

PCB Artist Tutorial

Table of Contents

Installing the Software

To install the PCB Artist software, go to http://download.cnet.com/PCBArtist/3000-18496_410712335.html?part=dIPCBArtist&tag=button and click the download now button. Once the download is complete, the software is ready to run.

Starting PCB Artist

During installation, a PCB Artist program icon is automatically added to your desktop and an Advanced Circuits program group is added to the **Start** menu and **Programs**. To start PCB Artist, simply double-click on the **PCB Artist** icon on your desktop or select the PCB Artist menu item.

Library Creation

PCB Artist is supplied with a set of libraries. These libraries can be used, modified and added to as required. You can mix and match new and existing library items together to create your own set of unique libraries. To insure the correct dimensions are used for the specific components, a new library with custom components is essential.

PCB Artist libraries are easy to understand and use once you grasp the basic concepts. The libraries are made up of three individual libraries. There is a **Component library** made-up of elements from the **Schematic Symbol** library and the **PCB Symbol** (footprint) library.

We only want to store a schematic symbol once and use it repeatedly in various configurations to produce different components. For instance, for the **INA321** amplifier we have a **PCB symbol library** to store our 8-pin INA PCB footprint plus any other footprints and a **Schematic symbol library** for our amplifier plus any other Schematic symbols. These can then be used on their own or brought together (Schematic symbol & PCB symbol) in many different permutations to create a **Component Library**. We also have multiple Component libraries in PCB Artist to make the categorization of components clearer, these can be categorized by manufacturer, type, functionality etc.

You can create a new Schematic symbol and store it in the Schematic symbol library. But before you can use it in a design you have to create a **Component** for the symbol and store this in the **Component library** for use in your designs. Typically, this is how power and ground symbols would be defined for example.

Working this way, any Schematic Symbols you produce are stored with an appropriate name in the Schematic symbol library. It can then be reused multiple times to create different components using the manufacturer's component name. Most components you produce will have the complete Schematic symbol and PCB symbol

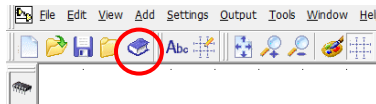
Schematic and PCB Symbol Libraries

Two types of symbol libraries exist, those for Schematic symbols with the extension .ssl, and those for PCB symbols with the extension .psl. There is no limit how many symbols may be kept in a given library.

Creating Schematic Symbols

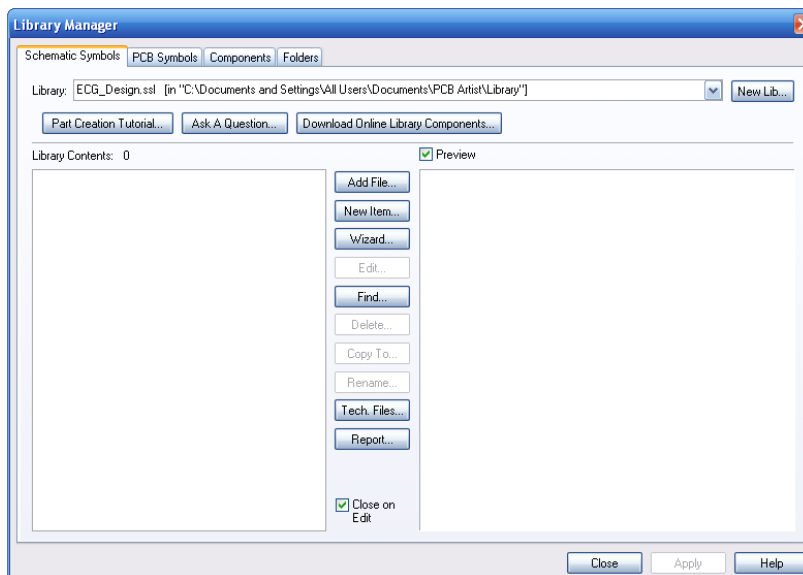
➤ **To open the library manager**

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



➤ **To open the symbol manager**

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

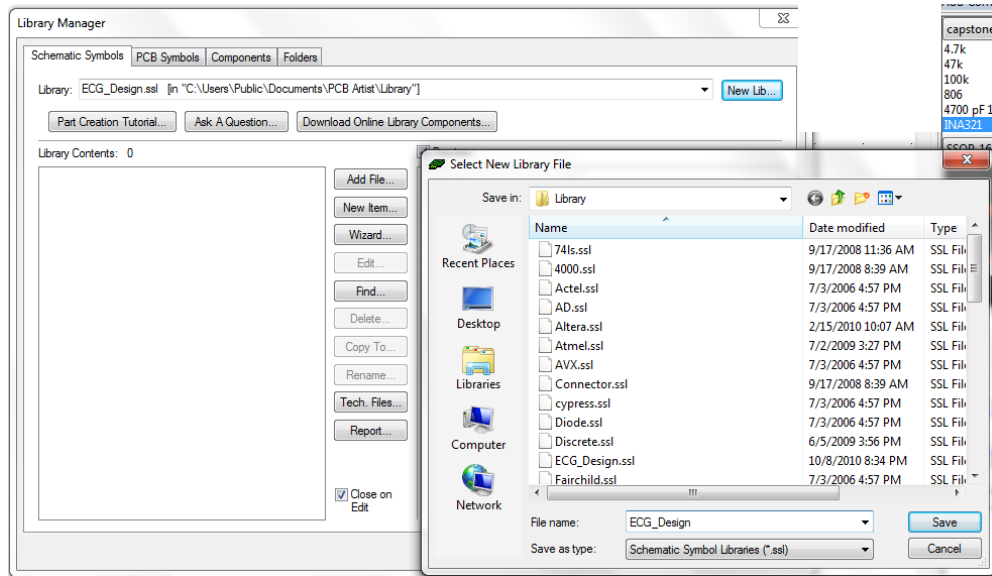


For this tutorial, we will create a new library for each symbol/component type. You could of course, add your new symbols to an existing library.

The Symbol Editor will take a set of default design settings, such as Styles, Colors, Grids, Units, etc. These are taken from a selected Technology file. Once you become familiar with PCB Artist, you can customize these files using your own selections. If you click on the **Tech Files** button on this dialog, you will see how to choose this Technology file.

For now we will leave the defaults set to the ones supplied in the standard technology file.

We will create a new library in which to save the new symbol into. Click the **New Lib** tab.

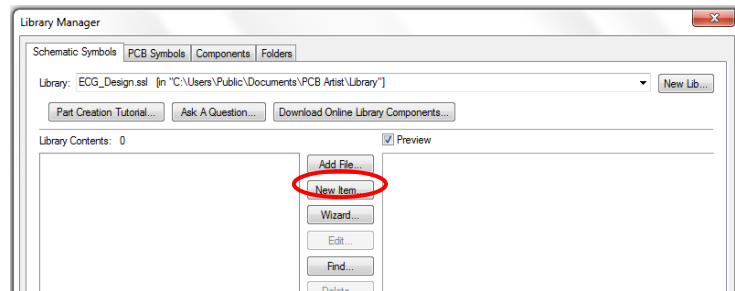


In the **File name:** box, type in the name **ECG_Design**. This will be the name of your new library.

Click **Save** to save this file name and exit the dialog.

The new Tutorial library is selected and is ready to have new symbols saved into it.

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



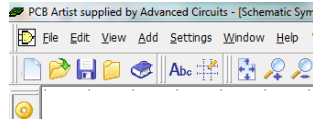
For this tutorial exercise, we are going to create an instrumentational-amp component. The component will contain two symbols, both different. It will use an intrumentational-amp gate plus a power gate, we will do this to demonstrate using multiple gates for one component.

You will create the op-amp gate only, the power gate (PWR) already exists in the supplied libraries.

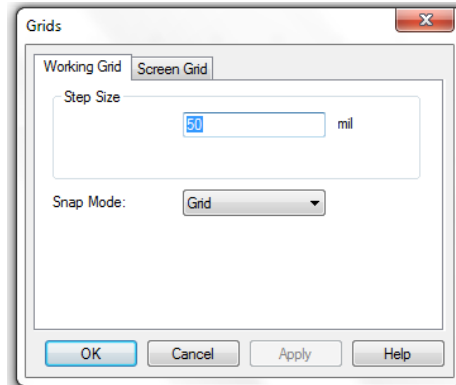
➤ **Checking the screen dots**

Firstly, zoom in (roll the mouse wheel or click the **<Z>** key) so you can see the **Screen Grid** displayed. The grid needs to be set to 50 mil so check this.

From the **File** toolbar select the **Grids** icon.



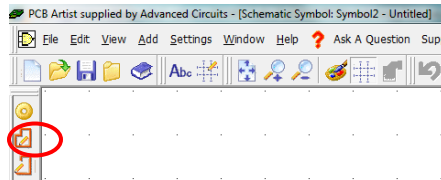
The Grids dialog is displayed.



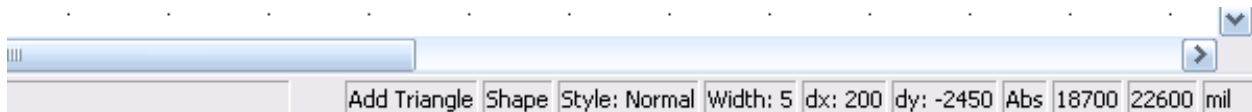
Ensure the grid **Step Size** is set to **50** mil steps and click **OK**.

