

Draw Electronic Schematics with CadSoft EAGLE

by **westfw** on August 6, 2006

Table of Contents

Draw Electronic Schematics with CadSoft EAGLE	1
Intro: Draw Electronic Schematics with CadSoft EAGLE	2
Step 1: Create new project	2
Step 2: Create new schematic in the project	2
Step 3: Find and place ("add") components	3
Step 4: Add Integrated Circuits	4
Step 5: Add resistors	5
Step 6: Add capacitors	6
Step 7: Add lamps	7
Step 8: Add other semiconductors	7
Step 9: Add Ground symbols	8
Step 10: Re-zoom the drawing	8
Step 11: Save your work often	9
Step 12: Fiddle with the layers a bit	9
Step 13: Start making connections	12
Step 14: Neaten things up	14
Step 15: Apply component values	16
Step 16: Do Rule Check!	18
Step 17: Done!	19
Step 18: A couple of other hints	19
Related Instructables	22
Comments	22



Author: westfw

Middle aged geek

username also works at yahoo.com, mac.com, comcast.net, wharton-10.arpa

Intro: Draw Electronic Schematics with CadSoft EAGLE

There are a couple instructables here on some of the finer points of Eagle (making your own library parts: <http://www.instructables.com/id/ERHQQ180Y3EP286NQY/> modifying the design rules: <http://www.instructables.com/id/EZVIGHUBGCEP287BJB/>) But feedback indicates that a lot of people could probably use an instructable on the more basic aspects of creating a schematic and board.

This instructable covers creating a schematic, presumably from a printed schematic in a magazine or image on the web. I'll start with schematic shown, which is from <http://www.bowdenshobbycircuits.info/555light.gif> It's got a "typical" collection of parts, and is vaguely useful as well.

Cadsoft EAGLE generic information:

Cadsoft EAGLE is available from <http://www.cadsoftusa.com/>

Cadsoft is a German company that is a veritable mecca of software distribution enlightenment. In addition to the reasonably-priced professional PCB design packages (\$1200), they have freeware, lite, non-profit, and other intermediate licenses. Their software runs under windows, linux, and MacOSX. It's slightly quirky, with a steep (but not too high) learning curve on the front end, but from most reports it is not any more so than other professional CAD packages. They have online support forums that are active from both the company and other users, the package is under current development and gets better with each release. A number of PCB fabricators will accept their CAD files directly. It's good stuff. Use it. Propagate it. Buy it when you "go pro."

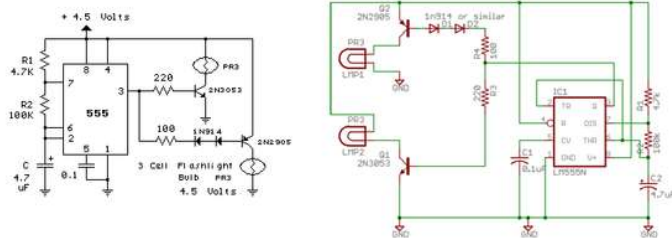
See also:

[Creating PCB from Schematic](#)

[Creating Library parts](#)

[Design rule modification](#)

[Send CAD Files to manufacturers](#)



Step 1: Create new project

Start up the Eagle control panel, and right-click on "projects" to create a new project. You'll get to name it whatever you want.

Step 2: Create new schematic in the project

Once you have created the new project, it will be "opened" automatically (which doesn't do much other than tell EAGLE that "this is the current project".) Right click on the new project and follow the popup menus to create a new schematic.

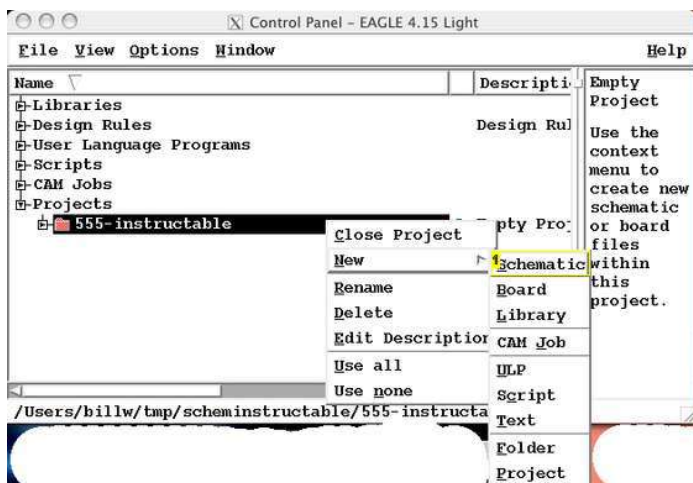


Image Notes

1. Create new schematic in the project

Step 3: Find and place ("add") components

Components are added to a schematic from the ADD dialog, which you get to by clicking the ADD button over on the GUI menu.

One of the major challenges of using Eagle is finding the components you want in the "official" libraries. These libraries are extensive, not particularly well named (and the components aren't so well named either), and seem to date back to a time when there was a different philosophy about multiple packages for a particular device. Resistors and capacitors have so many packages defined that picking the right one is difficult. Transistors, despite formable leads, tend to only have a single package defined. Many experienced EAGLE users don't use the standard libraries at all, copying common components and packages into private libraries or creating them from scratch.

The search capability of the add dialog is pretty good; you just have to be less specific about what you search for if you expect to find it.

When working from a published schematic, I prefer to add all the components in the approximately correct locations before I start connecting anything.

Once you've finished the add dialog, the part will get attached to your mouse pointer, and you can put it down wherever you want on the drawing page by clicking the left mouse button. The right mouse button rotates the part 90 degrees. The middle button, or left and right simultaneously (maybe) "mirrors" the part drawing, which may also be useful.

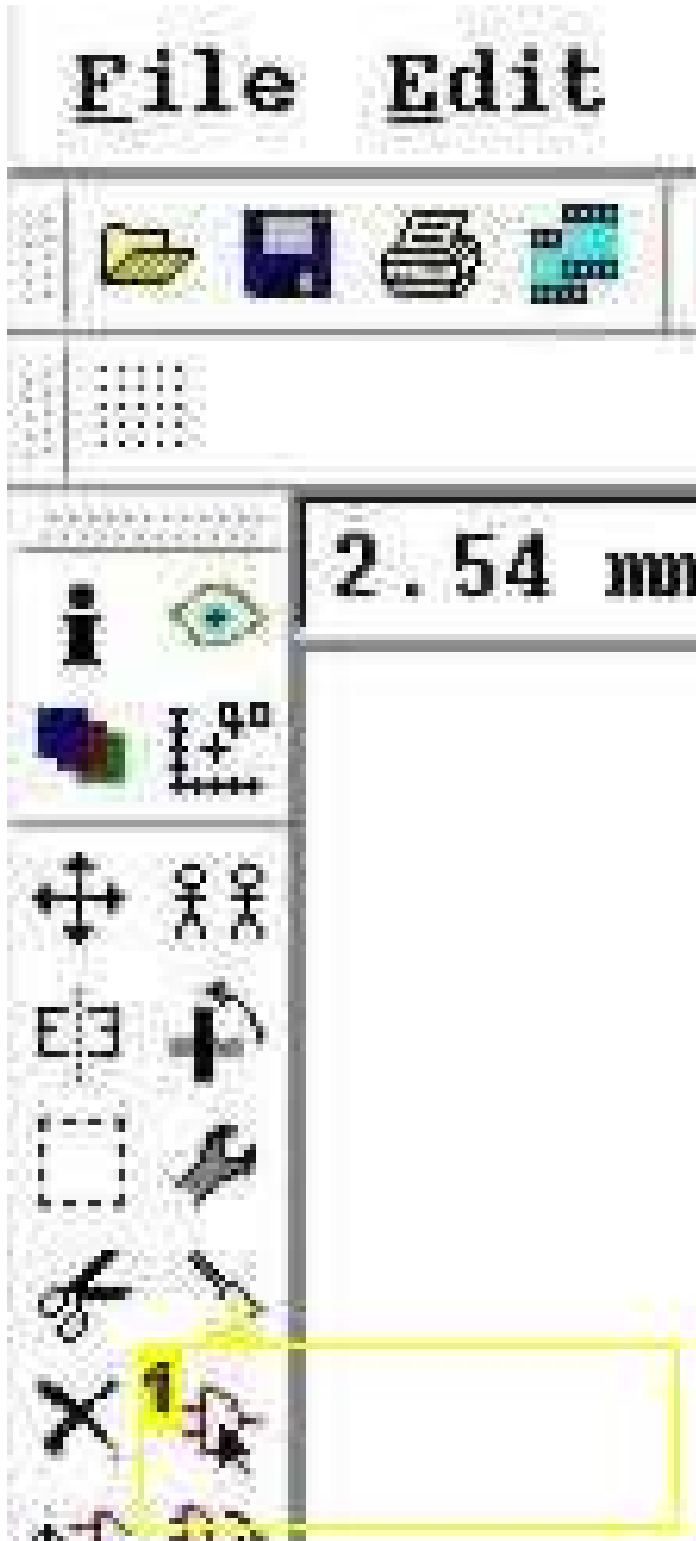
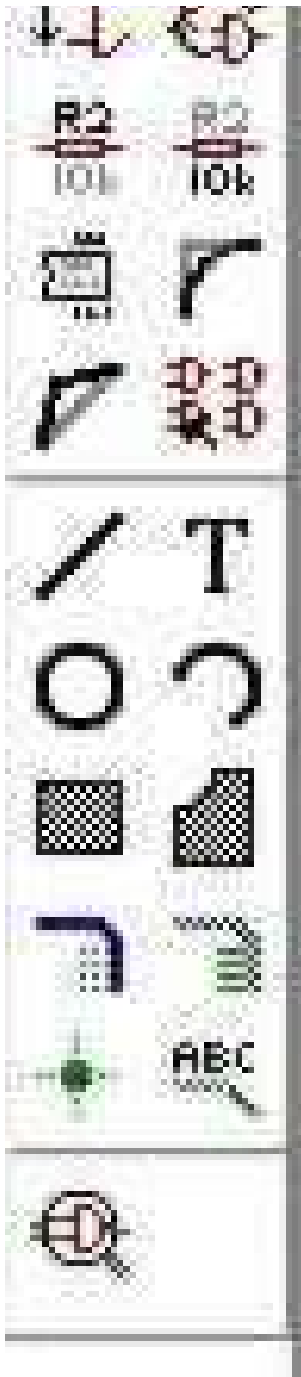


Image Notes

1. ADD menu button



1007114

Image Notes
1. ADD button

Step 4: Add Integrated Circuits

Let's search for our 555 timer. Lots of companies have manufactured the 555, and given it slightly different suffixes and prefixes. LM555N, SE555, TL555, NE555; all are pretty interchangeable. If we search for just "555", there are several results; we can check the preview panel to see which one looks most appropriate. In this case, we'll use LM555N from the "linear" library.

