AUTODESK.

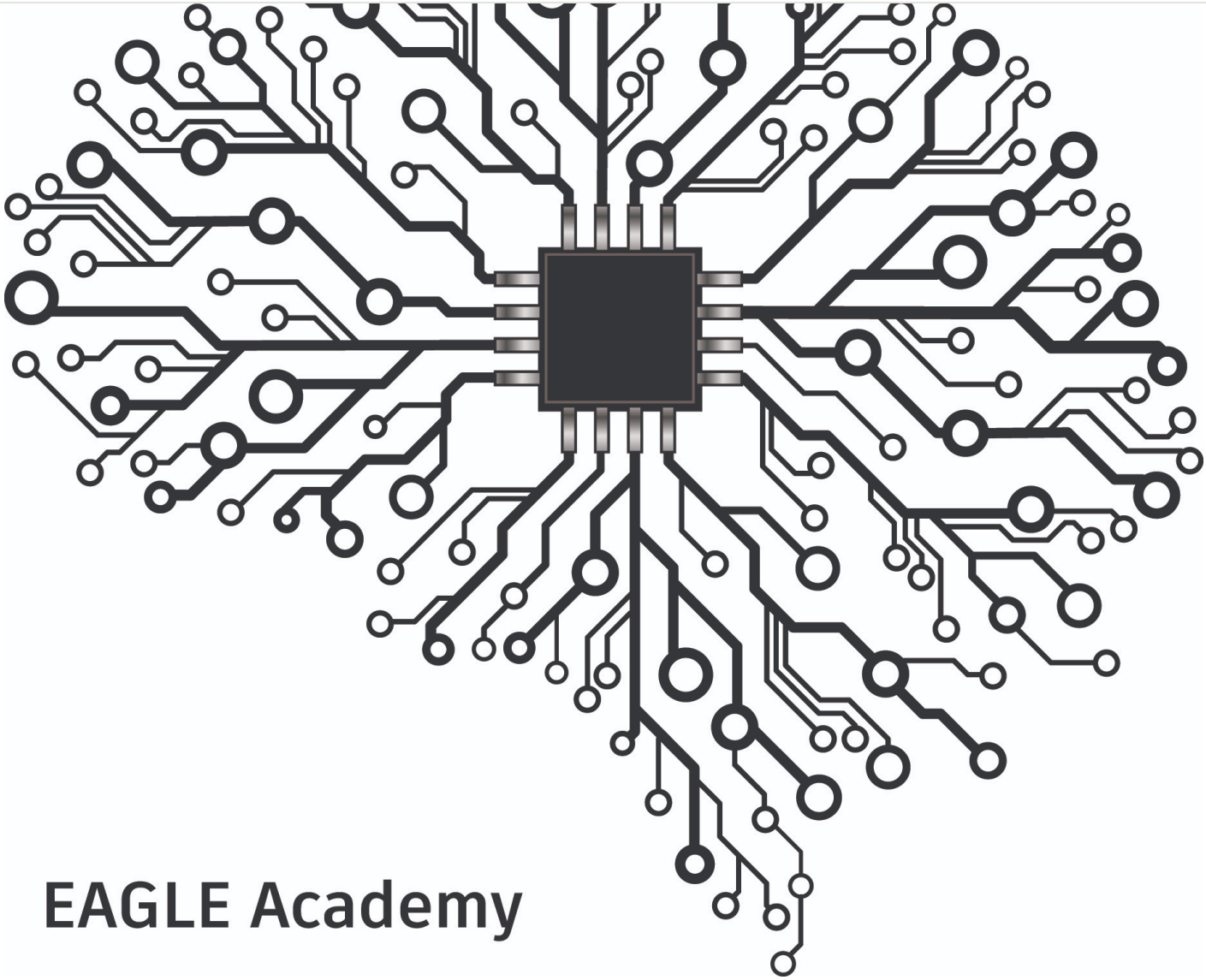
EAGLE

Library Basics Part 2: Creating Your First Symbol in Autodesk EAGLE

Sam Sattel

EAGLE Academy How To

AUTODESK.