➤ **Adding the symbol detail**

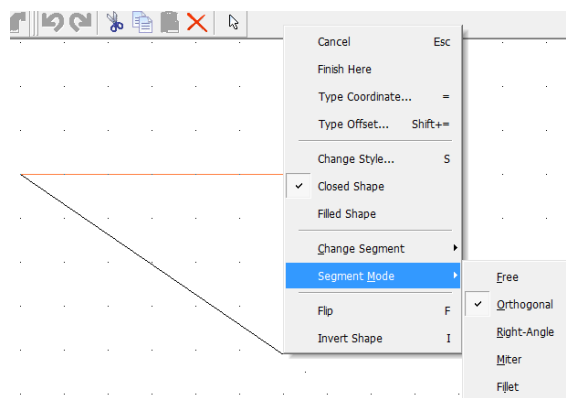
On the Schematic Symbol toolbar (the vertical toolbar on the left side of the screen) select **Add Shape** (polygon).



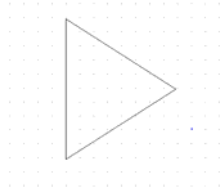
Choose a starting point and create a shape 500 mil long moving down. The status bar at the bottom of the screen will show you the size being drawn.



Notice the Symbol origin point '**S**' at the top left of the shape. We will move this later on to a more relevant location. This is always added to the original of the first symbol item added during creation. In order to get the triangle shape, you will need to take the editing mode out of **Orthogonal** and switch it into **Free** hand editing. To change to Free hand mode, click once to add a corner, 500 mil down (10 grid points) from the start point and then right click to display the shortcut menu. From the menu, select **Segment Mode>** and **Free**.



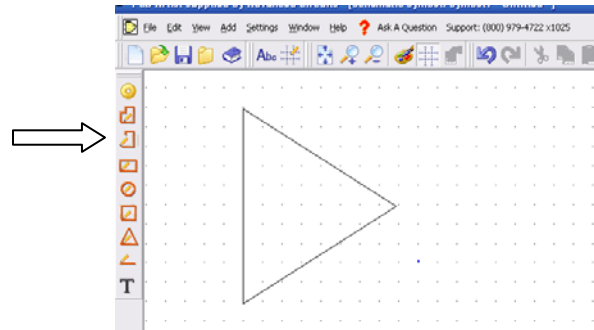
Now you can move the 'floating' second segment right 500 mil and up 250 mil to give you the point on the triangle. Because the grid is set to 50 mil steps you will be able to snap to 250 mils exactly.



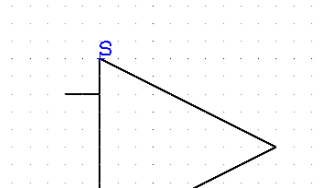
➤ **Adding the symbol detail**

You may need a bit more space around the existing shape to add some additional shape detail. Use the pan function <P> on the keyboard or press the middle wheel button, or use zoom out or zoom in.

On the **Schematic Symbol** toolbar, select **Add Shape Line**.

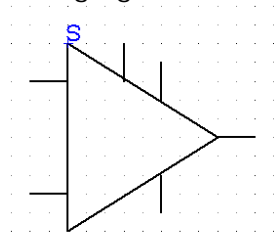


Select a start point 100 mils down from the symbol origin 'S' on the vertical segment (2 grid points), drag a line 100 mils (2 grid points) to the left. Like this:



Use a single click to finish.

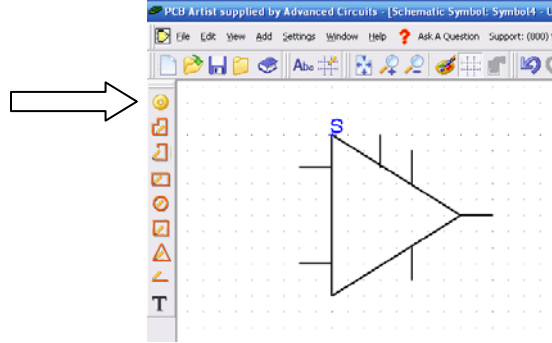
Add the additional lines required by selecting the line you have just inserted and clicking <Ctrl-C> to copy and <Ctrl-V> to paste. Do this twice. Of course you can always simply draw them in using the Add Shape Line tool as well. Your finished shape should look like the following figure.



➤ **Adding pins**

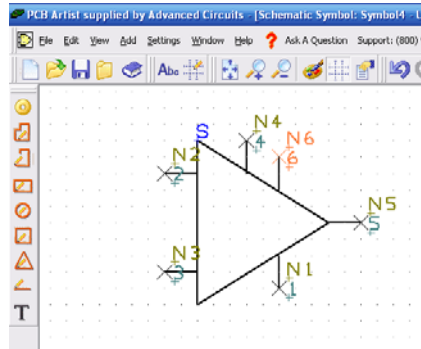
Pins need to be added to the Symbol to give PCB Artist 'real' electrical points on which to make connections. This gives PCB Artist its unique true electrical connectivity at all times.

On the Schematic Symbol toolbar, select **Add Pad**.



A pin will be available on the end of your cursor shown in the highlight color.

Position each pin on the end of the lines so that it looks like the example below. As each pin is positioned, the next one in the sequence is available on the end of your cursor.



You will see that these are auto-numbered as you proceed. After the last pin has been added, click the **<Esc>** key to cancel.

These pins must be sequential and for this example we will number them according to the picture shown above. In general, the sequence does not matter. They must also be unique and with no gaps in the sequence.

Click the **<A>** key to position the symbol to fit the window.

➤ **Information on the pins**

Each pin contains three pieces of information:

- The big cross marking the shape and location of the pin
- The Pin Number (1,2,3 etc.)
- The name position marker, N1,N2,N3,etc. which marks the location of the pin name when used on the Component in the design

The **Pin name position marker (N3)** will only be populated once the Component is used in the Schematic design, until then you only see the name position marker. This



You may also see the symbol origin 'S' character on Pin 1 if this pin was the first item to be added to the symbol (before the symbol shape).

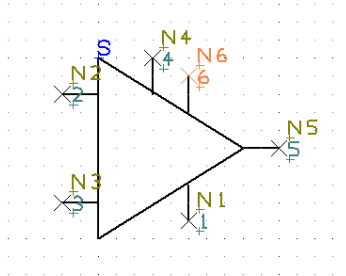
The **Pin Number** is then automatically numbered. This can represent the actual terminal number if required but can be renumbered in the Component definition. The number is shown in the position it will appear in when used in the design. This can be moved in the Symbol editor but cannot be moved once placed in the design.

➤ **Moving the symbol origin**

The Symbol origin is the Symbol's reference point used when moving it in the Schematic design (the small 'S' marker). For consistency we always try and place this in the same relative location, usually on Pin 1 of the symbol.

The symbol origin would have been added with the first design item. In our example, this was the symbol shape but it could have been the first pin added.

Select the origin on the symbol and drag the symbol origin to Pin 1 as shown below.



➤ **Moving Pin names**

Select and drag the pin numbers (pins 1, 2, 3, 4, 5 and 6) and pin names (pins N1, N2, N3, N4, N5 and N6) and move them into the required location making sure only the name is selected and not the pin itself (the big X).

To make sure the positioning of pin names and numbers easier and more refined, during the move, use the shortcut keys <Ctrl+G> to display the snap mode. Change the snap to a **Half Grid** or **Quarter Grid**.

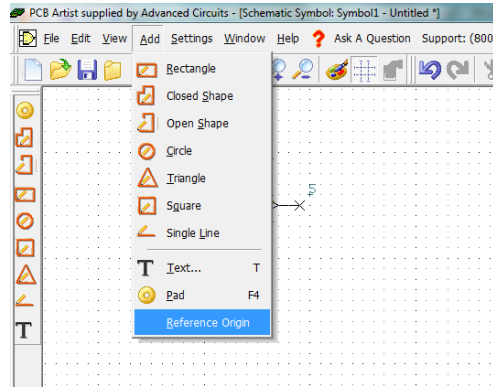
Position the pins to look like this:

➤ **Adding the reference origin**

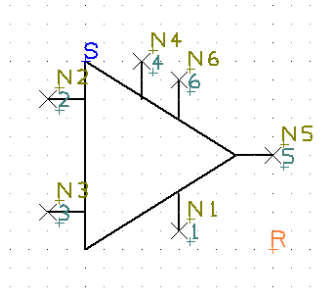
Without a **Reference Origin**, the component name (U1, IC1, etc.) used in the Schematic design will automatically be placed on the Symbol Origin. By adding a specific Reference Origin you can position the Component name where you like.

By default, the PCB Artist libraries have the Reference Origin placed at the bottom right side of the Symbol shape.

To add the Reference Origin, select **Reference Origin** from the **Add** menu.



Place the 'R' marker on the Symbol.

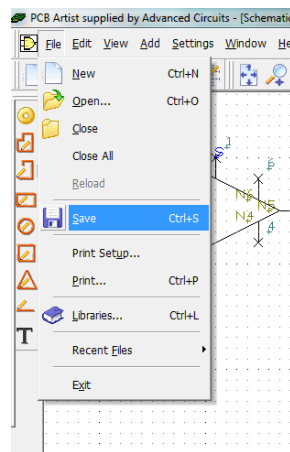


When the Symbol is used with the component in the design, the Reference Origin is also used for positioning the Component Name, Package Name and PCB Symbol Name if these items are selected as displayed.

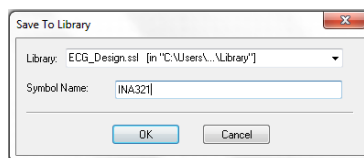
You are now ready to save the symbol into the library.

