

Making an Op Amp

[Make the Eagle Model](#)

[Find a datasheet](#)

[Make the symbol](#)

[Make the package](#)

[Make the device](#)

[Review the Eagle](#)

[Make the Morph Model](#)

[Create component](#)

[Add Octopart info](#)

[Add Connectors](#)

[Review Component](#)

Make the Eagle Model

For this tutorial we will focus on the Texas Instrument operational amplifier OPA344

Find a datasheet

A google search for 'OPA344' pulls up a link to [TI's datasheet for this component](#)

Make the symbol

- PCMCIA CARDS
- DATA ACQUISITION
- PROCESS CONTROL
- AUDIO PROCESSING
- COMMUNICATIONS
- ACTIVE FILTERS
- TEST EQUIPMENT

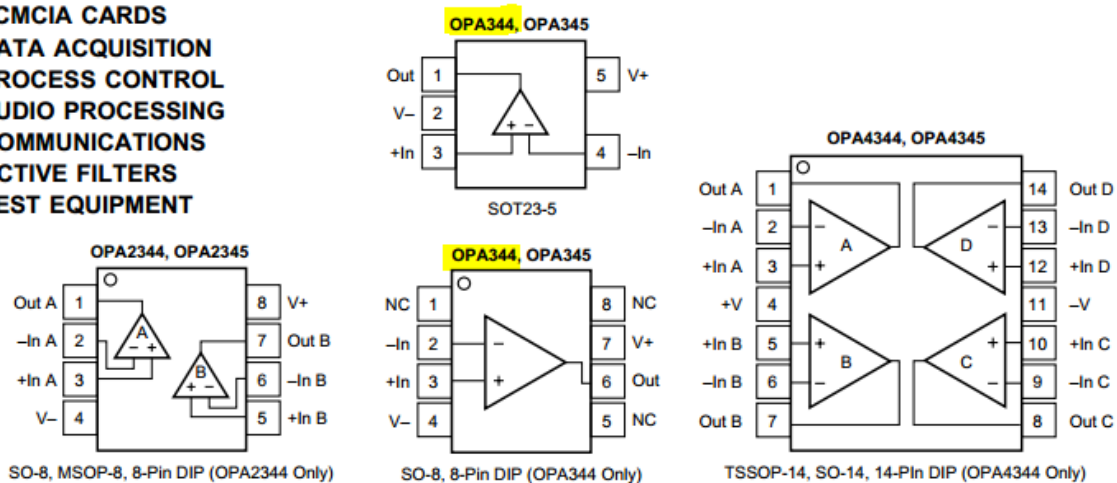


Figure 1: Datasheet excerpt showing block diagrams of the OPA344 component family.

Figure 1 indicates the OPA344 is a single, standard op amp. With the exception of IC chips, most electronic components have a standard symbol.

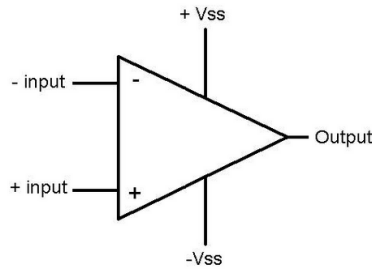



Figure 2: A standard symbol for a single operational amplifier. A component's symbol is what you see on a schematic.

1. Open up an Eagle library or create a new one: Eagle Control Panel: File > New > Library
2. Click the symbol icon  in the 'New: ' field type in "OPAMP" and click OK.

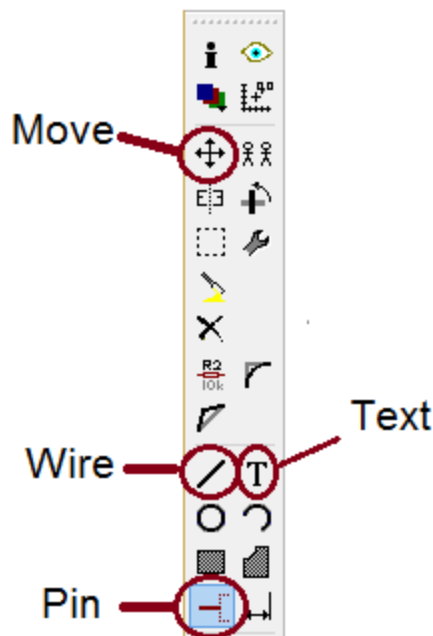


Figure 3: Eagle tool bar with move, draw wire, pin, and text tool labelled.

