

# EAGLE PCB TUTORIAL

## 2013

Making a Component



**Author**  
Ali Hadi Al-Hakim

e.quinox  
Department of Electrical and Electronic Engineering  
Imperial College London

## 1 INTRODUCTION

---

This guide will attempt to show you some of the basics of using the [CadSoft EAGLE](#) (Easily Applicable Graphical Layout Editor) software. The function of EAGLE is to provide the easy design of PCBs. The benefit is using EAGLE is you can get a relatively functional free version very easily.

This guide is broken into parts but in total will teach you the following using graphical step-by-step instruction.

- **Creating component libraries based off component data sheets**
- Producing a circuit schematic
- Designing a PCB suitable for manufacture

You can get your hands on a free version of EAGLE from <http://www.cadsoftusa.com/download-eagle/freeware/?language=en> and provided you have an internet connection you can start designing your own PCBs straight away.

### NOTE

Be very careful when using this PCB version. If you create or modify EAGLE project files using the EAGLE version (6.x) you install on your computer there is a high chance that it will not be compatible with the EAGLE versions on the university computers. EAGLE compatibility issues are highly frustrating, so be careful.

## 2 IMPORTANT BOOKMARKS

---

*(Click to jump to section)*

<a href="#">Symbol Design</a>	Page 3	Steps 1 to 10
<a href="#">[name]@[number] Naming</a>	Page 5 and 15	Steps 7 and 29
<a href="#">Package Design</a>	Page 7	Steps 11 to 26
<a href="#">Making the Device</a>	Page 14	Steps 27 to 30

### 3 CREATING YOUR OWN COMPONENT

It is very important to learn how to make your own components for use in EAGLE because this allows a great deal of flexibility in what you are going to use in your circuits. Using the EAGLE pre-built libraries is fine but sometimes it can be easier to design your own, say diode, with the correct dimensions rather than spend 30 minutes looking for the right one you need in pre-built libraries with hundreds of options. Other reasons for designing your own component may be: it doesn't exist otherwise; you need specific drill hole sizes; you want an easier-to-use symbol for your circuit schematic.

Designing a component in EAGLE is easy and consists of three steps:

1. **Symbol** to represent the component on the circuit schematic,
2. **Package** to determine how the component will attach to the PCB, and
3. **Device** the linking of the symbol to the package.

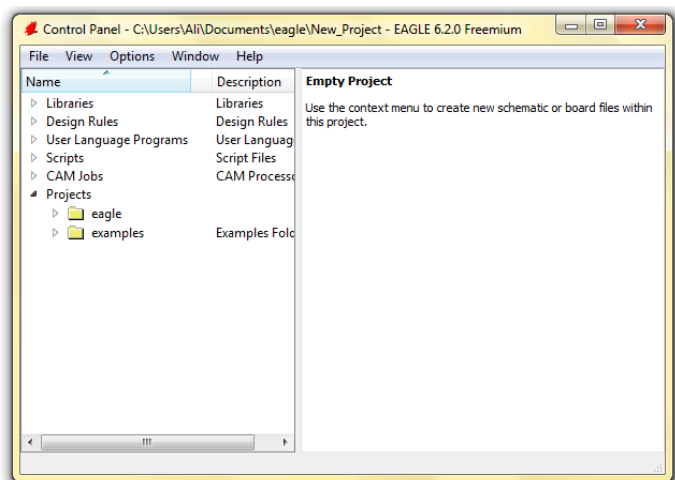
In this example we're going to design a rail-to-rail Op-Amp chip. We will use the [TI – NE555P](#).  
*Click the link to access the data sheet.*

Let's begin...

#### Step 1

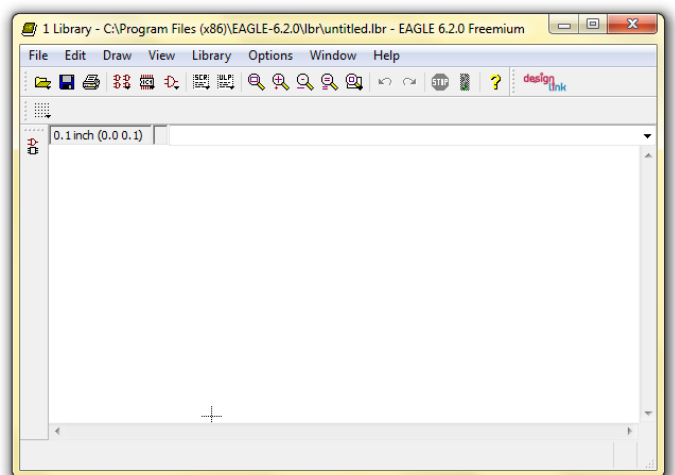
When you run EAGLE the window shown to the right will open. To begin creating your own library select:

**File → New → Library**



#### Step 2

You will be welcomed by a new window. Maximise it and click the "Symbol" button.

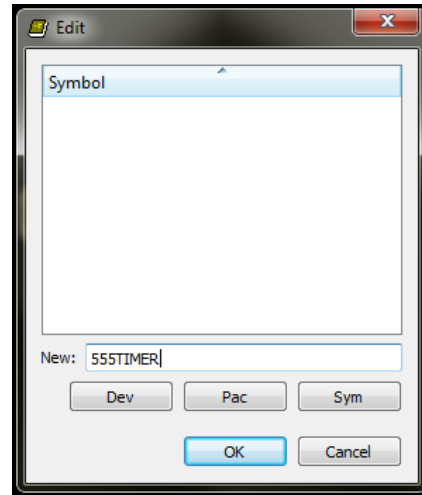


### Step 3

Your first task is to provide a name for your symbol. We could create a generic DIP8 symbol and name it DIP8. However, the Op Amp chip can be symbolised in a more convenient manner which we will use.