Library Basics Part 2: Creating Your First Symbol in Autodesk EAGLE

Welcome back to the Library Basics Series, Pirates! In Part 1 you got your hands dirty creating your very first package in Autodesk EAGLE. Now, it's time to continue on your journey by learning how to build your first symbol. Regardless of what kind of design you're creating, it all begins with symbols and schematics. You can think of these symbols as the map to the chest of schematic treasure! Symbols show off all the functionality of your design without having to get into the nitty gritty of physical dimensions and packages.

Let's begin!

AUTODESK.

Before we set sail, you'll need to gather your datasheet! In our example, we'll be making a symbol for a MOSFET switch from Texas Instruments. Feel free to follow along with the TPS92411 datasheet.

Within most quality datasheets you'll find both a table and drawing that gives a summary of all the pins and their intended functions for a device. This summary is important when it comes to making your symbol, as it tells you exactly what pins you need and what their labels are without having to guess. Lucky for us Texas Instruments makes some impressive datasheets, and if you look on page 3 you'll see the **Pin Configuration and Functions** heading. We'll be using the SOT23-5 package type, which requires 5 pins, including:

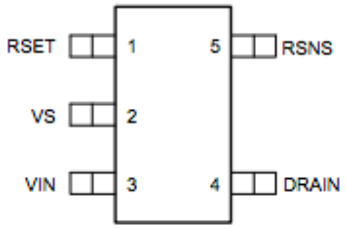
1. **RSET** – This pin serves as the reset of the device, allowing the switch to open.
2. **VS** – This pin serves as the internal switch and the device's floating ground.
3. **VIN** – This pin serves as the positive supply for our device.
4. **DRAIN** – As its name suggests, this pin serves as the drain for our device.
5. **RSNS** – This pin senses the VS voltage relative to our system's ground.

This is all the information you need to get started with your symbol.

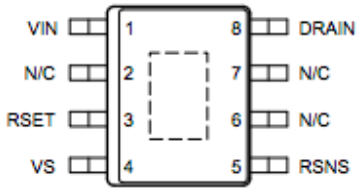
AUTODESK.

5 Pin Configuration and Functions

DBV (SOT23-5) PACKAGE
5 PIN
(TOP VIEW)



DDA (SO-8 Power-Pad) PACKAGE
8 PIN
(TOP VIEW)



Pin Functions

PIN			I/O	DESCRIPTION
NAME	NO.			
	DDA	DBV		
DRAIN	8	4	O	Drain of the internal switch.
N/C	2	—	—	Not internally connected.
N/C	6			
N/C	7			
VIN	1	3	I	Positive power supply for the device.
VS	4	2	I/O	Source of the internal switch. This pin is also the device floating ground.
RSET	3	1	I/O	A resistor connected between the RSET pin and the VIN pin sets the rising threshold to open the switch.
RSNS	5	5	I/O	A resistor connected between the RSNS pin to system ground senses the VS voltage relative to system ground.
Exposed Themal Pad				Connect to VS pin directly beneath the device.

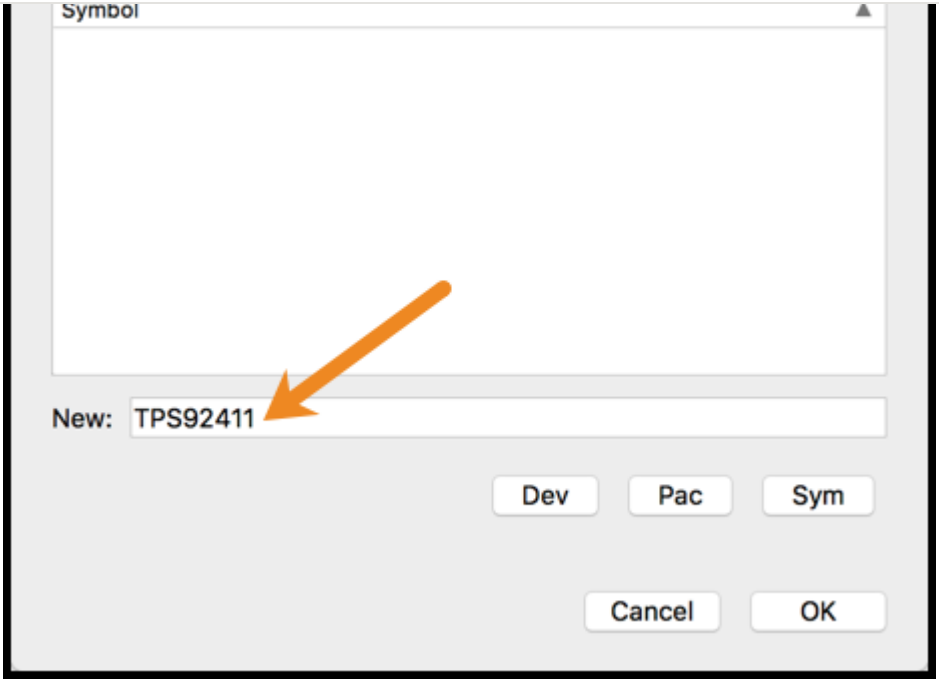
If only every datasheet was this easy to read! All the pins for our TSP92411 symbol.

