How to make a custom library part in Eagle CAD tool

by **kd7vnn** on March 10, 2006

Table of Contents

intro: How to make a custom library part in Eagle CAD tool . . . 2
step 1: Start the Eagle control panel . . . 2
step 2: Select or create a library . . . 3
step 3: The new library . . . 3
step 4: The easy way or the hard way. . . . 3
step 5: Time to get out the data sheet. . . . 3
step 6: The Package . . . 4
step 7: Building the package . . . 4
step 8: Setting the Grid . . . 5
step 9: Setting the grid (cont) . . . 5
step 10: Adding Pads . . . 6
step 11: Details for a cleaner look . . . 7
step 12: Name Pads . . . 7
step 13: Add name and value . . . 7
step 14: Building the Symbol . . . 8
step 15: Back to the data sheet . . . 8
step 16: Draw the symbol . . . 9
step 17: Naming Pins . . . 9
step 18: Make the device . . . 10
step 19: add package to devcie . . . 10
step 20: Make connections . . . 11
step 21: Save Device . . . 12
Related Instructables . . . 13
Advertisements . . . 13
Comments . . . 13

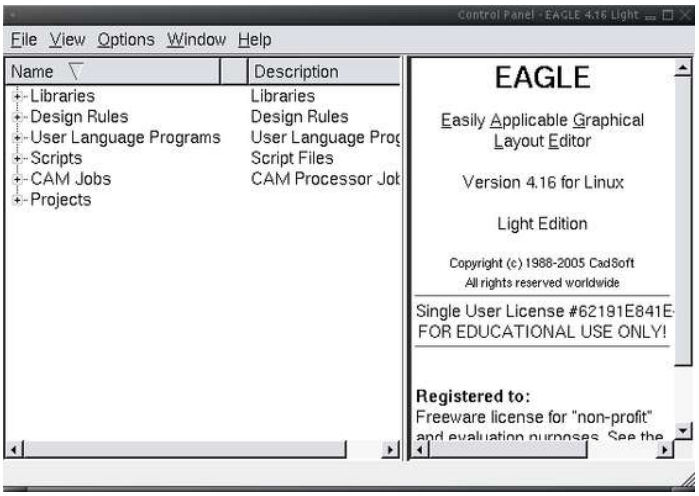
intro: How to make a custom library part in Eagle CAD tool

The eagle cad tool is a great thing. It does have something that I see as a draw back. That is that you need to pick a package for your part while you are still working on the schematic phase of a project. I assume Cadsoft, the makers of eagle, have their reasons. Although eagle comes with an extensive part library, some times the part you want is not in the package you want, and other times neither the package or part you want is in their libraries. In these cases you are you are left with two choices. First, pick a similar part that already exists. Second, make your own part. This instructable will focus on the later option.

step 1: Start the Eagle control panel

That step should be self explanatory.
In linux type eagle from the command line.
In windows double click on the eagle icon.
Or start->programs->eagle layout editor (version) -> eagle

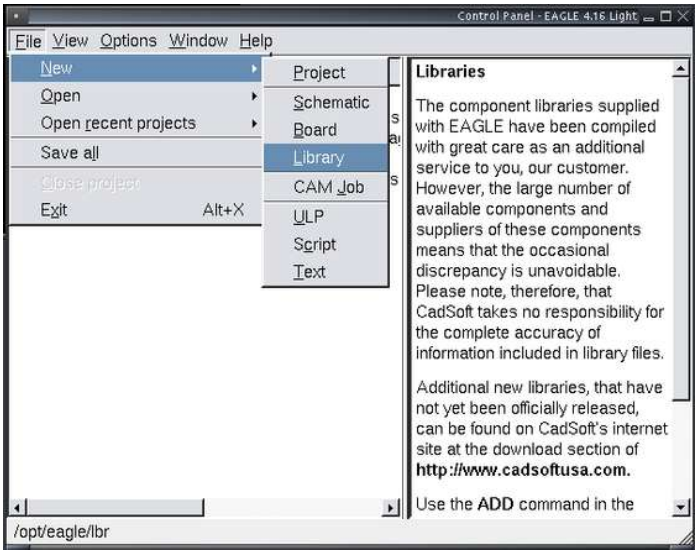
Your screen should look something like this now.

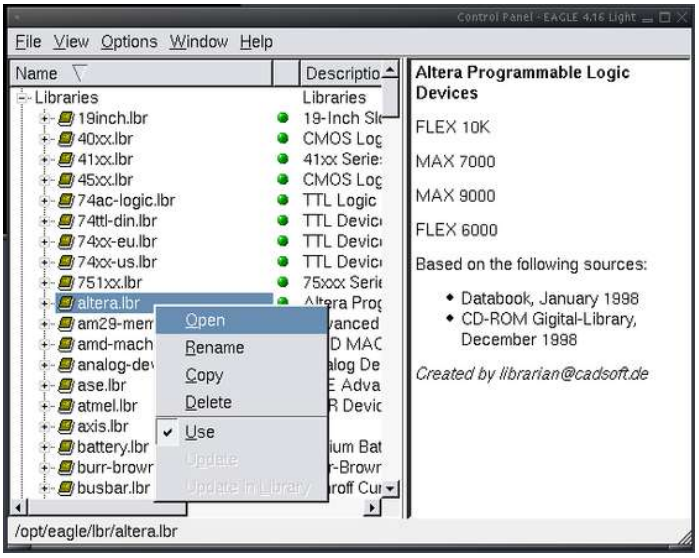


step 2: Select or create a library

Decide where you want your new part to be. I suggest creating your own library. If you have your own library it will be easier to share your work with others.
1) To create a new library go to the menu bar and select File->New->Library