(The string search in the add dialog will need wildcard designators around a substring, so you search for "**555*" to find all parts with a "555" in the name.)

You'll notice that the EAGLE part drawing doesn't look very much like the 555 drawing on the schematics we're trying to enter, the pins are in different places, and have funny names attached as well as pin numbers. And it's not like the pin layout matches the actual package shape either. We'll have to be careful when making our connections.

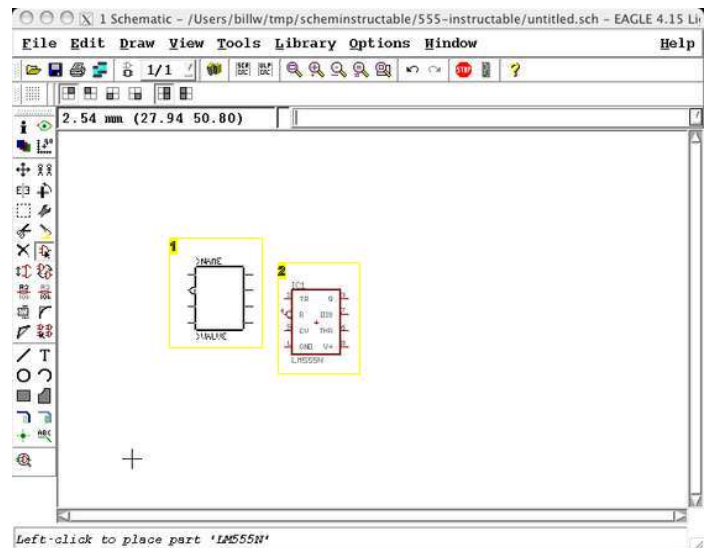
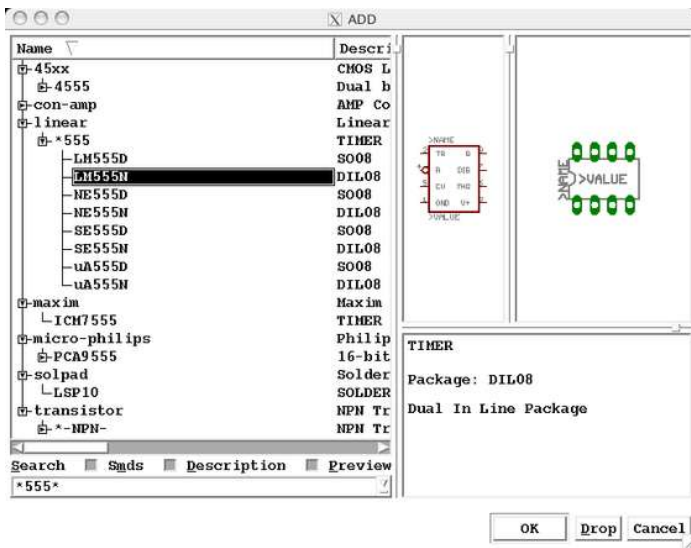


Image Notes

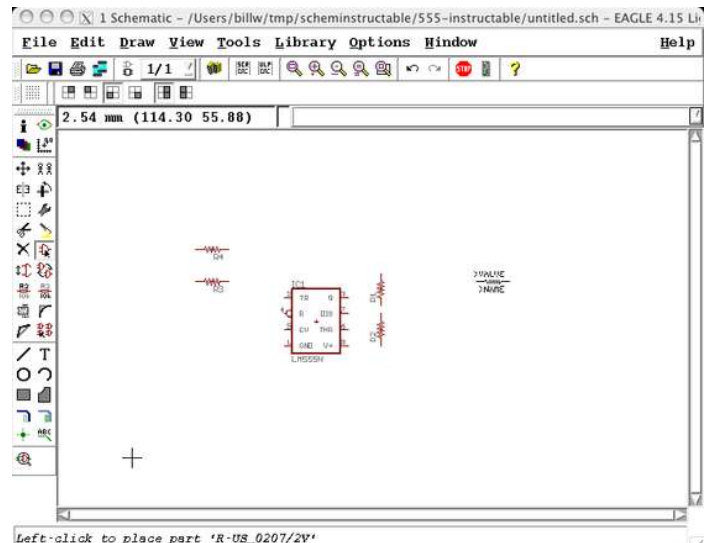
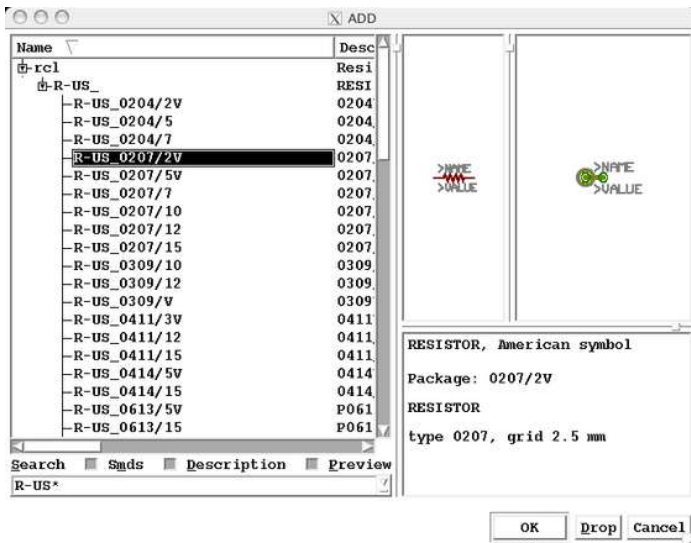
1. The editor is ready to add another 555
2. A 555 is added.

Step 5: Add resistors

Resistors, simple components that they are, cause a lot of confusion for Eagle beginners, perhaps because the eagle library contains 100+ different package/schematic options for generic resistors. Or perhaps it's because a search for short word like "R" isn't practical, and a search for "resistor" turns up a bunch of special purpose specific-manufacturer devices.

The general purpose fixed resistor devices are called either R-US (if you like the US zigzag line type of schematics symbol) or R-EU (if you prefer the European simple rectangle.) The package options are numerous, but make sense after you realize that there's a common format: "WWLL/SS" where WW is the body width, LL is the body length, and SS is the hole spacing, all in truncated millimeters. A typical 1/4W resistor measures about 2.5mm in diameter and 7mm long; hole spacing depends on how you bend the leads. So R-US_0207/10 is a 1/4W resistor with 10mm (actually 4*2.54, or 10.16mm, since we want to stay close to a 0.1 inch (2.54mm) grid.) R-US_0207/2V is the same resistor mounted vertically with 2.54mm lead spacing. 1/8W resistors are similarly designated R-US_0204/SS"

I'm going to pick a vertical package for use on our schematics, though of course it doesn't matter for the schematic drawing anyway. Perhaps I'll do the board layout in a "related" instructable later...



Step 6: Add capacitors

Capacitors are worse than resistors, largely because their bodies come in a wider variety of shapes (that are less standardized), and of course there are all those different types; disk, ceramic, mylar, film, electrolytic, tantalum, AC filter, etc (and those are just the FIXED value caps!) Again, there are slightly different US and European schematic symbols C-US and C-EU in rcl.lib. Again, there's a plethora of packages, but there's a standard format. In this case it's SSS-WWWLLLL, where SSS is the lead spacing (with an extra digit this time!), WWW is the body width, and LLL is the body length.

Polarized caps are similar (CPOL-US or CPOL-EU in rcl.lib), with a package name like TSSS-DD, where T is a type designator (E for electrolytics, TT for tantalum drops, for instance) SSS is the spacing again (only now it probably has an actual decimal point!), and DD is the diameter (for radial caps)

Most hobby projects can get away with either 2.5mm or 5mm lead spacing, and the designer "remembering" to leave enough space for the physical capacitor body; there might not be a silkscreen anyway.

(the "silkscreen" is the pictures and text describing the components, usually printed in white ink on the component side of the board (if you had it made professionally.)

Library designers spend a lot of time getting the silkscreen to look nice; all wasted if you make a board that doesn't have that printing.)