3. Using the Eagle tool bar as shown in Figure 3, create the op-amp symbol displayed in Figure 2.
 - a. First draw a triangle around the centroid with the wire tool. With the wire tool selected, cycle through different "bends" with right-click.
 - b. Create 5 pins using the pin tool. Use the move tool to right-click a pin and open its properties, give the pins a name matching the datasheet diagram in Figure 1. Try to make labels visible by staggering pins or hiding the default label (properties: visible > pad) and using the Text tool to create your own.
 - c. Finally, Use the text tool to create two separate texts just outside the symbol in the upper right hand corner: ">NAME" and "VALUE". Right click to open their properties and put them on layers 95 and 96, respectively.

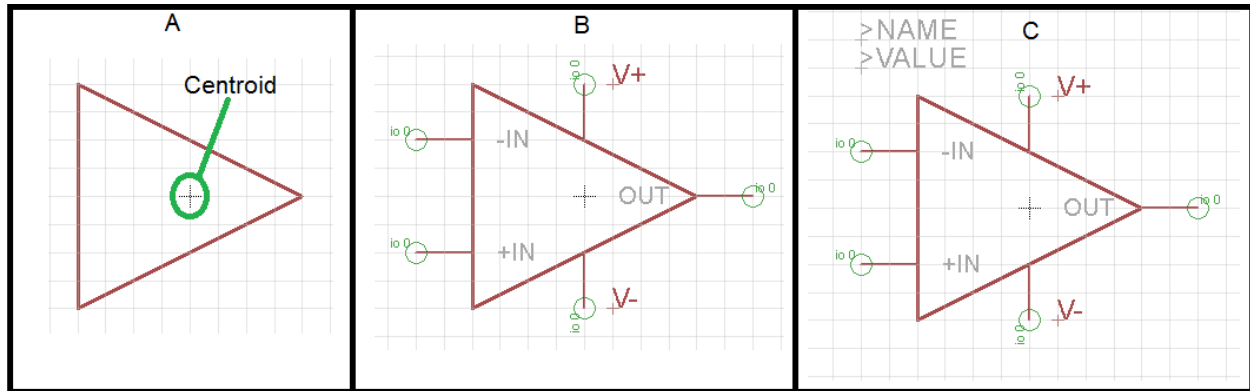


Figure 4: 3 step diagram showing the process of step 3 in making a symbol.




Make the package

The package is what appears on the circuit board itself, this step is crucial to the longevity and functionality of a printed circuit. Notice in Figure 1 there are two variants for the 344. Let's use size as a constraint here and pick the variant with the smallest form-factor. Note, we didn't include ("NC") in our symbol but that shouldn't matter either way as it stands for "No connection".

Device	Package Type	Package Drawing	Pins	SPQ	Reel Diameter (mm)	Reel Width W1 (mm)	A0 (mm)	B0 (mm)	K0 (mm)	P1 (mm)	W (mm)	Pin1 Quadrant
OPA2344EA/250	VSSOP	DGK	8	250	180.0	12.4	5.3	3.4	1.4	8.0	12.0	Q1
OPA2344EA/2K5	VSSOP	DGK	8	2500	330.0	12.4	5.3	3.4	1.4	8.0	12.0	Q1
OPA2345EA/250	VSSOP	DGK	8	250	180.0	12.4	5.3	3.4	1.4	8.0	12.0	Q1
OPA2345UA/2K5	SOIC	D	8	2500	330.0	12.4	6.4	5.2	2.1	8.0	12.0	Q1
OPA344NA/250	SOT-23	DBV	5	250	178.0	9.0	3.3	3.2	1.4	4.0	8.0	Q3
OPA344NA/3K	SOT-23	DBV	5	3000	178.0	9.0	3.3	3.2	1.4	4.0	8.0	Q3
OPA344UA/2K5	SOIC	D	8	2500	330.0	12.4	6.4	5.2	2.1	8.0	12.0	Q1

Figure 3: Datasheet excerpt showing the dimensions of each component variant in this family. Reel dimensions are not of interest here.