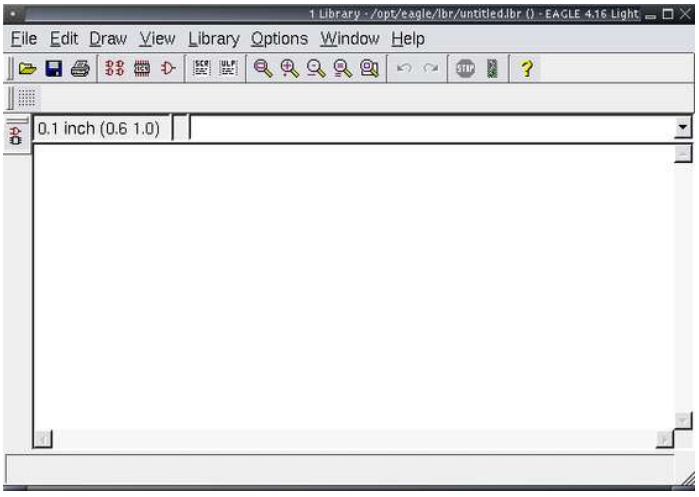
2) Add to existing library in the left pane of the control panel right click on the library you want to add the part to and select open.





step 3: The new library

Your screen should like like this. From here on out I will assume that you created a new library, but this really doesn't matter.



step 4: The easy way or the hard way.

To design a part in eagle you must define a device, package, and symbol. Each aspect has its own set of layers that you must keep straight. Again you are left with two choices. The easy way, in which you copy a similar part and tweak it to match your specifications. This is of course in contrast to making one from scratch. For this instructable we will design one from scratch.

step 5: Time to get out the data sheet.

For this instructable we will design a part used in the IMU for the PSAS rocket. The object of our affection is the ADXRS150 gyroscope from Analog Devices. To get all the parameters we need for the design we need not look any further then our trusty data sheet.

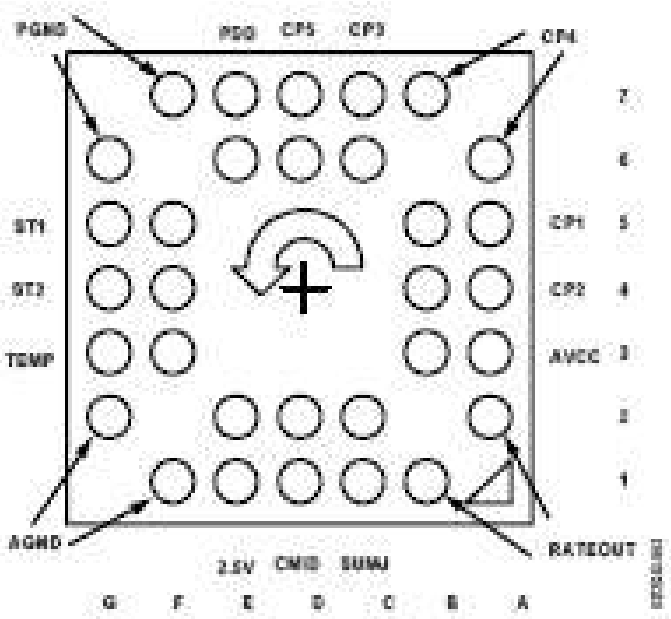


Figure 3. BGA-32 (Bottom View)

step 6: The Package

As I mentioned there are three aspects to a part in eagle. We will start with the package. We want to make a 32 lead BGA (Ball Grid Array). From the data sheet we can see that the balls are 0.55mm in diameter, and spaced 0.80mm on center appart. The far edges are 4.80mm appart on center.

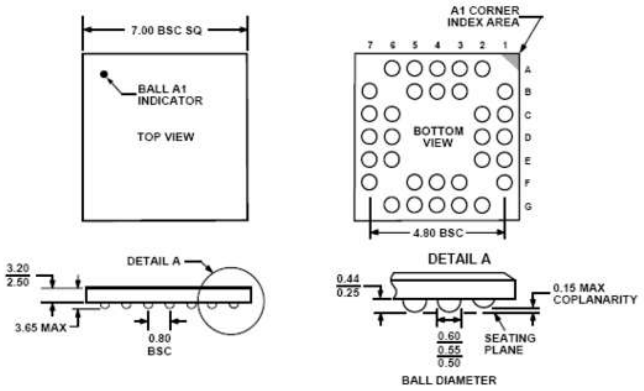


Figure 28. 32-Lead Chip Scale Ball Grid Array [CSPBGA] (BC-32)
Dimensions shown in millimeters

step 7: Building the package

click on the package icon in your library window. The edit box will pop up and in the "new" field type BGA-32 (remember we are making a 32 lead Ball Grid Array). and hit ok.
You will get a warning asking "Create new package'BGA-32', click "yes".

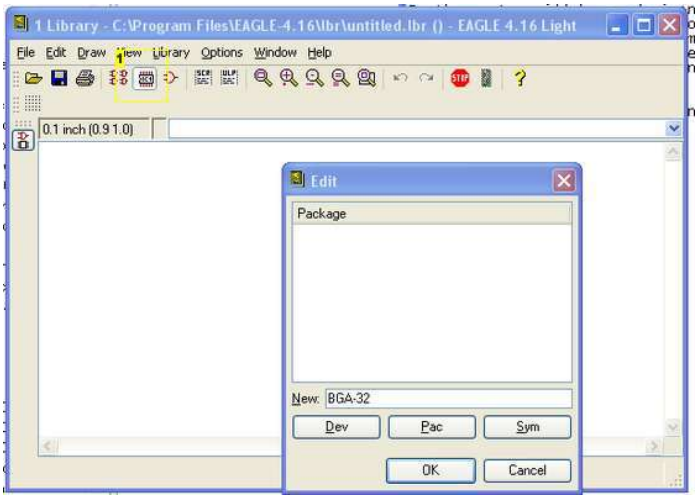


Image Notes
1. package button

step 8: Setting the Grid

The default eagle setup will create a black screen with a grid on it.
In the center will be a dominant white cross. This cross is the center of our package. It will be the point by which people will select/move the package around.
Placing our pads and other parameters wisely around this cross is important.
From the previous set we know we need some fine resolution make the grid half of what our smallest component is.