Step 1 – Creating Your First Symbol

If you haven't already done so, go ahead and open your library from the EAGLE **Control Panel**. You can do this by right-clicking on the name of your library and selecting **Open.** This will open the **Library Window**, which provides a summary of all the devices, packages and symbols in your personal library. To add a new symbol to your library:

1. Select the **Symbol** icon at the top of your interface to open the **Edit Dialog**.
2. Next, enter a name for your symbol in the **New:** field and select **OK.** In our example, we'll give our symbol a name of "TPS94211."
3. A warning dialog will open, select **OK** to confirm that you want to create the new symbol name.

AUTODESK.



Adding the name for our first Autodesk EAGLE symbol – TPS92411 from TI.

Now that you have a new symbol added to your library you'll be greeted with the **Symbol Editor** window.

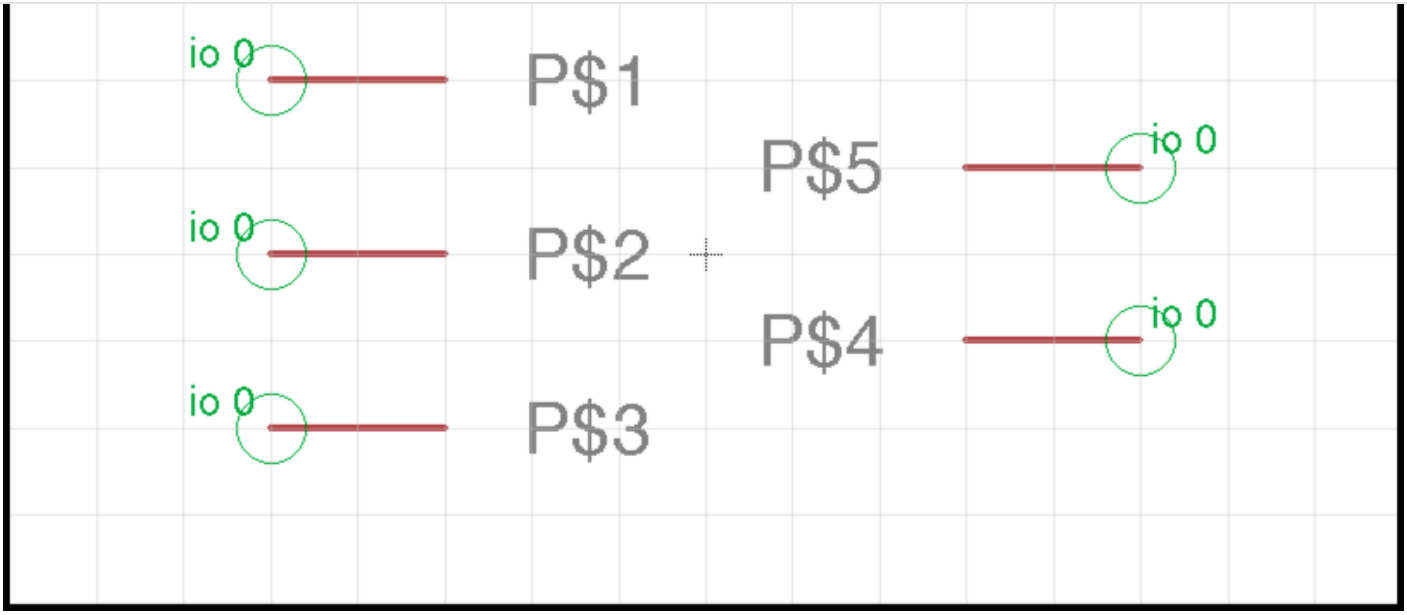
Step 2 – Adding Pins to Your Symbol

Let's insert some pins to your symbol. Based on the datasheet, we need 5 pins in total, 3 on the left and 2 on the right. Here's how to add them:

1. Select the **Pin** icon on the left-hand side of your interface.
2. Left-click in your **Symbol Editor** workspace to place your first pin.
3. Go ahead and repeat this process for the remaining pins on your symbol. To place the 2 pins on the right, **right-click** twice to rotate your pin 180° and then **left-click** to place.

