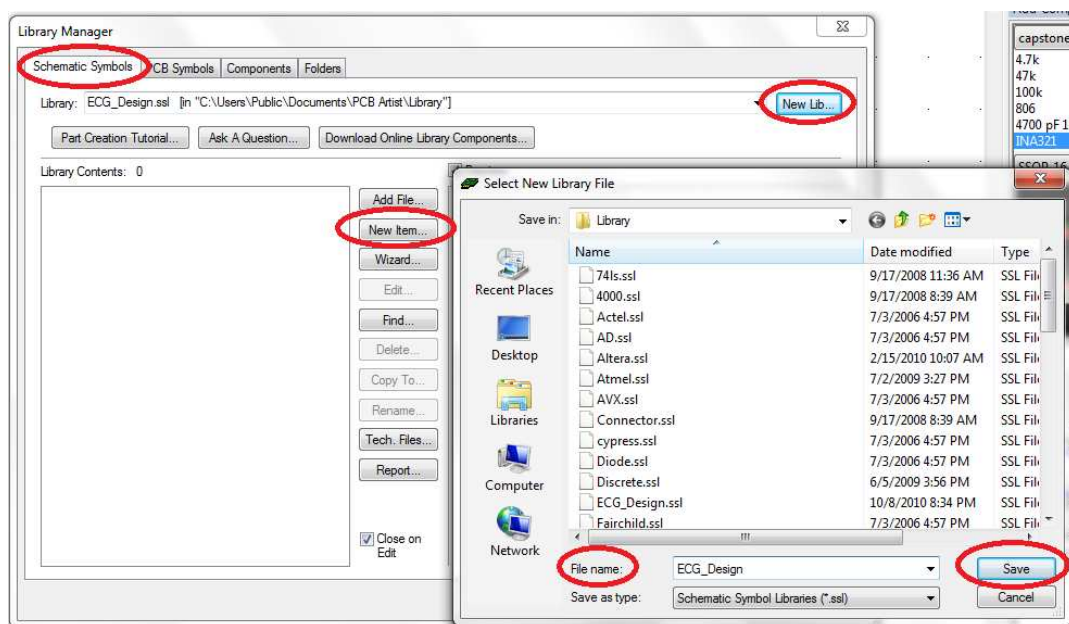


Figure 2

You will create a new Schematic Symbol so open the Symbol Editor using the **New Item** button.

Adding the symbol detail

First, ensure the grid **Step Size** is set to **50** mil from the **File** toolbar selecting the **Grid** icon so that we can the screen dots.

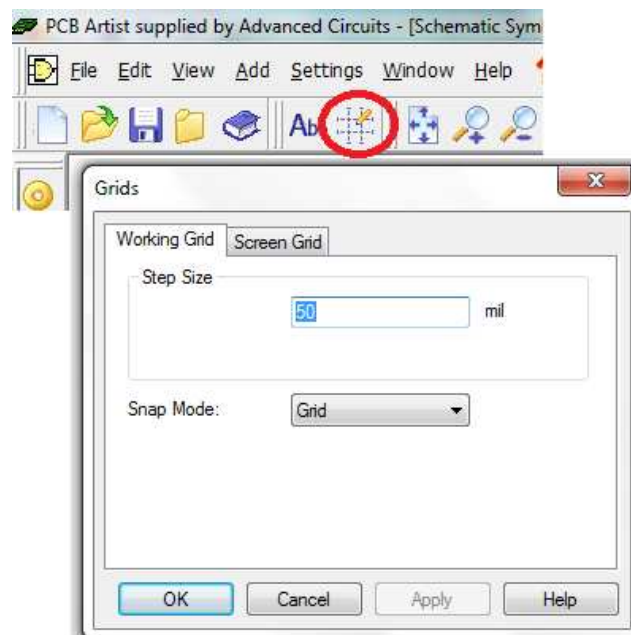


Figure 3

After checking the visible screen dots, we will draw the shape of instrumental operational amplifier. On the Schematic symbol toolbar (the vertical toolbar on the left side of the screen), select **Add Shape**. Choose a starting point and draw a triangular shape. The status bar at the bottom of the screen helps the size being drawn.



Figure 4

A few points are considered to add symbol details. The Symbol origin or "S" usually shows at the top left of the symbol, but it is movable. In order to get a triangular shape, the editing mode of **Orthogonal** needs to switch to free hand mode. To change to free hand mode, choose a starting point and right click to display the shortcut menu. From the menu, change **Segment Mode>** to **Free**. Once the shape and the lines are drawn, the pins are to be added to the symbol giving PCB Artist the real electrical connection. On the Schematic Symbol toolbar, select **Add Pad**. Position each pin at the end of the lines. As the pin is positioned, the next pin is in sequence. The last is adding the reference point which specifies the position of the component name. The **Reference Origin** is from the **Add** menu. Without add the reference origin, the component name will be automatically placed on the Symbol origin.