➤ **Saving the symbol**

From the **File** menu, select **Save**.



The **Save** dialog opens.



Choose the **ECG_Design.ssi** library and give the symbol and the name **ina**, click **OK**.

➤ **Viewing the symbol in the library**

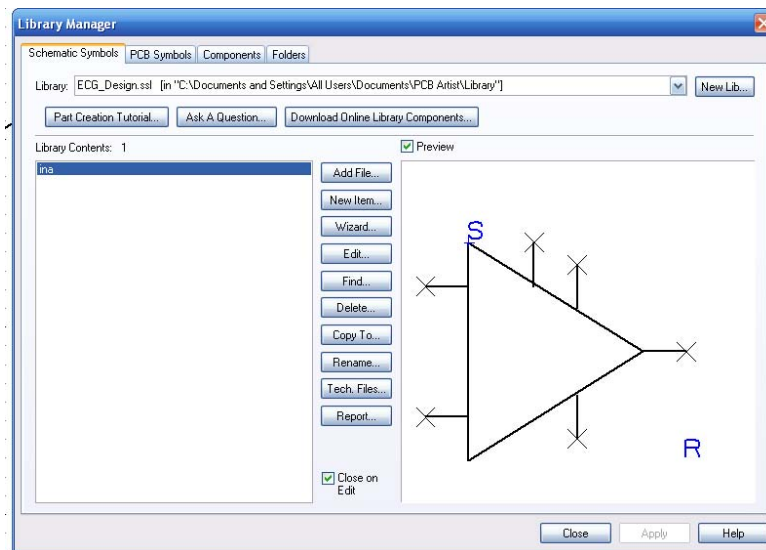
Now that the symbol has been created and saved, let's view it in the library manager.

From the **File** toolbar, select **Libraries**, you can also use the **<Ctrl-L>** shortcut key.

The Libraries dialog opens. Select the Schematic Symbols tab if it isn't already selected and select the **Tutorial.ssi** library from the drop down list.

Ensure the **Preview** box is checked.

From the **Library Contents:** list, select the **INAMP** symbol that you've created, it should look like the one below:

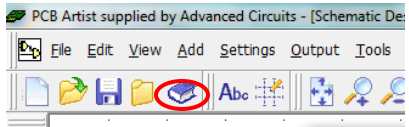


Once you are satisfied with this, move onto the next section to create a PCB Symbol (footprint).

Creating PCB Symbols (Footprints)

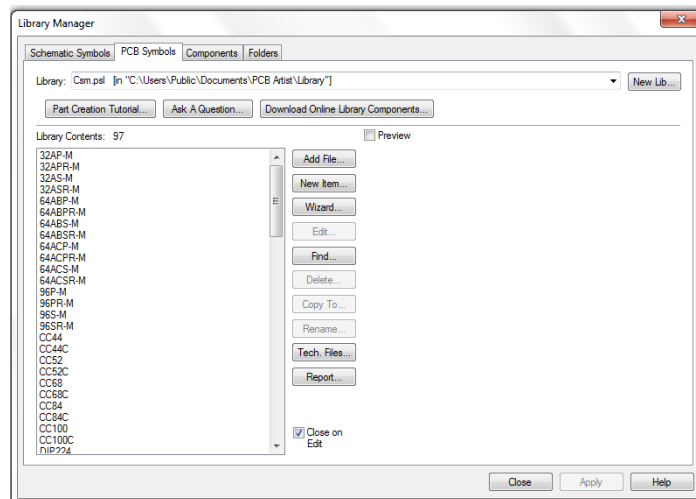
➤ Start the Library Manager

From the **File** menu, select the **Libraries** icon (Ctrl+L)



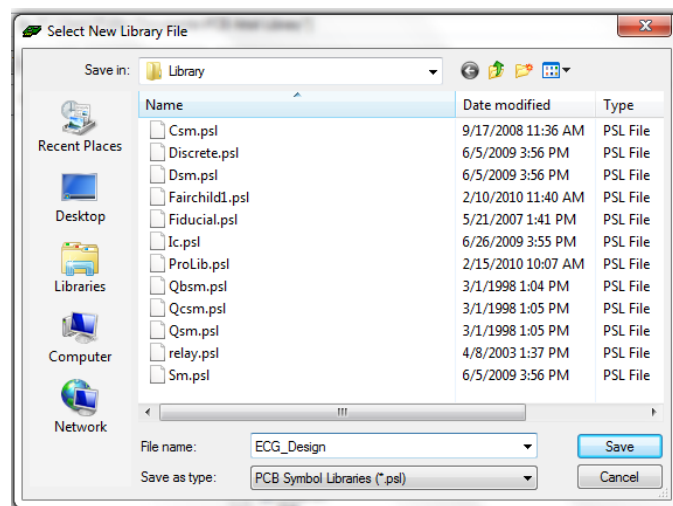
➤ To open the symbol editor

From the **Library** manager dialog select the **PCB Symbols** tab



For this tutorial we create a new library to add your new PCB Symbol to.

Click the **New Lib** button.

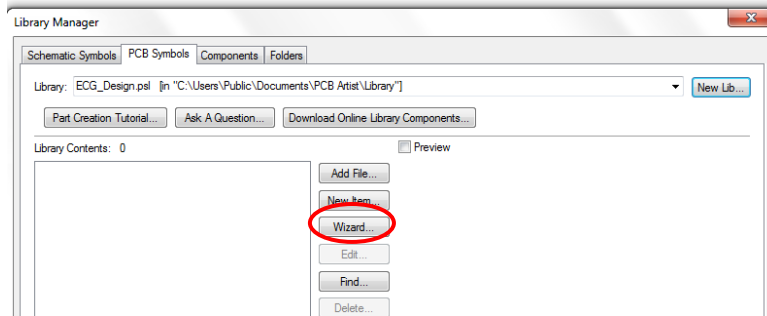


In the **File name** box, type **ECG_Design** as your new PCB symbol library. Click **Save** to accept this. The library manager is now ready to start entering new PCB symbols.

➤ **To Start the PCB symbol wizard**

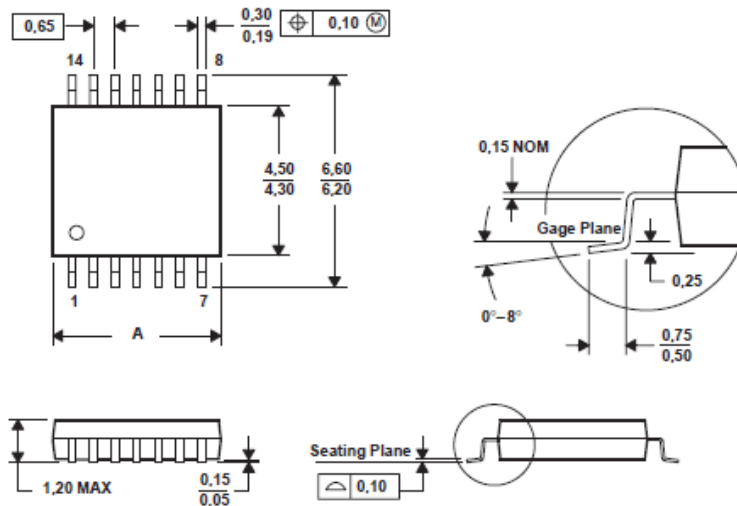
For this tutorial, we will use the PCB footprint wizard. For footprints that need calculating dimensions for, this is the easiest way to create them. Since the ECG board is an extremely small and design specific board, this step should be done for each individual part on the schematic and PCB board.

From the **Library Manager** window, click the **Wizard** button. Ensure that you are on the **PCB Symbols** page.



➤ **The Footprint**

For this tutorial, we will design the PCB Symbol for the INA321. The manufacturer's datasheet, which can be found at <http://focus.ti.com/lit/ds/symlink/ina321.pdf>, provides us with footprint pattern dimensions. The footprint, from the datasheet, is shown below.



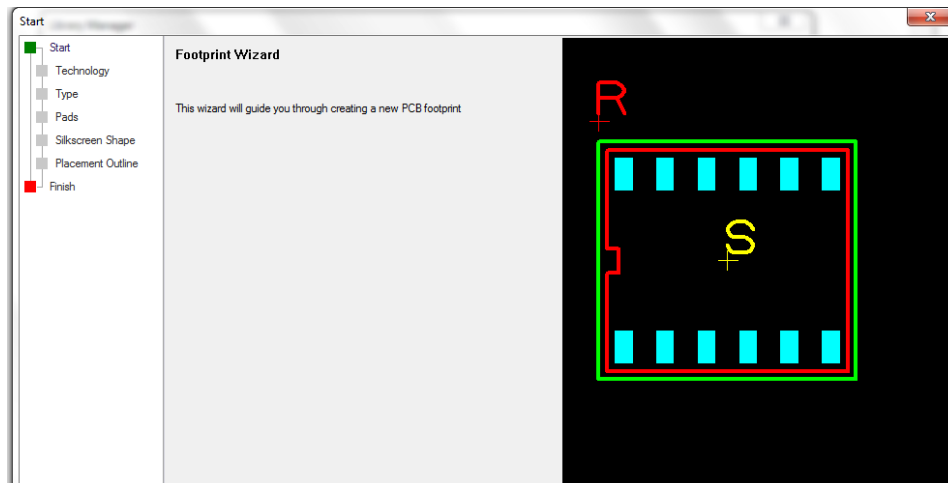
DIM \ PINS **	PINS **					
	8	14	16	20	24	28
A MAX	3,10	5,10	5,10	6,60	7,90	9,80
A MIN	2,90	4,90	4,90	6,40	7,70	9,60

➤ Using the footprint wizard

Click the **Wizard** button.