We've left a gap between ours for easier readability as shown in the image below. But those pin names aren't right; we will change those now.

AUTODESK.



Step 3 – Changing Your Pin Names

You probably noticed that Autodesk EAGLE names all of your pin names by default as you place them. But we need to change these to match what it shows on our datasheet. To do this:

1. Select the **Name** icon on the left-hand side of your interface.
2. Select the first pin you placed, and the **Name** dialog will open.
3. Enter "*RSET*" in the **New Name:** field and select **OK**.

One down, four to go. Change all of your pin names to:

- Pin 1 – **RSET**
- Pin 2 – **VS**
- Pin 3 – **VIN**
- Pin 4 – **DRAIN**
- Pin 5 – **RSNS**

If you named everything correctly, then your pins should look similar to ours below.

AUTODESK.

All of our pin names are now in place to match our datasheet.

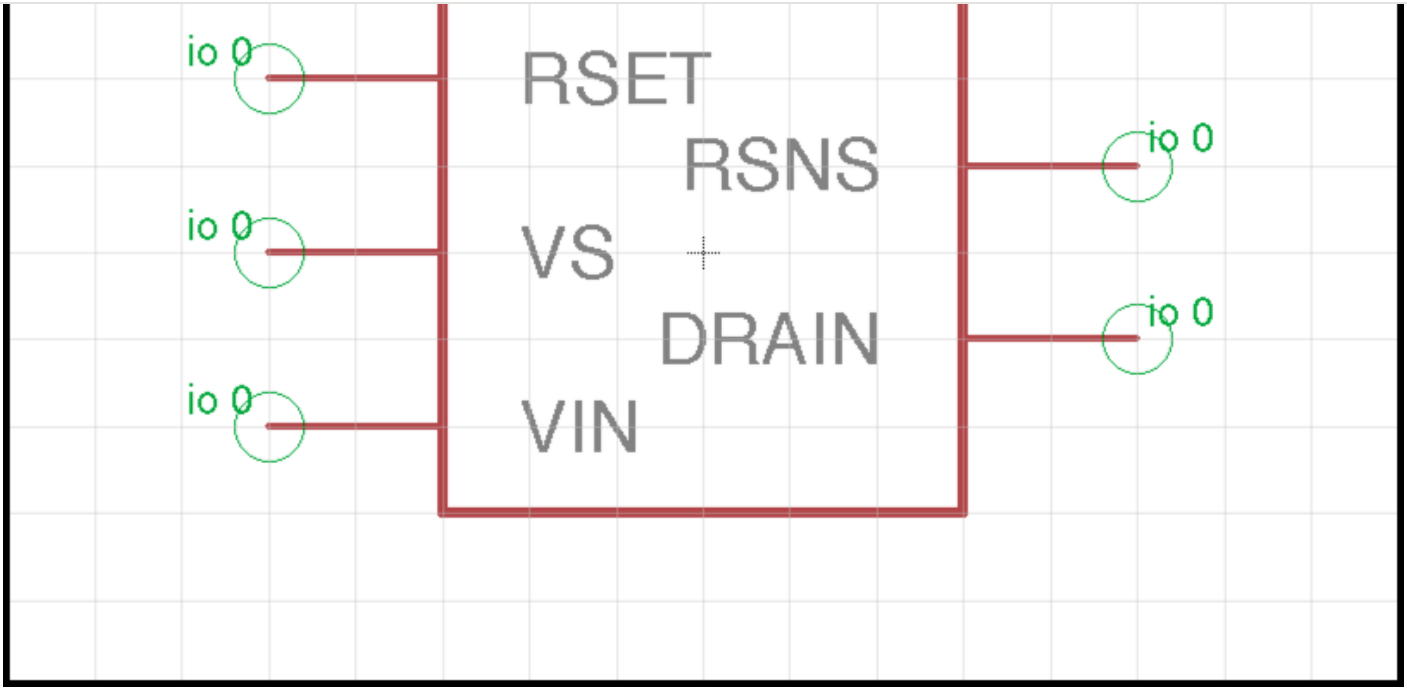
Step 4 – Adding Your Symbol Outline

To complete the visual of your symbol, you'll now need to add an outline to group all of your pins together. Here's how to accomplish this:

1. Select the **Line** icon on the left-hand side of your interface.
2. Next, left-click on a starting point to begin drawing your outline.
3. Continue to drag and left-click as needed to complete each side of your outline to draw a full box.

If you make any mistakes along the way, feel free to use the **Delete** icon on the left-hand side of your interface which can delete individual wire segments. Does your outline look like ours?

AUTODESK.



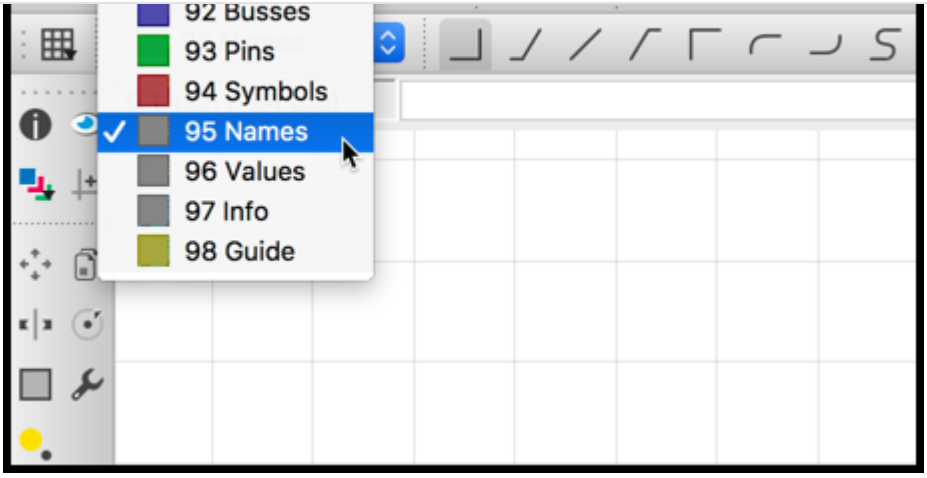
We've now got a visually complete symbol with correct pin names and an outline.

Step 5 – Adding Your Name and Value Placeholders

The final step in this process is a necessary one, adding placeholders for both the name and value of your symbol. Having a name and value will come in handy when cross-referencing information between your schematic and PCB layout, and will also help if a friend has to review your schematic. Here's how to add these:

1. Select the **Text** T icon on the left-hand side of your interface to open the **Text Dialog.**
2. Next, type in "*>Name*" in the **Enter text:** field and select **OK**.
3. Before placing your new text, you need to switch to it's placement layer. To do this, select the **Layers** drop-down at the top of your interface, and choose **Layer 95 Names**.

AUTODESK.