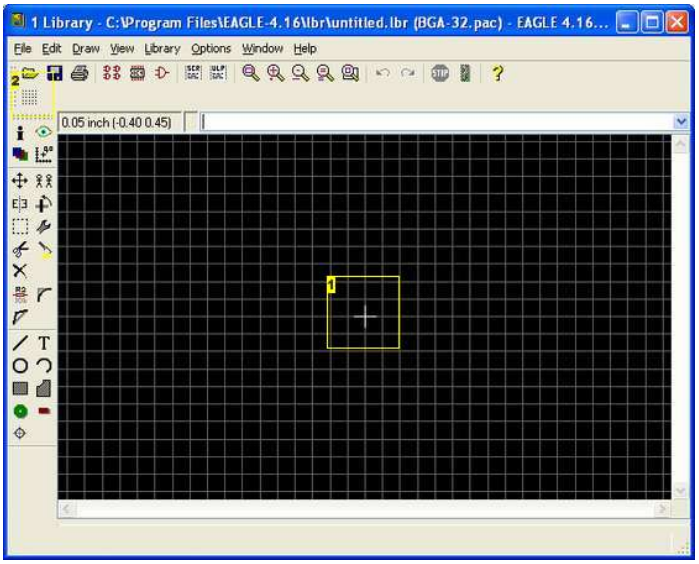
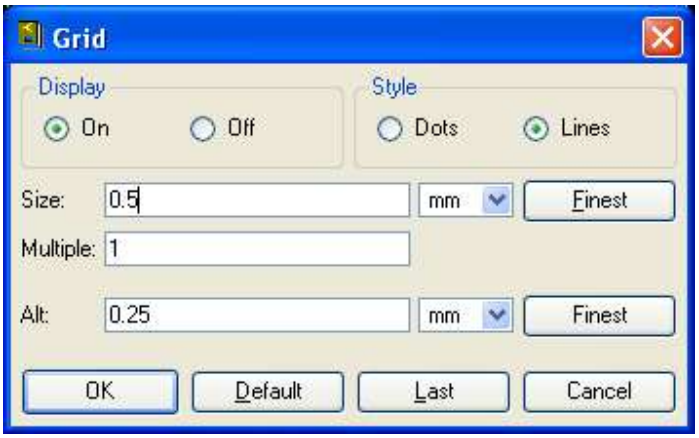


Image Notes
1. proposed center point for device
2. Grid button

step 9: Setting the grid (cont)

Recall the data sheet has balls that are .55mm in diameter and are spaced .8mm apart on center.
The centers of the balls on parallel outer edges are 4.8 mm apart on center. So we want a grid size that will make it easy for us to place these balls.
From the "view" menu select grid, or simply type grid into the command window.
The grid tool will open up make the size 0.2 units mm Alt: 0.2 and multiple of 5. Without the multiple the grids are too small to be displayed. Note the lines will now be 1 mm apart. leave the display on
and the style lines. Your screen will have a dizzying amount of grid lines on it.

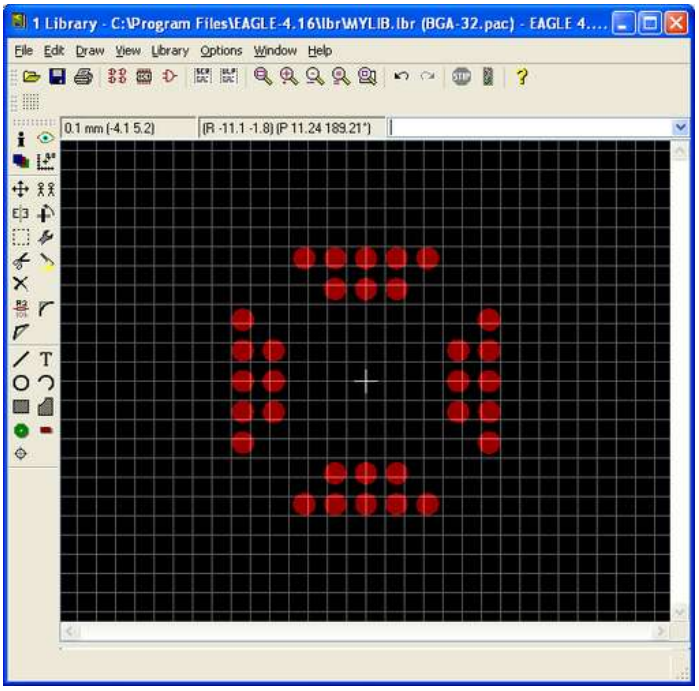


step 10: Adding Pads

At this point if you want to copy another package from an alternate library you can use the copy command with the following syntax in the command window.
copy packagename@libraryname
and the package will magically appear, but being man a first principles I'll show you the long way.

As mentioned earlier one must be careful to make sure one adds elements to the appropriate layer. Our pads (i.e. balls) for example will belong to the top layer. In the command window type smd, this command will be used to create the pads. By default the top layer will be selected. In the Smd drop down box will not have a circle by default in that box type "0.55 x 0.55", and make the roundness 100%. I also placed a second cross hair as a reference guide 7mm up and 7mm over know that is the over all size of the chip. One measurement that is missing is how far from the edge are the pins. Being a slave to symmetry I made the assumption that the center of the ball would be .8 mm away from the edge. With properly spaced grids, using the mouse to place pads can be very quick accurate. Alternativly, in the command window if you can type (x-cord y-cord) and the pad will be placed where you want it. Place the pads as well as you can, and it should look like this when you are done.