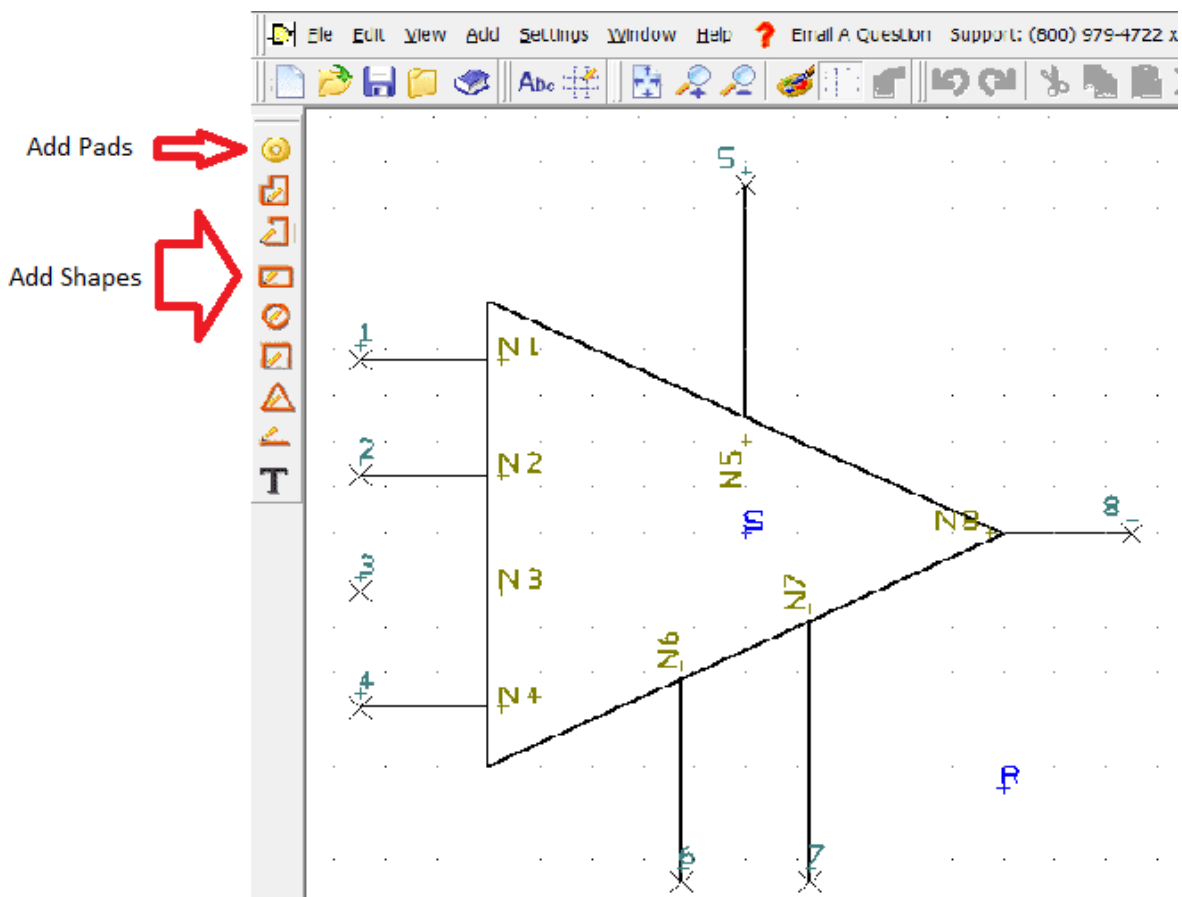


Figure 5

Save the symbol

From the **File** menu, select **Save**. The **Save** dialog opens and choose the **ECG_Desing.ssl**

library and give the symbol name.

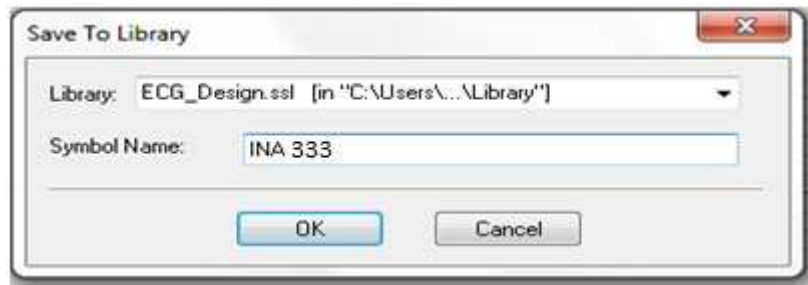


Figure 6

Open the Library Manager and click on the saved component in the Library Contents. Ensure the **Preview** box is checked.

