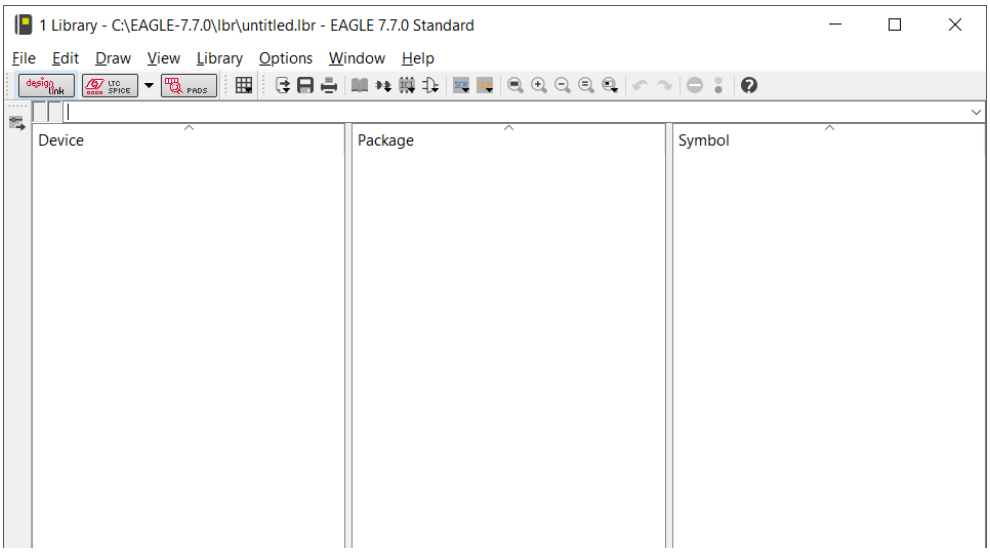
Eagle Tutorial: Making Libraries

Libraries in EAGLE contain all the necessary information for various electronic components (parts). Parts are split into three categories within a library: a symbol, package, and device. The symbol corresponds to how the part is viewed in a schematic layout. Passive components are expressed with common shapes you've seen in EE211 (resistors, capacitors, inductors, etc.) and Integrated Circuit (IC) chips are shown as rectangles with pins. The part package corresponds to the physical footprint of the component. The package dimensions can be found in the component's datasheet. However, most packages are standardized throughout different manufacturers so you can repurpose already-made packages as needed. Finally, the device category links connections from the symbol to package connections.

For this tutorial, we will create a 0603 package resistor (RC0603JR-0710KL) and an SOT23-5 package voltage regulator (MIC5219-3.3YM5-TR).

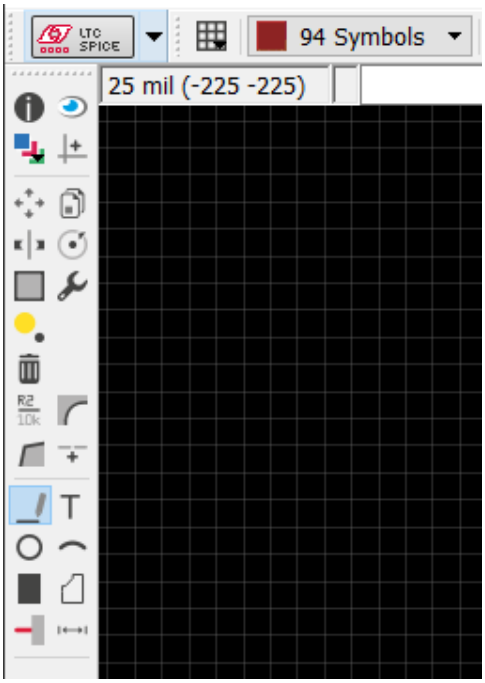
Resistor

1) Navigate through the EAGLE control panel and create a new library. Save the library in the C:/EAGLE-7.7.0/lbr file path and name it 296.lbr

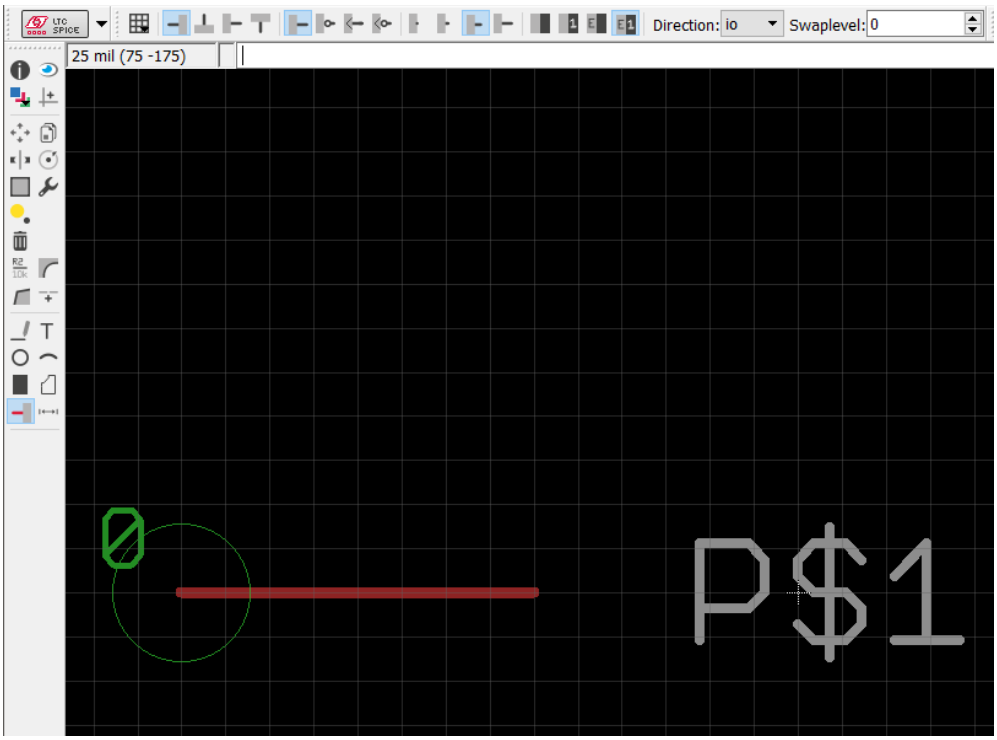


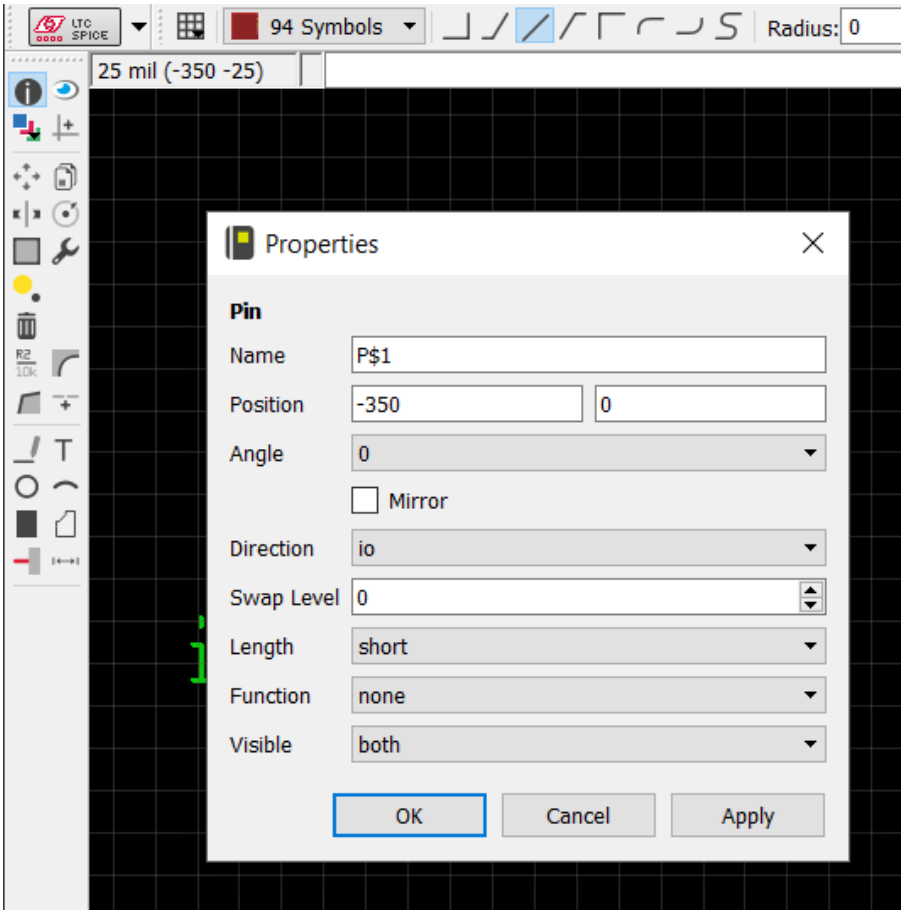
2) Create a new symbol. Name the symbol appropriately. For components with multiple package sizes (usually passive components), you can give a more generic name. In this case, we will name the symbol "RES".

3) Type “grid mil” followed by “grid 25” in the command bar. Make sure that the units are now 50 mils. If not, repeat the command.