Name it "**555Timer**" and click OK.

Click yes on the Warning popup that follows.



### Step 4

Your new window will now be gridded and a toolbar will be present on the left. The functions of the most important tools are as follows:



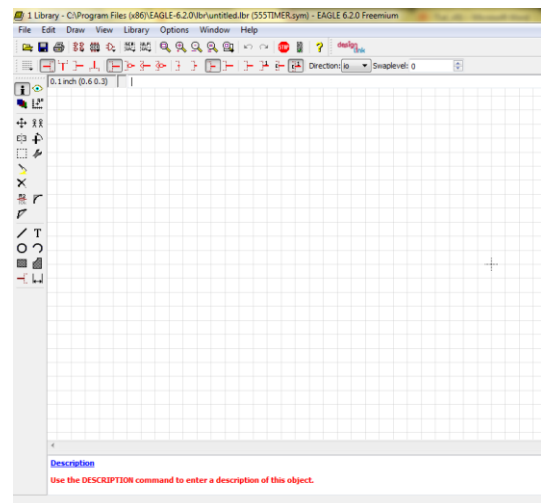
#### TOP- Information

- Show object properties
- Show available layers

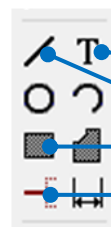


#### MIDDLE- Cursor Modifiers

- Copy object
- Move object
- Rotate object
- Group object
- Delete object
- Name object



#### BOTTOM- Place Object



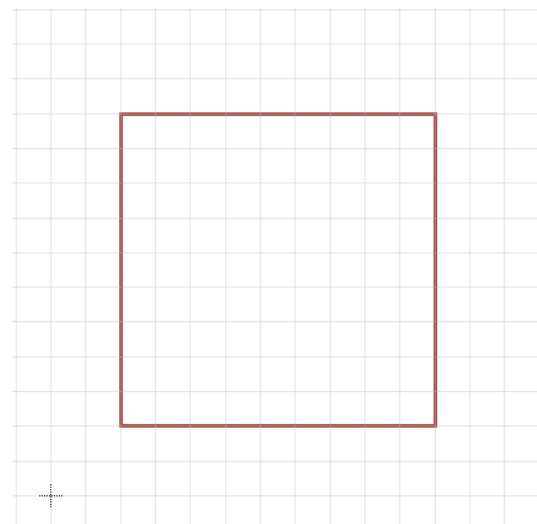
- Place text
- Draw lines
- Draw a rectangle
- Draw a pin

### Step 5

To start, click the 'Draw lines' button OR type "**wire**" and press return. With the new cursor draw a 9 × 9 square using the default grid.

**NOTE: You should never modify the grid size in the "Symbol" workspace as this will cause serious issues during schematic design.**

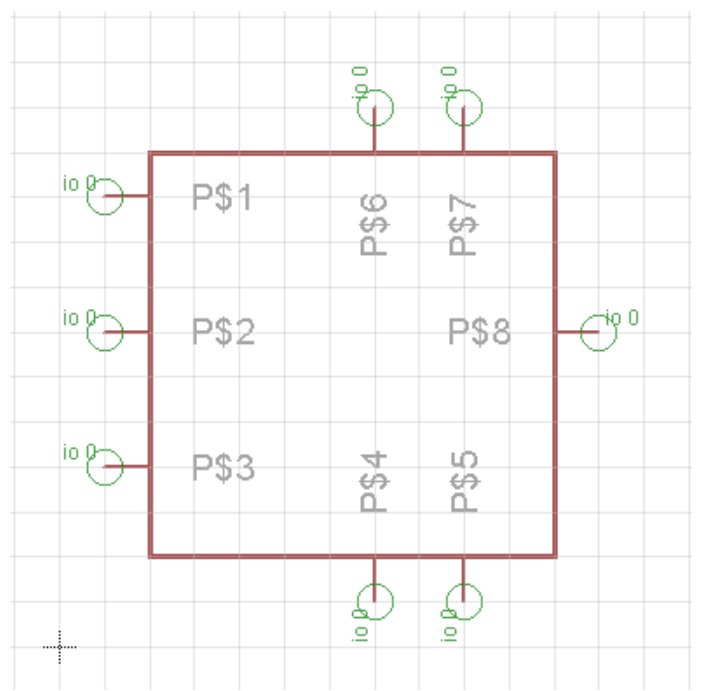
To delete any mistakes type "**del**".



### Step 6

Draw 8 pins onto the square as shown. There are options on the top toolbar which let you modify the pin appearance. Use these to make your pins shorter.

To rotate the pins, **right-click** when the pin is on your cursor. You can do this with all objects.



### Step 7

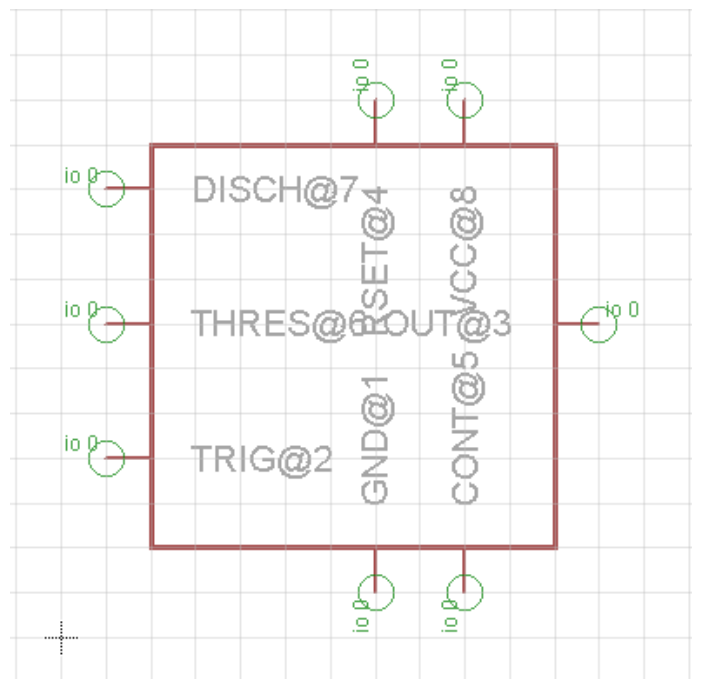
Click the 'Name object' button OR type "**name**" and press return. Click on each pin and name them as shown.