The SOT-23 has a smaller form factor than the SOIC variant. (Notice we are only concerned with devices starting with OPA344). Page 19 and 20 have the mechanical dimension information for the SOT-23 package. Fortunately, TI provides an "example board layout" on page 20 which we will use as our reference in creating the package.

1. Click the package icon  and in the 'New: ' field type in "SOT-23".
2. Click the grid tool  and change the size units to "mm" and the size itself to the pad pitch (0.95 mm in this case) [datasheet pg 20].
3. Notice there are some new tools in right-hand side eagle tool bar. Click the pad tool  and change the size to 0.6 x 1.05 [datasheet pg. 20].



- a. A smart way of placing pads is placing them in order of their number (place 1 first, then 2...etc). Datasheet pg. 19 shows how the numbers are oriented (1, 2, 3 from left to right on the bottom and 4, 5 from right to left on top). Place the pads one grid mark apart (and one grid mark up or down from the centroid) in this order.
4. Keep a calculator handy and using the the origin as the center of the board layout[datasheet pg. 20], determine the center of the pads are 1.35mm(2.7/2) up or down from the centroid. Simply go into each pad's properties and alter the rightmost field (y) of "position" to be +/- 1.35. (+ for 4 and 5, - for 1, 2, and 3). Hit OK.

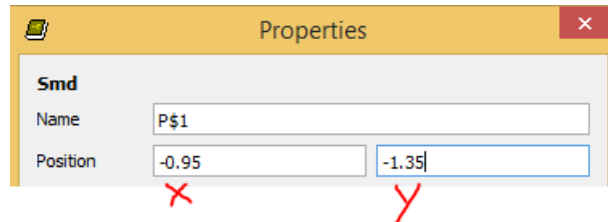



Figure 4: Properties dialog for pad 1, with x and y positions indicated.

- a. Alternatively, the pads all need to be moved just 0.4 mm (1.35 - 0.95) up or down from where they were after step 3. Change the grid size to 0.4mm. Use the group tool  and select either the top or bottom row of pads, then select the move tool and while holding down the ctrl key on your keyboard, drag with right-click the pads up or down one grid mark. (verify you did this correctly by checking the pad's properties, this technique can save lots of time for larger components).

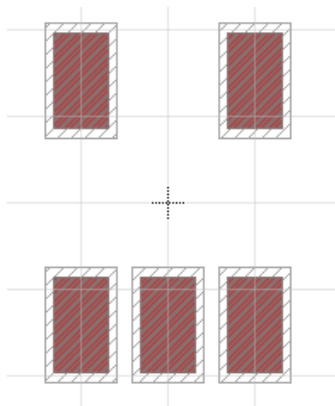




Figure 5: Picture of SOT-23 package after step 4 of process with 0.95mm grid setting

5. Change the grid size to 0.2 mm and use the wire tool (on layer 21 - tPlace, setting located next to the grid tool. to draw lines between the gaps of the pads but atleast one grid mark away from any red "copper".
6. Then use the circle tool  to draw a pin 1 indicator (also on layer 21) off to the side of pin 1 (bottom left). Make the circle big enough to alter it's properties: change the

width to 0 and the radius to 0.1542. Move the circle so that it lies between 1 and 3 grid marks away from the red “copper”.

7. Finally, Use the group tool to drag and select everything you just created, then click the rotate tool  and while holding down the ctrl-key, right-click in the center of the components until pin 1 is oriented in the upper left (3 - clicks).
8. Use the text tool to insert “>NAME” on layer 25 (tNames) just about the pin 1 marking.