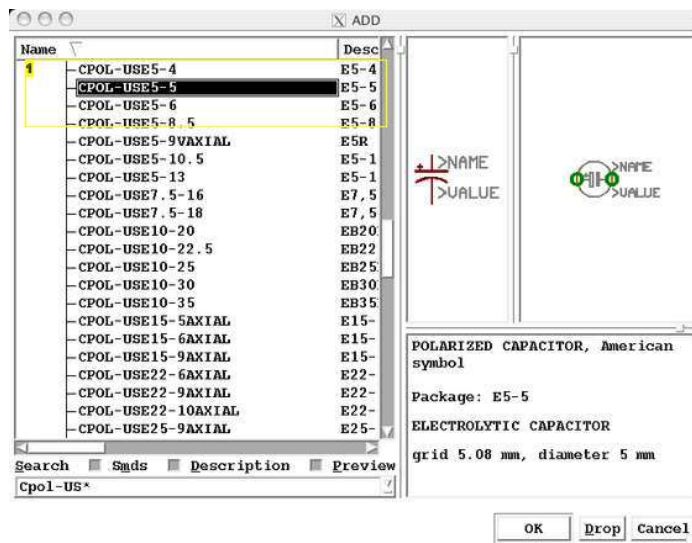
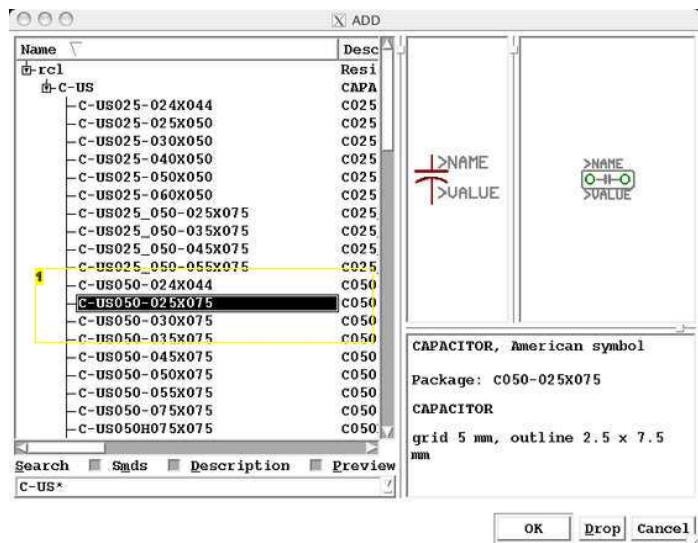


Image Notes

1. Add one disk or ceramic capacitor.

Image Notes

1. And a typical small electrolytic.

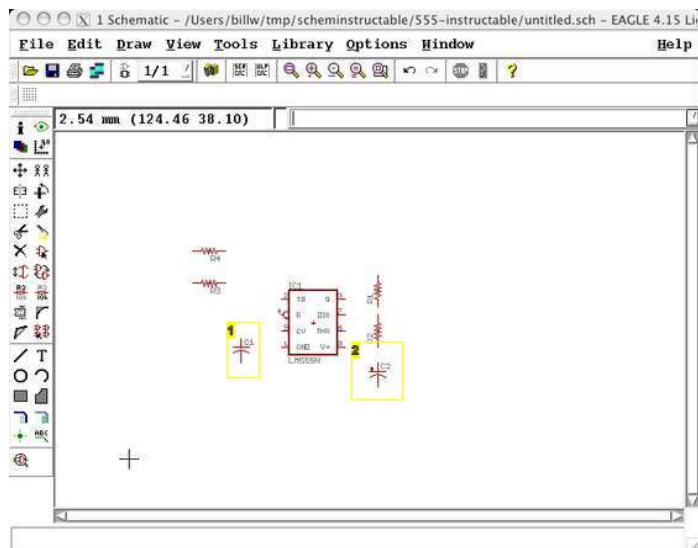


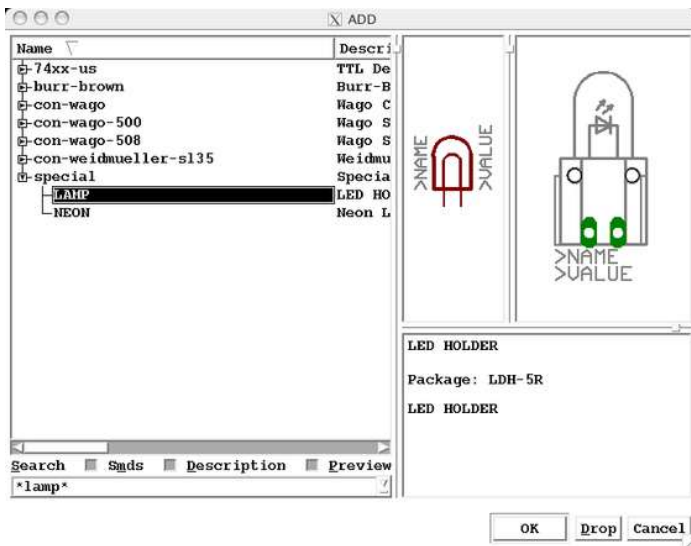
Image Notes

1. disk cap.
2. electrolytic cap

Step 7: Add lamps

When you have parts that aren't going to mount on your actual PCB anyway, such as control knobs and battery packs and switches and lightbulbs and such, you of course have a lot of flexibility in how they are portrayed on the schematic and PCB. You can use single pins for each wire, or find a part whose drawing isn't too obnoxious that has pins of appropriate size and shape for attaching wires as well as an actual component.

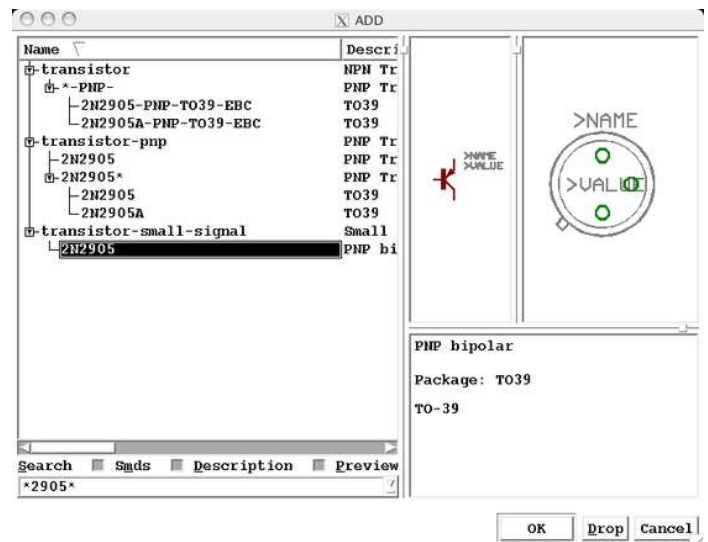
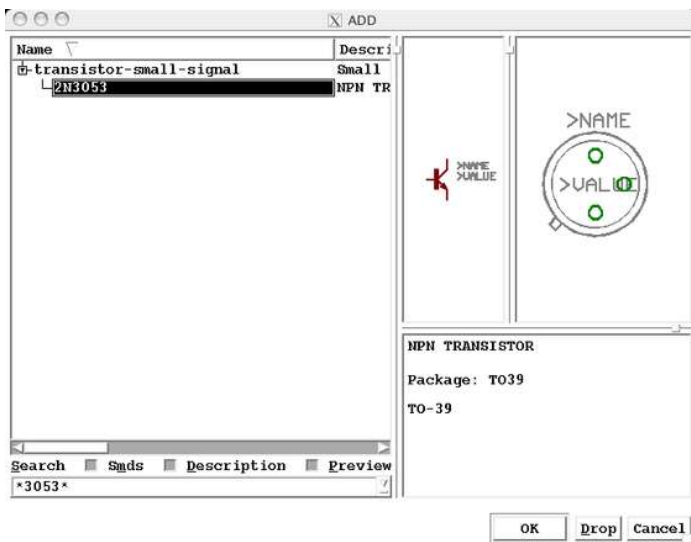
The eagle library has *A* "lamp" part. It says it's actually an LED holder, but the drawing is OK and the part has pins suitable for attaching wires that go to off-board lamps, so it looks fine for our purposes.

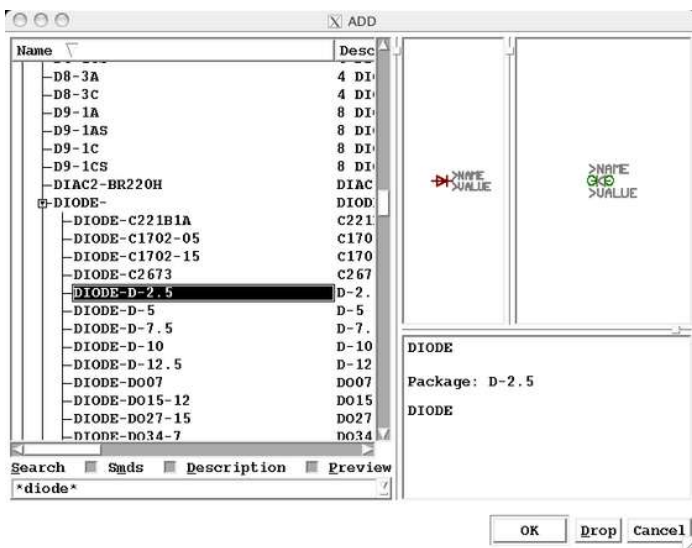


Step 8: Add other semiconductors

We luck out because the transistors called for actually exist in the EAGLE library, and there isn't much choice of package. There are some choices for the PNP transistor, but they're all the same anyway.

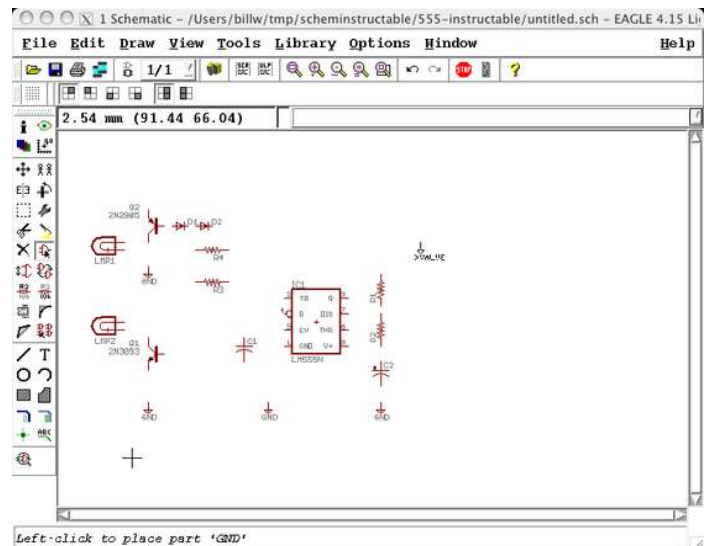
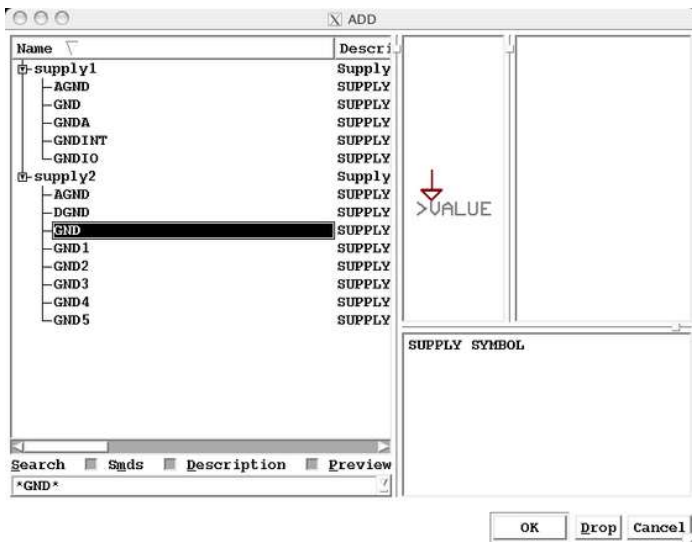
The 1n914 diode isn't in the library, but a search for just "diode" turns up some generic diode footprints that will work fine. I'll mount the diodes vertically to match the resistors.