You are using the [name]@[number] method where [name] corresponds to the pin names on the component datasheet. You will see later why we use this method.

Ignore the fact it's a bit messy now and some of the text overlaps. This will not be the case later when you use the symbol.

Going counter-clockwise the labelling is as follows:

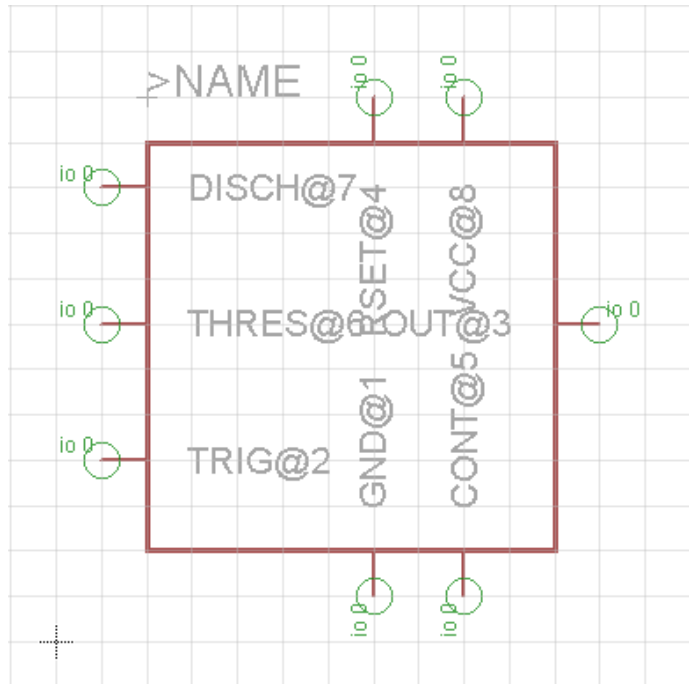
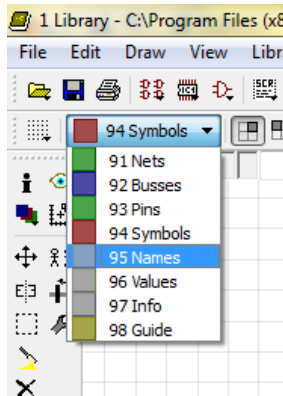
DISCH@7  
THRES@6  
TRIG@2  
GND@1  
CONT@5  
OUT@3  
VCC@8  
RSET@4



### Step 8

Click the 'Place text' button OR type "**text**" and press return. Type ">NAME".

Before placing the text on your cursor, make sure you change the layer type to "**95 Names**" on the upper toolbar. Place the text on your symbol drawing.

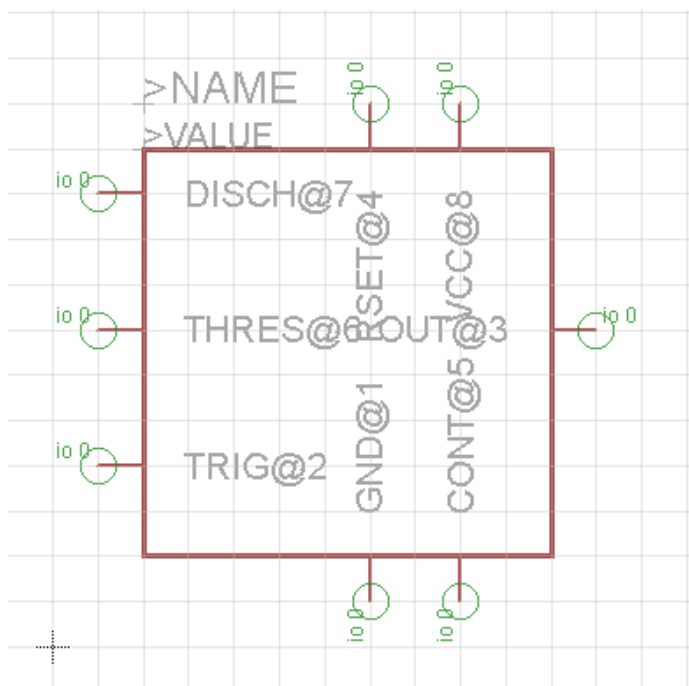


### Step 9

Place more text, but this time call it ">VALUE" and change the layer to "**96 Values**". You can also change text size on the upper toolbar.

Placing a value object is not always necessary.

The reason you use the ">[name]" syntax will become clearer later.