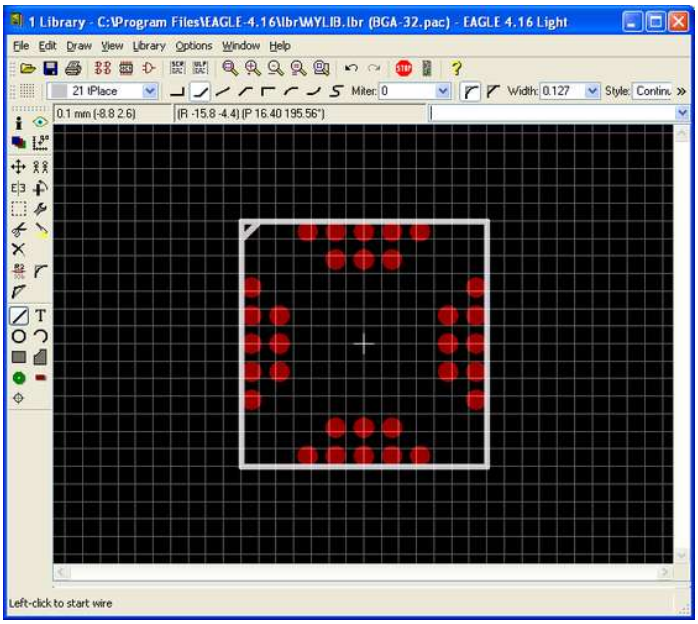
Hints: It may be easier to make the origin the center of the device and just give the coordinates to place the pads (3.2 0) (-3.2 0) ... etc



step 11: Details for a cleaner look

On the tPlace layer put an outline of the chip's foot print and make the Ball A1 indicator visible with the wire tool.

Type wire in the command window. Select 21 tPlace for the layer. Now draw a 7mm box around all the pads you placed in step 10. Either trace it out or type the coordinates in the command window.



step 12: Name Pads

To make our life easy in the furture its a good idea to name the pads. Type name in the comand window, and double click on each pad. A dialog box will appear and simply type in the new name. Its good to go off of what the data sheet uses for names as you will have to repeat this process for the symbol. Following this advice will make the final step (matching package with symbol) much easier, however, it does not make for a generic package (i.e. when you want to use this package for a different device).

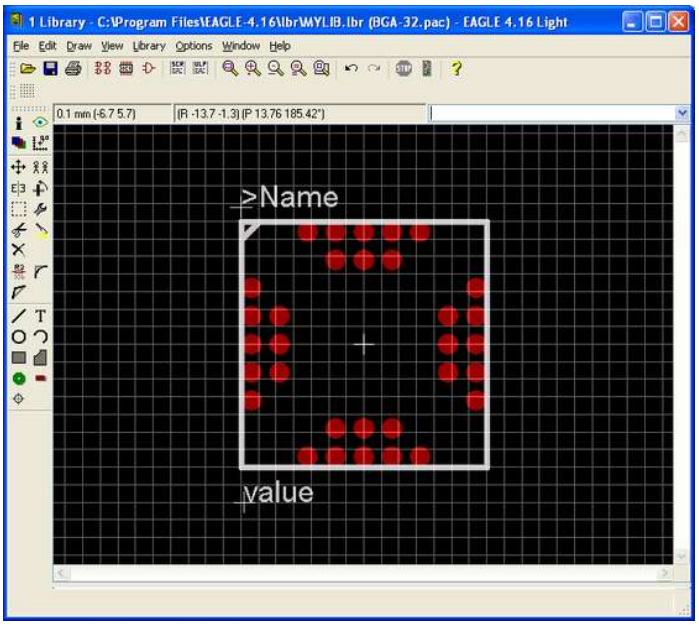
step 13: Add name and value

name and value parameters are added on separate layers tname and tvalue respectively. These will be named later on by who ever is using the package so just put in generic headings like "name" and "value"

Select the text tool or type text in the command window. Select the tName layer, and an appropriate size and place on the top of the drawing.

Repeat this process for the value but use the value layer.

Test to make sure you have the right elements on the right layers by selecting the layer tool and turning off all the layers except the one you want to check.



step 14: Building the Symbol

Click on the symbol button and add a new symbol. This step is identical to step 7 except its for a symbol not a packge. The symbol is what will appear when you are drawing your schematic. The schematic is a fundamentally different representation of your circut then the layout (or package view). The package needs to match the datasheet as it represents the physical entity and has a huge impact on the baord layout. The schematic should be designed so that it is easy to read and neeed not be a prefect representation of the devcie (in terms of size). For examplep pins without connections dont need to be placed on the schematic.

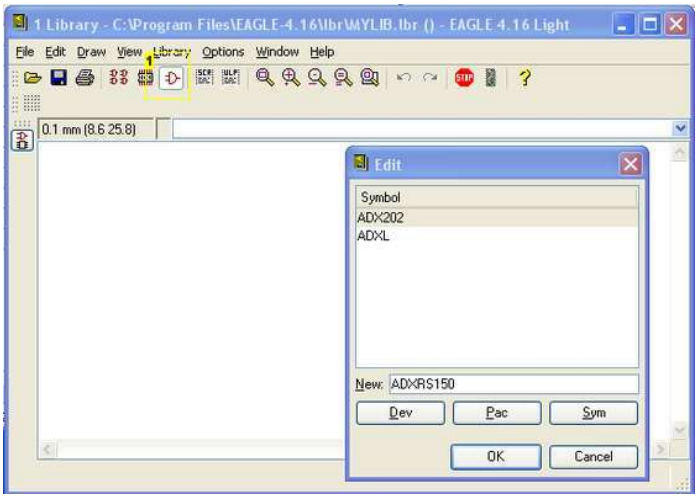


Image Notes
1. symbol button

step 15: Back to the data sheet

On some devices not all pins are used. However for this device all the pins are doubled up. We can also see that all the pins have names. To make life easier it is a good idea to name the pins that are placed on the symbol.