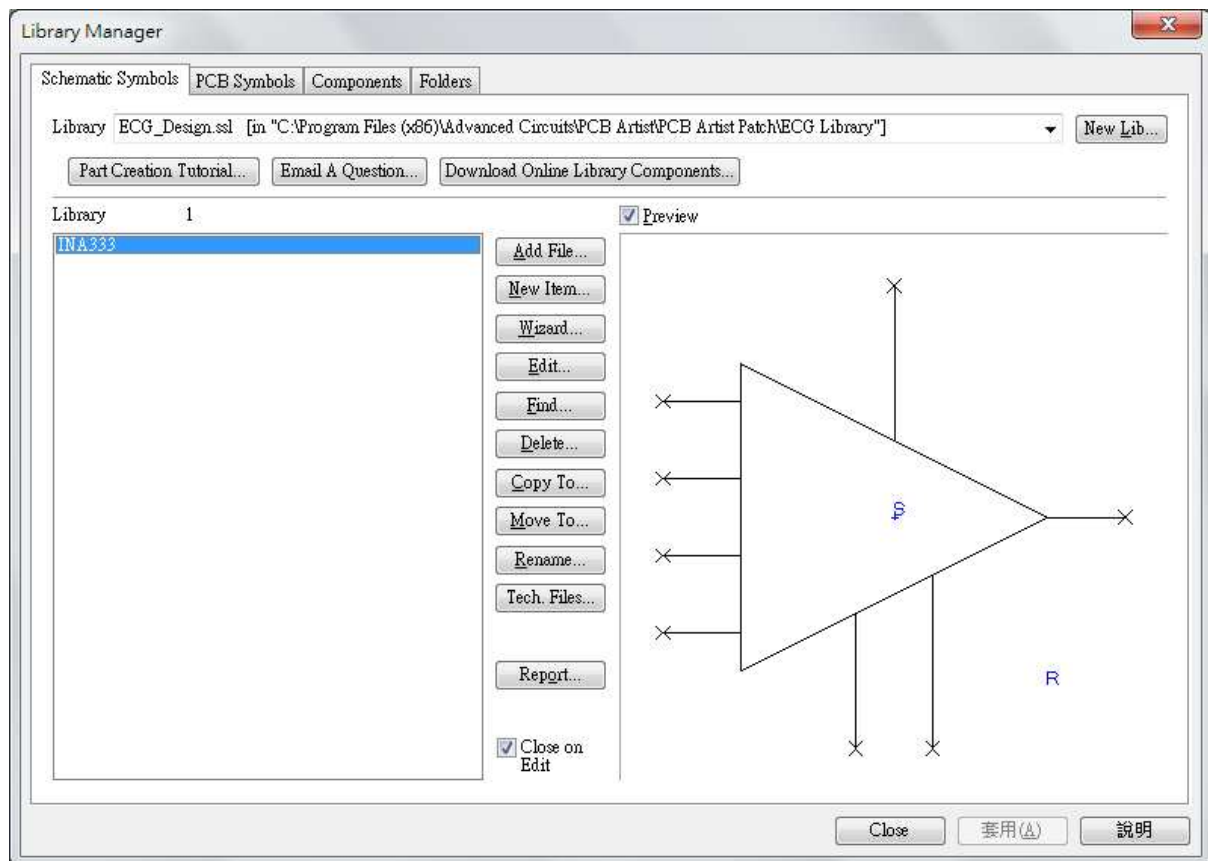


Figure 7

Once the schematic symbol is ensured, we move on to the creation of footprint.

Creating Footprints

To open the symbol editor

Open the Library (refer to Creating Schematic Symbols) and select the **PCB symbols** tab in Library manager dialog. Again, we need to start a new library, named **ECG_Design**, by clicking the **New Lib** button.

The footprint

Texas Instrument provides the datasheet for INA333, which can be found at <http://www.ti.com/lit/ds/symlink/ina333.pdf>. This offers the physical dimensions shown below. All the measurements are in millimeters.