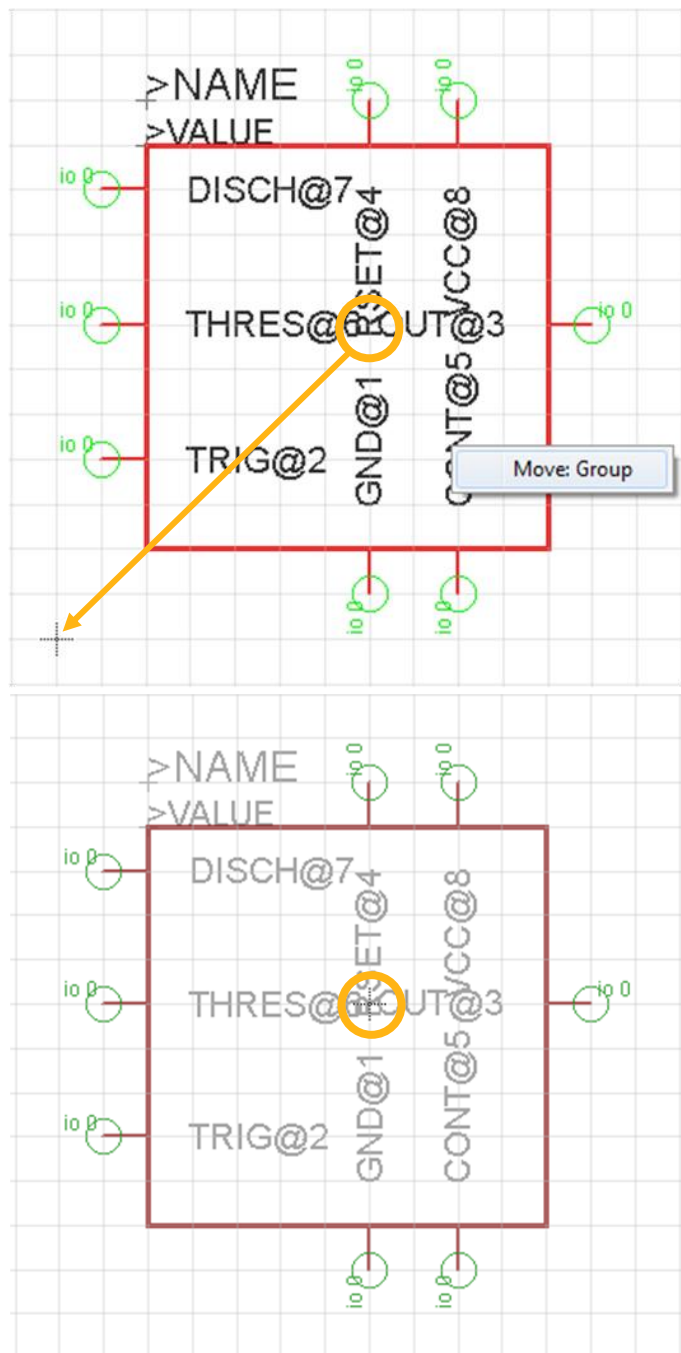


### Step 10

The symbol is now complete enough to be used. However, you may have drawn it far from the crosshair representing (0,0) on the grid. When you come to use your symbol this origin will become the focal point of the symbol so it should be more central!

Click the 'Group object' button OR type "**group**" and press return. Drag your cursor across the entire symbol so that it is all highlighted.

Now, select the 'Move object' button OR type "**move**" and press return. Right click anywhere and choose "**Move: Group**" and then place the symbol central over the origin, as shown.



**Your symbol is now complete.**

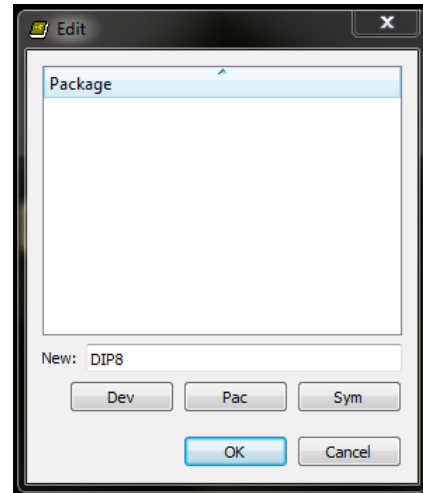
### Step 11

We will now create the package. Thus, this time select the "**Package**" button from the upper toolbar.



### Step 12

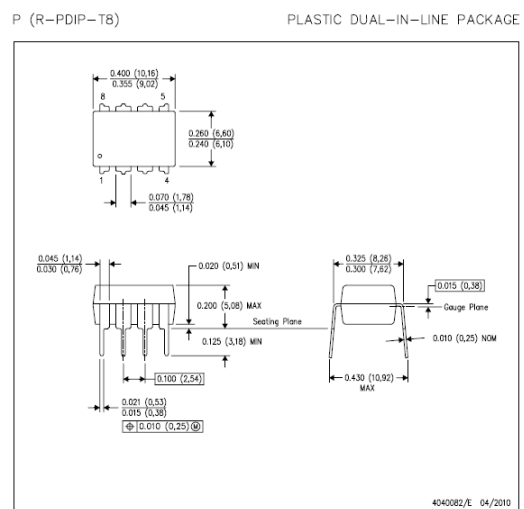
Name the package type. The package we are using is the common DIP8 form. Therefore, we should just use the name "DIP8". Click yes on the warning popup that follows.



### Step 13

We will now learn to use a components datasheet for physical information on the package. Open the [TI – NE555P](#) datasheet and go to page 24.

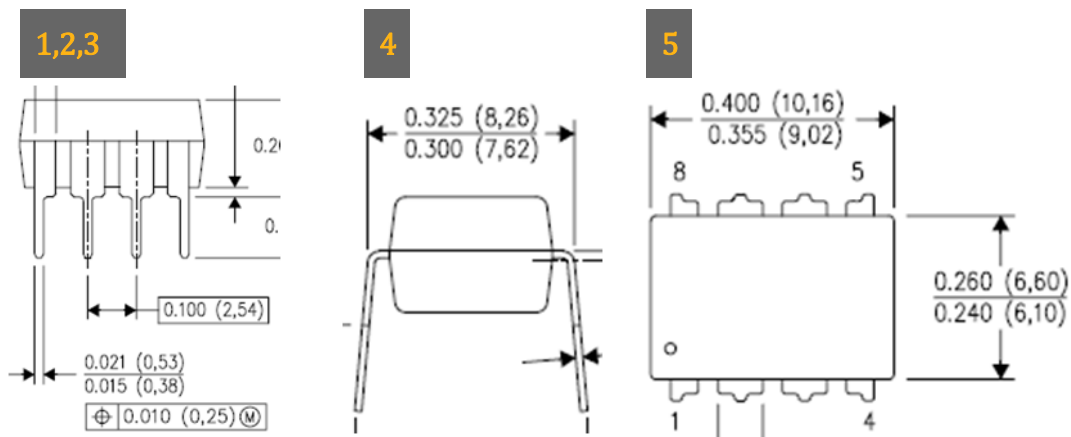
We are interested to know the relative locations of the pins, the type of pins (through-hole or surface mount) and the casing dimensions.