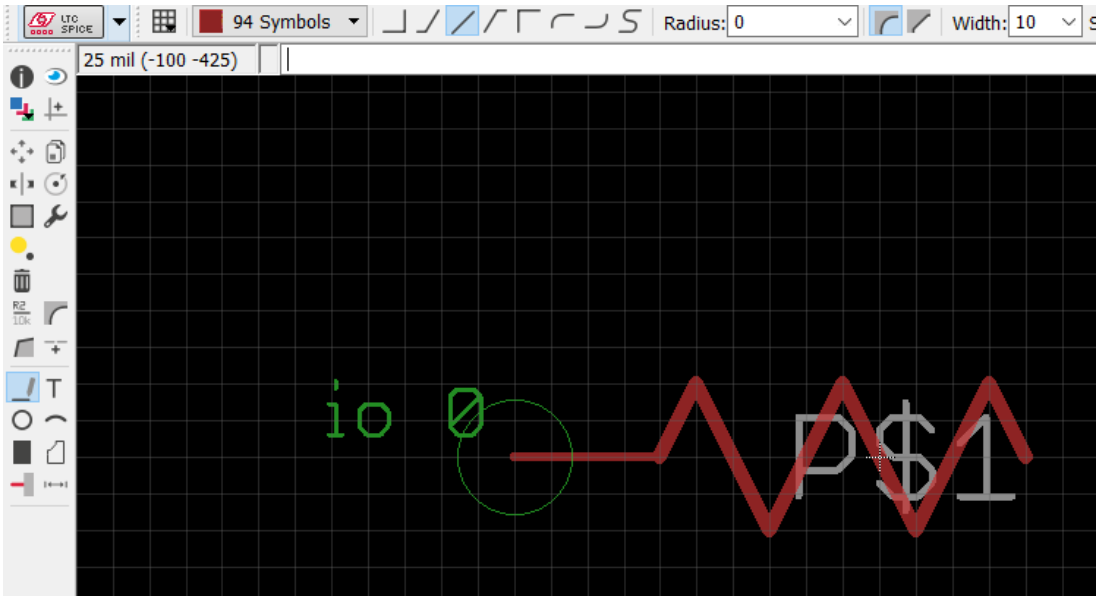


4) Select the pin tool and place a pin on the grid close to the origin (denoted by a white crosshair). After placing it, use the information tool to change the pin length to short.

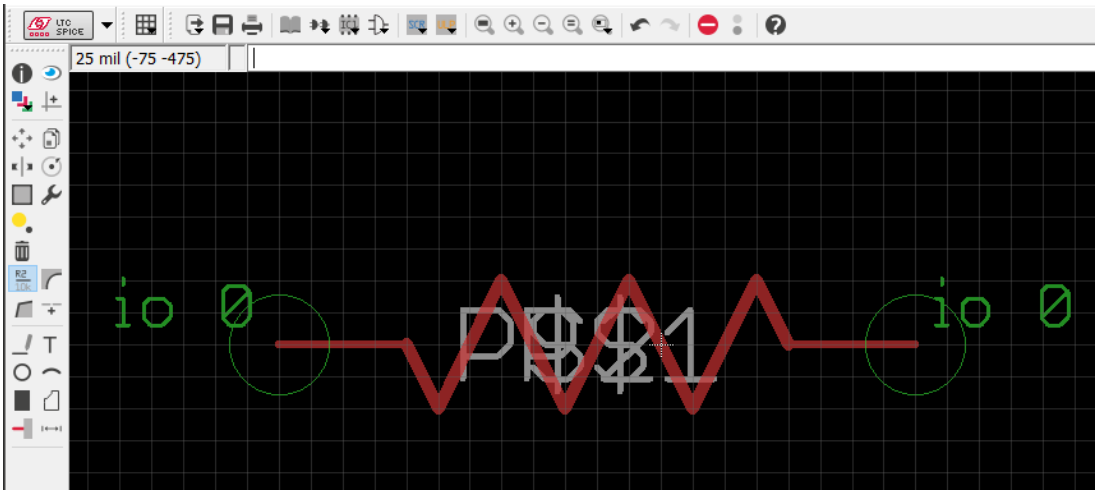




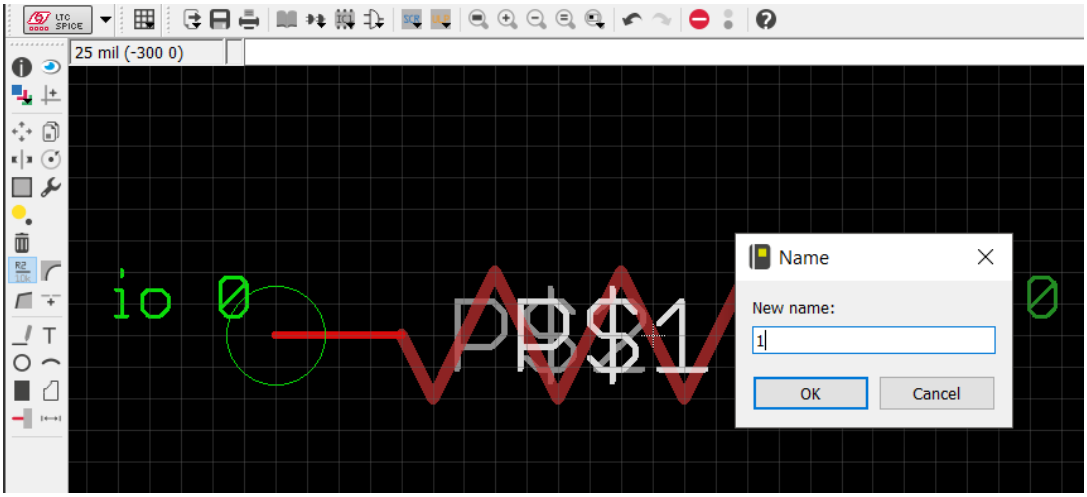
5) Use the wire tool to draw the component. Make sure that the current layer is "Symbols". Any wires drawn should be red. As you are trying to draw the resistor symbol, you can toggle through the different wire routing styles by right clicking on your mouse or touchpad. If you mess up, you can use the trash can tool to delete any unwanted wires or use Ctrl + Z to undo.

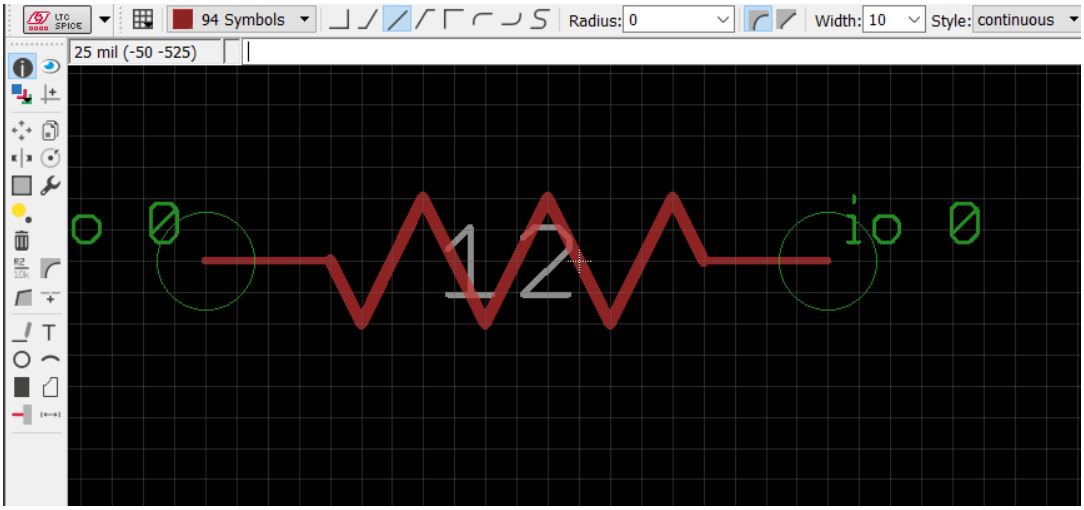


6) Place another pin at the other end and change its length to short.

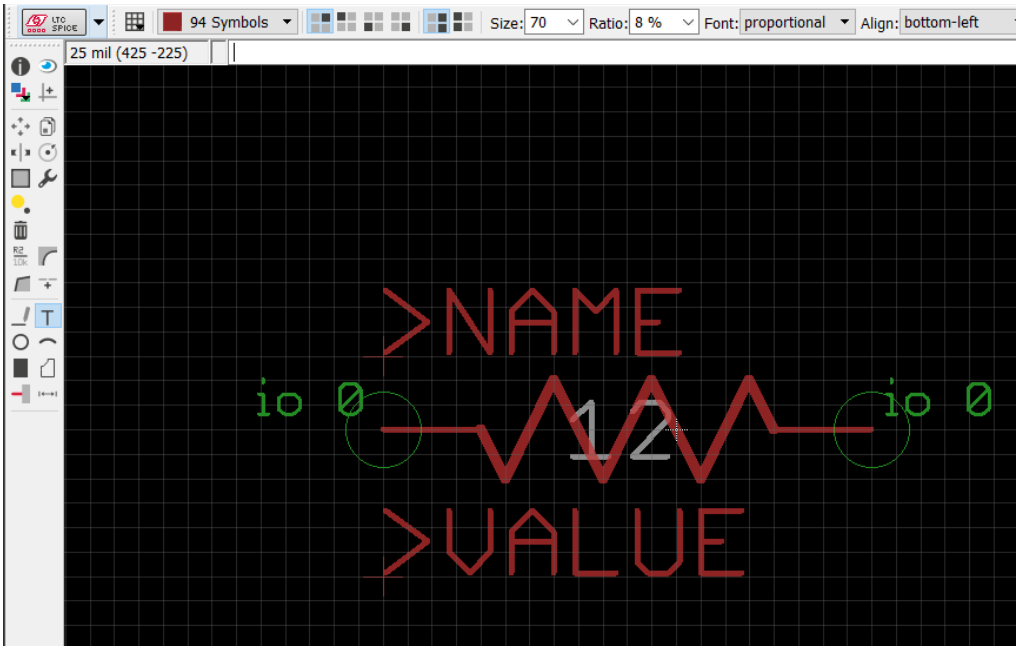


7) Select the name tool and name the pins appropriately. For resistors, we will name one pin “1” and the other one “2”.