Step 9: Add Ground symbols

There are some symbols for various supply off in the "supply1" and "supply2" libraries. They're handy, make your schematic look nicer, and have magical properties when it comes to connecting things; the pin on a supply symbol has a magic name so that all the pins on all the GND symbols are connected together, whether you draw them or not.



Step 10: Re-zoom the drawing

Since we have all the components added, we can use the zoom-to-fit button to fit things better in our window...

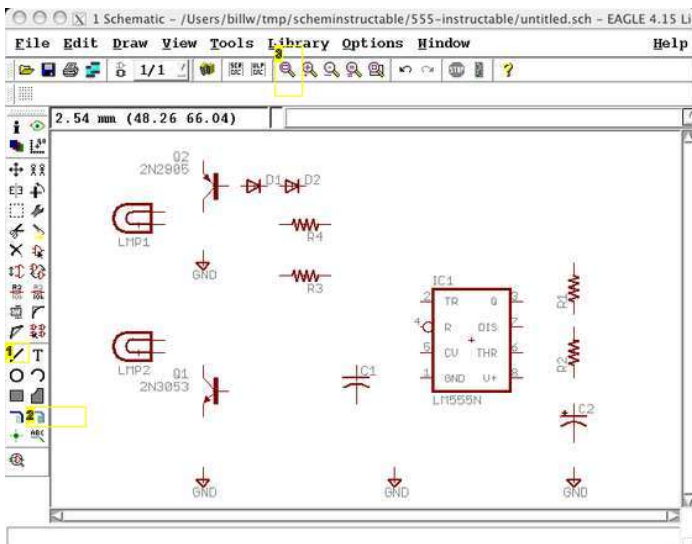


Image Notes

1. This is the WIRE command. Don't use it.
2. The NET command is here!
3. Re-zoom to fit

Step 11: Save your work often

Now is a good time to save our work. This is where you get to attach a name to your schematic, as well. You might need to use "save as" for your first save, to prevent it from saving "untitled.sch".

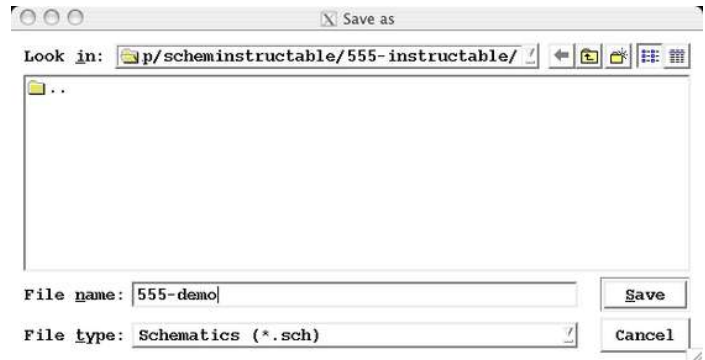
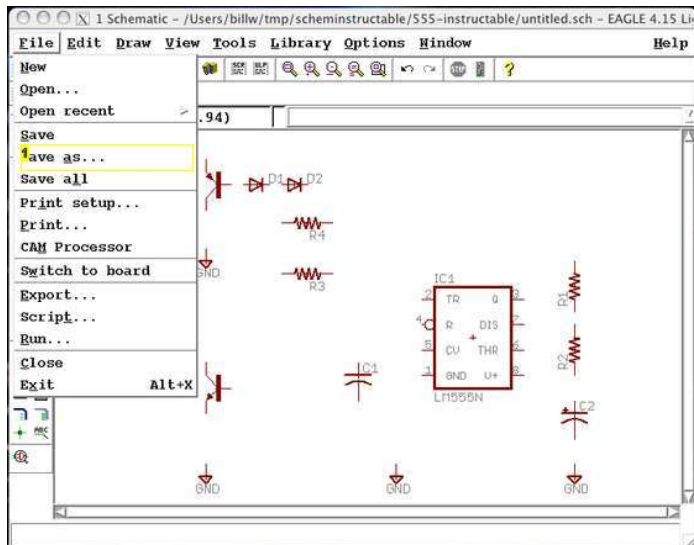


Image Notes

1. Save with new filename

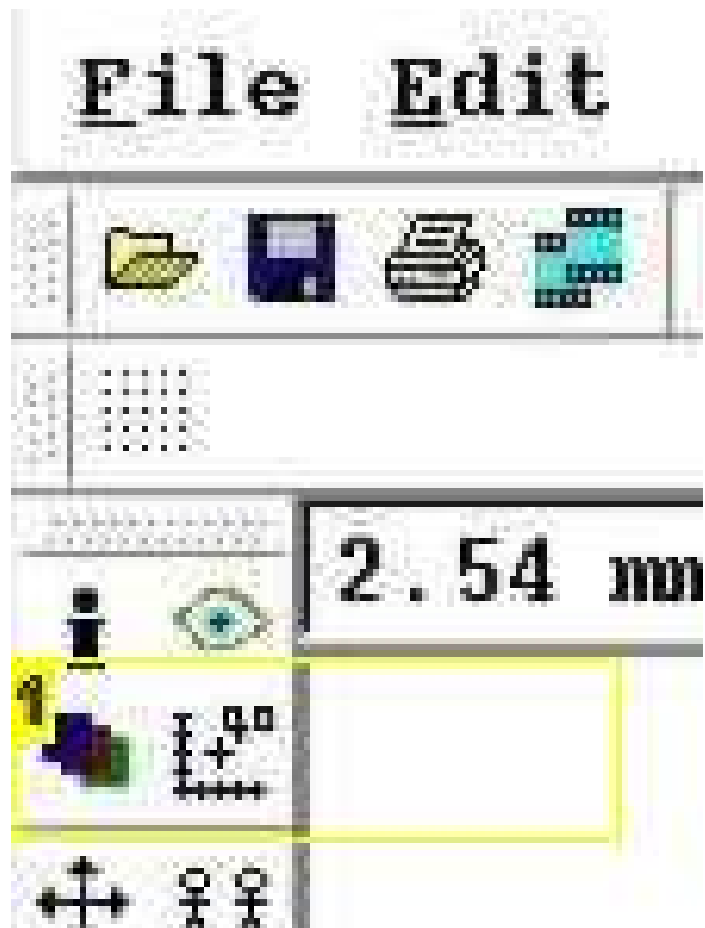
Step 12: Fiddle with the layers a bit

It can be helpful to turn on the "pins" layers so you can see exactly where you are supposed to make connections on the various components.



Image Notes

1. Layers button





PCB

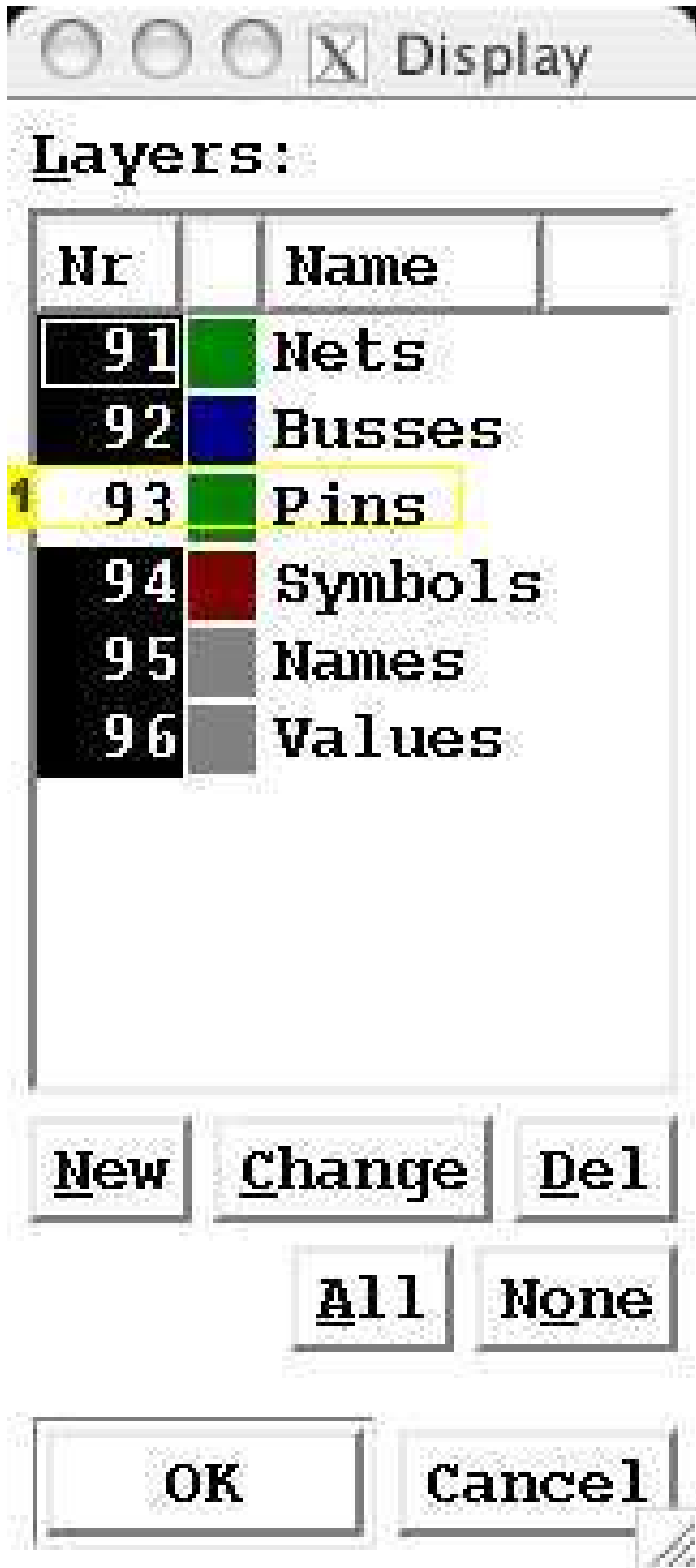


Image Notes
1. Click over the number to display PINS layer.