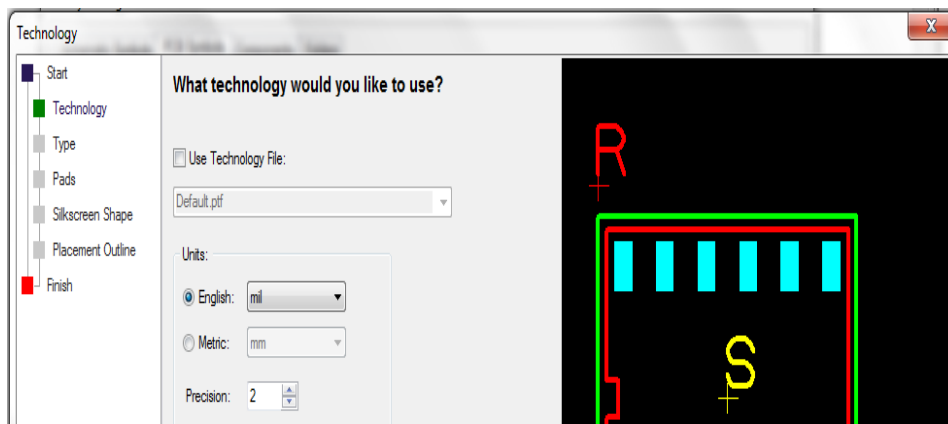
From our example footprint shown above, the other aspect you'll notice is how exact the range of dimensions is. This exactness also gives you a level of vagueness as you need to know how the footprint will be manufactured before being able to complete it.

The first page you are presented with is **Start**.



Click the **Next>** button to move forward and start the new symbol. Across the bottom of the page are always located the **<Back, Next>**, **Cancel** and **Help** buttons, which may be used at any time during the wizard.

The next page is used to choose a possible **Technology** file and the **Units** to be used.



No technology file will be used. We will change the **Units** to **Metric** and the **Precision** of **2**. We will use a precision of 2 because our datasheet measurements are within that same precision. **Our footprint is very small and will require a pin pitch of .5mm.**

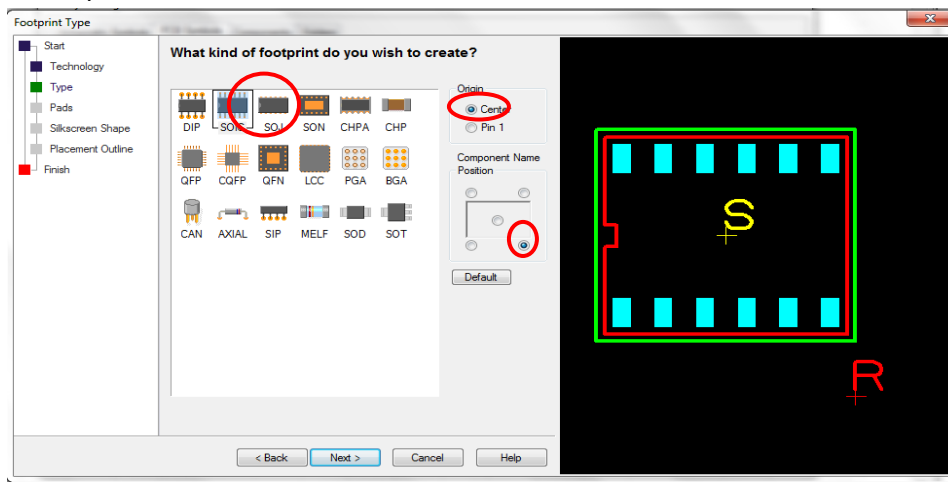
The preview pane (shown on the right in the **Footprint Type** window) will display a symbol. This symbol is a graphical representation of the current shaped and differ depending on the component you are designing.

Click **Next>** to move on.

On the type page, select the **SOIC** shape. Your preview will now change to represent a **SOIC** device.

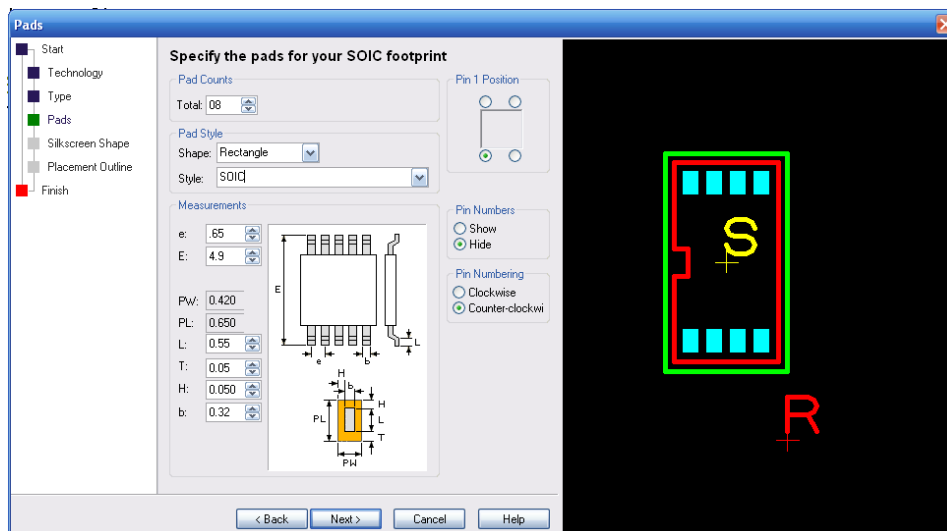
On **Origin**, select **Center**. Since the ECG board has such tight space requirements, all components for the ECG PCB design will be surface mount. For a surface mount (SM) component, having the **Origin** at **Center** is normal.

For **Component Name** position, click the bottom right radio button. You'll see the preview update.



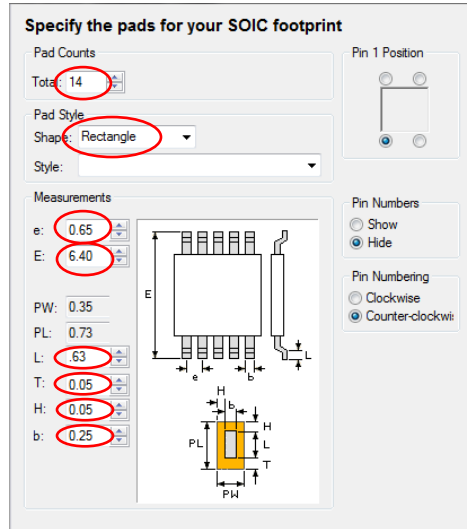
Click **Next>** to move on.

The **Pads** page is used to define the overall footprint layout. This is probably the most important page of the wizard. The default page setup is shown below, we need to change the parameters defined here.



From the given datasheet, observe that the INA321 has 8 pins. Since the INA321 has 8, the **Pad Count** should be set to a **Total** of 8. You can use the spinner or type

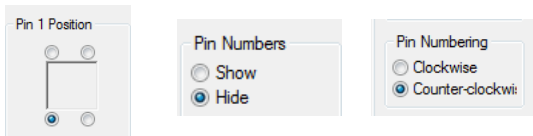
in the number to change it. You must type in the name of the pad **Style** that will get saved once you've edited the dimensions. Use **Rectangle** for the specific pad style.



In **Measurements**, you need to enter the dimensions to be used. These dimensions are located on the datasheet provided, or you can find it at the Texas Instruments link provided: <http://focus.ti.com/lit/ds/symlink/ina321.pdf>. The correct dimensions have been highlighted for our footprint; all are in mm.

e	0.65
E	4.90
PW	Automatically calculated
PL	Automatically calculated
L	0.55
T	0.05
H	0.05
b	0.32

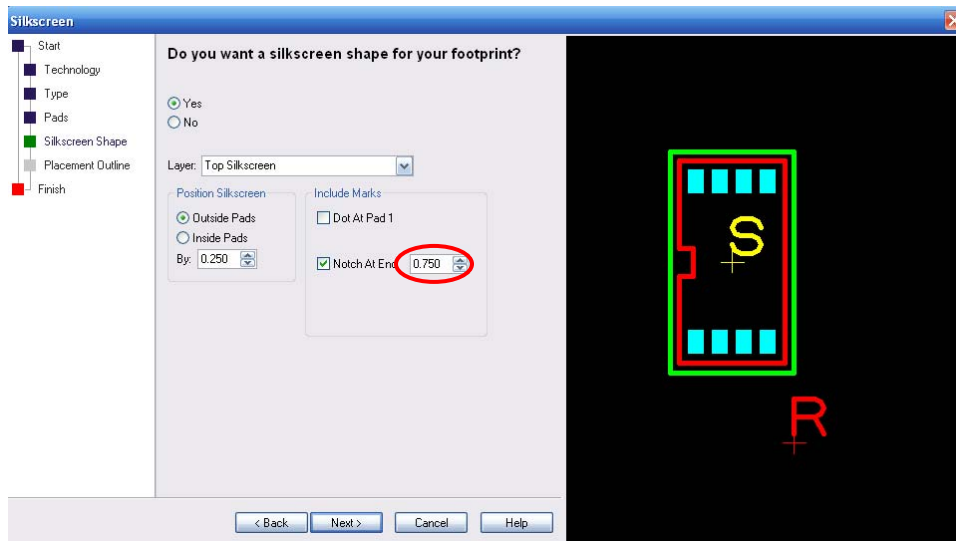
You must now choose the position for **Pin 1** to be the bottom left corner, to **Hide Pin Numbers** and the **Pin Numbering** to be **Counter-clockwise**.