DGK (S-PDSO-G8)

PLASTIC SMALL-OUTLINE PACKAGE

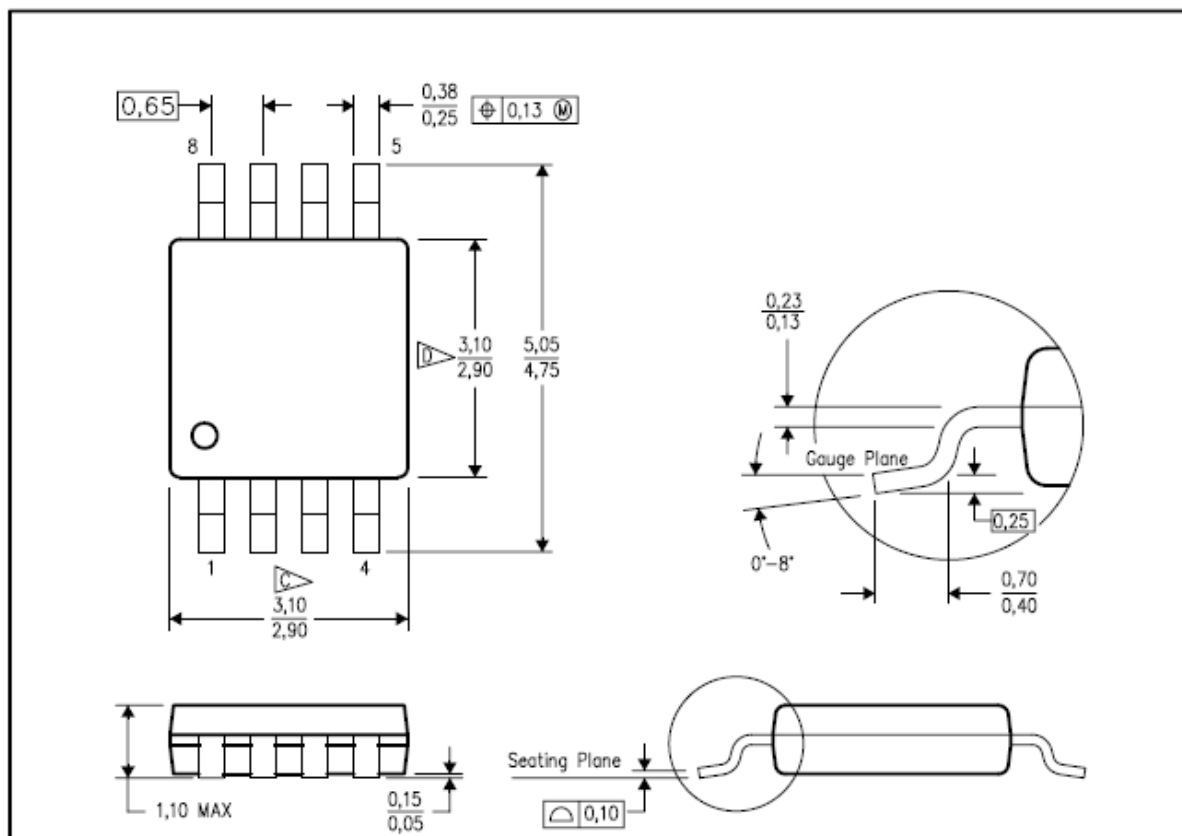


Figure 8

Using the footprint wizard

Clicking the **Wizard** button opens the footprint creating wizard. The first page you are presented with is the **Start**. Click **Next** button. The next page is to choose the **Technology** file. We will use the default technology file and change **Units** to the **Metric** and use the **Precision** of **2**. The measurements have a precision of 2 so we use the same precision. Click **Next>** to move on. In the Type page, select **SOIC** type, select **Center** to **Origin** and position the **Component Name** to the bottom right. For the surface mount component, it is common to place the origin at the middle. So, you will see the update preview on the right. Click **Next>** to move on.