Image Notes
1. LAYERS button

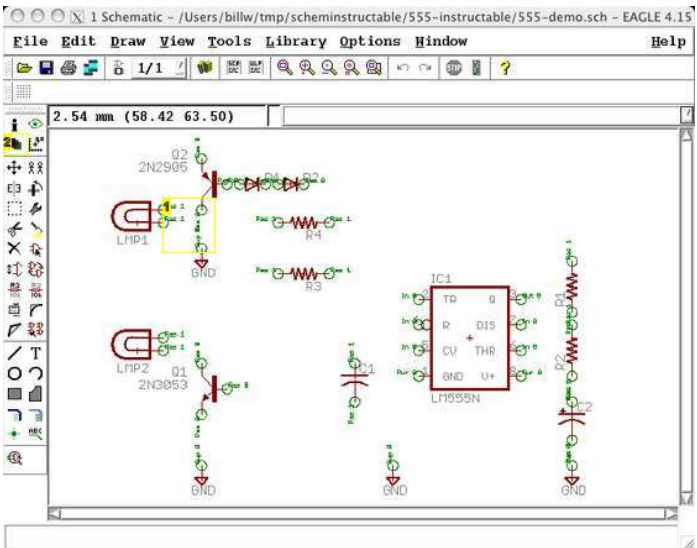


Image Notes
1. The little green circles are the connection points.
2. The LAYER button is here

Step 13: Start making connections

You might think that you'd use the "wire" command to make connections in your circuit.

However, a wire in EAGLE is just a line; to properly make actual electrical connections, you need to use the "net" command. Nets will automatically form junctions when terminated on pins or other nets; I don't think wires are so nice.

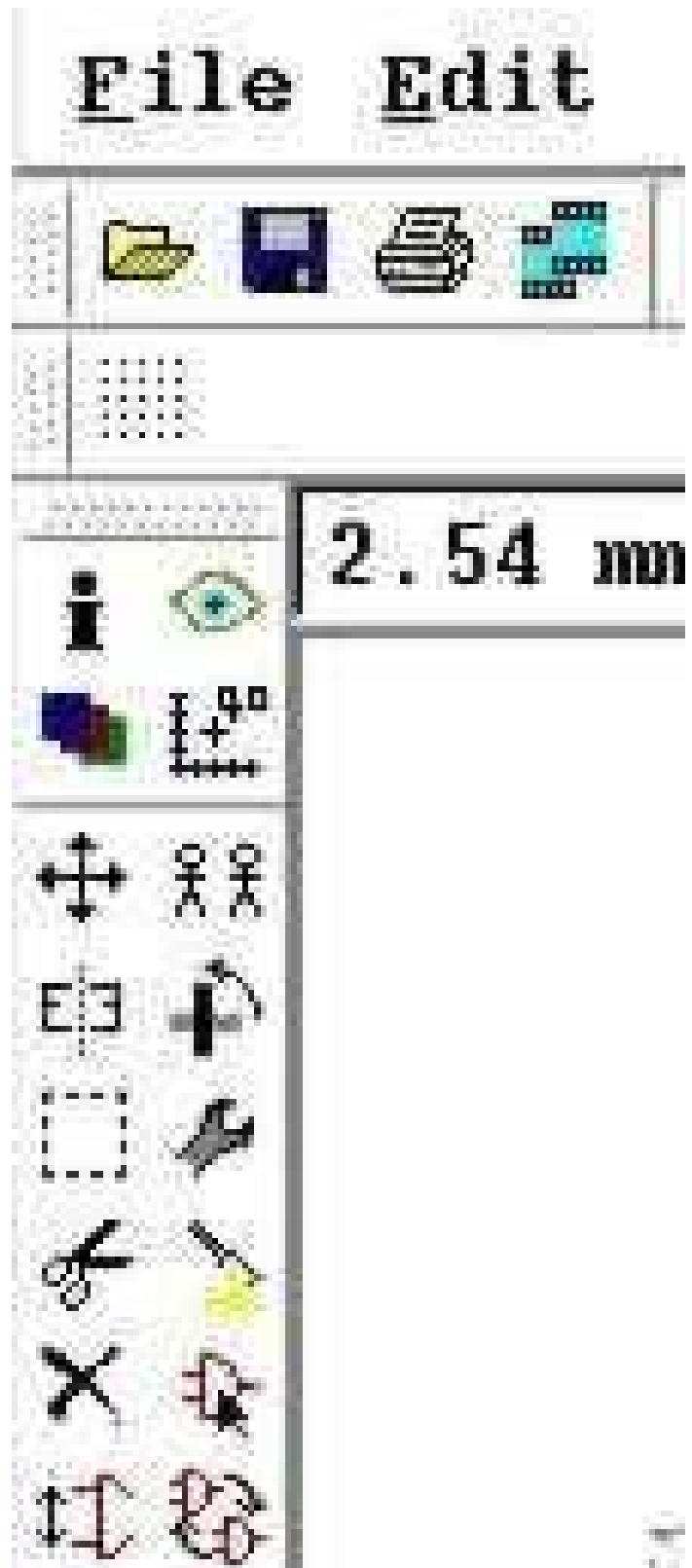
Once you've clicked the NET button, your mouse pointer becomes your drawing tool. Click on a pin (or anywhere, actually), move your mouse somewhere else and you'll see a line. The actual route that the line takes is controlled by the "wire bend" setting for the line, which you can control with your right mouse button. Schematic designers tend to like nice straight lines with right angles.

So first I draw a nice V+ line sort of over the top of the schematic, starting at the top of R1. Next, similar for a ground wire (you don't actually need to connect the GND symbols, they're "magic" in they name the net that connects to them, and all nets of the same name are connected whether you can see lines between them or not.) Then I go wild and connect all the other pins appropriately, either to other pins to the existing nets. Junctions (big solid dots) should get inserted