Click **Next>** to move on.

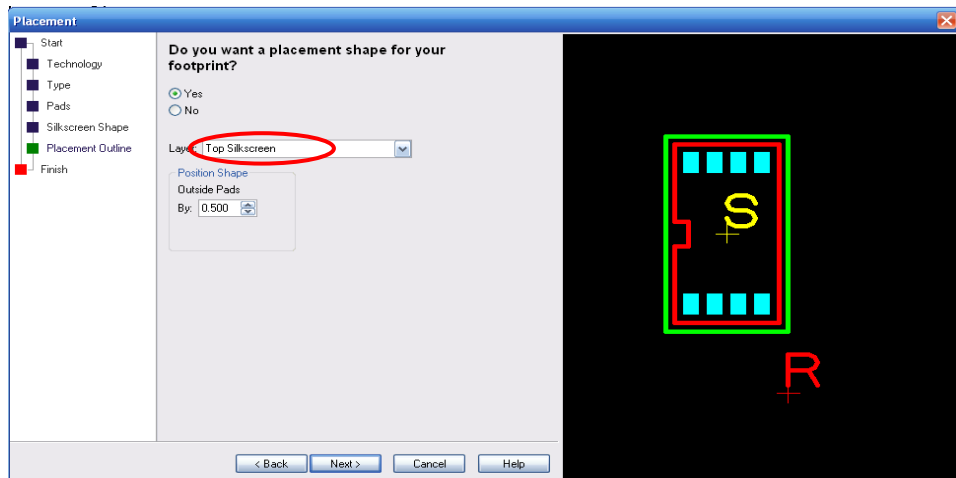
You should now define the **Silkscreen** outline.

This dialog is setup by default to assume you will always require a silkscreen outline. The only parameter to change here is the **Notch At End** value, change this to **0.50**.



Click **Next>** to move on.

We will use the placement shape and add it to the **Documenta1** layer. Select this layer from the drop down list.



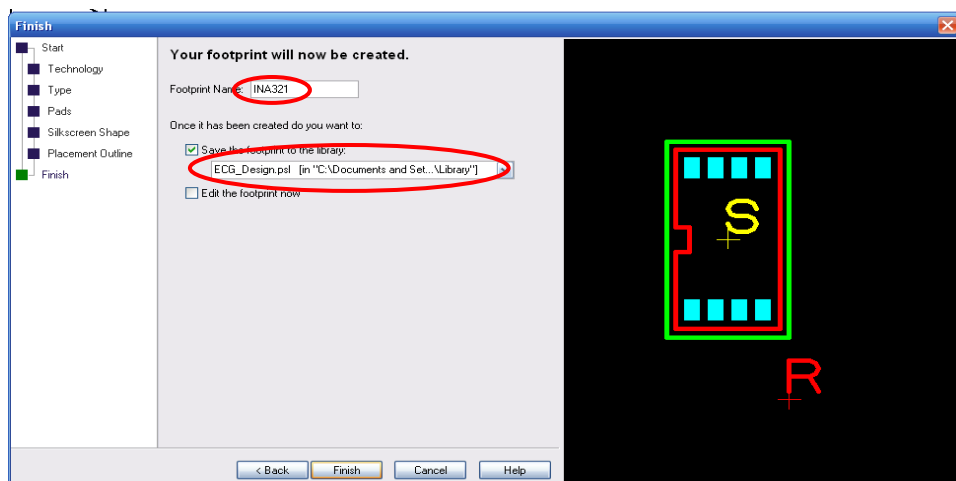
Click **Next>** to move on.

In the **Footprint Name:** box, type **INA321**.

Check the **Save the footprint to the library** box.

Select your **ECG_Design.psl** library from the drop down list.

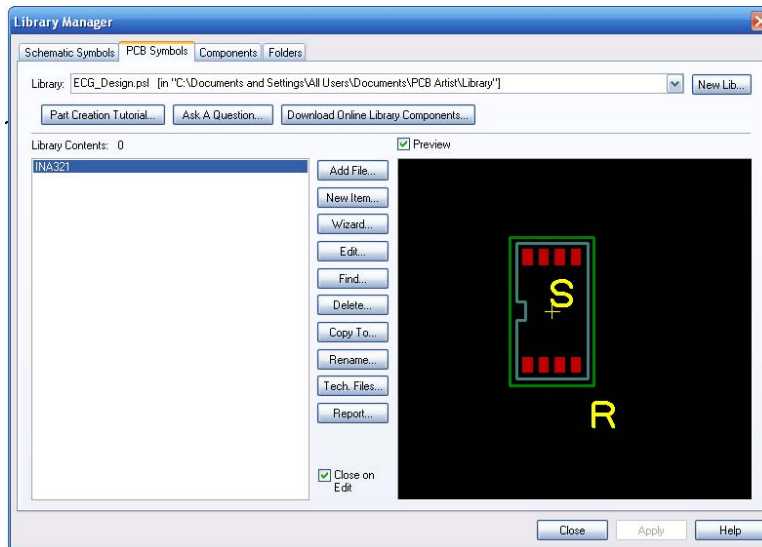
Leave the **Edit the footprint now** check box unchecked.



Click **Finish** to complete the footprint creation.

➤ **Viewing the symbol in the library**

Now that the symbol has been created and saved, let's view it in the library manager. From the **Libraries** button on the **File** toolbar, select **Libraries**, shortcut key **<F11>**. Select the newly created footprint from the contents list (INA321). Check the **Preview** box if not already selected so you can view the footprint. You can also edit it from here as well.



The footprint is now ready to be used in a component.

Creating Surface Mounted Footprints

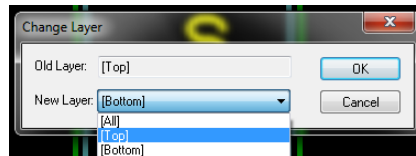
Surface Mounted pads of the PCB Symbol are easy to create, and will be necessary for most of our ECG components. The only consideration needs to be for the side that the pads reside on. For normal pads, this would be the **[All]** layer. This means that the pad occupies all of the layers in the design. Whereas a surface mounted (SM) pad would only occupy the outer layers **[Top]** or **[Bottom]**.

All surface mounted pads are defined on the Top layer. When used in the PCB design, the component is flipped to the other side of the board (Bottom). For the ECG PCB design, all components will be mounted on the top layer of the board.

➤ Change the layer of the pad

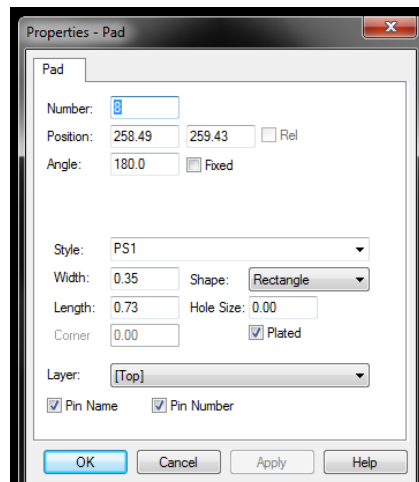
During the addition or once the pad has been added, the layer can be changed. You may also find that you need to change the pad style as well.

Select the pad(s) and right click. From the menu, select **Change Layer <L>**.



Select the **New Layer** from the drop down list. The **Old Layer:** box will display the current layer used.

An alternative method is to use the **Pad Properties**, also on the shortcut menu.



The **Layer:** drop down list will contain legal layers for which to swap your pad to. You should be aware that there may also be a **Pad Style** change required, especially to note is a drill hole and plated status of the current pad style.

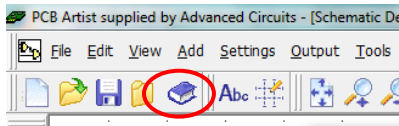
Click **OK** to accept the change.

Creating Components: Using the Component Editor

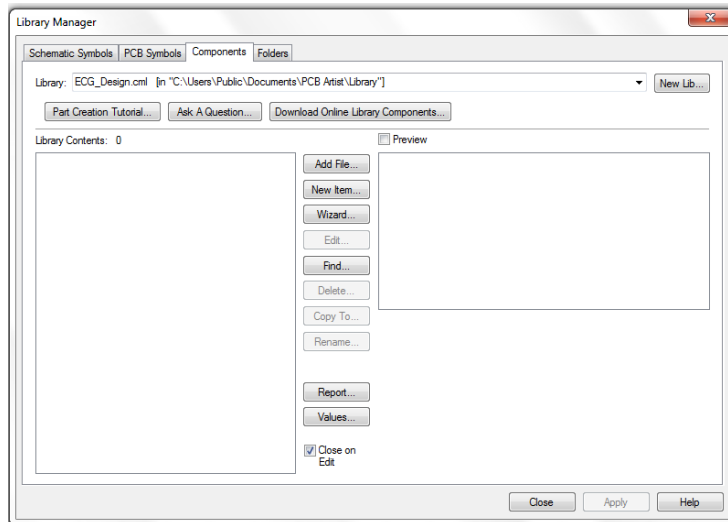
This tutorial uses the Schematic Symbol and PCB Symbol (Footprint) created in the previous tutorials, but equally applies to creating a component using other Schematic Symbols and PCB Symbols.