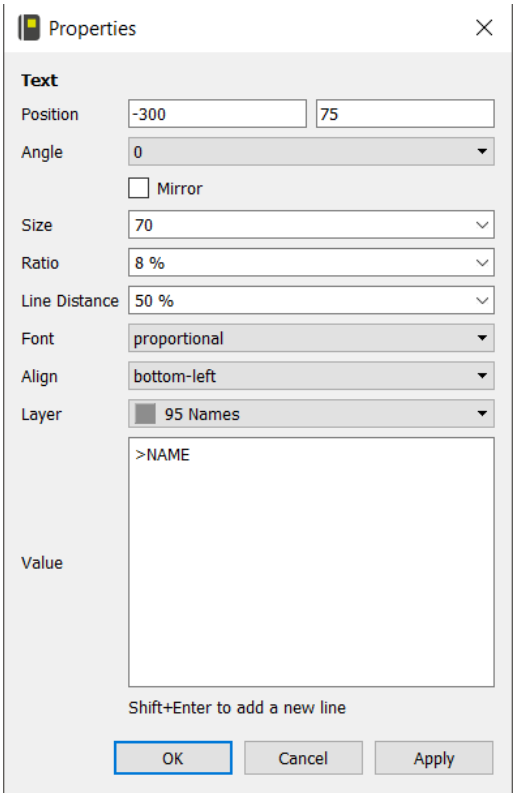


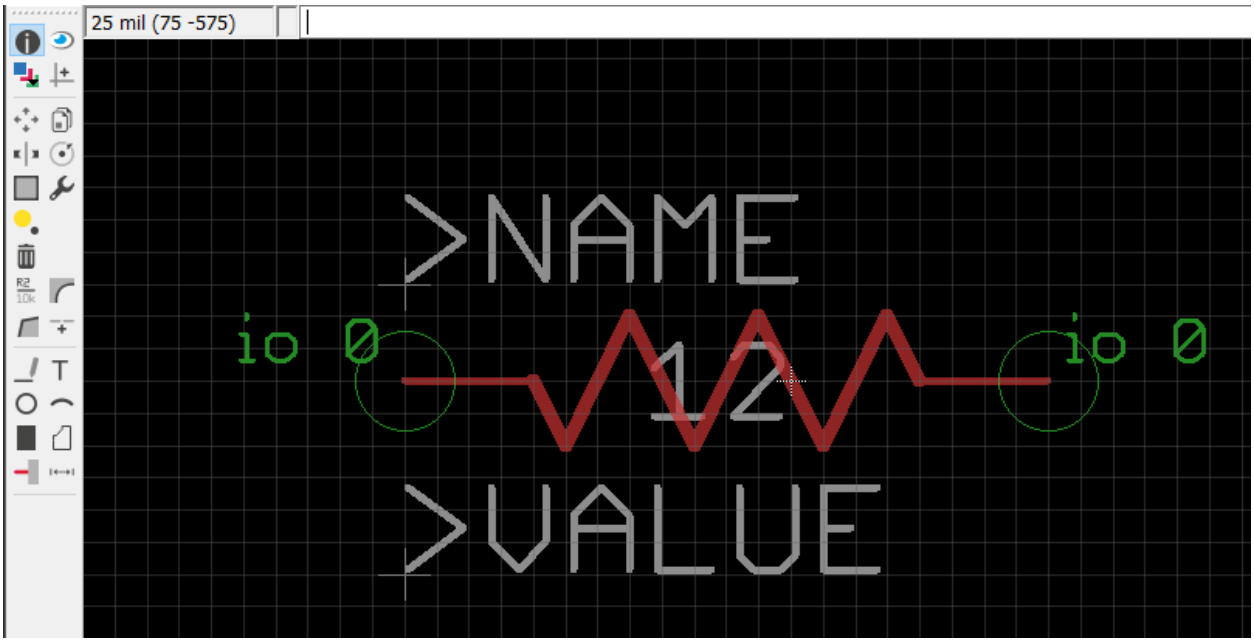


8) Select the text box and type in “>NAME”. Place this label slightly above the resistor symbol. Repeat the process and place a “>VALUE” label slightly below the symbol.

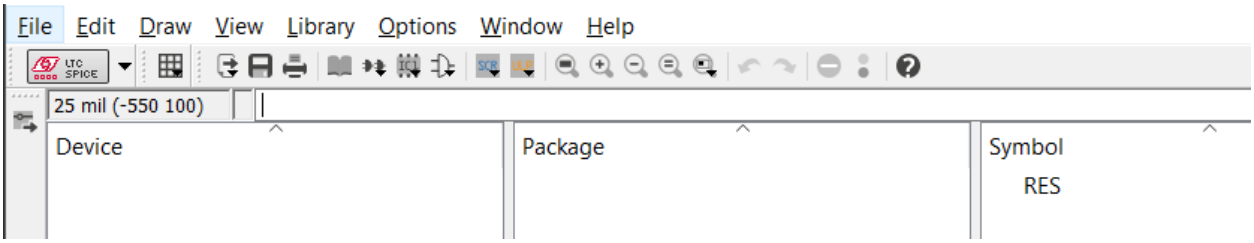


Click on the information tool and then on the label to view its properties. Make sure that the layer for the “>NAME” label is Names and that the layer for “>VALUE” is Values. Both should be gray in color.

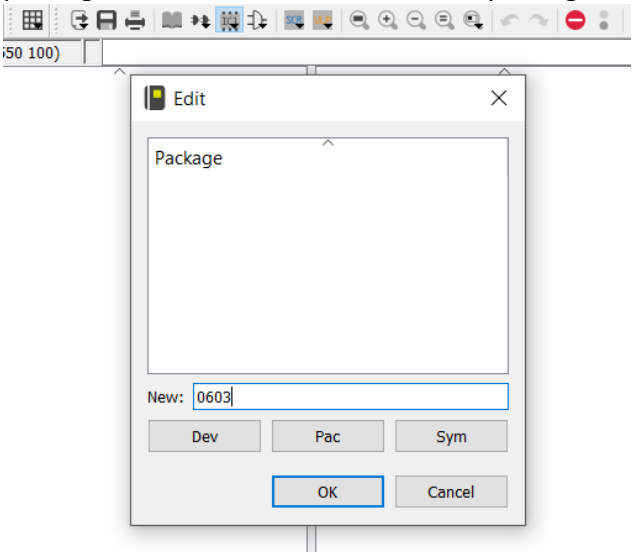




9) Save and return to the library menu.



10) Create the package for the component. As mentioned above, we will be creating a 0603 package resistor so we will name the package “R0603”.



11) Create the package for the resistor:
Digikey Link: https://www.digikey.com/product-detail/en/yageo/RC0603JR-0710KL/311-10KGRCT-ND/729647
Datasheet Link: http://www.yageo.com/documents/recent/PYu-RC_Group_51_RoHS_L_8.pdf
It is important to remember that 0603 is a standardized size so this package should work for other components with 0603 packages.

12) Navigate to the package dimensions and/or PCB layout recommendation section of the datasheet. The package dimensions can be seen below. Make sure to keep the units consistent when creating parts. For packages, we will use millimeters.

Outlines

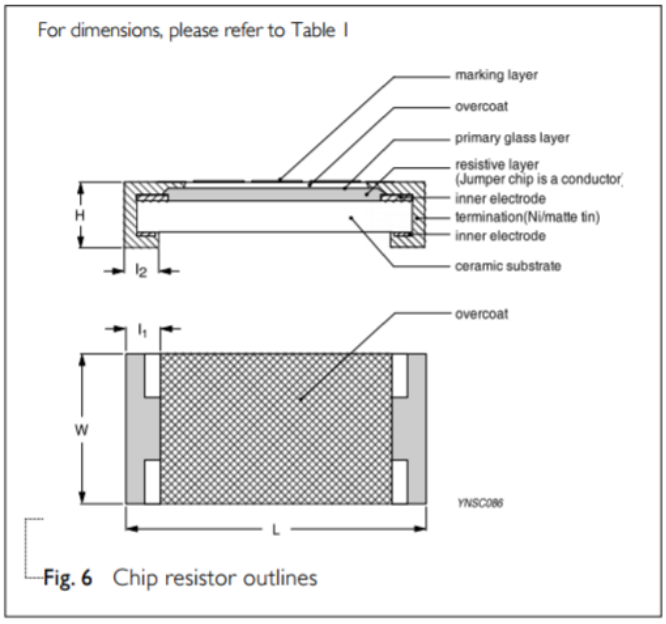


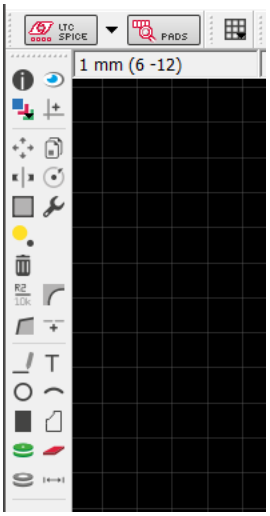
Fig. 6 Chip resistor outlines

DIMENSION

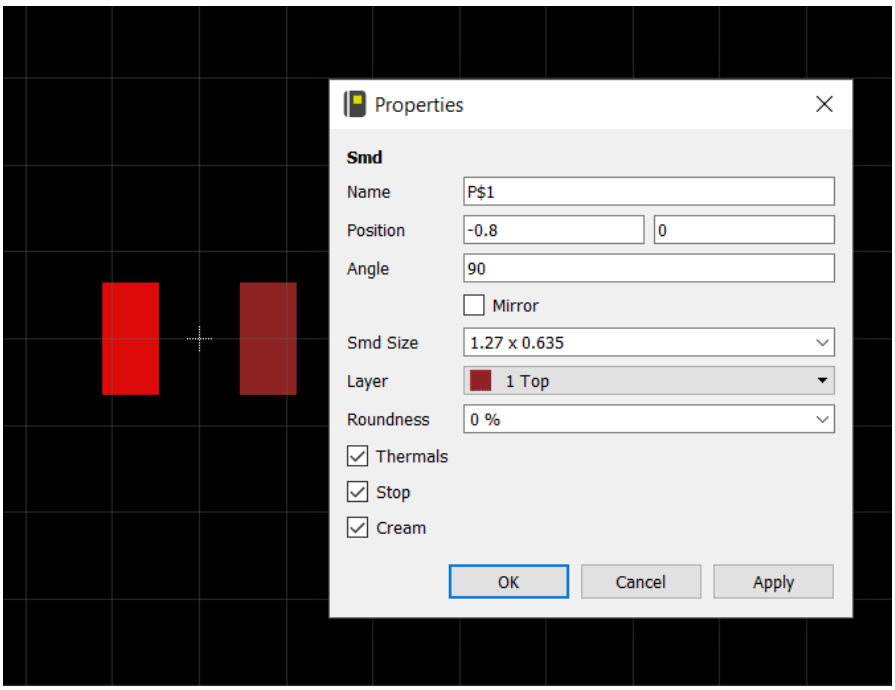
Table 1

TYPE	L (mm)	W (mm)	H (mm)	I_1 (mm)	I_2 (mm)
RC0075	0.30±0.01	0.15±0.01	0.10±0.01	0.08±0.03	0.08±0.03
RC0100	0.40±0.02	0.20±0.02	0.13±0.02	0.10±0.03	0.10±0.03
RC0201	0.60±0.03	0.30±0.03	0.23±0.03	0.10±0.05	0.15±0.05
RC0402	1.00±0.05	0.50±0.05	0.35±0.05	0.20±0.10	0.25±0.10
RC0603	1.60±0.10	0.80±0.10	0.45±0.10	0.25±0.15	0.25±0.15
RC0805	2.00±0.10	1.25±0.10	0.50±0.10	0.35±0.20	0.35±0.20
RC1206	3.10±0.10	1.60±0.10	0.55±0.10	0.45±0.20	0.40±0.20
RC1210	3.10±0.10	2.60±0.15	0.55±0.10	0.45±0.15	0.50±0.20
RC1218	3.10±0.10	4.60±0.10	0.55±0.10	0.45±0.20	0.40±0.20
RC2010	5.00±0.10	2.50±0.15	0.55±0.10	0.45±0.15	0.50±0.20
RC2512	6.35±0.10	3.10±0.15	0.55±0.10	0.60±0.20	0.50±0.20