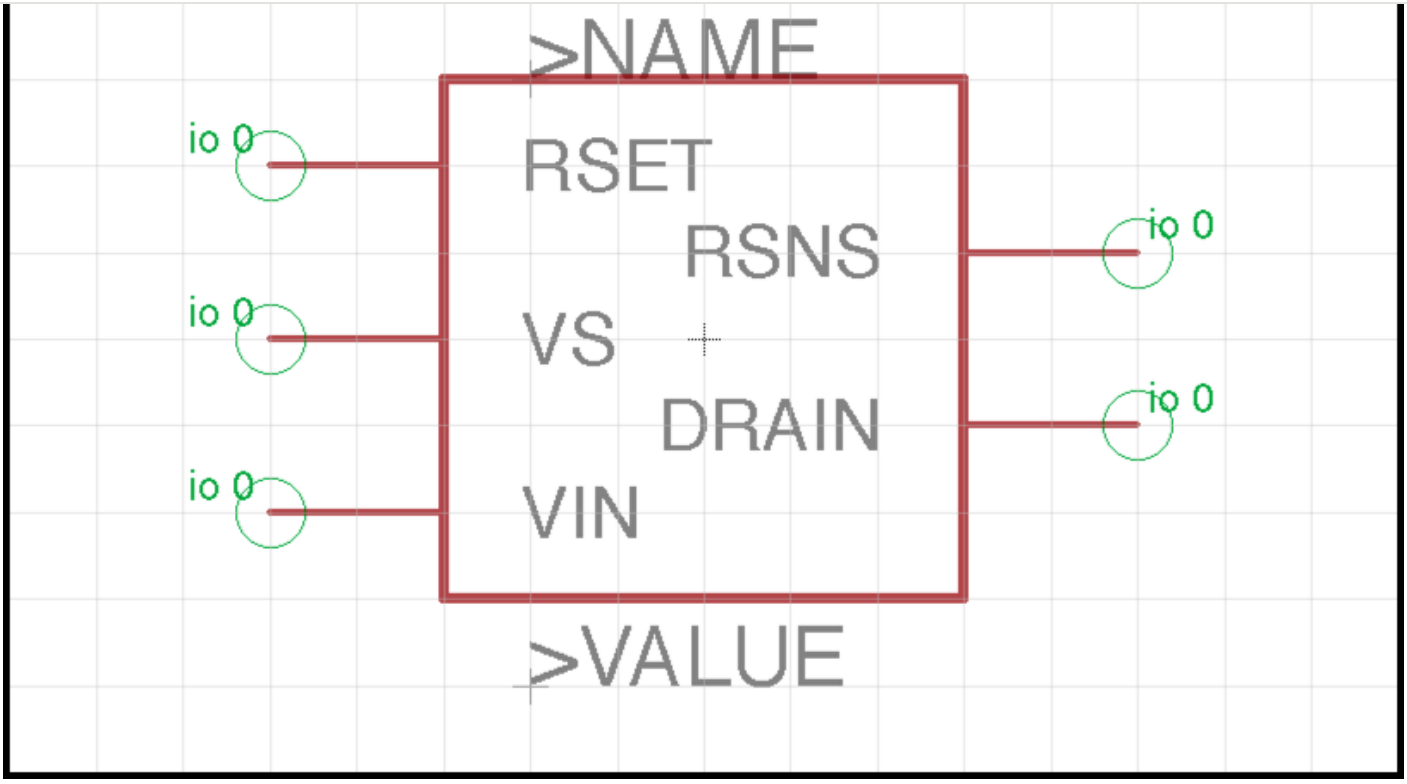


Remember to change your layer when placing your Name text.

4. Now drag your text to the upper part of your symbol, and left-click to place it. You'll know if it's on the correct layer if the text shows up in gray.

Go ahead and repeat the same set of steps above to place text for your symbol's value, this time using **Layer 96 Values**. When finished your symbol should look like ours below.

AUTODESK.



At last, our symbol is complete with placeholders for a Name and Value!