➤ **To open the Component editor**

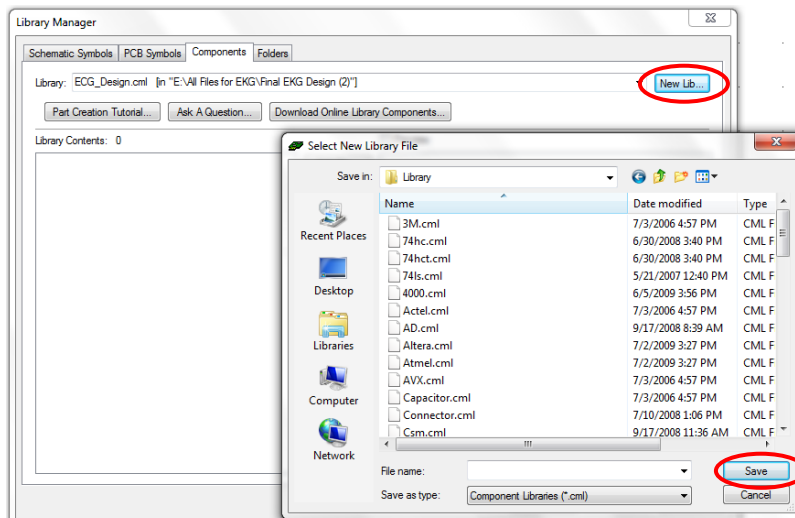
On the **File** toolbar click **Libraries** <F11> or <Ctrl+L>.



From the **Library** manager select the **Components** tab. For this tutorial, we will use the same library we have used for our Schematic Symbol and PCB Symbol (Footprint).



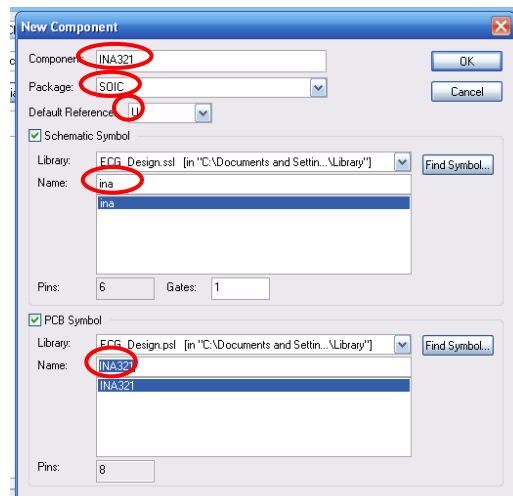
Click the **New Lib** button.



In the **File name** box, type **ECG_Design** as your new component library.

Click **Save** to accept this. The library manager is now ready to start entering new components. We need to create a new component so click on **New Item**. The **New Component** dialog box opens. There are 8 pieces of information required:

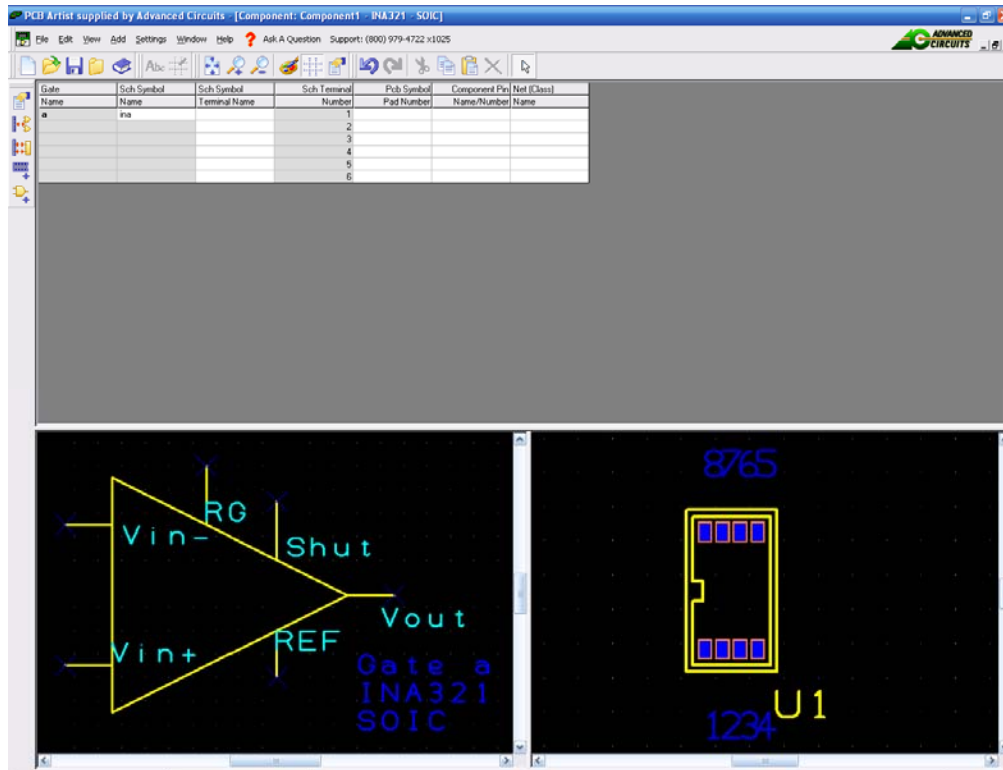
- In the **Component:** box type the component name **INA321**
- In the **Package:** box type **SOIC**, this identifies the package type, multiple package types can be used for selecting alternative footprints.
- In the **Default Reference:** box type **U**. This will be the component name used in the actual design (or seen on the silkscreen of the PCB)
- For **Schematic Symbol**, ensure that the library is set to **ECG_Design.ssl**. Select your Schematic Symbol previously created **ina** from the list.
- The dialog will automatically show **Pins:** as **6** and **Gates:** as **1**. **Change the Gates: to 2. This will add another to the component.**
- For the PCB Symbol, ensure that the library is set to **ECG_Design.psl**. Select your PCB Symbol previously created **INA321**. Ensure that the Pins box shows 8 to verify that your PCB Symbol is correct.
- Your dialog is complete and should look like the one below:



Once it does, click **OK** to continue.

The **Component Editor** opens showing the items added from the **New Component** dialog.

The **Component Editor** now looks like this:



So far we've given the Component a **Name**, selected the name **default reference** name stem to be used and selected the Schematic and PCB Symbols to be used. We now need to add more detail to the component to give it more intelligence.

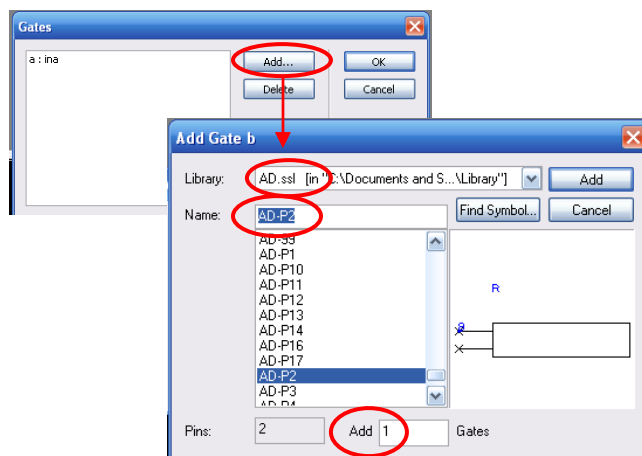
➤ **Adding an extra gate**

We need to add a power gate to the component. There are various methods in PCB Artist to adding power and ground to the component, for now this is one of the easiest.

Right click anywhere in the component grid. From the list, select **Edit Gates**.

From the **Gates** dialog, select the **Add** button. Notice your existing 2x OPAMP gates are already shown in the list as gates a & b.

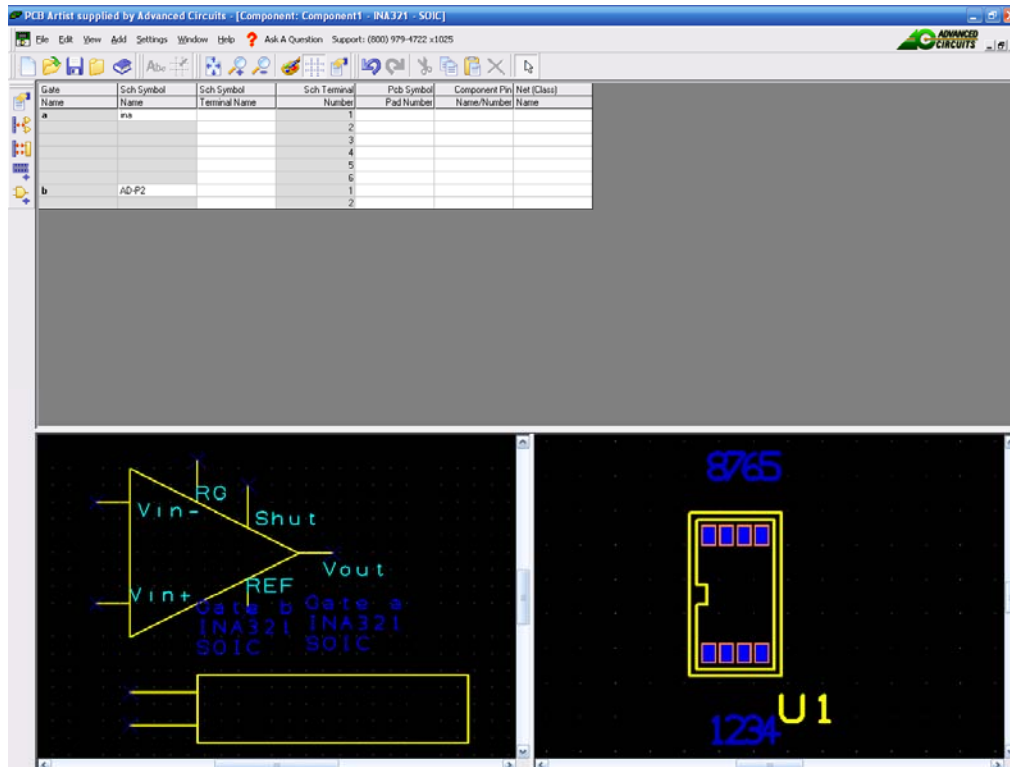
From the **Library**: list select the **AD.ssi** library.