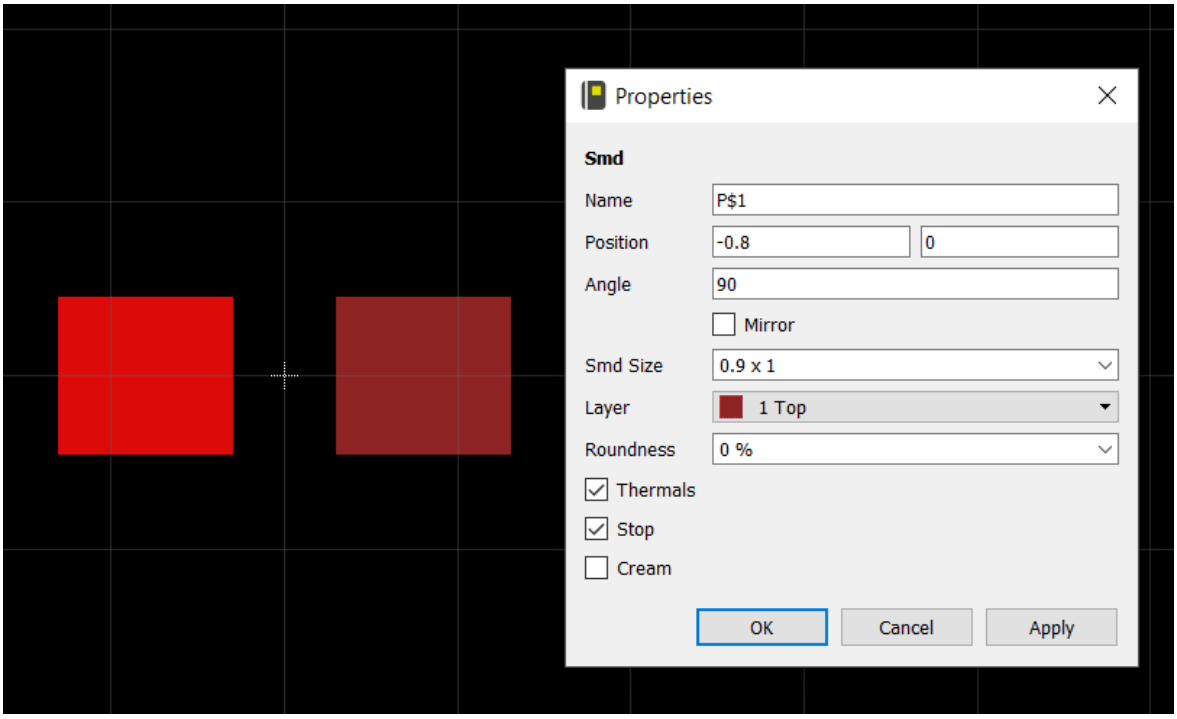
13) Change the grid size to 1 mm. Enter commands: “grid mm” and “grid 1”. Verify that the grid settings are correct before proceeding.



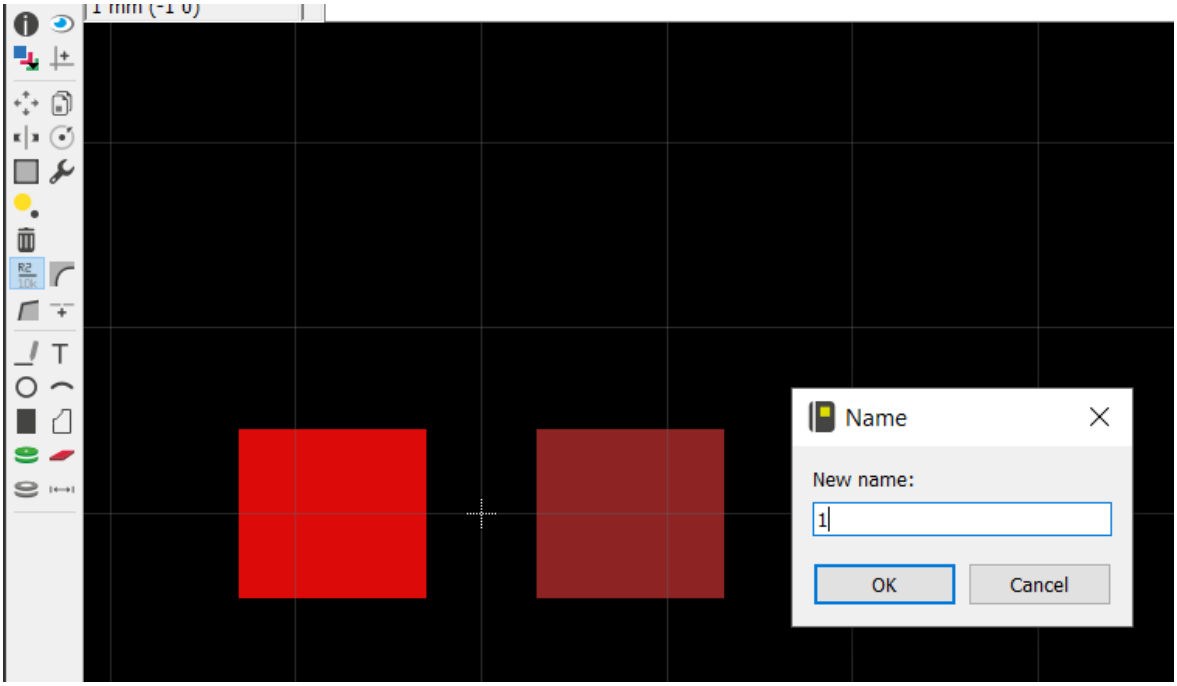
13) We will use the origin crosshair as the center of our package. Select the SMD pad tool and place a pad close to the origin. Looking at the datasheet, the resistor dimensions are denoted by L and W. Using table 1, the part we want is RC0603 which corresponds to the package size. Therefore, L = 1.6 ± 0.1 mm and W = 0.8 ± 0.1 mm. For this tutorial, we will ignore the tolerances and assume exact sizing. The surface mounted pads are 0.8 mm (L/2) from the origin. Since the origin is being used as the center of the package, place two pads, each in-line with the x-axis and 0.8 mm away from the origin. Use the info tool to verify the correct placement of pads. The coordinates for the left and right pads should be (-0.8, 0) and (0.8, 0), respectively.



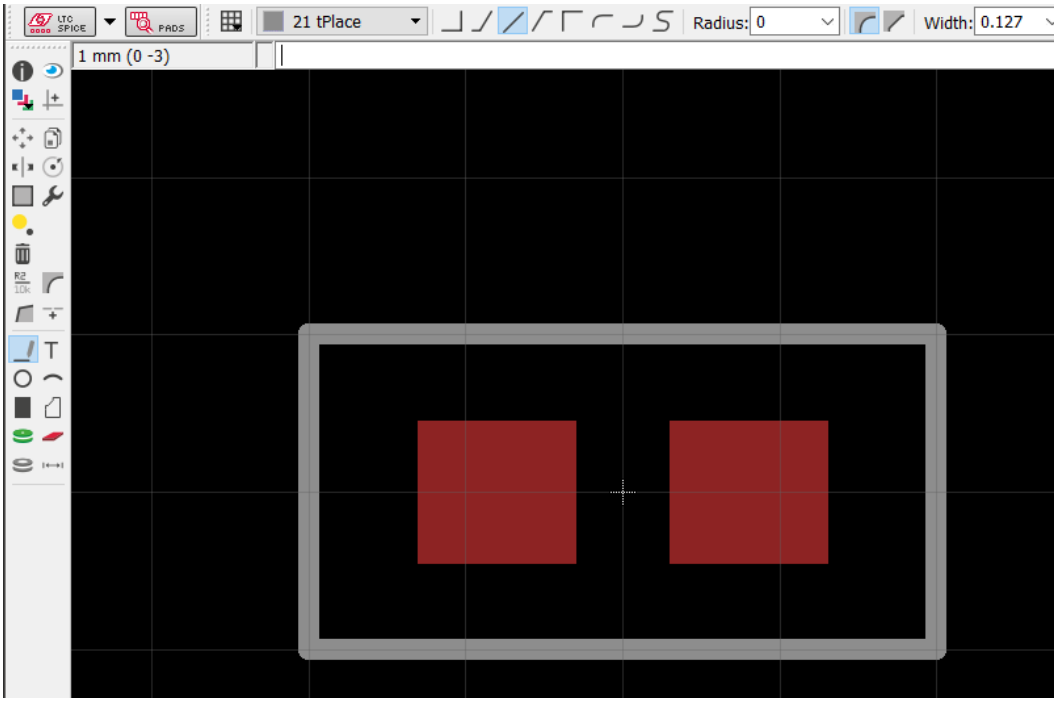
14) Once the pad position is verified, the next step is to modify the pad size to match the datasheet. Once again (ignoring tolerances), W = 0.8 mm and I1 = 0.25 mm. Using the info tool, change the smd size to 0.9 x 1 and check the thermals and stop boxes.