PIN CONFIGURATION AND FUNCTION DESCRIPTIONS

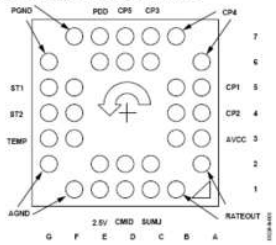


Figure 3. BGA-32 (Bottom View)

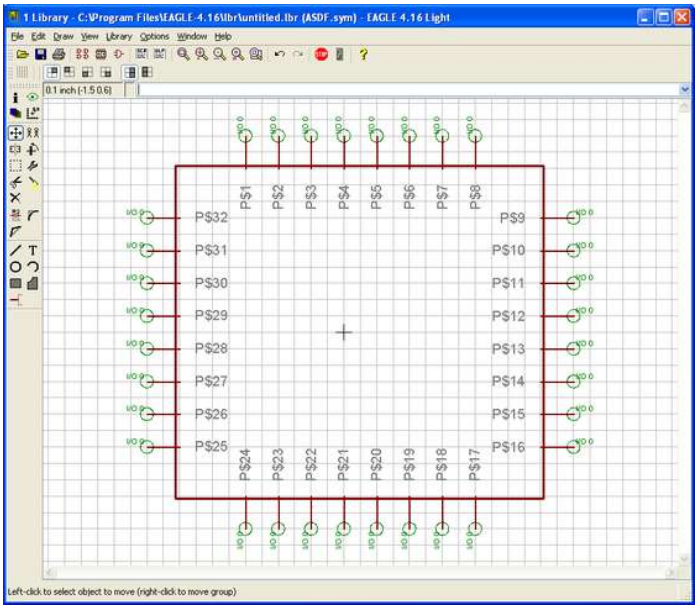
Table 3. Pin Function Descriptions

Pin No.	Mnemonic	Description
6D, 7D	CP5	HV Filter Capacitor—47 nF
6A, 7B	CP4	Charge Pump Capacitor—22 nF
6C, 7C	CP3	Charge Pump Capacitor—22 nF
5A, 5B	CP1	Charge Pump Capacitor—22 nF
4A, 4B	CP2	Charge Pump Capacitor—22 nF
3A, 3B	AVCC	+ Analog Supply
1B, 2A	RATEOUT	Rate Signal Output
1C, 2C	SUMJ	Output Amp Summing Junction
1D, 2D	CMID	HF Filter Capacitor—100 nF
1E, 2E	2.5V	2.5 V Precision Reference
1F, 2G	AGND	Analog Supply Return
3F, 3G	TEMP	Temperature Voltage Output
4F, 4G	ST2	Self-Test for Sensor 2
5F, 5G	ST1	Self-Test for Sensor 1
6G, 7F	PGND	Charge Pump Supply Return
6E, 7E	PDD	+ Charge Pump Supply

step 16: Draw the symbol

Use the wire tool to draw a box that will represent the symbol on the schematic. By default you will be drawing on the symbol layer. Double check to make sure by looking in the upper left corner after the wire tool is selected. The layer drop down menu should have "94 Symbols" selected.

Once the box is drawn, type "pin" in the command window, and start placing the 32 pins evenly around the box.



step 17: Naming Pins

As great a names as P$1-p$32 are it will make our lives easier when we connect pins on the symbol with pads on the package if we use a more intelegent naming shceme. We will assign the names of the pins based on, you guessed it, the data sheet.

Type name in the command window and double click on the pin to remain. A small dialog box will appear with the current name. Change the name and click "Ok". Repeat 32 times.

By default the name on the pin and the symbol will show up in the device. This makes for a very cluttered look. Click on the "change" button and select "visable" from the drop down menu, and then select "Pin". Then click on every pin. It will not be obvious what you are doing but trust me the final design will be easier to use.

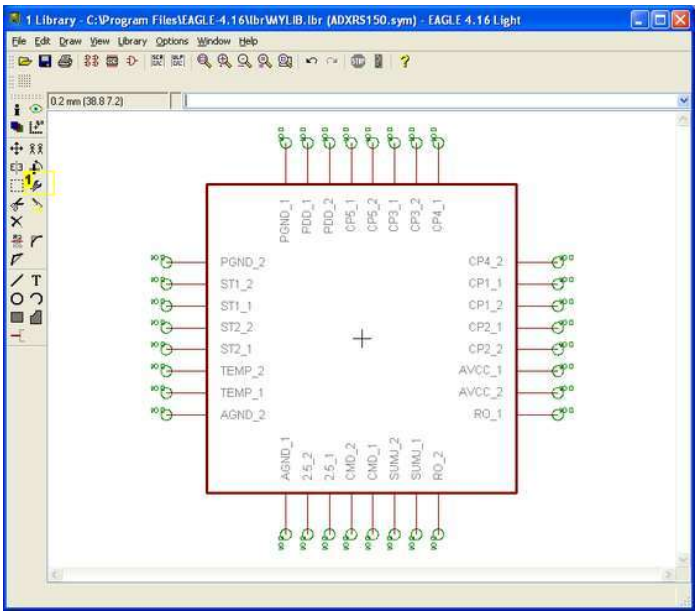


Image Notes
1. Change button

step 18: Make the device

In this step the association between the symbol and the package is made. Click on the devcie icon, add the name of your device, and your screen should like like this.