Optional Add-Ons

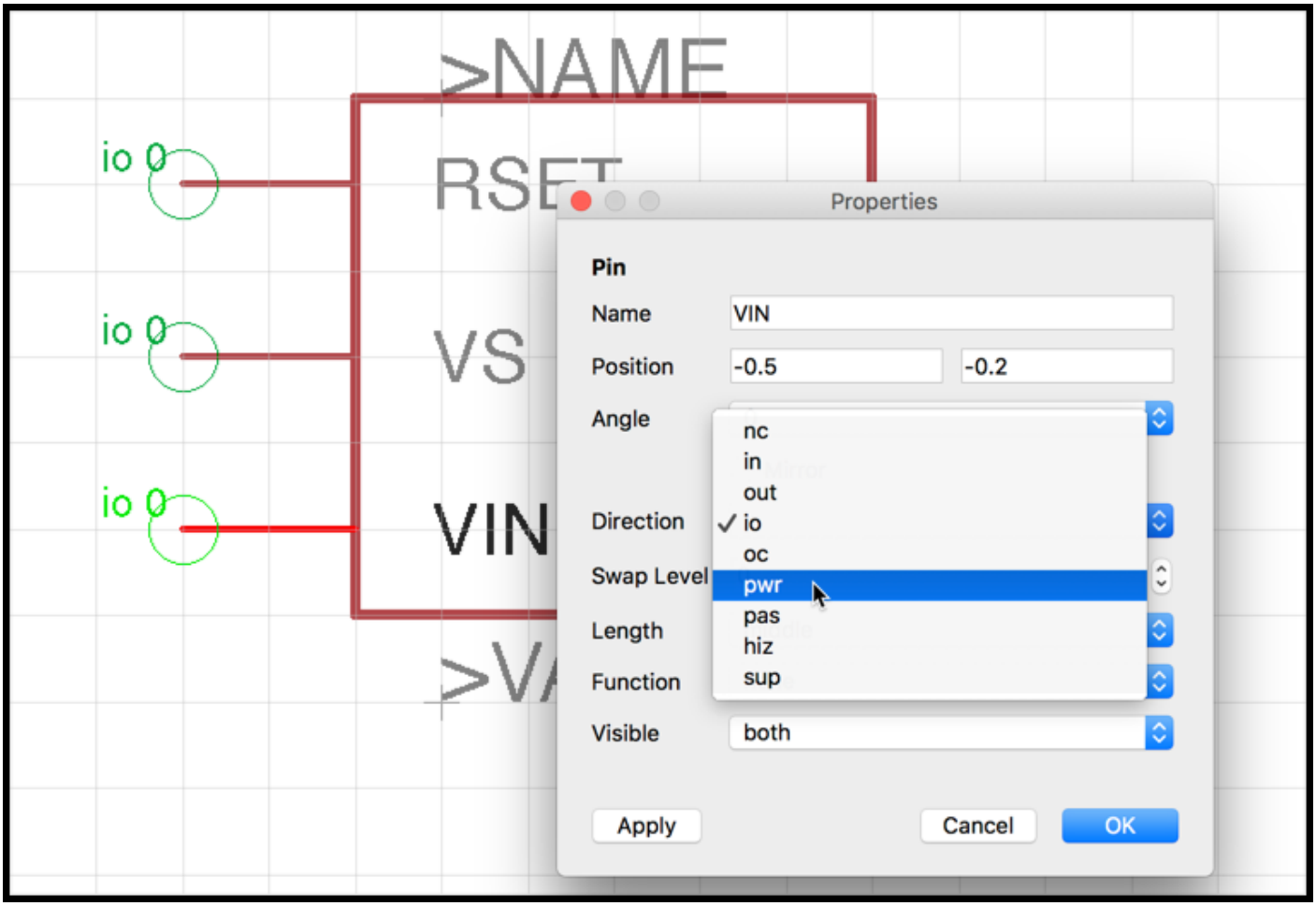
At this point your symbol is considered fully complete, and you can go ahead and save your changes. However, there are two other helpful additions that we'd like to cover which are entirely optional – setting pin directions and changing pin lengths.

Changing Pin Directions

Setting your pin direction will help to give Autodesk EAGLE some context about the pin usage. This is especially handy if you have a pin that is a no connect on your schematic. To set a pin direction:

1. **Right-click** one of the pins on your schematic and select **Properties.**
2. Select the **Direction** drop-down and choose one of the available direction options.

AUTODESK.



We can change the direction of our VIN pin to match its purpose, power (pwr).