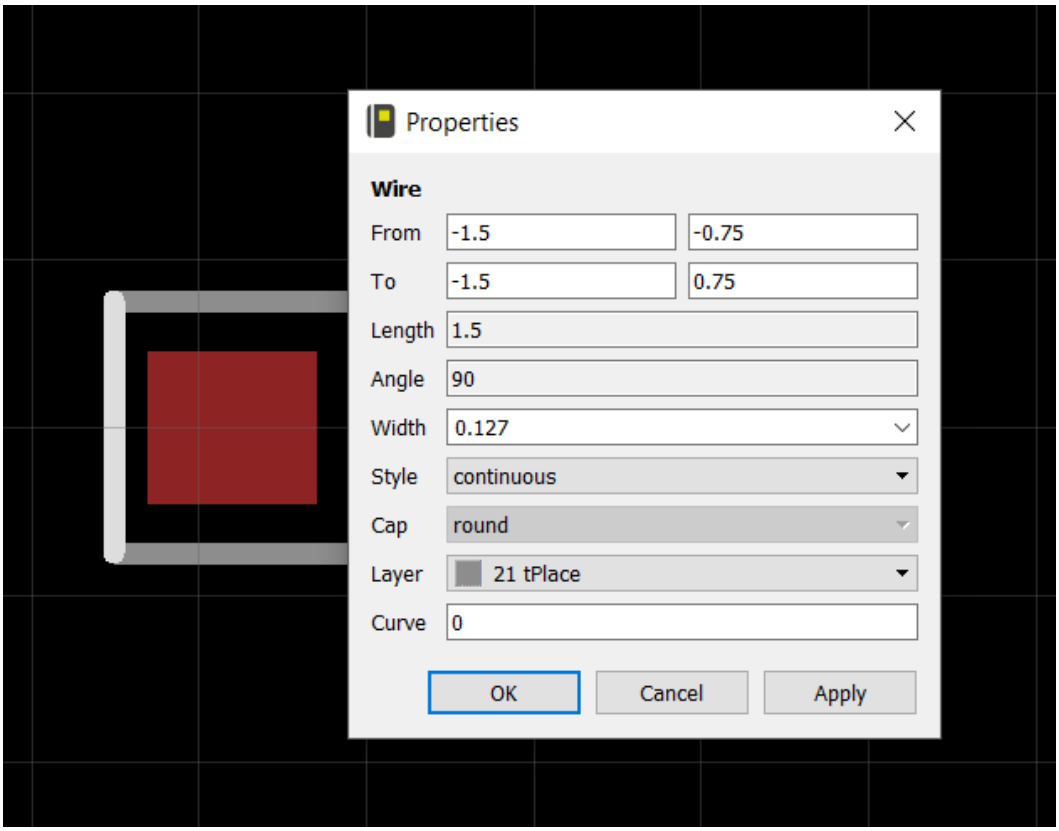


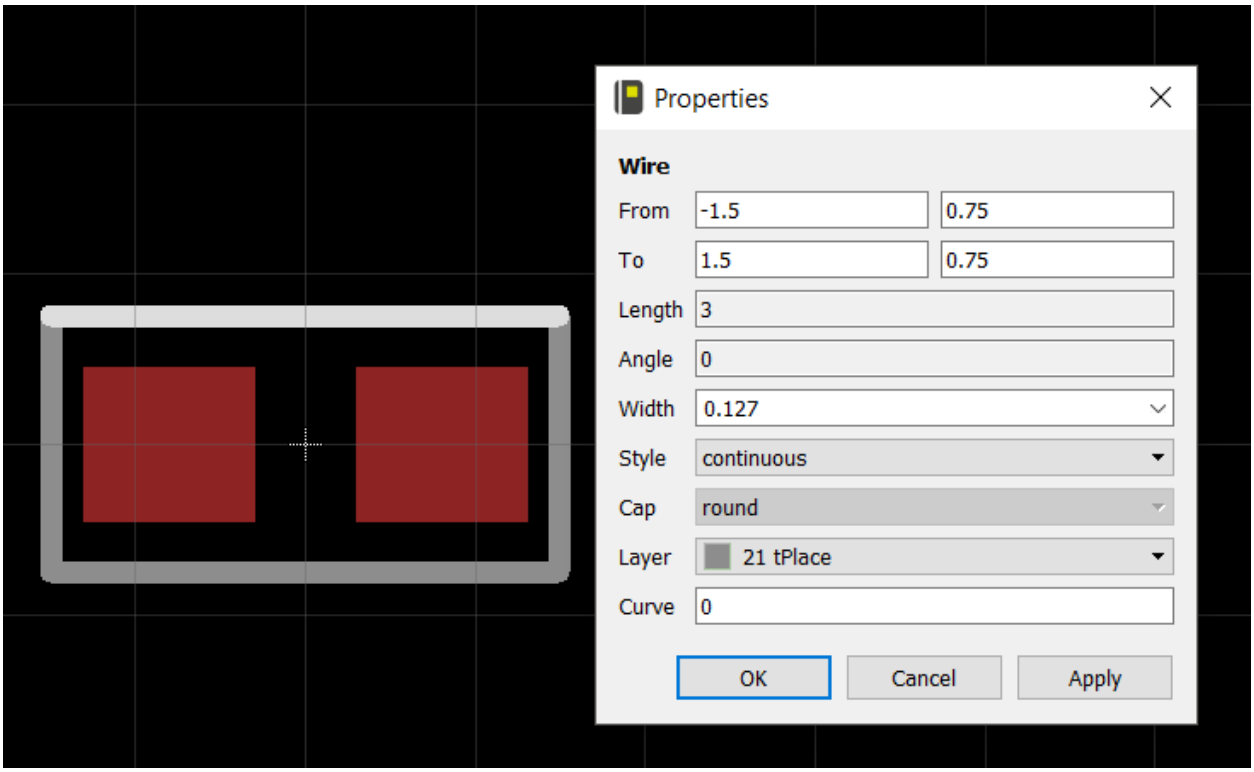
15) Use the name tool to change the names of the pads to “1” and “2”.

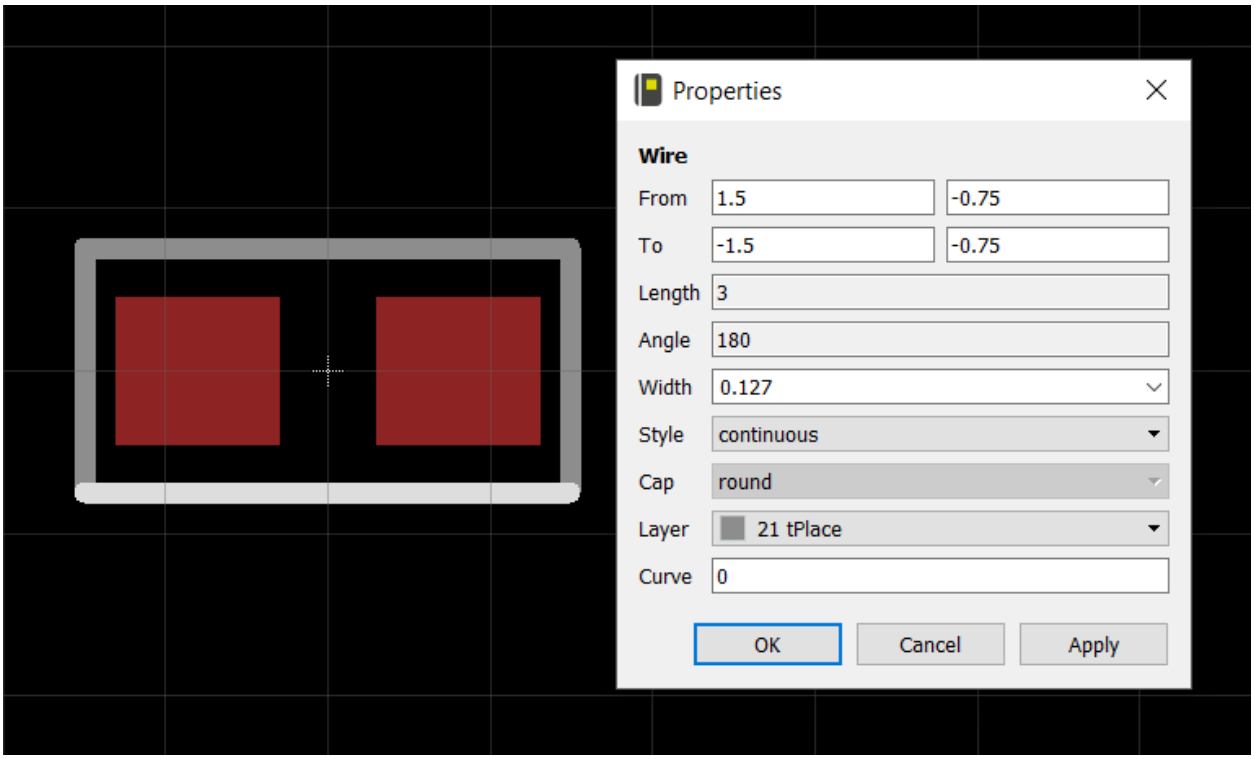


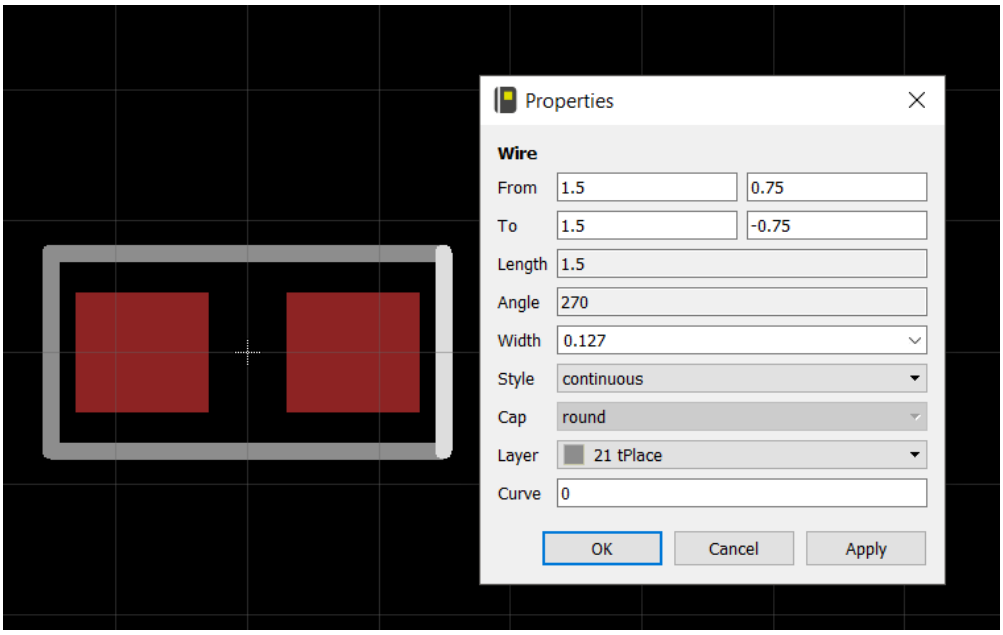
16) Create the outline of the package. Using the wire tool, trace an outline around the pads. Using the info tool, change the layer of these wires to tPlace. Use the info tool to change the positions of the wires to be 0.7 mm away from the pads horizontally (-1.5 mm and 1.5 mm) and to be 0.75 mm away from the origin vertically (-0.75 mm and 0.75 mm).