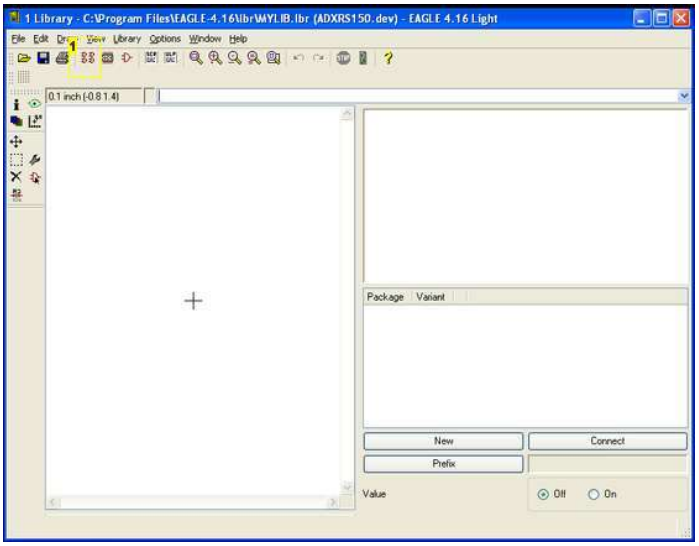


Image Notes
1. device button(sorry about image quality)

step 19: add package to devcie

In the lower right hand corner click on the new button and select the package. Your package will show up in the upper right pane.

On the left vertical tool bar click on the symbol icon, and place your symbol in the left pane.

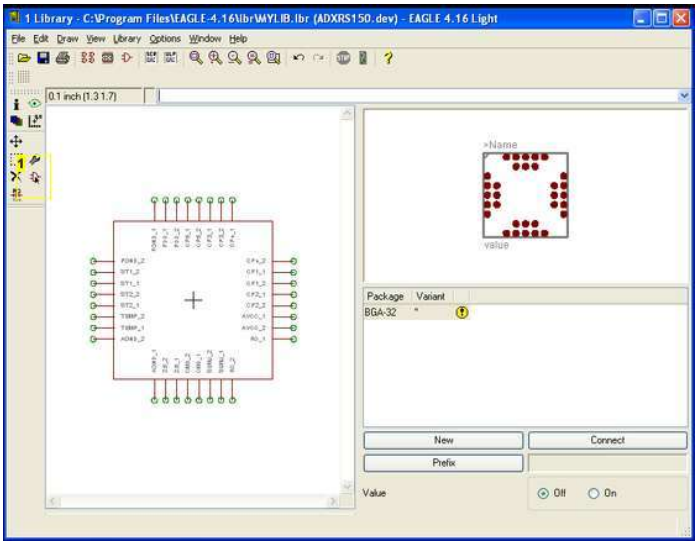
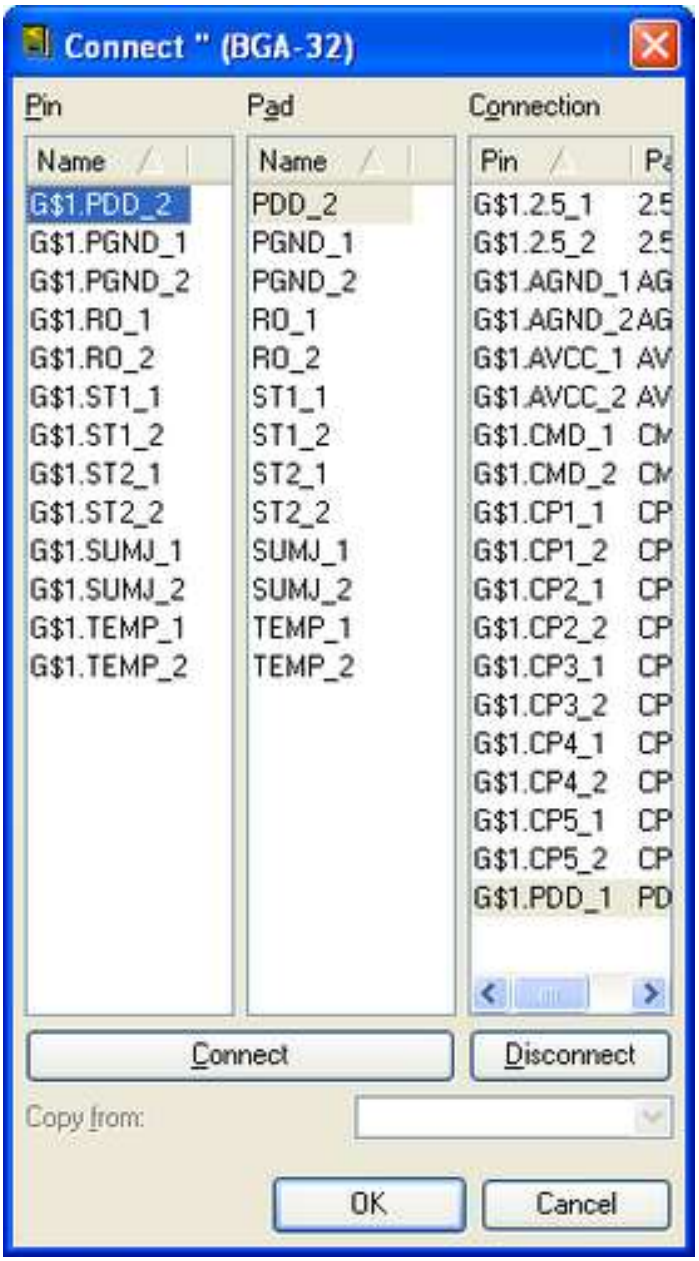


Image Notes
1. left tool bar symbol icon

step 20: Make connections

If you have followed my advice and named the pins on the symbol and the pads on the package the same this step should be easy.