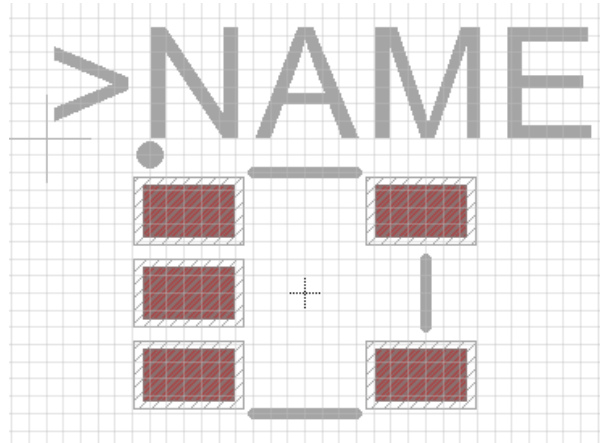




Figure 6: Finished package of OPA344 SOT-23 variant.

Make the device

The device is where the package and symbol are linked together. Specifically, you link a package pad with a symbol's pin.

1. Click the device icon  and in the 'New: ' field type in “OPA344”.
2. Click the “Add a symbol” icon on the left-hand side  and select the “OPAMP” symbol made earlier in this tutorial. With the opamp selected, click right in the middle on the canvas on top of the centroid.
3. In the bottom, right corner of the screen click “New”. Highlight the “SOT-23” package created in this tutorial and select OK.
 - a. Optional: you can type in the variant name “NA” (as seen in Figure 2).
4. Next, click “Connect” in the bottom-right corner and it opens up a window showing the pins of the symbol and pads of the package (most likely named, P\$_). If you followed step 3a of “Make the Package” then select a combination of matching pin and pad (refer to the SOT-23 block diagram on page 1 of the datasheet or Figure 1 of this tutorial).
 - a. Click “connect” when you have highlighted a matching combination

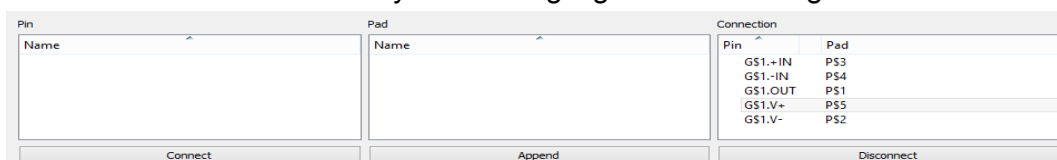


Figure 7: Connection Dialog with correct pin and pad match-ups.

- Once you are done, click OK and save the library in a familiar directory.