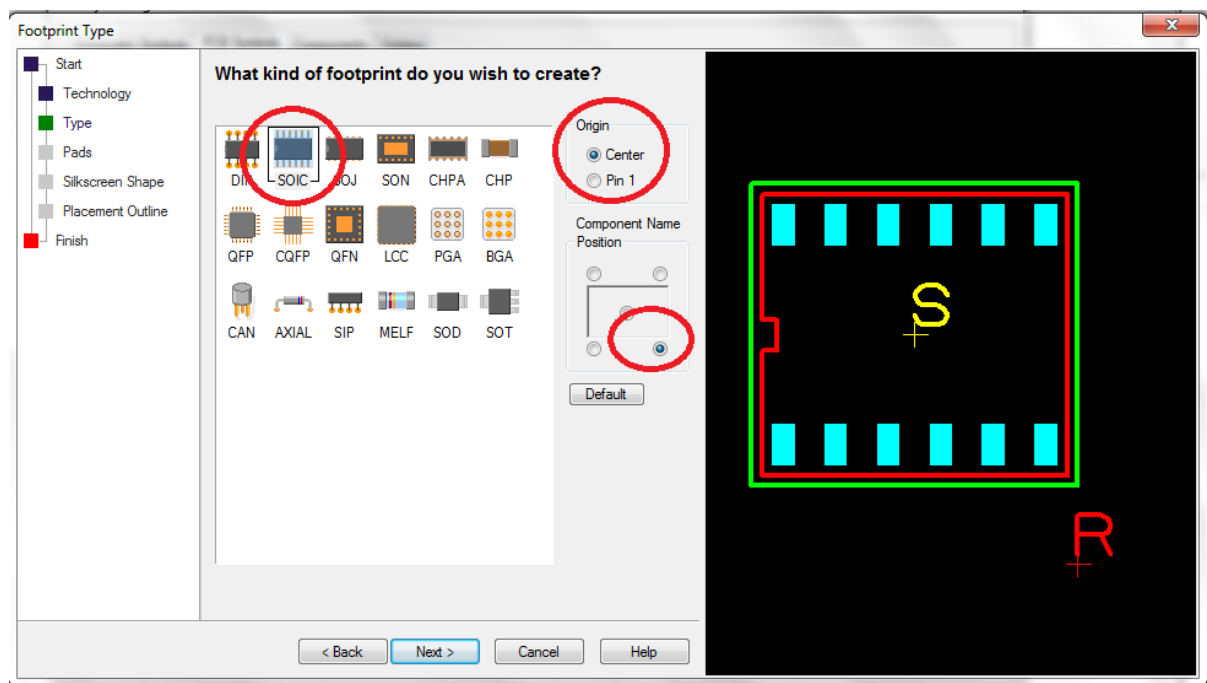


Figure 9

In the Pads page, you need to enter the measurements according to the datasheet. The name of pad style must be entered and saved. Choose the **Pin 1** position to be the bottom left **Hide Pin Numbers**, set **Pin Numbering** to be **Counter-clockwise**. Click **Next>** to move on. The **Figure 10** shows the setting.

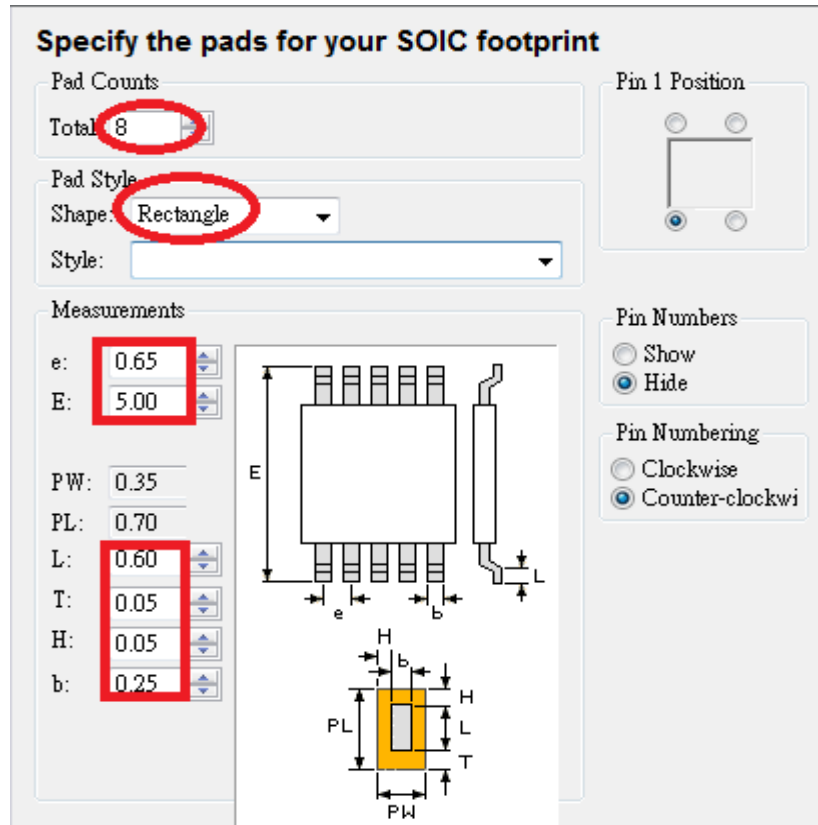


Figure 10

In the **Silkscreen Shape** page, the only parameter needs to be changed is the **Notch At End** value. Change this to **0.50**. Besides, the dialog is setup by default assuming that a symbol requires a silkscreen outline. Click **Next>** to move on.

In the **Placement Outline** page, add a placement shape, add to the desired layer and define the size of outside pads. For the surface mount device, the pad layer must be the top layer. Click **Next>** to move on.

In the **Finish** page, insure the **Footprint Name** is type, save to **ECG_Design.psl**, and uncheck the **Edit the Footprint now**.

After creating the footprint, preview the PCB symbol in the Library manger. The **Figure 11** illustrates the previewed PCB symbol.