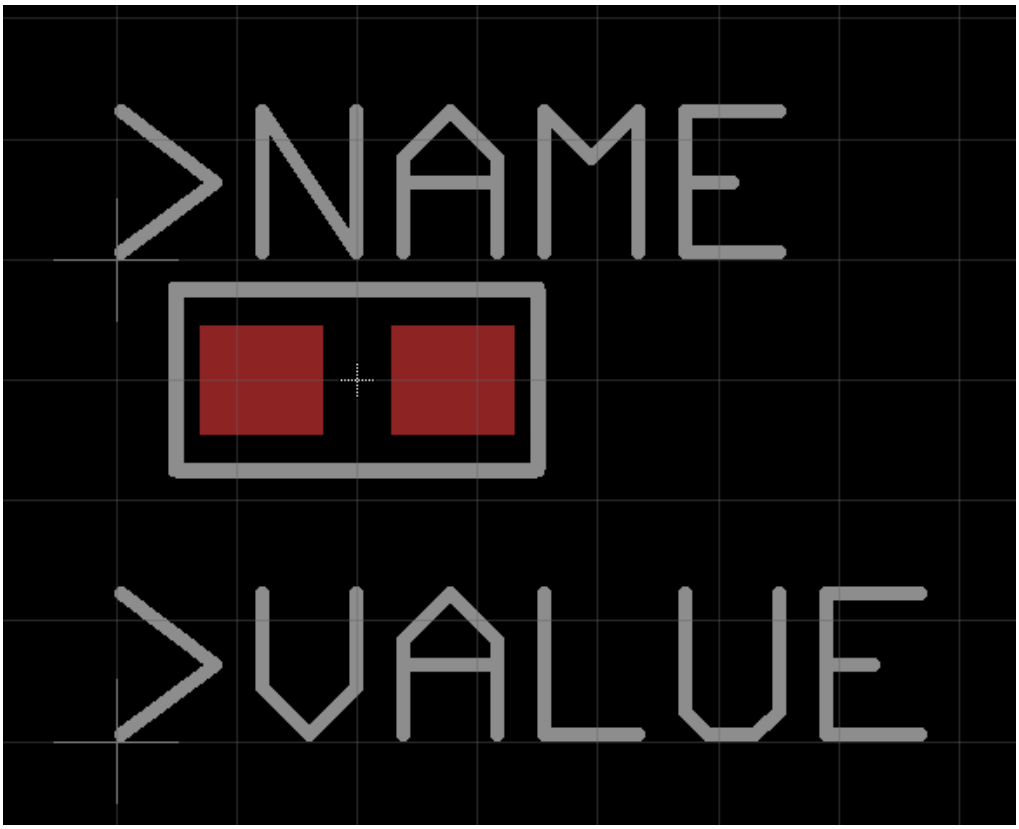



Properties
Wire
From -1.5 0.75
To 1.5 0.75
Length 3
Angle 0
Width 0.127
Style continuous
Cap round
Layer 21 tPlace
Curve 0
OK
Cancel
Apply


Properties
Wire
From 1.5 -0.75
To -1.5 -0.75
Length 3
Angle 180
Width 0.127
Style continuous
Cap round
Layer 21 tPlace
Curve 0
OK
Cancel
Apply

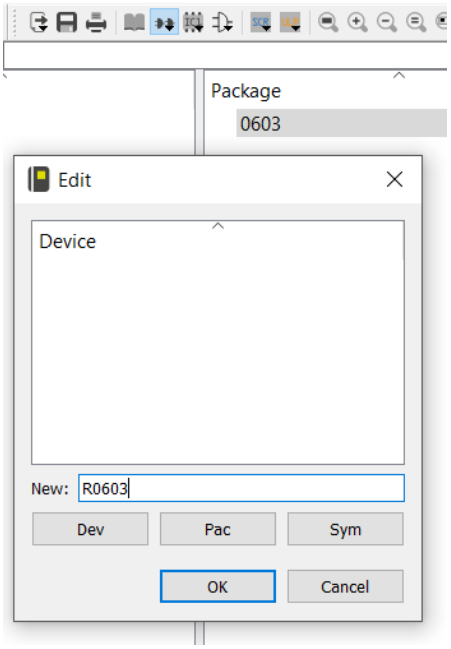


17) Using the text box, create >NAME and >VALUE labels. Move the name label on top of the package and the value label on the bottom. Use the info tool to verify that they are on the tNames and tValues layers, respectively. When finished, the package should look like this.

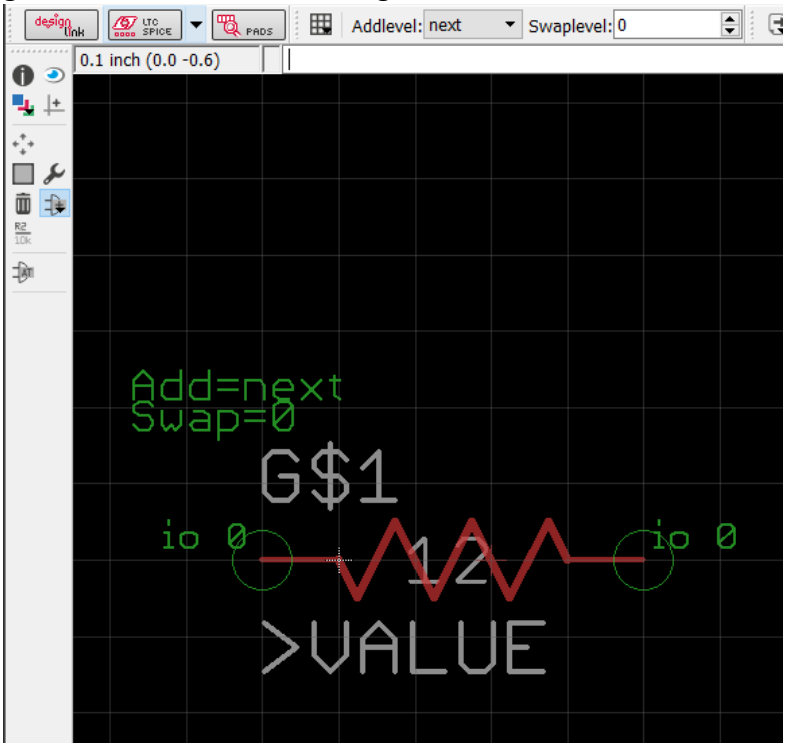


18) Now that the package is made, the next step is to create a device for the part. This will tie together the pins from the schematic symbol to the pads on the package. Navigate back to the table of contents and create a new device.

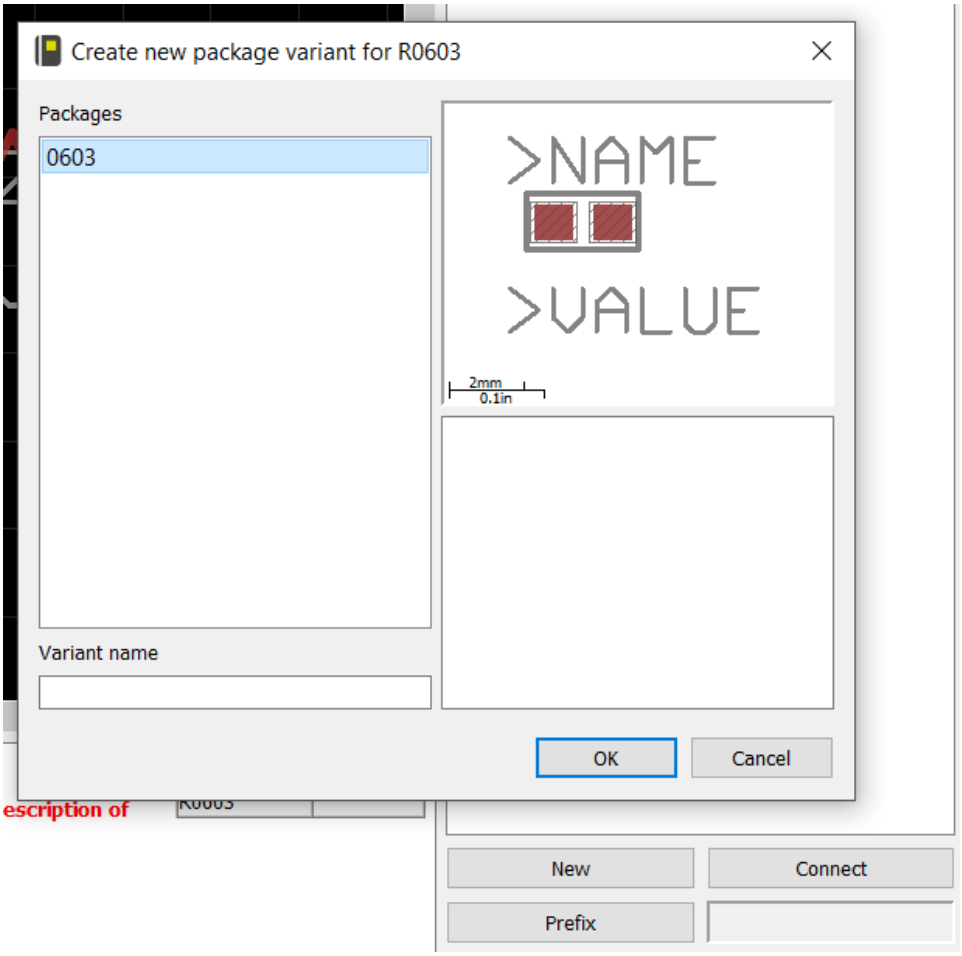
19) Name the device R0603.