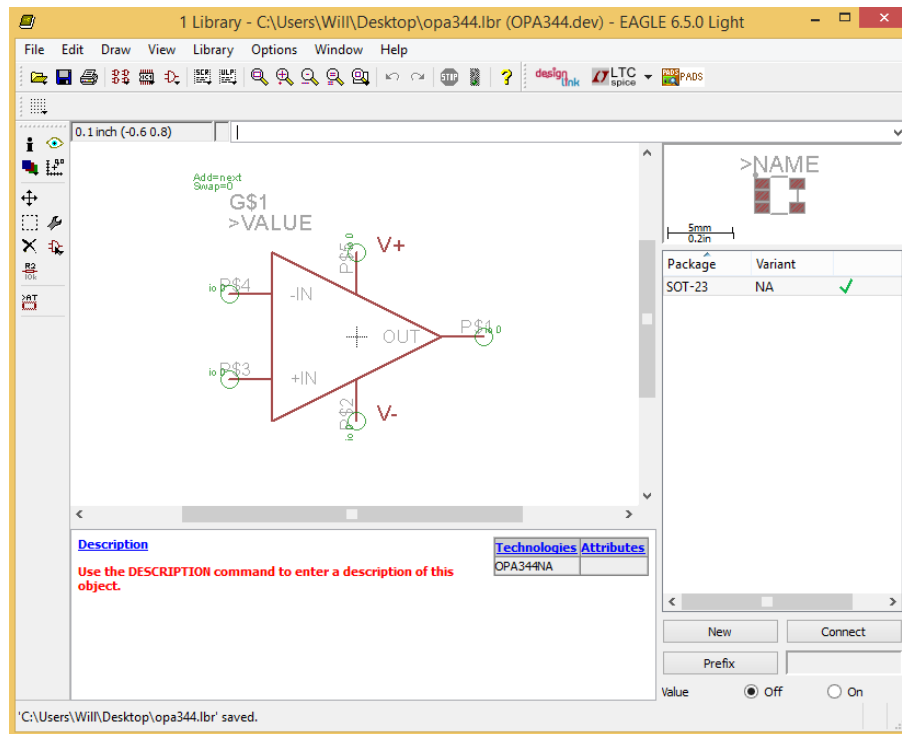


Figure 8: OPA344 device at stage of completion.


Review the Eagle

A couple things to make sure of:

- Symbol:
 - “>NAME” and “>VALUE” text are on layers 95 and 96, respectively
- Package:
 - Copper pads are placed at proper locations with correct numbers for names
 - Layer 21 objects do not interfere with copper pads
 - “>NAME” is on layer 25
- Device:
 - Connections are made matching the datasheet’s specified pinout.
 - The symbol and package chosen is a small form factor and is a true existing combination on the datasheet (“Does this symbol go with this package?”)

Make the Morph Model

Create component

- Create a folder for components by right clicking the “RootFolder” and selecting “Insert Folder > Components”
- Right click the components folder created in Step 1 and select “Insert Model > Component” and type in “OPA344NA” and hit the Enter key.
- Click the CAT tool  on the top of the screen and a window will pop up.

- a. Locate the library created earlier in this tutorial and select the OPA344 device.
- b. Close out the CAT window.

Add Octopart info

4. Next, select the EDAModel and in the Object inspector (lower-left corner), change the “DeviceSet” field to “OPA344NA/250”. (Note that OPA344NA/3K is an equally valid octopart number for this component. It is simply a difference in shipment packaging.)
 - a. Rename the component to “OPA344NA_250” (‘/’ character is forbidden)

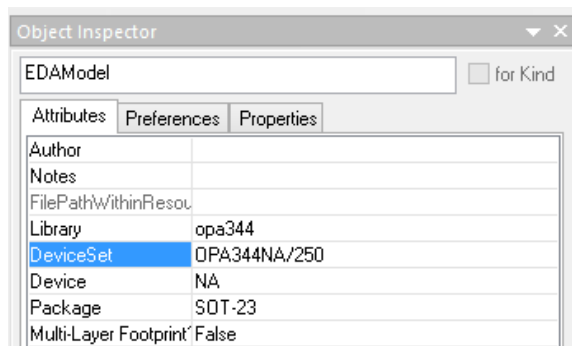



Figure 9: Object inspector of EDAModel with DeviceSet set to Octopart number

5. Select the CAT tool again and choose “Add OctoPart Information” and when prompted choose “NO” to run on all components.
 - a. This step uses the DeviceSet field of the EDAModel to find matching information on the Octopart database.

Add Connectors

6. Select all of the red pins branching out of the EDAModel (be sure not to select the EDAModel itself) and click the “AddConnector” tool  at the top.
 - a. Rearrange the connectors (using left click-and-drag) so they roughly follow the standard opamp symbol in Figure 2.

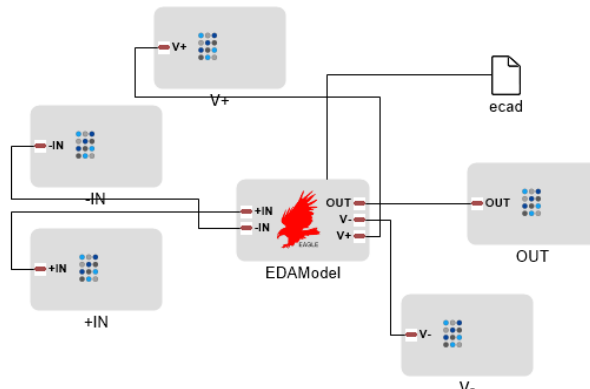


Figure 10: Screenshot of EDAModel with connectors roughly resembling the standard opamp.