Image Notes

1. NETS button



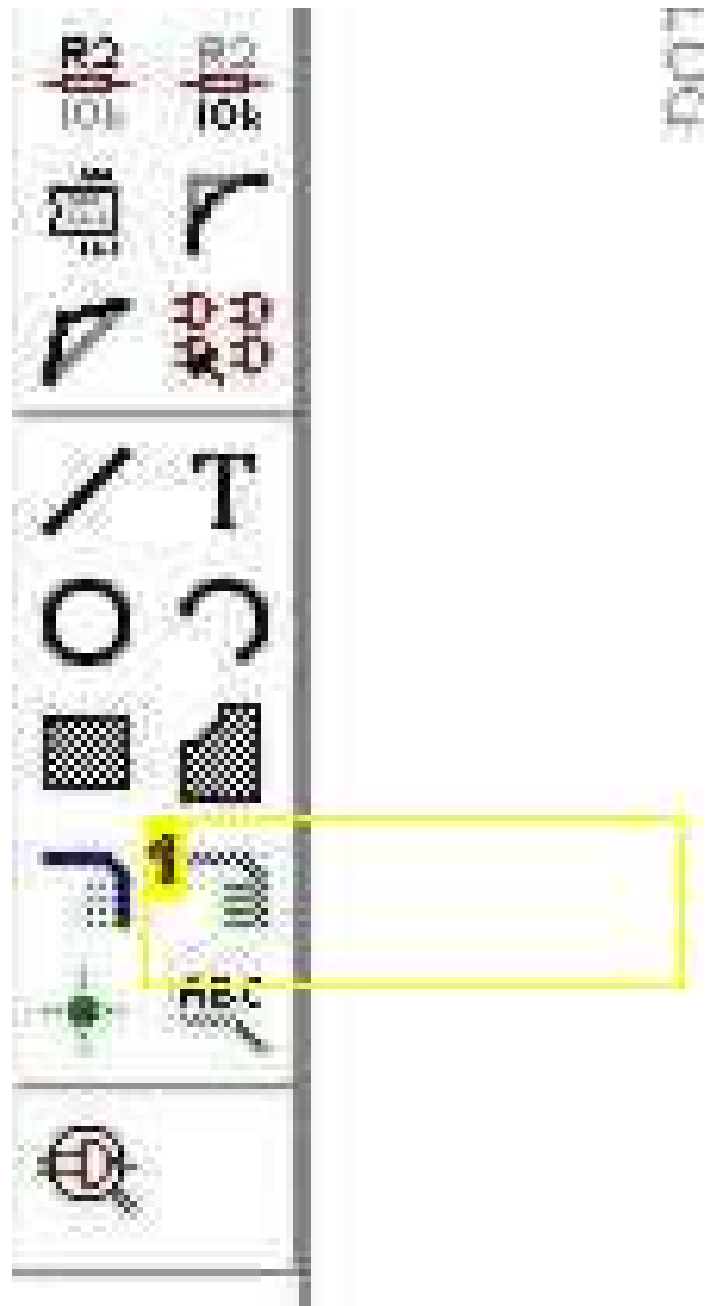


Image Notes
1. NETS button



Page 4

Image Notes

1. MOVE button

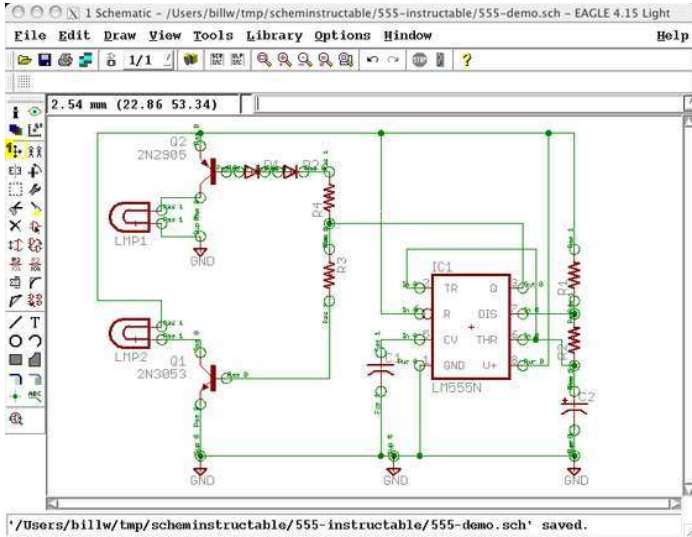


Image Notes

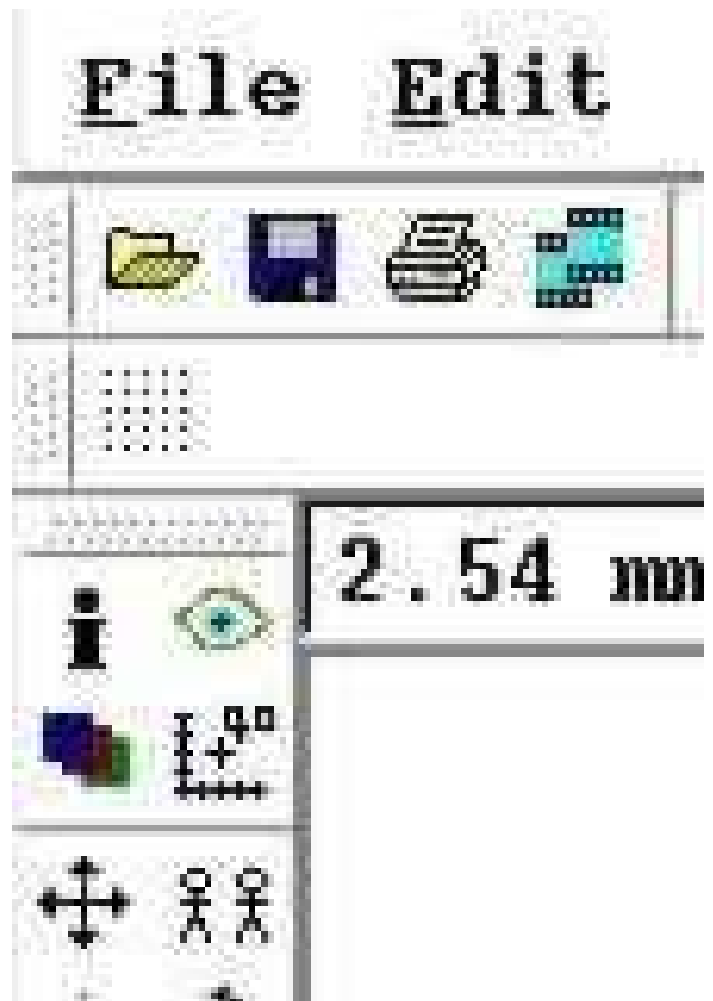
1. MOVE command button

Step 15: Apply component values

When we plopped down the components during the "add" phase, we didn't assign specific values to any of them. Some of the components (ie 555, transistors) have inherent values that don't need to change. But the resistors, capacitors, and diodes should all have their values filled in appropriately.

Values are assigned using the "Value" button. After selecting the button, click on each component near its origin (little "+" sign), and you should be presented with an opportunity to change the value.

R2
10k



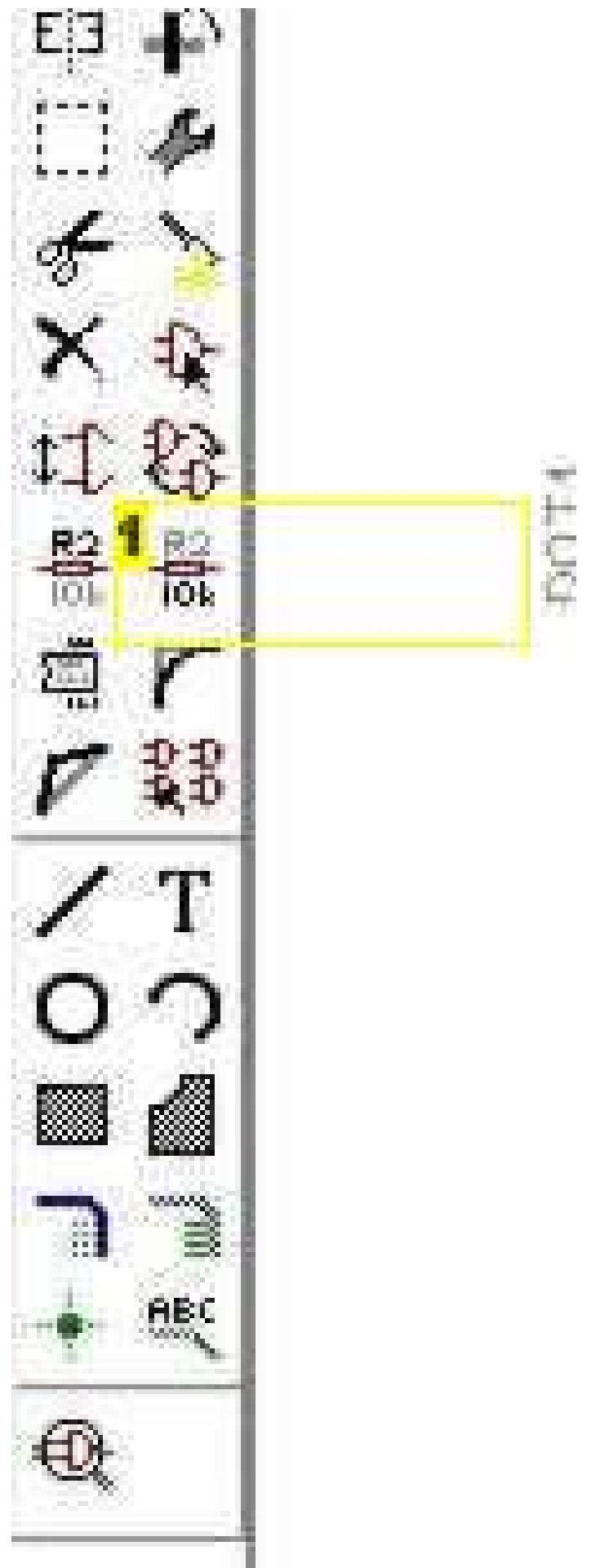


Image Notes

1. VALUE button

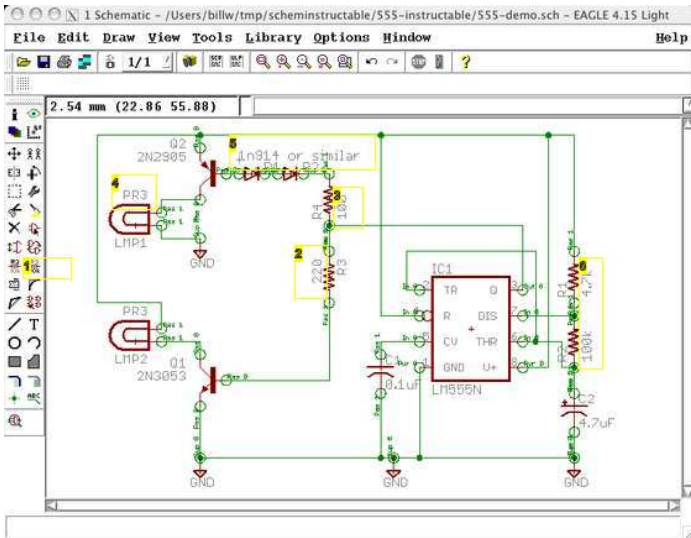


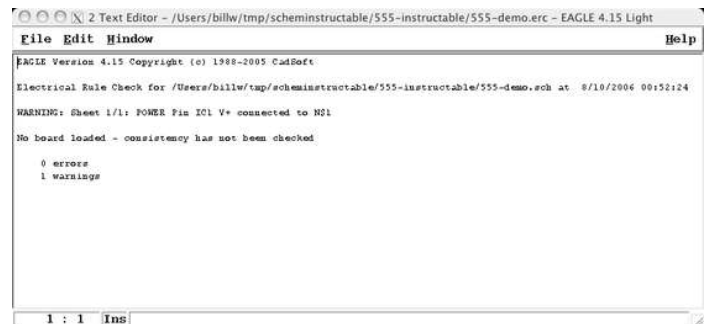
Image Notes

1. The VALUE button
2. Another resistor value
3. resistor value
4. Lamp value
5. Values can be complex text.
6. Resistor values

Step 16: Do Rule Check!

The button shown does an electrical rule check. It will check whether the pins designated outputs are connected to inputs, whether there are obvious missing junctions, and stuff like that. When we run it on our schematics so far, we get a warning that the power pin named V+ on the 555 is connected to a net that DOESN'T have the name V+. We could fix that with the name command, or just leave it as is.

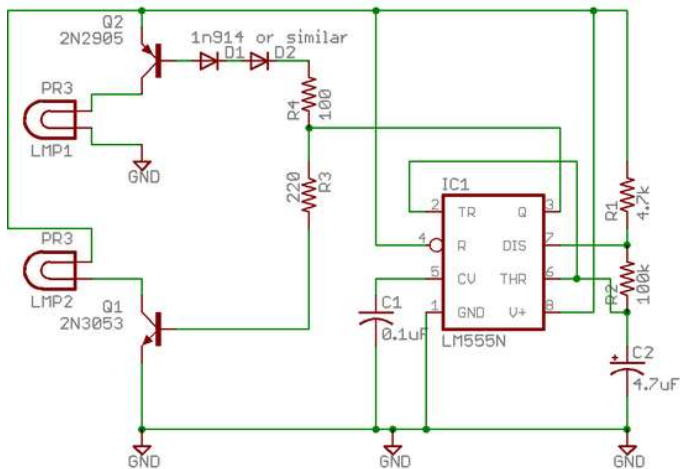
In general, you don't have to FIX every warning that shows up in a rule check, but you should understand what the complaint is, and have a good justification WHY you don't need to fix it.



Step 17: Done!

That's about it, as far as this schematic goes. Turn off the PINS layer to make it less busy; Looks OK, eh? There are a couple things that have been left out that will become apparent when we try to make a PCB from the schematics, but that's a topic for another instructable .