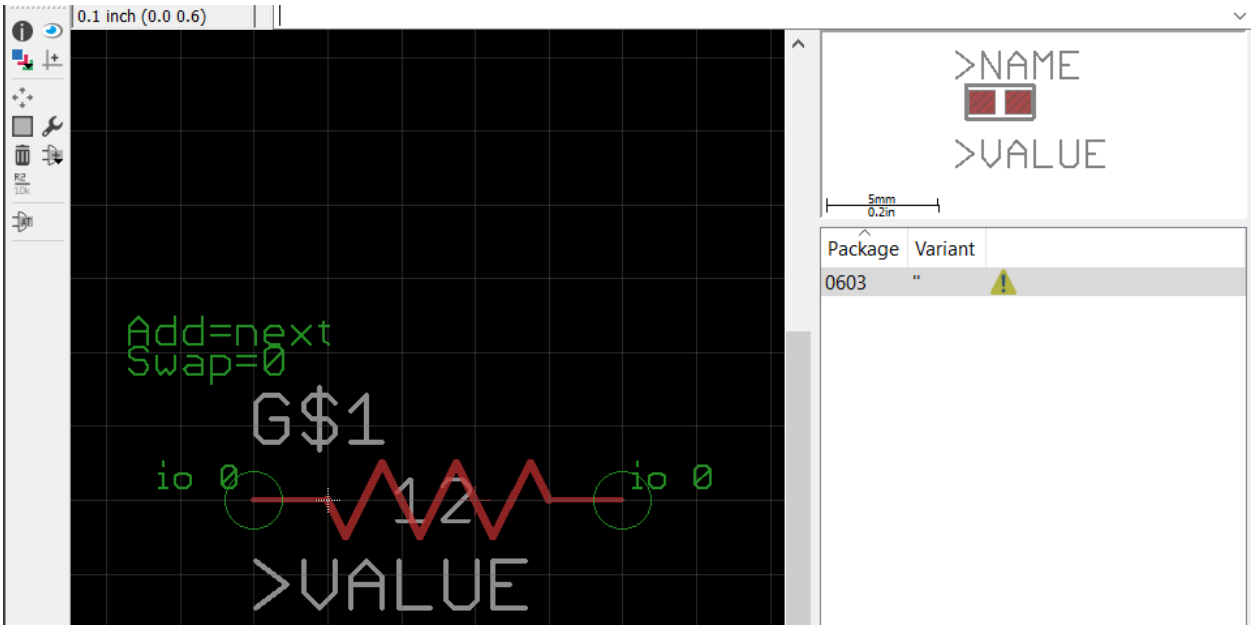


20) After opening the device, type in “grid on”. Add the schematic symbol and place it on the grid somewhere near the origin crosshair.

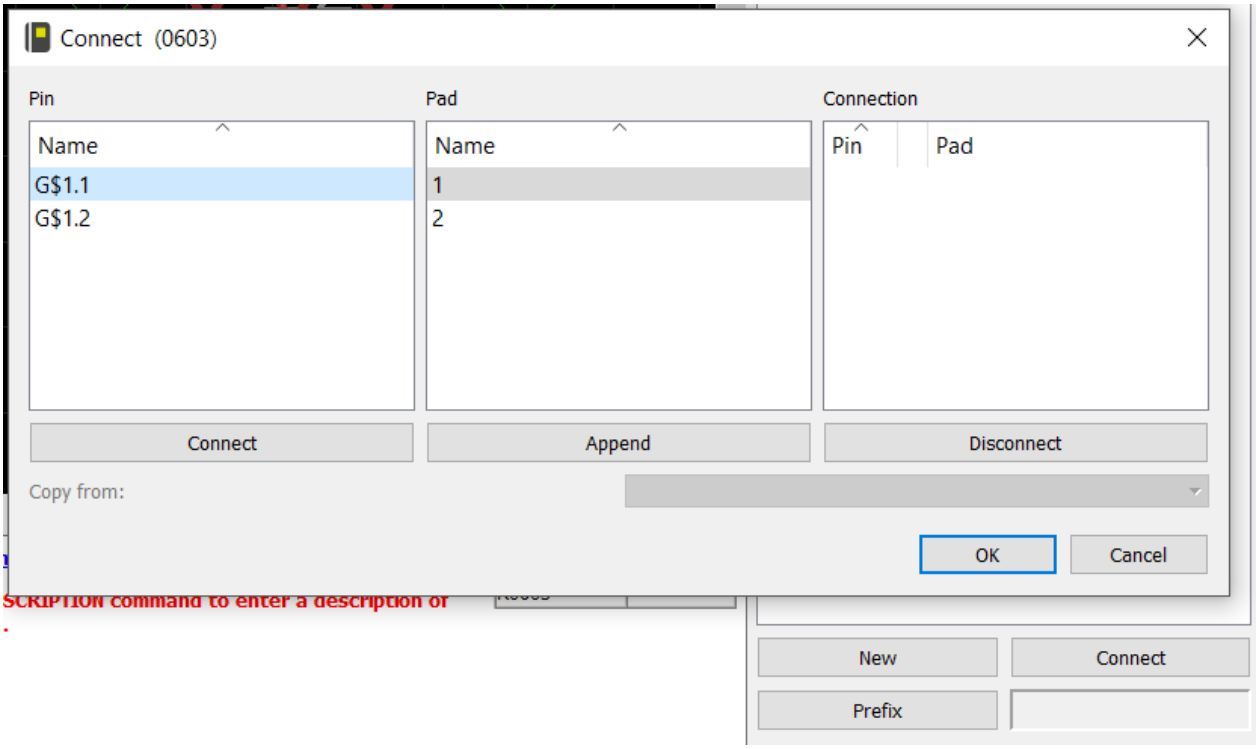


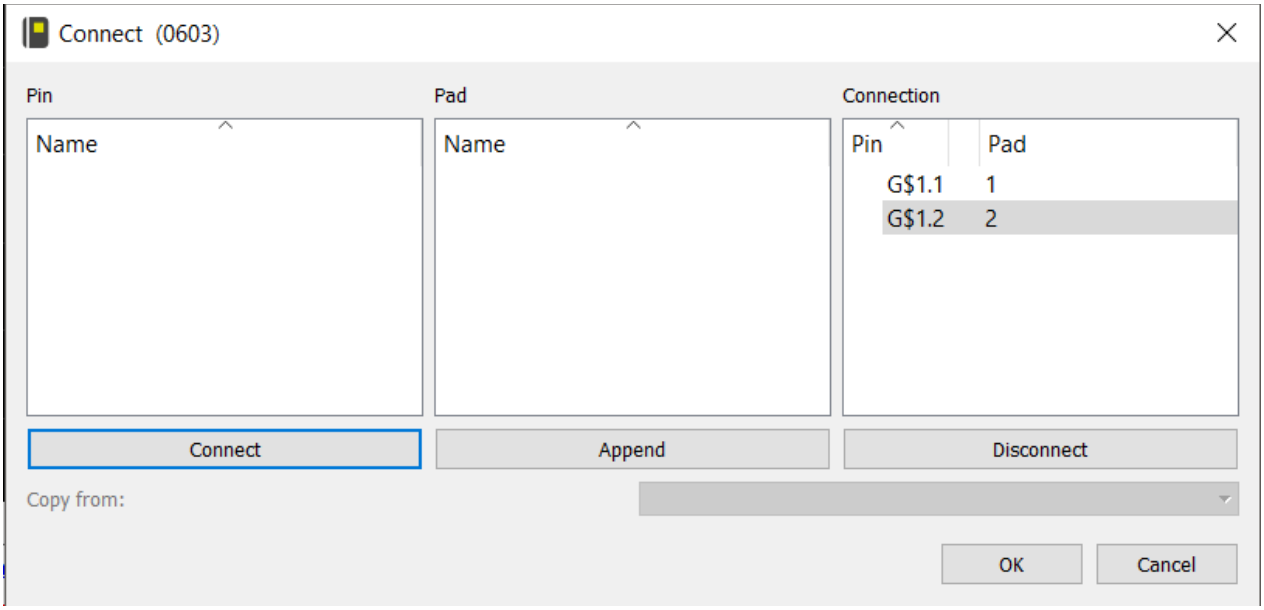
21) Select the New option and select the package you've just made.

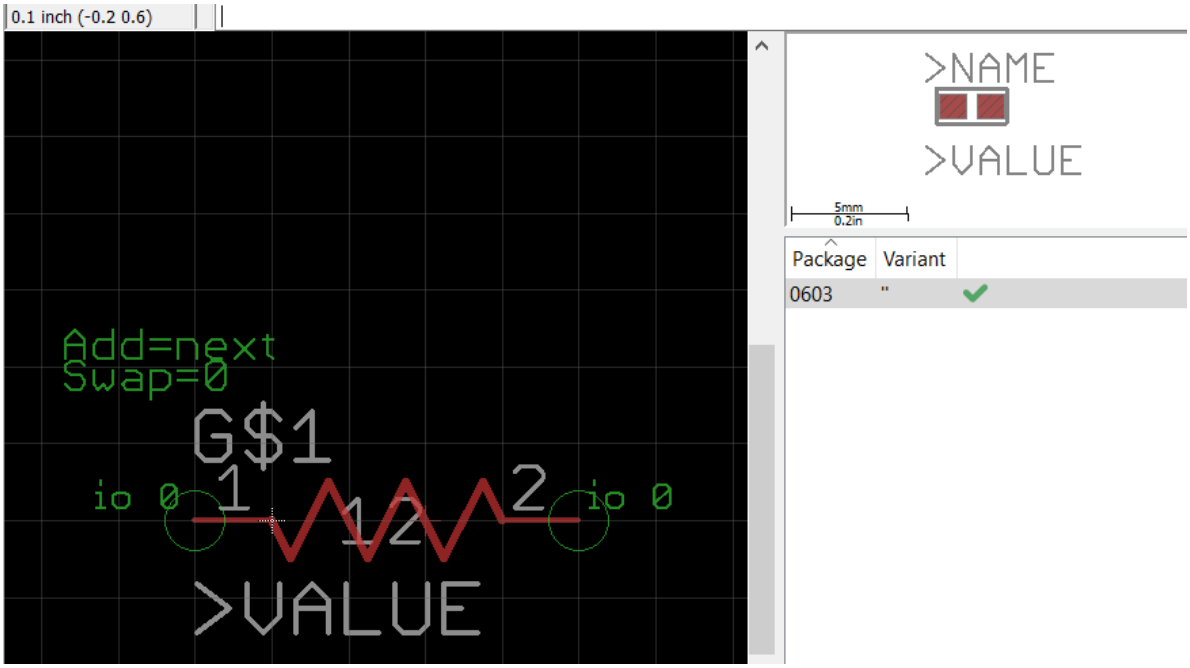




22) Select the Connect option and connect the appropriate pins and pads together (1 should be connected to 1, 2 should be connected to 2).