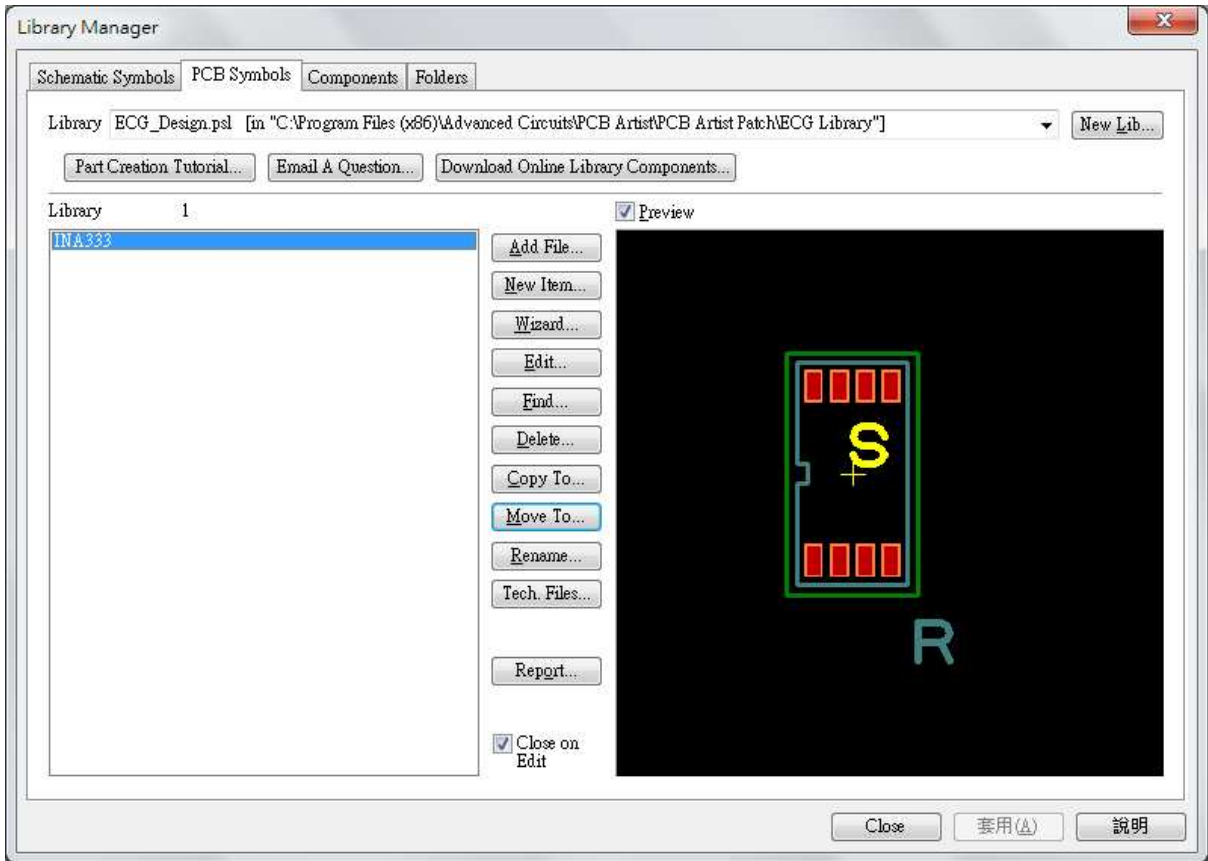


Figure 11

The footprint is ready to use in a component.

Creating Component

In this application note, a component is created using the previous Schematic and PCB symbols, but equally applied to create using other Schematic and PCB symbols.

To open the component editor

Open the Library manager, select the **Components** tab and start a new component lib by clicking **New Lib** button. In the **File name** box, type **ECG_Design** as new component library. Clicking save button lets the library be ready to enter new components. The **New Component** dialog would open by clicking the **New Item** button. Some information is required before moving on.

- In the **Component** box, type a component name INA333.
- In the **Package** box, select **SOIC**.
- In the **Default Reference** box, type **U**. This will be seen on the silk screen of the PCB or the component name used.
- For the **Schematic Symbol**, make sure the library is located to ECG_Design.ssl. Select the previously created Schematic Symbol from the list. The number of pins will automatically show in the dialog.
- For the **PCB Symbol**, ensure the library is located to **ECG_Design.psl**. Select the previously created PCB Symbol. The pins box should be 8.
- The dialog should look like the **Figure12**.