### Step 14

We can see that:

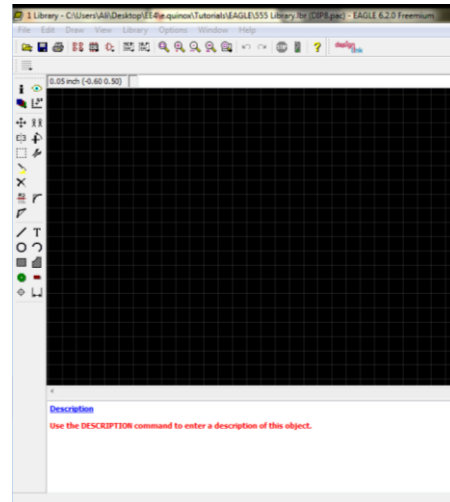
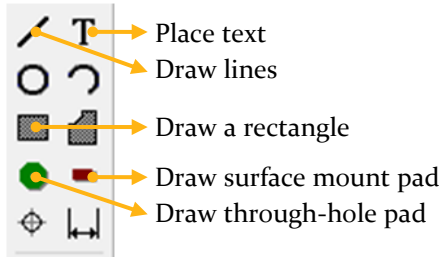
1. Our device is through-hole.
2. The leg diameter is 0.25mm.
3. One line of pins has a pitch (distance between pins) of 2.54mm.
4. The distance between each row of legs is 8.26mm.
5. The package dimensions are max 10.16mm x 6.60mm (W×L).





### Step 15

Back to the EAGLE package design workspace we see some new tools and a new look.



**When designing a package it is very important that all of the dimensions are correct. If they are not then when you get your PCB fabricated you may find your component won't fit as expected. i.e. the component legs don't match up to the drill holes quite right. Thus, it's very important to take some time here and get your dimensions and relative object positions right.**

There are different ways to layout a package. Perhaps the most obvious are:

1. Change the grid size to place items in the correct places.
2. Directly modify object properties and coordinates/dimensions.

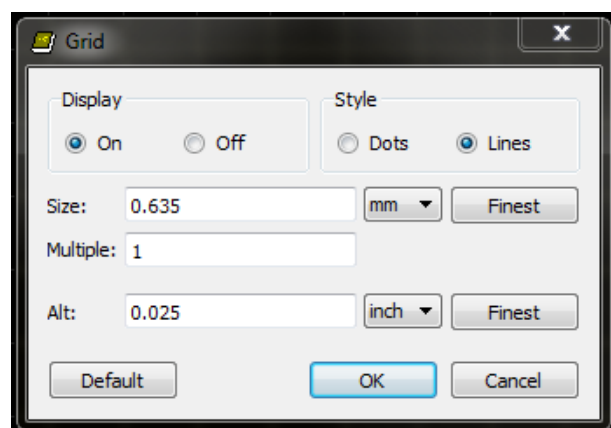
We will use both methods at our convenience. using the grid to place pins is normally the most efficient way but the the second method as it can result in far swifter and more accurate package outline layout as you will see.

### Step 16

Although we're not using the grid method, lets still change the workspace dimensions into mm. To do this go to the menu bar and choose:

**View → Grid**

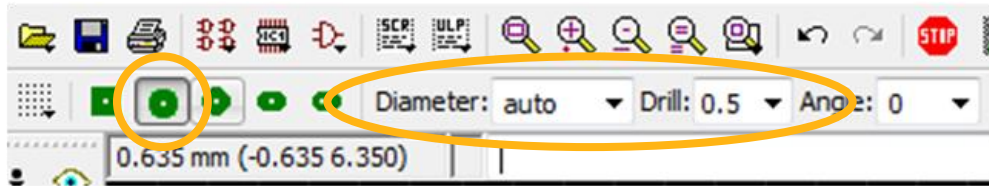
Change the dimension to "**mm**" (not "**mil**"), change the grid size to 0.635mm and select OK.



### Step 17

Click the 'Draw through-hole pad' button OR type "**pad**" and press return. Change the properties to as shown.

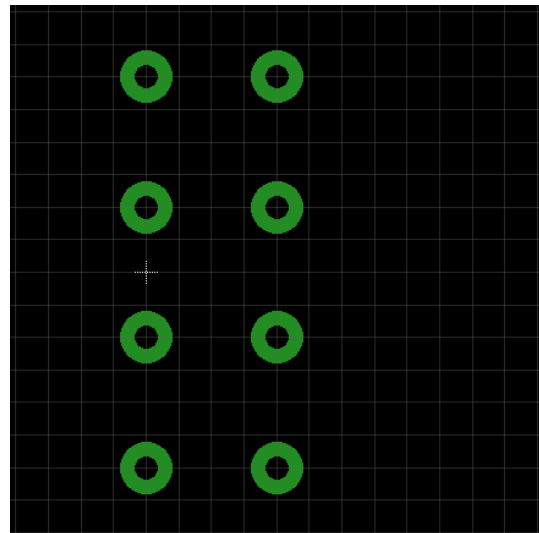
The pad shape is in fact quite arbitrary in most cases but some shapes are preferable in some situations. In general, leave "diameter" as auto, this is the diameter of the metal pad and will automatically adjust to be a decent size with regards to the drill hole diameter. A "Drill" diameter of 0.5mm is chosen as a hole diameter about 0.2mm bigger than the component leg diameter will generally fit well.