- While holding the ctrl-key, select all of the blue connectors (and only the blue connectors) and in the Object Inspector (Lower-left corner) enter “AnalogSignal” into the Definition Field.

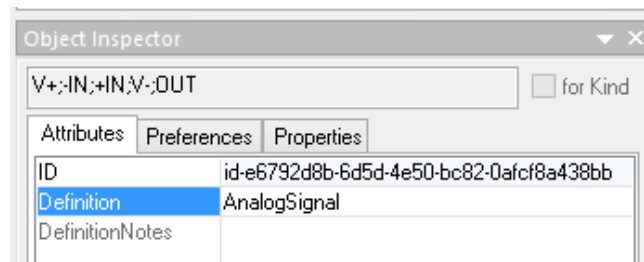


Figure 11: Object inspector of all five connectors with “AnalogSignal” set as Definition.

Review Component

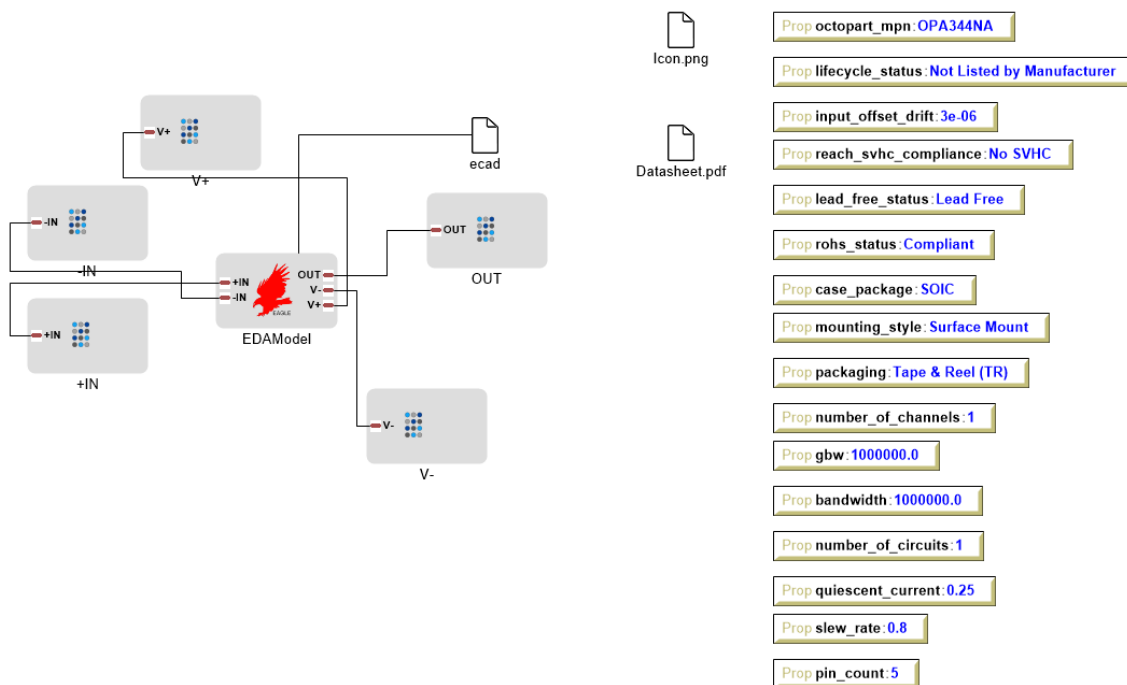



Figure 12: OPA344 Model at the stage of completion.

A couple of things to make sure of:

- All of the connectors are defined as “AnalogSignal”
- The deviceset name of the EDAModel is entered as a valid octopart number (using octopart.com to get track down the octopart number if necessary).

Export Component

- Use the component exporter tool  to export the selected component and use it in another GME project.