Scroll down the **Name:** list and select the **AD-P2** gate symbol. Ensure that you only have **1** selected in the **Add 1 Gates** box. Click the **Add** button.



The Power gate will appear at the bottom of the list. Click **OK** to add it to the component.



The power gate will now appear at the bottom (as the last gate) in the component grid and in the Schematic Symbol preview.

Now we need to add more detail to the component and to 'map' the pins used in the Schematic Symbol to those that will actually be used in the PCB Symbol.

➤ **Mapping pins to pads – why?**

For a Component to work, it needs to know how the pins defined in the Schematic Symbols relate to the pads used in the PCB Symbol. This mapping is presented in a 'grid' format in the component editor.

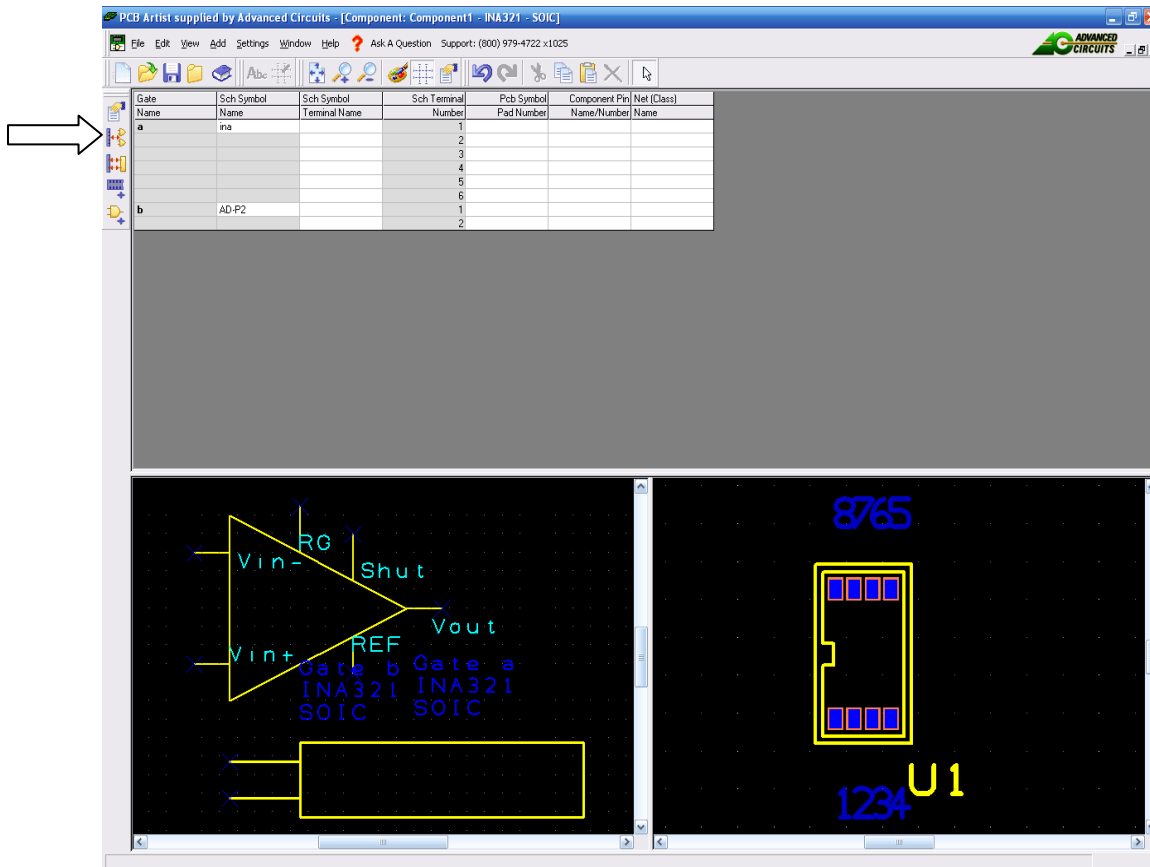
You can map the pin relationships manually or automatically by using the tools provided. On complex devices like a 256 pin BGA, the automatic method may be more preferable than the manual method which may be better for resistors for example.

You can use three methods: you can directly type the pad numbers into the respective grid cells or you can use the **Assign Pins** modes that let you click on the Schematic Symbol pin first and then the equivalent PCB pad. You also have the Assign Pins 1:1 which automatically assigns pins on a 1 to 1 basis, Schematic to PCB symbol.

➤ **To map pins to pads**

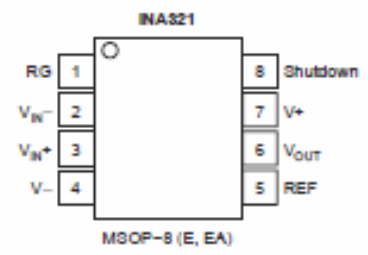
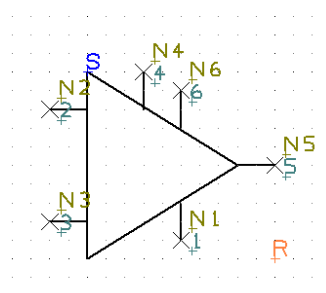
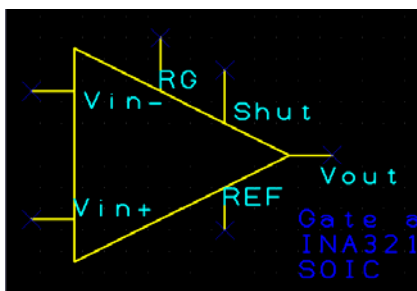
Generally speaking, you will use the **Assign Pins** method most times to do this task. This enables you to manually click on the Schematic symbol pin and then the corresponding PCB pin.

Click on the **Assign Pins** button on the Component toolbar.



The natural pin mapping of the Schematic Symbol and the PCB Symbol during creation will not be the same and will not necessarily be the correct sequence for mapping. PCB pins generally drive the mapping. However, for this specific example, we have considered the number sequence of the Schematic Symbol during the creation, which allows us to follow more closely the PCB pin sequence.

Consider the component and schematic symbol shown below. We will number our pins according to these pictures.



Select **Pin 1** on the **Schematic Symbol**.

Select **Pad 1** on the **PCB Symbol**.

The component grid will display the number **1** for **Sch Terminal Number 1** on gate **a**, it will also display a number **1** in the corresponding **Component Pin Name/Number** cell.

As you see, we have chosen to map the first pin on the Schematic Symbol to the first pin of the PCB Symbol because they correspond to each other. Map the other pins accordingly.

Once assigned, a number 1 will be shown on the Schematic Symbol output pin (pin 1). The color of the Schematic Symbol and PCB Symbol pins will also change color to reflect this. A different color will be shown for assigned and unassigned pins, and the currently selected pin.

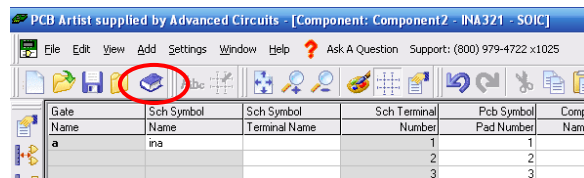
You should complete all pin to pad mapping so that looks like this:

Gate Name	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Class) Name
a	ina		1	1	1	
			2	2	2	
			3	3	3	
			4	5	5	
			5	6	6	
			6	8	8	
b	AD-P2		1	7	7	
			2	4	4	

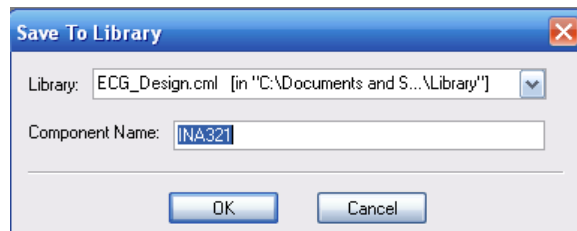
With the assignment of all pins between the Schematic and PCB symbols, this would be considered a minimum configuration to be able to save the component to the library and then use it in a design. Other information can be added and is discussed below after saving the component.

➤ Saving your Component

From the **File** toolbar, select **Save <Ctrl-S>**



The **Save To Library** dialog opens.



Choose the **ECG_Design.cml** library from the list if not already selected.

You entered the Component editor through the New Item dialog and have already given the component a name. This name will be displayed.

Click **OK** to confirm the library name and component name. You have now saved the Component in the Library. The component will still be visible in the editor. We will edit it some more so keep it open for now.