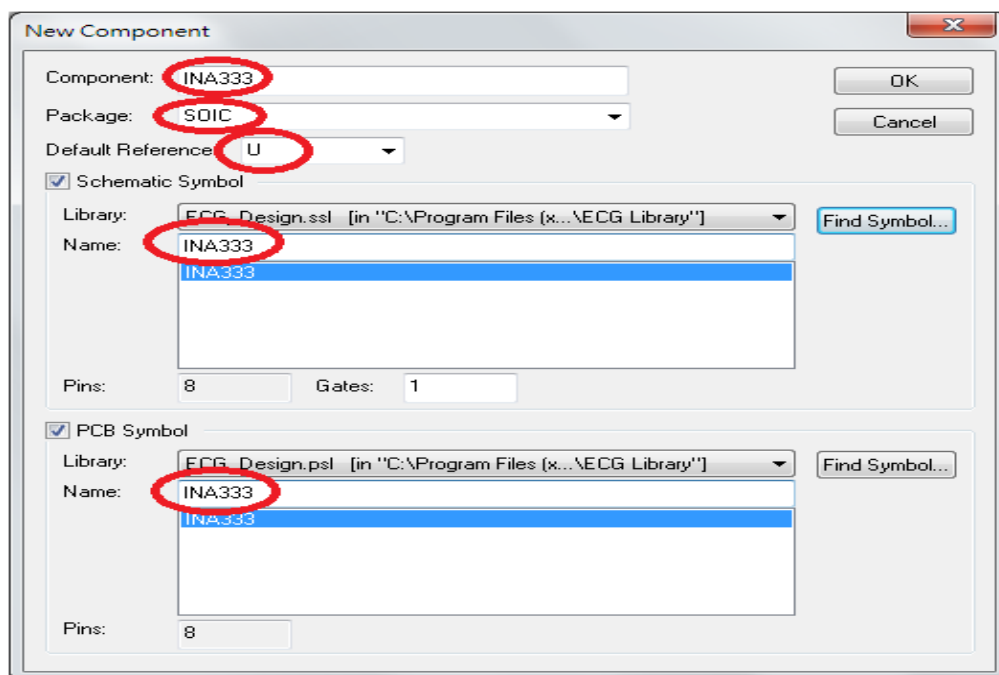


Figure 12

Click **OK** to continue. The **Component Editor** opens that shows the items added from the **New Component** dialog.

To map pins to pads

For a component to work, the pins need to be defined in the Schematics symbol and PCB symbol. The mapping exists in the 'grid' format in the component editor. Three methods of mapping can be used: 1) directly typing the pins and pads number in the grid cell; 2) in the **Assign Pins** modes, click on the Schematic Symbol pin and the equivalent PCB symbol pin; and 3) automatically assign pins on a 1:1 basis, Schematic to PCB symbol. If the pins of 256 BGA needs to be assigned, the automatically assign pins is preferred. We only require manually assign pins considering the pin configurations from the **Figure 13**.



Figure 13

After the pins are assigned, the component editor may look like the **Figure 14**.

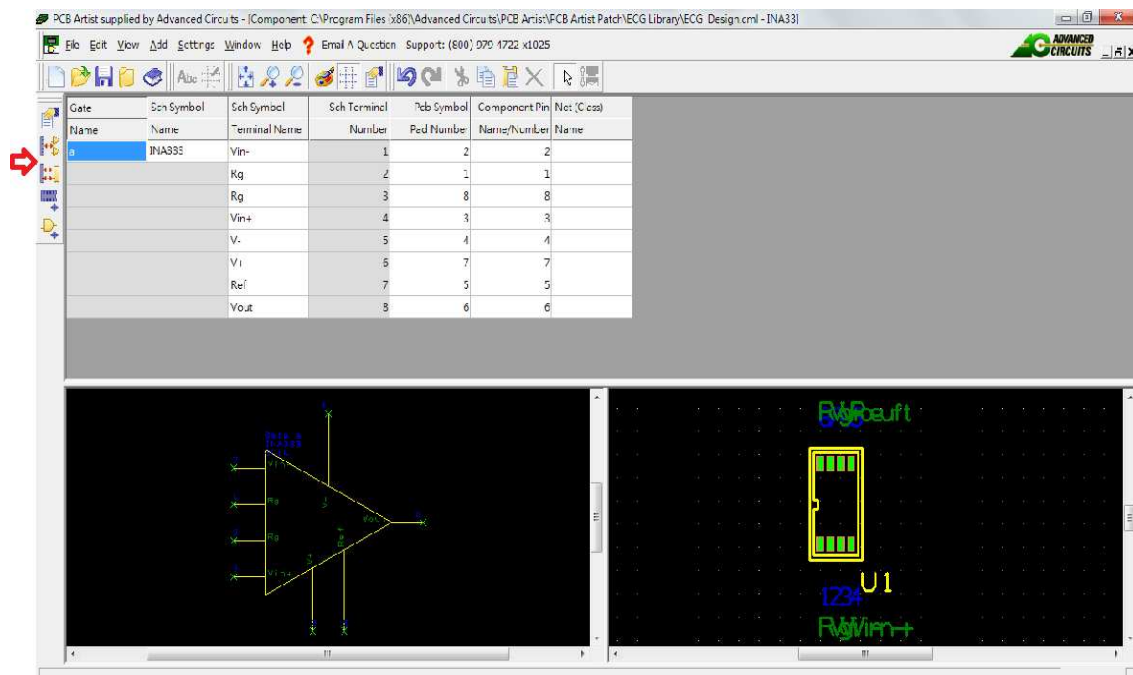


Figure 14

From the **File** toolbar, select **Save <Ctrl+S>** will open the **Save To Library** dialog. Click **OK** once the library is chosen to **ECG_Design.cml** and the component is given. Then, the component can be previewed in the Library manager.