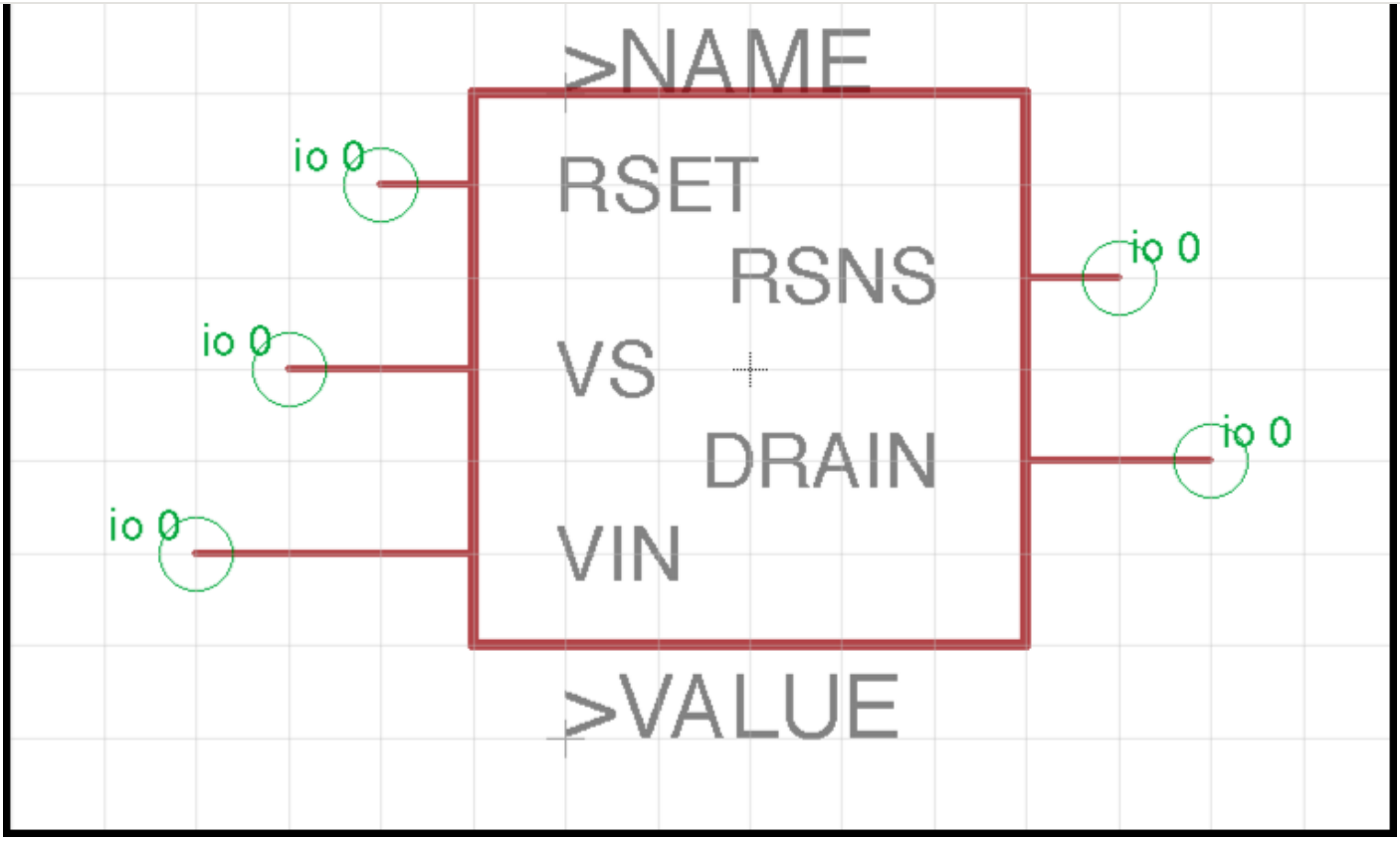
Changing Pin Lengths

Changing the length of your pins is another useful setting that comes down to personal preference. To change lengths:

1. **Right-click** one of the pins on your schematic and select **Properties**.
2. Select the **Length** drop-down and choose one of the available lengths (point, short, middle, or long).

Select **OK** to finalize your changes. You'll notice the visual change immediately on your schematic with either a shorter or longer pins.

AUTODESK.



You can make your pins short, medium or long, your choice!

Onwards We Go

Way to go, you created your very first schematic symbol in Autodesk EAGLE! Compared to packages, symbols are easier to make, and with a quality datasheet in hand, you shouldn't have any trouble. If you go back to your **Control Panel** after saving your changes, then you should have your very first symbol listed in what will soon be a growing personal library. The last part of this whole part creation journey is to put both your package(s) and symbol together in a Device.

Stay Tuned for Library Basics Part 3!

Ready to Subscribe? Head over to the eStore to start your Subscription for the easy-to-use PCB Design Software package, Autodesk EAGLE.

Share

POST A COMMENT

Log In to leave a comment

RELATED ARTICLES

Top 10 Design for Manufacturing Mistakes

Ever wonder what kind of DFM mistakes you can make that will drive your PCB manufacturer absolutely crazy? Learn about the top 10 DFM mistakes now, so you don't make them in the future!

AUTODESK.

How Your PCB Is Manufactured

Ever been curious about how the PCB you just finished designing is actually made? Read on to learn how it all happens in plain English, no technical jargon needed!

How Schottky Diodes Work

Need to control current flow in low voltage applications? Sounds like you need a Schottky Diode! Here's what you need to know before using one in your next project.