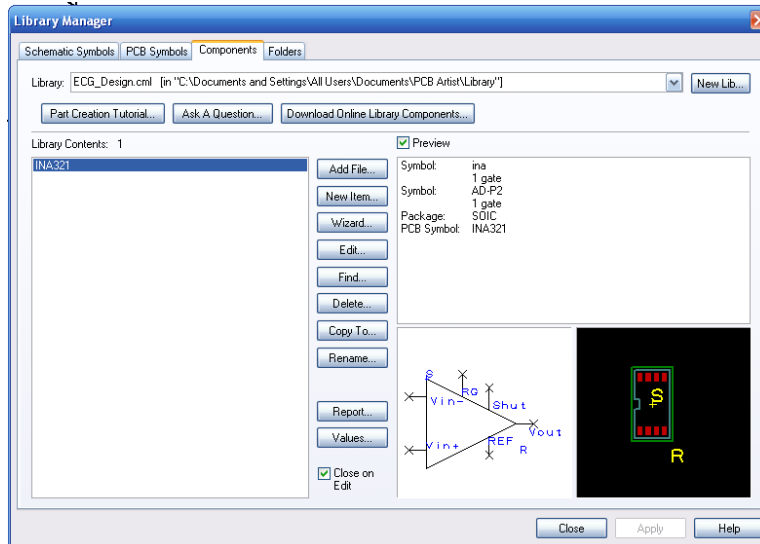
➤ **Viewing the Component in the library**

From the **File** toolbar, select **Libraries**, <F11> or <Ctrl-L>

The **Libraries** dialog opens. Select the Components tab if it isn't already selected.

Make sure the **Preview** box is checked.

Select the **INA321** Component that you created, it should look like the one below.



A quick check to see whether you have created is correctly is the existence of both the Schematic and PCB Symbols in their respective Preview windows.

At this stage, you would add this component to a Schematic or PCB design and it would work.

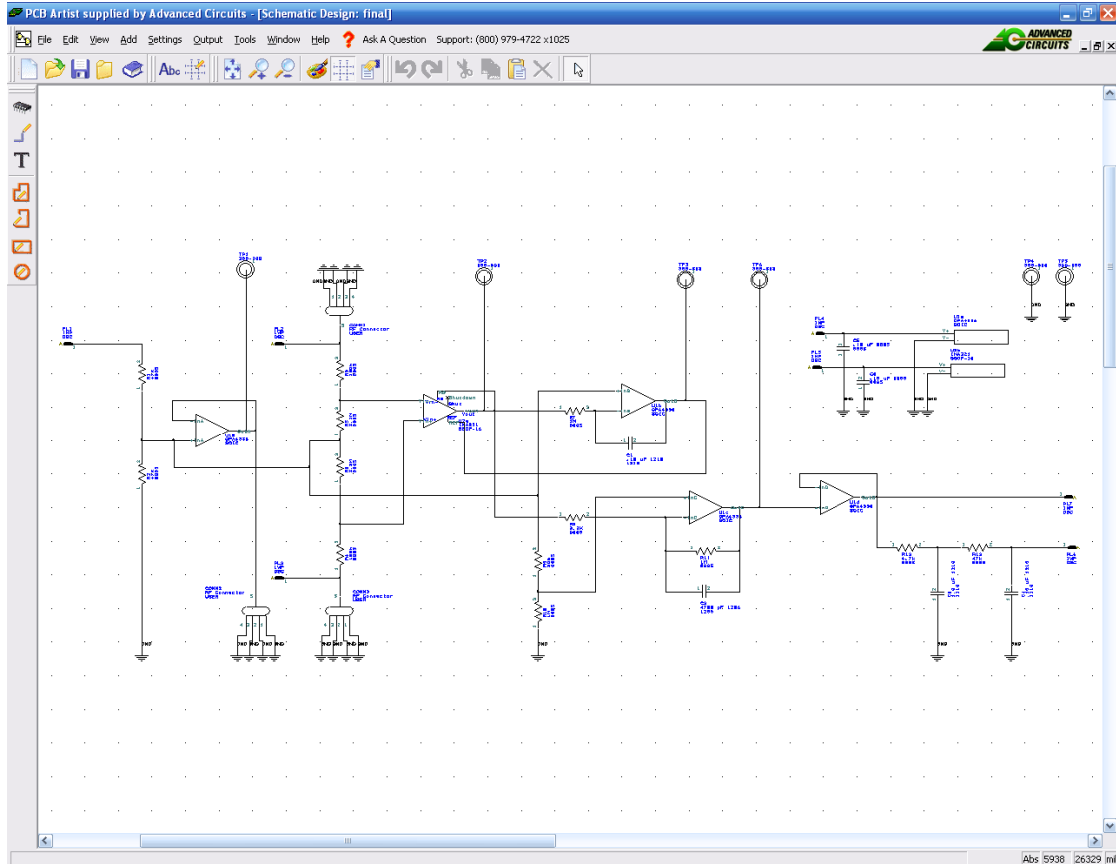
Creating the Rest of Your Components

The rest of the components, with the exception of the OPA4336, are very simple to create. The Schematic Symbol for both the resistors and capacitors do not require designing, they can be added from the **Discrete.ssi** schematic library, using the **Add File** button in the Schematic Symbol window.

The PCB Symbol does require using the **Wizard**. This allows you to design each resistor and capacitor to the size of the actual component that you will be ordering. For further help go to the PCB Artist Library Creation Tutorial, which is located at the following web address: <http://www.4pcb.com/downloads/pcbartist/PCBArtistLibraryTutorial.pdf>

Schematic Design Tutorial

Fortunately, the Schematic from which the ECG PCB Design will be made has been supplied to us by Dr. Fei Hu. Open the file from the extension you have it saved at. Your PCB Artist window will look like this ready to start:



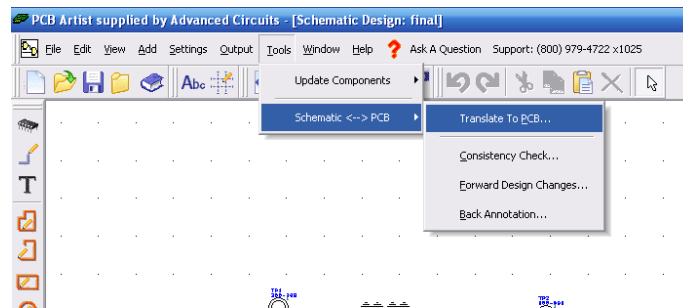
The next stage is to convert the design into the PCB design environment

Although we will not cover how to create this schematic in this tutorial, there is some useful information located in the PCB Artist tutorial, which can be found at the following web address: <http://www.4pcb.com/downloads/pcbartist/PCBArtistTutorial.pdf>

Converting the Schematic to PCB

If you created a **Schematic** in PCB Artist, the final process before starting on the PCB design itself is to translate the Schematic design into the PCB design editor.

When you are in a position where you wish to go-ahead with this translation you should select **Schematic <- - > PCB >** and use the **Translate To PCB** option from the **Tools** menu.



This will display the **New PCB** wizard.

Starting a New PCB Design

You do not have to create a PCB design from a Schematic design. If you intend starting a new PCB design without using a Schematic design first, from the **File** menu select **New** and check the **New PCB Design** radio button. Click the **OK** button and this will start the **New PCB** Wizard. However, we already have our Schematic design, and will definitely be creating our PCB design from this.

Getting Started with the Design

PCB Artist PCB is flexible in that you can start your layout in two ways:

- Use PCB Artist Schematic Capture as the 'front-end' to the PCB system. The nets and components are converted to PCB in the integrated environment.
- Use the PCB Artist PCB design editor interactively creating the design on-the-fly without the need for an initial netlist. You just add components and connections as required to create the PCB design.

Converting the Schematic to PCB

If you are following the tutorial, you would have already processed your 'logical' Schematic design into the 'physical' PCB design layout editor using the **Schematic To PCB** wizard. If you didn't do this, go back to the previous chapter (*Chapter 3 - Converting your Schematic to PCB*) and convert your schematic into a PCB design.

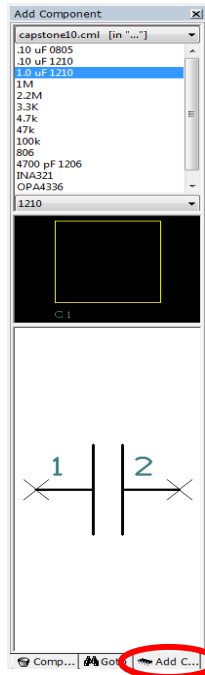
At this point you are ready to start laying out the PCB design. You will see the board outline ready to place the components in. We previously chose to leave the components unplaced and the nets unrouted. These two features can both be selected during the final conversion process.

Adding Components

➤ To add a component

There are two methods for adding components; we will use the **Interaction bar** to start with, you can also use the Add Component option as well. The Interaction bar can be displayed by clicking the **<F9>** shortcut key.

Select the **Add Component** browser tab at the bottom of the Interaction bar.



From the list of available libraries, select the **ECG_Design** library. Click on the small down arrow to reveal the drop down list and then make the selection. Notice, in this list, all the required components to design the schematic have been made. **As said in the last part of the Creating Components tutorial, you will have to make these parts on your own.**

What's left:

- Mapping the Pins to the correct pads so the functions of each gates are correctly defined
- A brief statement to address using these three steps to create all of the components and using them to create a PCB design
- Wiring and layout issues:
 - o Making sure the pad is 20 mils larger than actual component
 - o No 90° angles for the wire routing
 - o Advantages of having an entire ground plane/layer
 - o Others?
- How to verify?