### Step 18

Now, using the coordinates shown in the top left of the package editor workspace, create two aligned columns of through-hole pads which are spaced 2.54mm (4 squares) apart. This will align them perfectly in relation to each other on the vertical axis. Do this around the workspace origin as shown.

This is a prime example of how using the grid can make very light work of some layout.



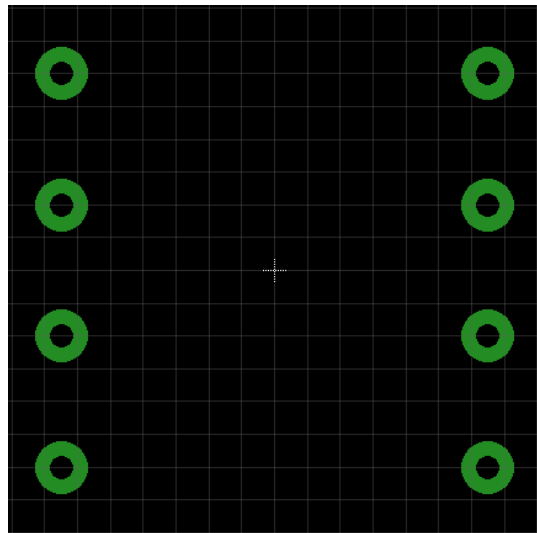
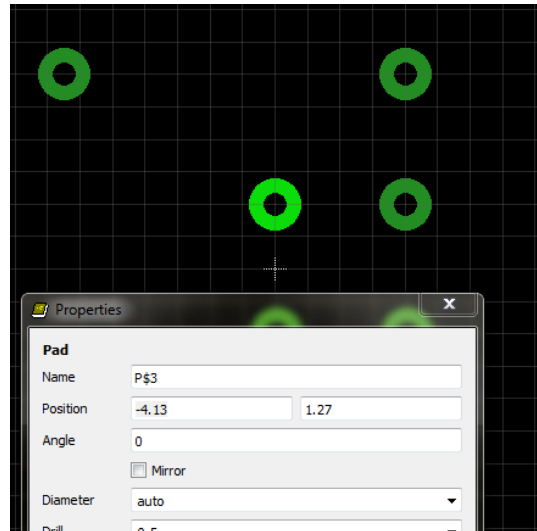
### Step 19

Now centre your pins on the x-axis. The best way to do this is to use the pin properties.

Right-click on each pin and select "**Properties**". Change the first "Position" field to **-4.13mm** for the left column, and **+4.13mm** for the right column.


(From the datasheet you know the pins should be in columns at the x-coordinate  $\frac{\pm 8.16}{2} = \pm 4.13\text{mm}$ ).

You now also have perfectly aligned through-hole pads in the x-axis.

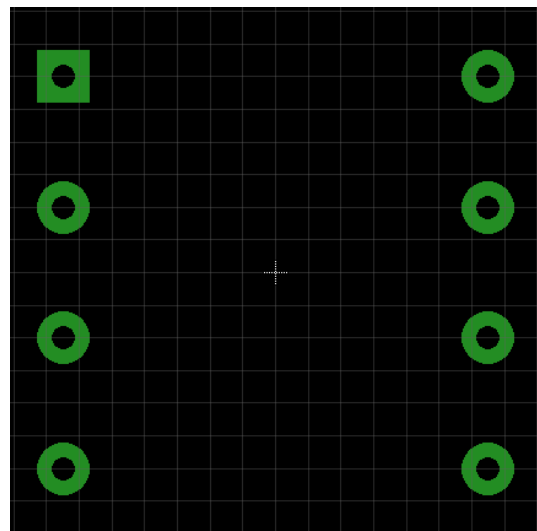


### Step 20

To create a very clear indicator of the first pin of the package we will change pin#1 to a square shape.

In your toolbar, click the spanner (  ) which is general change settings button. Choose "**Shape → square**".

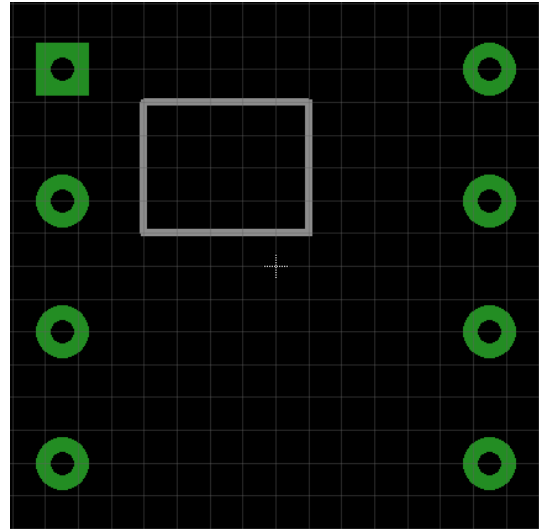
Your cursor will now change any through-hole pad you click on into a square. Change the shape of the top-left pad.



### Step 21

For package outline information we will use only the properties method of layout design. This will consist of drawing some lines and then changing their coordinate properties to place the lines exactly where we want them.

Draw a box of any size using the "**wire**" tool. Make sure the layer is "**21 tPlace**".