Here's the finished, exported schematic drawing.



Step 18: A couple of other hints

Some of the other buttons that are useful to beginners are described in the pictures.

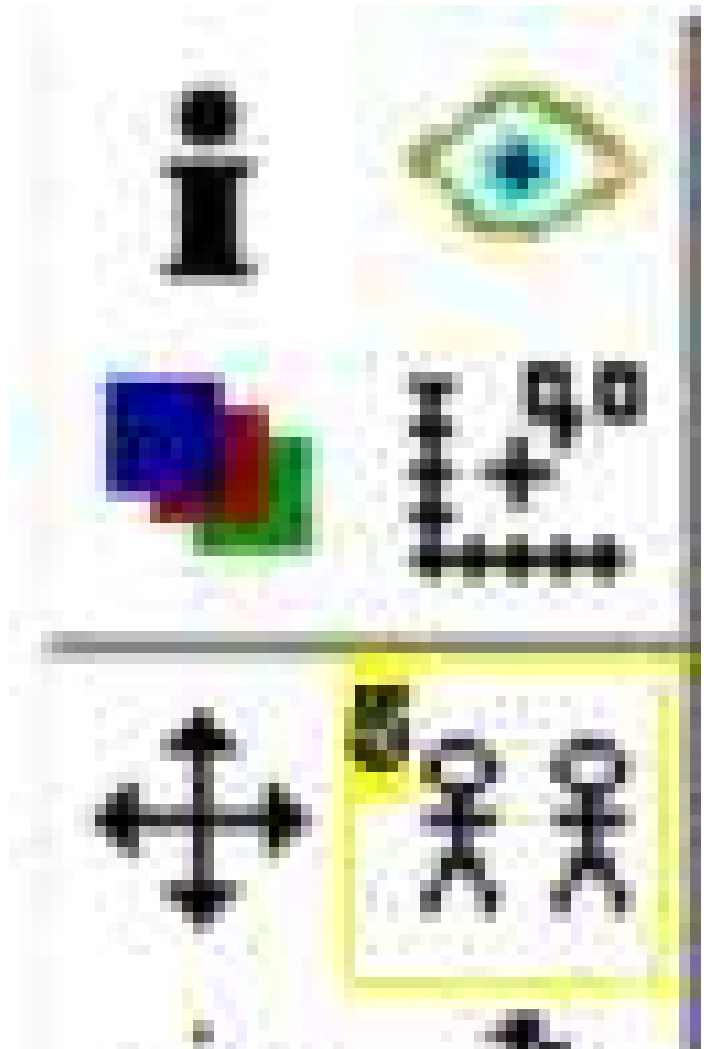
Many EAGLE commands can be entered three ways:

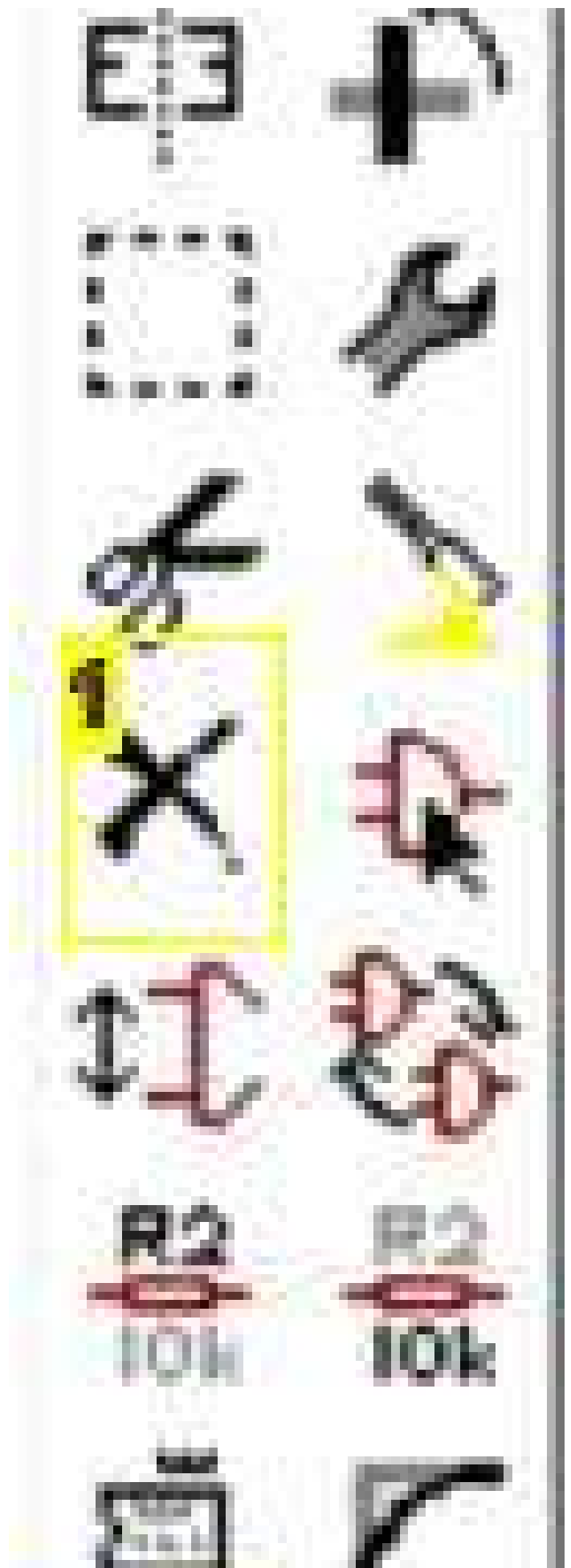
- 1) via the buttons on the left-hand vertical toolbar
 - 2) via the text menus on the top of the window
 - 3) via the text command input area
- Don't get confused...



Image Notes

1. Zoom control.
2. UNDO and REDO. Eagle has pretty infinite UNDO capability for simple commands. This can be very handy!
3. Options that go with the current draw command. For instance, you can change your wire bend style by clicking here if you don't have a right mouse button.
4. text command entry area.





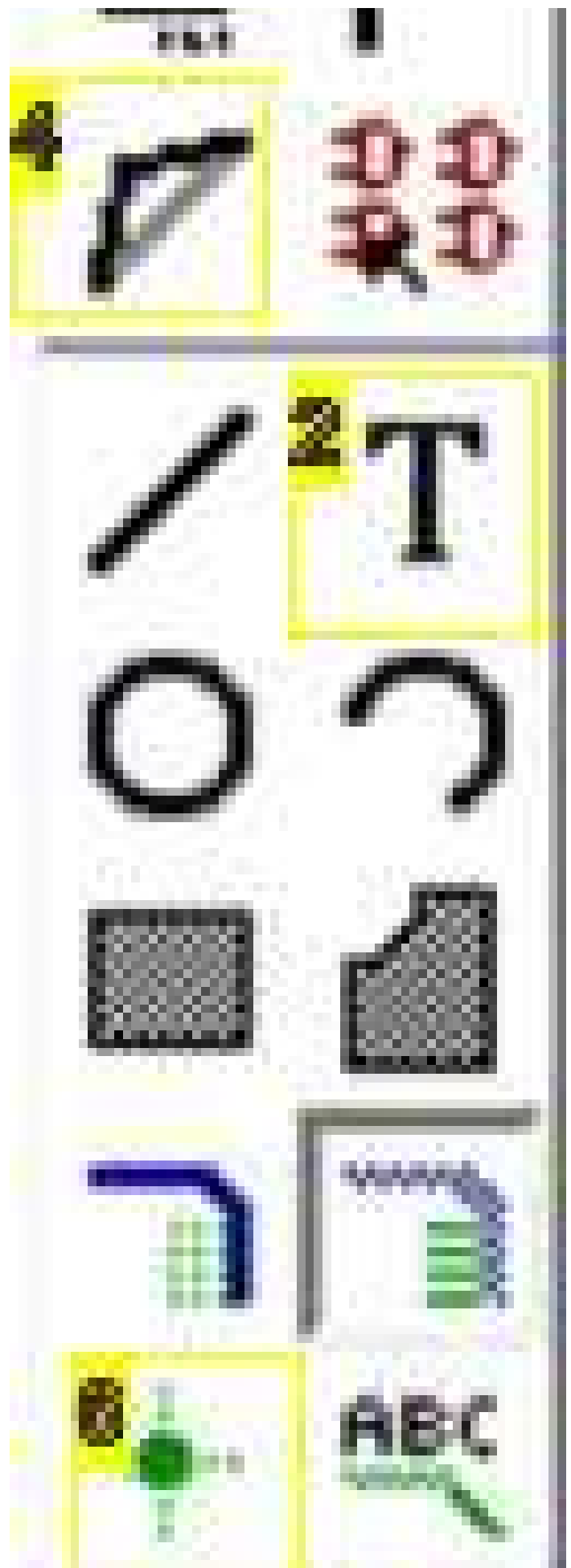




Image Notes

1. Delete an object
2. Add text to drawing.
3. Electrical Rule check
4. Add a bend to a wire. Useful for making things look neater.
5. Duplicate object. Duplicating an existing component may be easier than adding it from the ADD dialog.
6. Create an explicit junction.

Related Instructables



Eagle-ize Leevonk's PIC protoboard by westfw



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



Make hobbyist PCBs with professional CAD tools by modifying "Design Rules" by westfw



How to make a custom library part in Eagle CAD tool by kd7vnn



Create a NOR Gate! by rtty21



Guitar Tube Pre Amp by Lenny24

Comments

50 comments

[Add Comment](#)

[view all 105 comments](#)



badbad214 says:
Thank you!

Mar 13, 2011. 7:33 PM [REPLY](#)



stainsor says:
DO NOT click the "Drop" button in the Add dialog. My first time using Eagle I thought that was the button to drop a part into a schematic. It turns out that's the button to remove a set of parts from the libraries! If you do click this button and accidentally remove some of the libraries you can get them back by reinstalling Eagle.