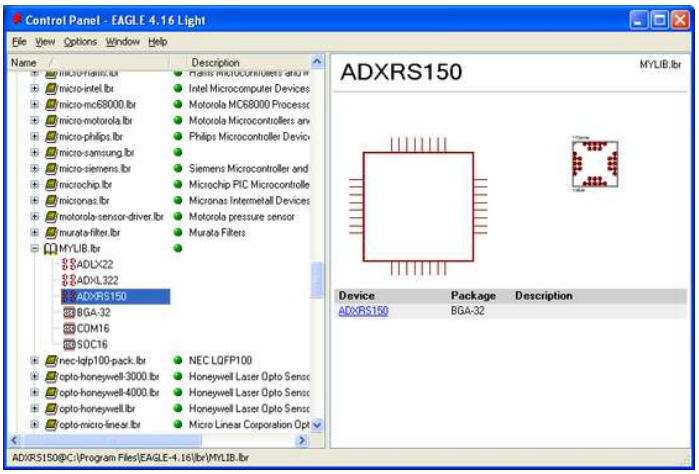
Click on the connect button and the connect dialog box will appear. Keep clicking the connect button untill all the connections are made.



step 21: Save Device

CONGRADULATIONS!! YOU ARE DONE. Click on the save button. It always a good idea to check all is well, so navigate to your library, and expand it by clicking on the plus sign. You should see your device listed. Hightlight it and it will appear in the right hand pane.

Now get to work using your new device.



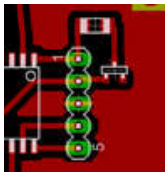
Related Instructables

Eagle by neelandan

High Power LED Head or Bar Mount Light by raintonr

How to Build a Bench-Top Power Supply by greyhathacker45

Render 3D images of your PCBs using Eagle3D and POV-Ray by ongissim



Understanding ICSP for PIC Microcontrollers by ian

Make an RGB Mixer by purduecer

JDM2 based PIC Programmer by ian

LED Hanukkah Menorah by barney_1

Comments

39 comments **Add Comment**

SRChiP says: Jun 14, 2009. 10:42 PM **REPLY**

??

thermoelectric says: May 2, 2009. 3:31 PM **REPLY**

I'm having a little trouble with this part, the measurements are on the second last page but I can't figure out which ones to use...

Can anyone figure out which measurements I use?

amtekdesign says: Jan 9, 2009. 9:26 AM **REPLY**

This step is confusing. Should item (1) be done, then item (2), or is it (1) OR (2)? I'm guessing the latter, based on the title. However given that creating your own library is recommended, why even mention adding to an existing library?

Doktor Jones says: Apr 28, 2009. 11:59 PM **REPLY**

If you have your own custom library already, it might make more sense to add new devices to that library rather than creating a new library for each custom device you add :)

It could be clearer though that this is an either/or step.

gandalfsz says: Mar 21, 2009. 5:50 AM **REPLY**

Very good !
Made a display in about half an hour

amtekdesign says: Jan 9, 2009. 9:50 AM **REPLY**

The screen capture doesn't seem to match the instructions. Which is correct?

amtekdesign says: Jan 9, 2009. 9:35 AM **REPLY**

The "data sheet" link is broken. The new link is ADXRS150. However note that this part is to be obsoleted, replaced by ADXRS613.

beazleybub says: Sep 27, 2008. 9:21 PM **REPLY**

The image covers instructions in step 7.

forrealhomie says: Aug 21, 2008. 10:17 PM **REPLY**

great tutorial! good job dude

everything says: Jul 7, 2008. 4:11 AM **REPLY**

Nice tutorial!
but can someone give a link to the other tutorial, where it explain how to copy a library part in eagle, and just modified it?
I have seen it before, but i can't find it again

sorry for my bad English, I'm from norway...

tgdavies says: Jul 6, 2008. 3:51 AM **REPLY**
Excellent tutorial -- thanks for taking the time to do such a good job!

justy says: Jun 29, 2008. 8:38 PM **REPLY**
I've done this (excellent!) instructable before with no probs. This time however I seem to create a package that can't be selected in the board editor. I re-did the part, and it worked for a while, then I re-dimensioned the part and the problem occurred again! weird...

justy says: Jun 29, 2008. 8:40 PM **REPLY**
D'oh- solved my own issue- it was a layer not turned on. tOrigins .. :P

Sparks86 says: May 16, 2008. 7:17 PM **REPLY**
Thanks! Its taken less time to make a device from scratch than search through the existing library trying to find one to copy!

However: Is the package correct in your example? In step 10 you mention (3.2, 0) which is 6.4 across, but the datasheet says 4.8?

xehpuk says: May 16, 2008. 10:52 AM **REPLY**

Great instruction! It was just what I needed.
It took me only a little over an one hour to do the device I needed.

praetorious says: Apr 22, 2008. 5:20 PM **REPLY**
When i am printing the design for a single sided board (bottom layer) from eagle, do i need to mirror the image(printing to pdf) or can it remain as is?

osembedded says: Apr 4, 2008. 12:00 PM **REPLY**
Wow Thanks for the tutorial! I just made my first part in eagle. Eager to see the results when the PCB comes back!

Keep posting good stuff like this!

zachninme says: Jan 17, 2008. 4:47 PM **REPLY**
Does anyone have any idea how one would make a ring, about 1/2 inch width, but divided into 18 segments? I have the ring, I just have no clue how I can separate it...