### Step 22

Right-click on the box edges and select "**Properties**". You will modify the "To" and "From" fields. They are structured as:

From point	$x_A$	$y_A$
To point	$x_B$	$y_B$

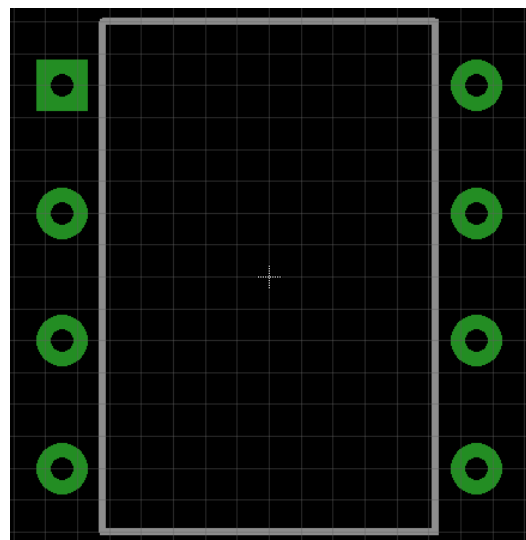
Again just use the method of halving the casing dimensions and place the lines directly around the workspace origin. So, where the casing is 6.60mm  $\times$  10.16mm then the (x,y) should only ever be of the values ( $\pm 3.30$ mm,  $\pm 5.08$ mm).

Try this on the left edge. You want to draw it from the top left ( $-3.30, 5.08$ ) to the bottom left ( $-3.30, -5.08$ ).

### Step 23

Correctly modify the properties of both the left and right edge using the method described above to get an package similar to that shown.

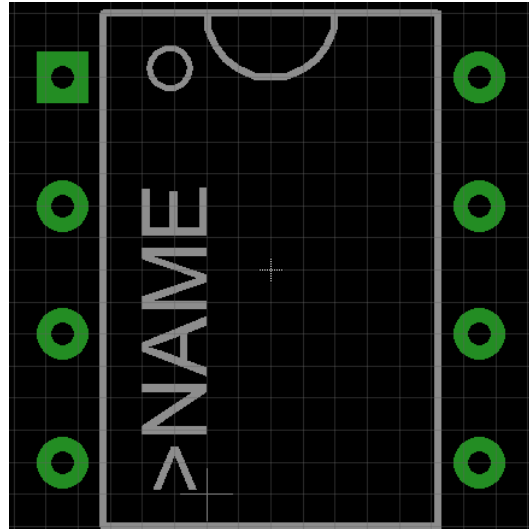
You may sometimes have to reverse the minus sign of the non-identical coordinate depending on how the lines were originally drawn.



### Step 24

Place the text ">NAME" by your package design and if necessary add ">VALUE" (type "text" and change layer to tName / tValue).

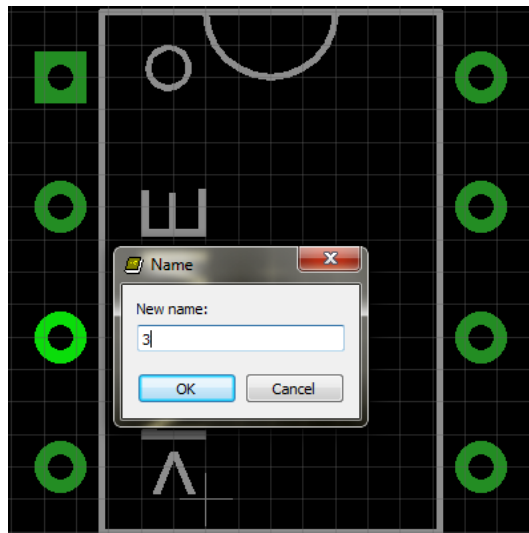
Feel free to also add other "21 tPlace" objects on the design too, to make the pin 1 location clearer.



### Step 25

You must again name the pads (type "name" and click on the pads). This time you are just going to number the pads by their pin number. As you should know and from what the datasheet shows you, the pin numbers are as follows.

1	8
2	7
3	6
4	5

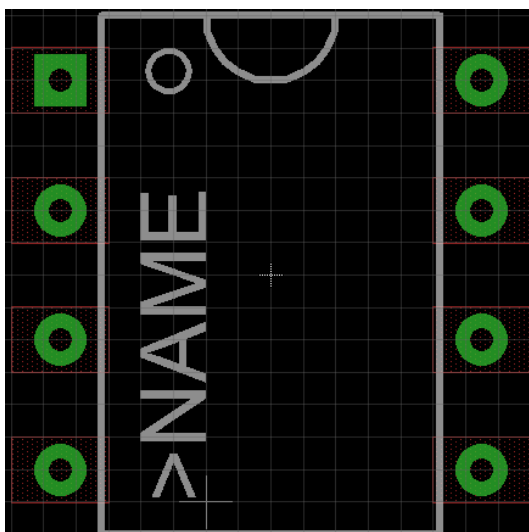


### Step 26

The final step to this package design is to add some restricts which control where PCB tracks are allowed to go.

Select the 'Draw a rectangle' button OR type "rect" and press return. Change the layer type to "41 tRestrict" (top restrict) and place a rectangle roughly around each pad.

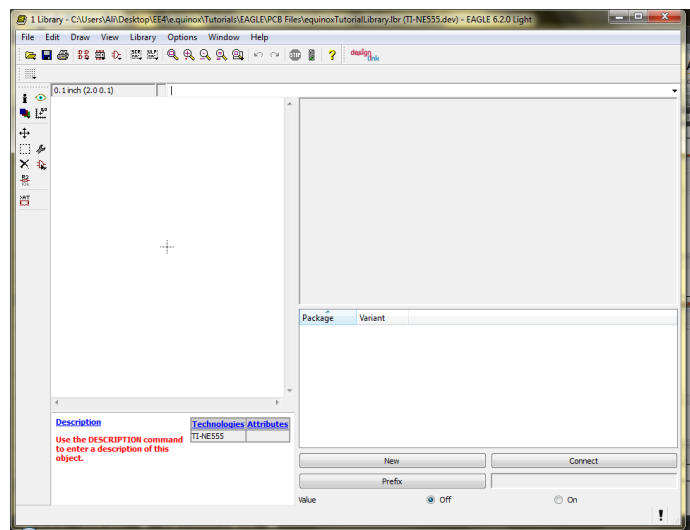
**NOTE: Restricting is only necessary for components which may be difficult to solder from one side (i.e. because casing gets in the way). For example, a resistor would not need this, but some chips might!**



**Your package is now complete.**

### Step 27

The last part of making a component is linking the symbol to the package. This is called making a Device. To enter this workspace click the **"Device"** button. When prompted name the device **"TI – NE555P"**.