Feb 25, 2009. 5:14 AM [REPLY](#)

If I knew anything about UI design (which I don't) then I would probably say that this was a horrible choice for both the placement and naming of the button. I would probably think that it should be changed immediately.



webster32 says:
It's very misleading, but all you've actually done is "disable" that particular library. The Control Panel window that opens on startup allows you to re-enable that library. Just expand the "Libraries" and click the dot beside the library you disabled.

May 31, 2010. 11:52 AM [REPLY](#)



badbad214 says:
Thank you webster32

Mar 13, 2011. 6:28 PM [REPLY](#)



Cobalt59 says:

Thank you. I did the same thing as Stainsor. :)

Jan 28, 2011. 7:30 PM [REPLY](#)



tamentis says:

That's also what I initially thought, I tried it on a random weird component by curiosity.. It doesn't warn you, there is no tool-tip.

Jul 11, 2009. 2:29 PM [REPLY](#)



rachel says:

Ha! I was just about to add this same comment, as I had the exact same wrong thought about what "DROP" meant. It's a terrible design choice and doesn't even have a confirmation dialog! Danger Will Robinson!

Mar 3, 2009. 10:32 AM [REPLY](#)



joie2vivre says:

Looks like you can get the library back by clicking on library/use, then select the library you want

May 5, 2009. 4:27 PM [REPLY](#)



amando96 says:

haha i pressed that aswell, but never figured out what it did xD

Feb 14, 2010. 3:14 PM [REPLY](#)



Russ1234 says:

You have 2 diodes on Q2. Thanks for these tutorials.

Feb 26, 2011. 7:04 PM [REPLY](#)



westfw says:

There's an explanation for the two diodes with the original schematic here: <http://www.bowdenshobbycircuits.info/555.htm#555leds.gif>
Basically, the 555 is not "rail to rail" on its output, and otherwise would not be sure to turn off the "upper" transistor when in the high state.

Feb 28, 2011. 8:19 AM [REPLY](#)



mauselous says:

Thank you so much for the helpful tutorial. The steps you illustrated really helped me get a handle on getting from circuit to schematic to layout.

Do you have any tips on where to find additional or alternate libraries that are more organized?

Feb 1, 2011. 8:28 PM [REPLY](#)



westfw says:

The EAGLE user community forums are active and helpful, and frequently people will share libraries for newer parts.

I don't know about "more organized", though. I haven't seen any organized attempt to modernize/fix/organize a comprehensive set of component libraries for EAGLE, but it's likely that such a thing would be in the "costs money" realm where I wouldn't notice.)

(there is the recent Element14/Newark/Farnell effort to tie EAGLE directly to parts purchases; I haven't paid much attention to that since they're not one of my preferred dealers.)

Feb 2, 2011. 11:26 PM [REPLY](#)



Cobalt59 says:

Where to find a + to GND power supply in the ADD dialog?

Jan 28, 2011. 7:50 PM [REPLY](#)



mrmizuno1 says:

Thank you. very helpful.

Jan 14, 2011. 11:09 AM [REPLY](#)



crogers13 says:

"your mouse pointer BECOMES your drawing tool"

Jan 13, 2011. 5:41 PM [REPLY](#)



westfw says:

if you insist!

Jan 13, 2011. 6:58 PM [REPLY](#)



ihart says:

Thanks for doing this. Now I need to check out your PCB instructable.

Dec 2, 2010. 7:21 PM [REPLY](#)





klarkg says:


Does anyone know of a website that has a good picture library of parts with the eagle cad names? Im having problems finding basic stuff sometimes like headers I need and the names in the eagle cad library confuse me , this instructable helped me alot thanks!


Oct 6, 2010. 8:31 PM [REPLY](#)

 **Taher81** says: Sep 28, 2010. 8:02 AM [REPLY](#)
Do you have any schematic capture of an ADSL Modem?
Thanks for sharing with us.


 **abraxas2** says: Aug 17, 2010. 7:42 PM [REPLY](#)
OK but how do we move a whole cluster of components. ? I tried group followed by move but no joy. By some miracle and a lot of attempts, I managed to combine two schematics and need to move one relative to the other. Thanks for this great tutorial, by the way. It is appreciated.


 **adrenalynn** says: Aug 21, 2010. 2:45 PM [REPLY](#)
Did this get answered for you? In 5.x, you use the group (select box, rubber-band box, marching ants, what-have-you) to select the components, then select the move tool. Now - _look at the status line_ - it tells you that CNTRL + RIGHT-CLICK moves the group. Hope that helps!

 **abraxas2** says: Aug 17, 2010. 8:08 PM [REPLY](#)
OK I got it but damned if I remember how. I think I used the group command, highlighted the cluster of components and then right clicked on the selected cluster and saw a pop up saying move and right clicked again. I don't want to mess with it by doing it again. Maybe you can verify the steps ??


 **westfw** says: Aug 18, 2010. 12:06 AM [REPLY](#)
After you use the group command and right-click somewhere within the group, you should either pick up the whole group for moving, or get a pop-up menu with separate options for "move" and "move group." (when do you get one, and when do you get the other? I'm not sure...)


 **pepegrillo** says: Jul 6, 2010. 5:31 PM [REPLY](#)
thank u very much! i can now draw pcb. THANK U!


 **Marnick** says: Apr 12, 2010. 4:16 AM [REPLY](#)
Thanks for this tutorial, it helped me getting along this fantastic, but comprehensive program!


 **jimmy** says: Apr 1, 2010. 9:22 PM [REPLY](#)
Fantastic tutorial -- got me up to to speed very quickly. Thanks for putting this together.

 **lodewijkadlp** says: Feb 1, 2010. 4:50 AM [REPLY](#)
Any tips on a basic library?


 **irritant#9** says: Aug 27, 2008. 8:00 AM [REPLY](#)
Great writeup. I've taken a few stabs at EAGLE and this is the one that cleared up a lot of "things" I didn't understand. Trying to figure out if there is a way to undo Autoroute? Other than saving before and discarding unfavorable Autoroutes as you rearrange devices.

 **Jon Peterson** says: Jan 11, 2010. 6:33 AM [REPLY](#)
type "ripup;" in the command line (blinking text cursor) w/o the quotes and then press enter.

 **westfw** says: Aug 27, 2008. 11:19 AM [REPLY](#)
I have not found a way to "undo" autoroute other than the save/revert sequence.

 **samr371** says: Jan 3, 2010. 12:45 PM [REPLY](#)
The libraries didn't load into my Eagle cad. Does anyone know how to load all the libraries at once?
Thanks,
Sam R.

 **westfw** says: Jan 3, 2010. 3:18 PM [REPLY](#)
"use *" in the cli window

 **samr371** says: Jan 4, 2010. 3:46 PM [REPLY](#)
Yah, but I can only add one at a time. (I'm using a MAC by the way) Is there a way that I could add all of them at once?
Thanks,
Sam R.



westfw says:

the "***" part of the command should cause all the libraries in the defined path to be "used"

Jan 4, 2010. 4:05 PM [REPLY](#)



samr371 says:

I don't get what you mean by the "***" part of the command. What do I need to do? I'm a first time user to Eagle cad (this is the first time I opened it) so I have no idea about anything in eagle cad.
Thank you so much!
Sam R.

Jan 4, 2010. 6:29 PM [REPLY](#)

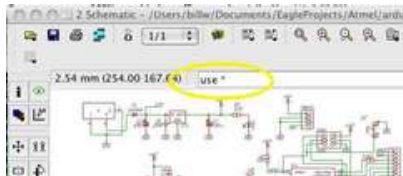


westfw says:

Type in the string

use *

including the asterisk, followed by enter, in the text command line area as shown here. Omitting the asterisk causes a menu of files to pop up. Including a filename or * bypasses the menu...



Jan 4, 2010. 11:50 PM [REPLY](#)



samr371 says:

ok it worked. Thanks for all the help!
Sam R.

Jan 5, 2010. 3:11 PM [REPLY](#)



AlamoRobotics says:

Thanks for your instructable. It helped me be able to read the component codes in the libraries!

Dec 24, 2009. 6:22 AM [REPLY](#)



jimby says:

When I tried to search I would get no results. It seems in version 5 at least, you have to load the libraries you want first, otherwise you will not find any components.

Dec 11, 2009. 2:37 PM [REPLY](#)

If you type 'use *' in the command box at the top it will load them all.



westfw says:

I've had it act like that; but usually it behaves as though "use *" is the default. I don't know if that's because it's "sticky" somewhere, or because of some previous configuration, or what.

Dec 11, 2009. 2:57 PM [REPLY](#)

Remembering that you might need to issue the "use *" is a good idea.



aramja says:

thanks - very informative. I've seen a number of eagle schematics from different internet sites and they all appear to have a standard way of displaying them with the "title, document number, date, revision number, and sheet..." in the right hand corner - is there an option in eagle to do this?

Aug 13, 2009. 11:57 PM [REPLY](#)



westfw says:

Huh. Somehow some of the pictures got disconnected from where they were supposed to be. This should now be fixed!

Apr 4, 2009. 1:26 AM [REPLY](#)



LittleTyke says:

Sorry, normally I do try to be constructive, but I've just installed and attempted to use the Freeware edition of Eagle 5.4 and it's come off my computer faster than a knife through butter. I don't think I have /ever/ come across more user-unfriendly software than this. Simply awful. Awful.

Mar 5, 2009. 6:52 AM [REPLY](#)



thermoelectric says:

Hi, I'm trying to make a schematic with the 74LS47N, I have found the part but the Ground and Vcc pins aren't present on the pinout when I "drop it into the schematic, How do I connect things to the pins that aren't present?

Jan 16, 2009. 7:25 PM [REPLY](#)



btxzer0 says:

use invoke command to show pin that are hidden

Jan 31, 2009. 8:52 PM [REPLY](#)



thermoelectric says:

Oh, Where's the invoke command button located Thanks!

Feb 1, 2009. 12:34 AM [REPLY](#)



btxzer0 says:

above Text button on toolbar or you can access it from Edit menu

Feb 1, 2009. 6:06 AM [REPLY](#)



thermoelectric says:

Cool, Thanks!

Feb 1, 2009. 12:56 PM [REPLY](#)



biggej says:

Great job explaining Eagle Cad. This is just what I needed to get started. Thanks

Jan 5, 2009. 12:32 PM [REPLY](#)

[view all 105 comments](#)