Spokehedz says: Mar 18, 2008. 8:19 AM **REPLY**
Wait, what?

Dr_Acula says: Jan 2, 2008. 3:46 AM **REPLY**
This is a brilliant instructable. There are lots of relays that don't exist in the library but with this instructable it was possible to take an existing one and modify the pins.

One thing that didn't work straight away is that in the schematic view/add part, the new library would not come up in the list so I couldn't use my new device. I tried all the options in the Library menu at the top of the screen (Library Use, Library Open, Library Update, and Library Update All) and it still wouldn't come up, but then after a few shutdowns and more experiments it now does appear. But I don't know exactly which step worked! Any suggestions?

josheeg says: Jul 4, 2007. 7:07 PM **REPLY**
I think it would be useful to have the keys pressed or how to do some things. rather than do this kind of show how. Video tutorials for eagle would be great. But a new user would not know how to fallow this completely.
I apreseate it was done thow.

Arx says: Jun 10, 2007. 5:08 PM **REPLY**
Excellent instructions. Thanks.

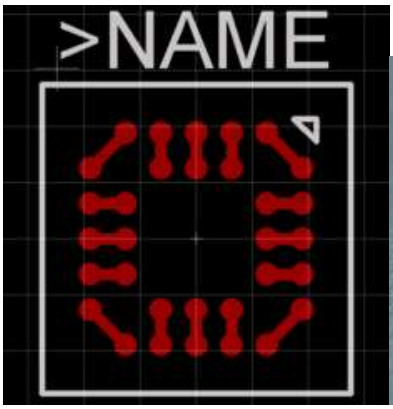
sureshundley says: May 3, 2007. 12:20 AM **REPLY**
Hi all, I m suru, i've a corel draw file now i would like to make pdf file with the help of corel draw software and like to put a hyper link on the text, how it has to be done i m not having idea, anybody has any idea so plz share ur knowledge.

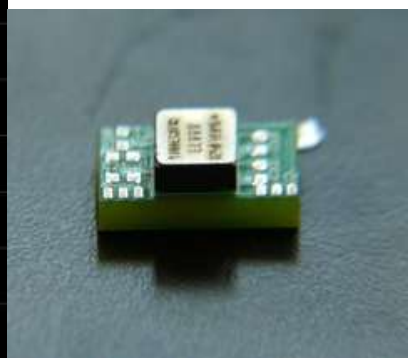
waiting for +tive response.

colin says: Mar 10, 2006. 4:23 PM **REPLY**

Awesome instructable.

I actually have been working with this exact same chip. For my library part I went ahead and hardwired the pairs of pads that share the same signal. It simplified the layout later, although it also caused the DRC to freak out somewhat.





---

zanfar says: Mar 28, 2007. 6:02 PM **REPLY**

You have the right idea Colin, the trick to get around the DRC, is to not use the PAD command for the duplicate pads, instead, just draw circles on the TOP layer. This way, the DRC won't think you have unconnected pins, and you don't have to manually connect the duplicates in the schematic.

Making circles that match the pads is a bit difficult, but you can use the command:
CIRCLE 0 (0 0) (0.275 0)
to do so. This command will create a single circle at the origin of the right dimensions, copy it to wherever you need it.

The CIRCLE command syntax is:
CIRCLE <WIRE_WIDTH> <CENTER_POINT> <CIRCUMFERENCE_POINT>
Where a wire width of 0 is a filled circle.

---

xoxota says: Mar 22, 2006. 10:11 PM **REPLY**

How did you end up getting it past the DRC finally?

---

kd7vnn says: Apr 3, 2006. 1:47 PM **REPLY**

I didn't have the DRC problems Colin had because I went the long way and didn't connect the pins together.

---

kd7vnn says: Mar 13, 2006. 2:06 PM **REPLY**

Colin:
Thanks for your comment. I just put this up because I didn't have a lab notebook and thought this would be cool place to put stuff. I'm working on an IMU, so anything you would like to share about how to calibrate one of these little buggers, I'd LOVE to hear about it!!

---

jcomuzzi says: Feb 4, 2007. 12:18 PM **REPLY**

Great contribution, thank you.

So I created a part for a connector, but to my untrained eye - I don't think I have any holes (at least I didn't see a place to specify the hole size!) Can you suggest how to add a part that needs holes?

---

jcomuzzi says: Feb 7, 2007. 7:17 AM **REPLY**

OK - I figured it out and I'll admit I'm a turkey. You need to use the "pad" command rather than the "smd" command if you want holes!

---

jcomuzzi says: Feb 7, 2007. 7:16 AM **REPLY**

Something that took me a while to figure out here: If you want a pad with holes, use the "pad" command rather than the "smd" command!

---

woop says: Dec 24, 2006. 7:40 AM **REPLY**

i am new to eagle, never would have figured it out. i made a part for the LT1399 op amp. thanks!

---

mkmckenzie says: Aug 22, 2006. 1:35 PM **REPLY**

So, there is no built-in way to connect multiple package pads to a single symbol pin?

kd7vnn says: Oct 15, 2006. 2:42 PM **REPLY**

Not that I have found, and I have looked.

led555 says: May 2, 2006. 1:30 PM **REPLY**

I just followed your procedure to create a part for MIC2981, an 8-channel, high-voltage, high-current source driver array.

led555 says: Apr 22, 2006. 5:44 PM **REPLY**

very informative!

joelotz says: Mar 13, 2006. 1:44 PM **REPLY**

Great instructable!! I find this helpful and will definitely use it.

dworden16 says: Mar 12, 2006. 12:19 AM **REPLY**

By far one of the best instructables.

alceste says: Mar 10, 2006. 3:53 PM **REPLY**

Thanks! I just had to do this the other day, and I'm sure I would have forgotten how to do it again.