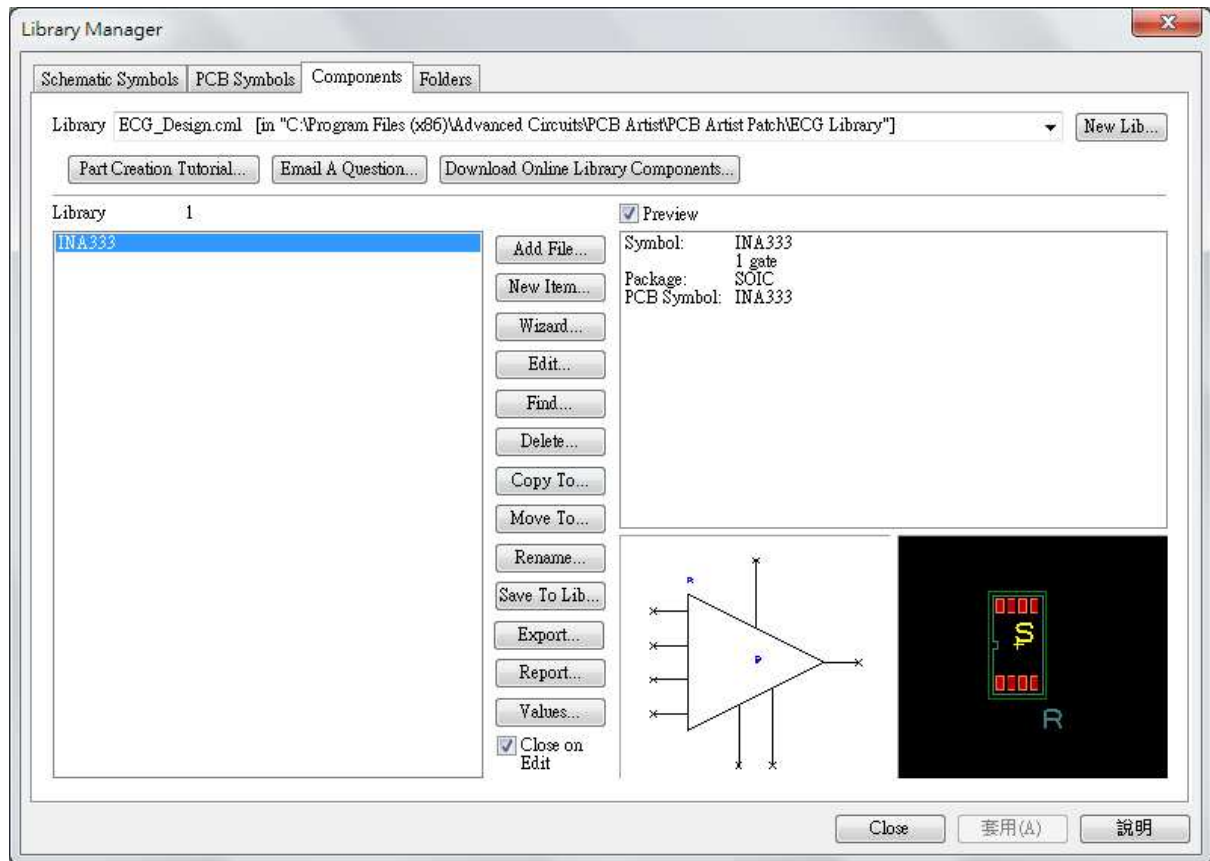


Figure 15

At this stage, the component is created and can be added to the schematic or the PCB layout.

Recommendations

Most of passive components are in the **Discrete** library. If the capacitor or inductor has a different PCB symbol, it can be easily created using the **Wizard** with existing PCB symbols. This will allow a user to design each capacitor and inductor for the actual ordered component.

A few points need to be kept in mind when creating the library for the schematic and PCB symbol. The package type and size must be found before the library creation. One can be lost and stuck in the creating the library forever. Also, mapping the pins to the correct pads is important to have the exact functions of each component. Inaccurate pad assigning makes the component not being solderable on the printed circuit board. Moreover, keeping the library in the same folder is essential when converting schematic-to-PCB. PCB Artist would not be identified the components while the loss of the library