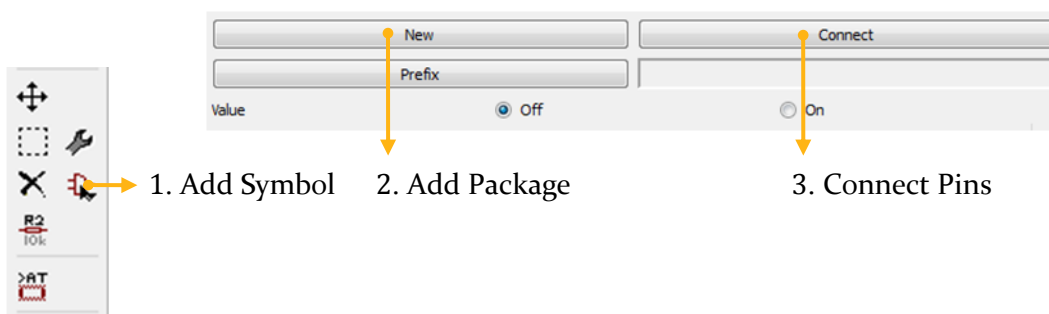
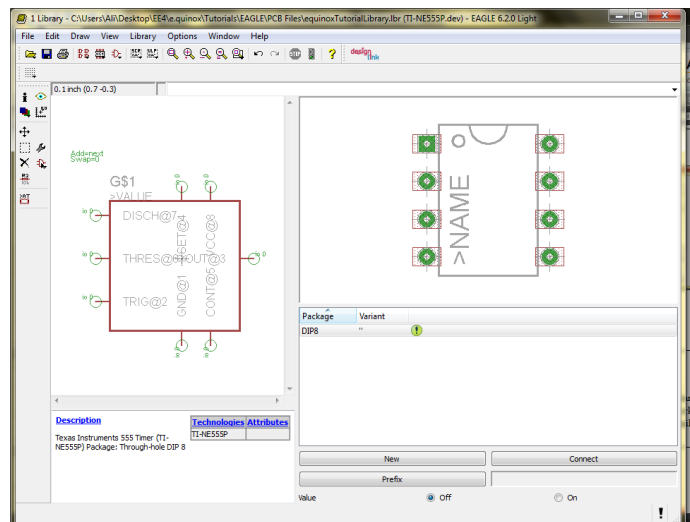


### Step 28

Click the **"Add Symbol"** button and place the **555TIMER** symbol over the centre point of the left workspace.

Then, click the **"New"** button and add the **DIP8** package.

If you want add a description of the device in the bottom left window. This is useful if sharing your library with other people or if you re-use this library in the near future where you've forgotten what everything is.



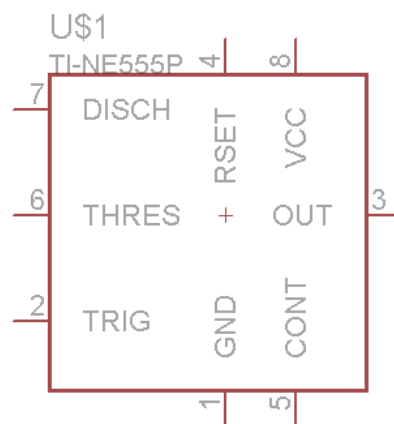
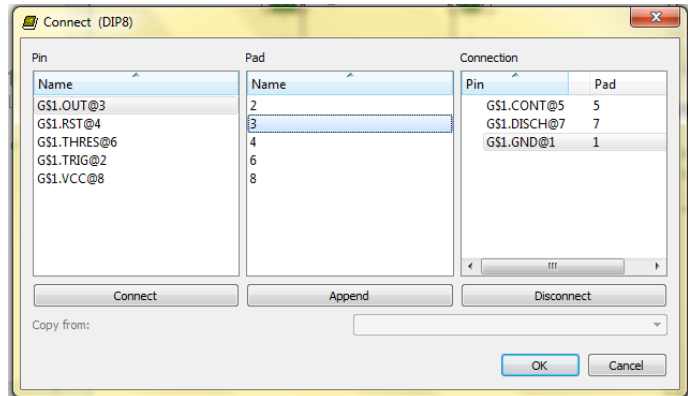
### Step 29

Now press the "**Connect**" button to assign symbol pins to package pads.

This is where you can see the usefulness of the [name]@[number] convention used in the symbol design. It is very clear which symbol pin connects to which package through-hole pin.

Once everything is connected to its pair, press "**OK**".

*The [name]@[number] will actually show to be more useful later when using the component in a schematic. What actually happens is the @[number] part will disappear leaving only the pin name behind. For complicated chips this is very powerful and neat. An example of this is shown to the left for the 555 Timer symbol. The pin numbers are also shown because of the names you provided in the package design.*



### Step 30

Save your library somewhere sensible and name it "**TimerChips.lbr**".

Normally a good naming convention would be to name it after the PCB the components in the library will be used for. For example, EE3Boost.lbr for the third year 2013 power electronics coursework. Then any components you need to make for this PCB, put them in this library and they're easily accessible from one place and easily shared amongst peers.

**You have now completed the creation of a component in EAGLE.**

**AND REMEMBER**

***Build multiple components in one library, not one component in multiple libraries!***