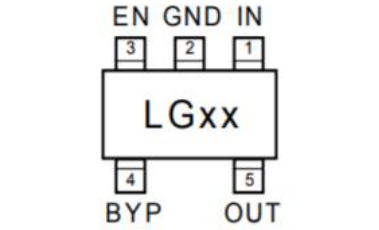
23) Save the device and navigate back to the table of contents. Congratulations! You have just finished creating your first part.

Voltage Regulator

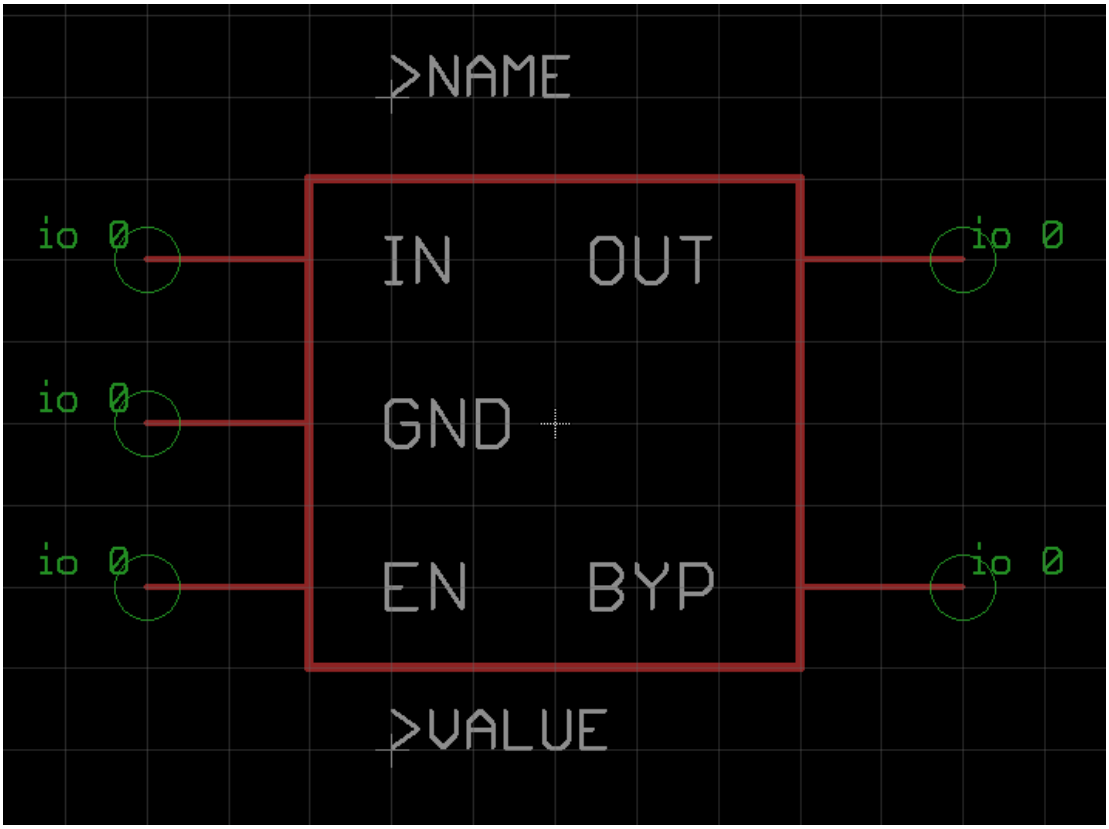
1) Using the same library as above, create a new symbol. We will name it “MIC5219-3.3” to reflect the part number.

2) Look over the datasheet and determine the amount of pins and the names of each pin.

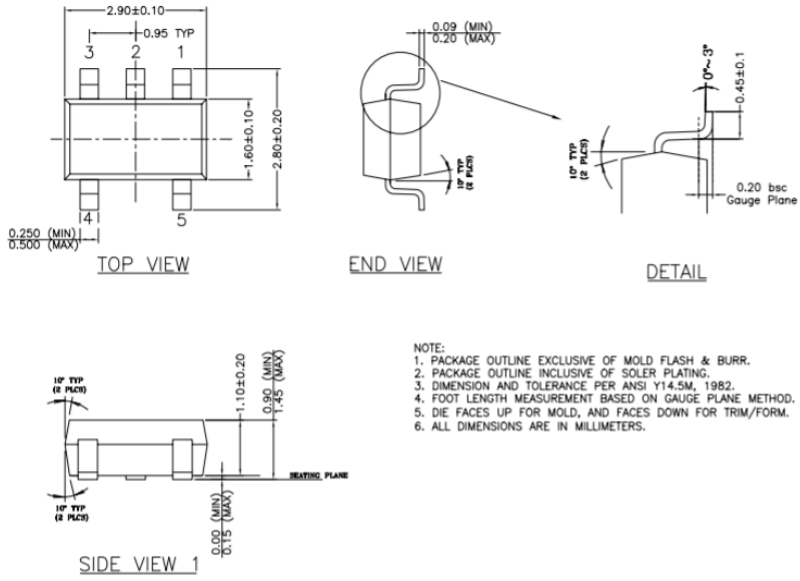
3) Set the grid to 0.1 in and draw the symbol. The wires should be on the Symbols layer (red in color). Following the datasheet, replicate the chip pin placement. Name the pins accordingly to match the datasheet. Create name and value labels for the symbol. When finished, your symbol should roughly match the symbol shown below.



MIC5219-x.xBM5 / SOT-23-5
Fixed Voltages
(Top View)

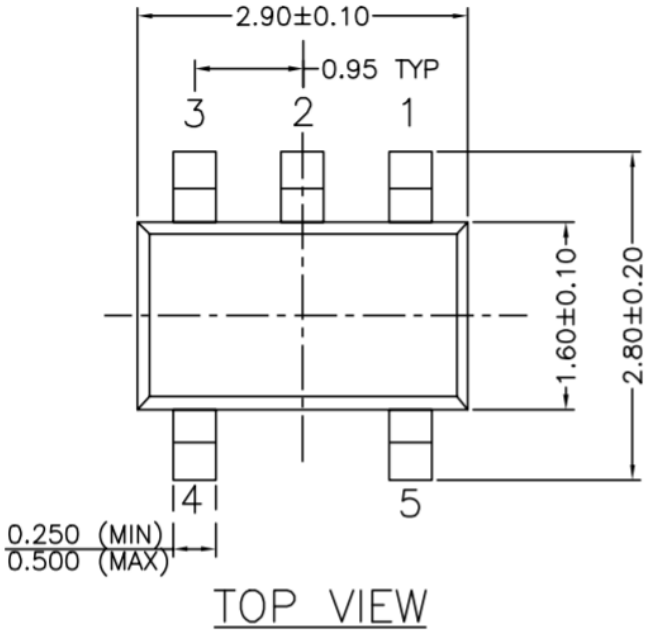


4) Save and navigate back to the table of contents. Create a package and name it “SOT23-5” which is the standardized package for this chip variant. Follow the datasheet to find the pad sizing and positioning.

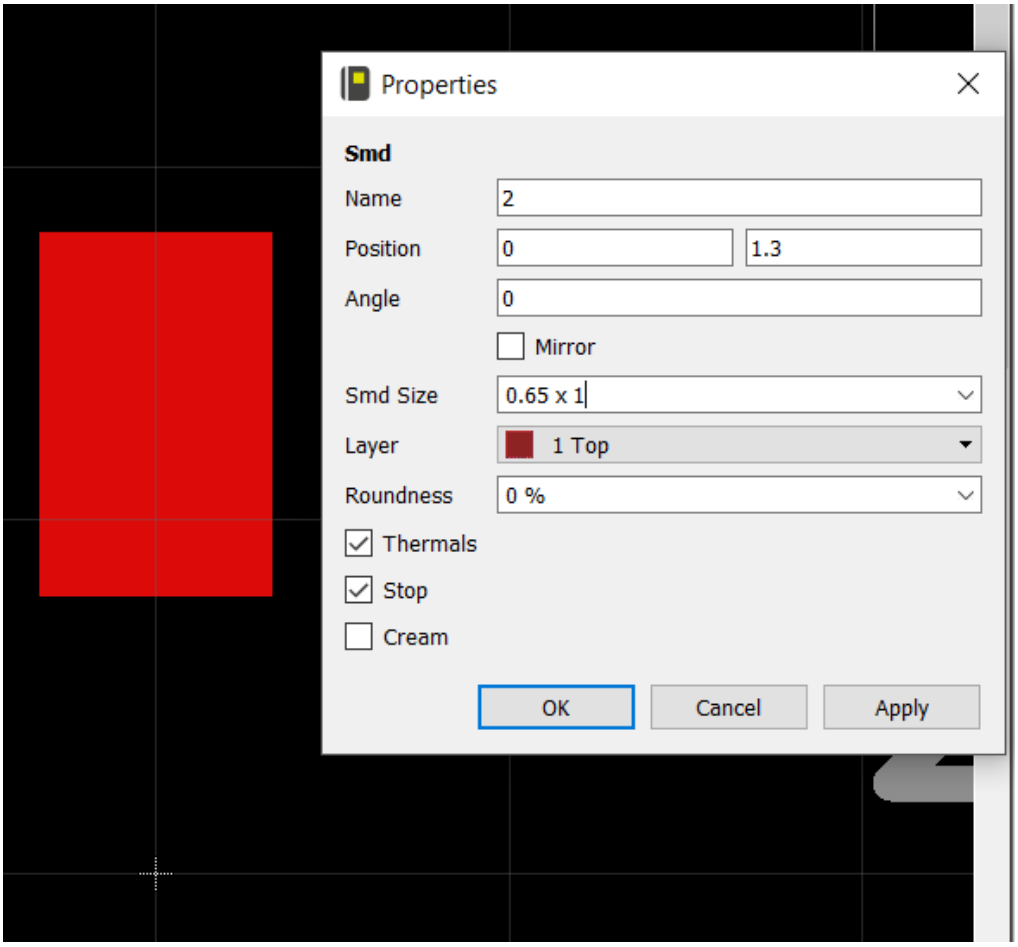


SOT-23-5 (M5)

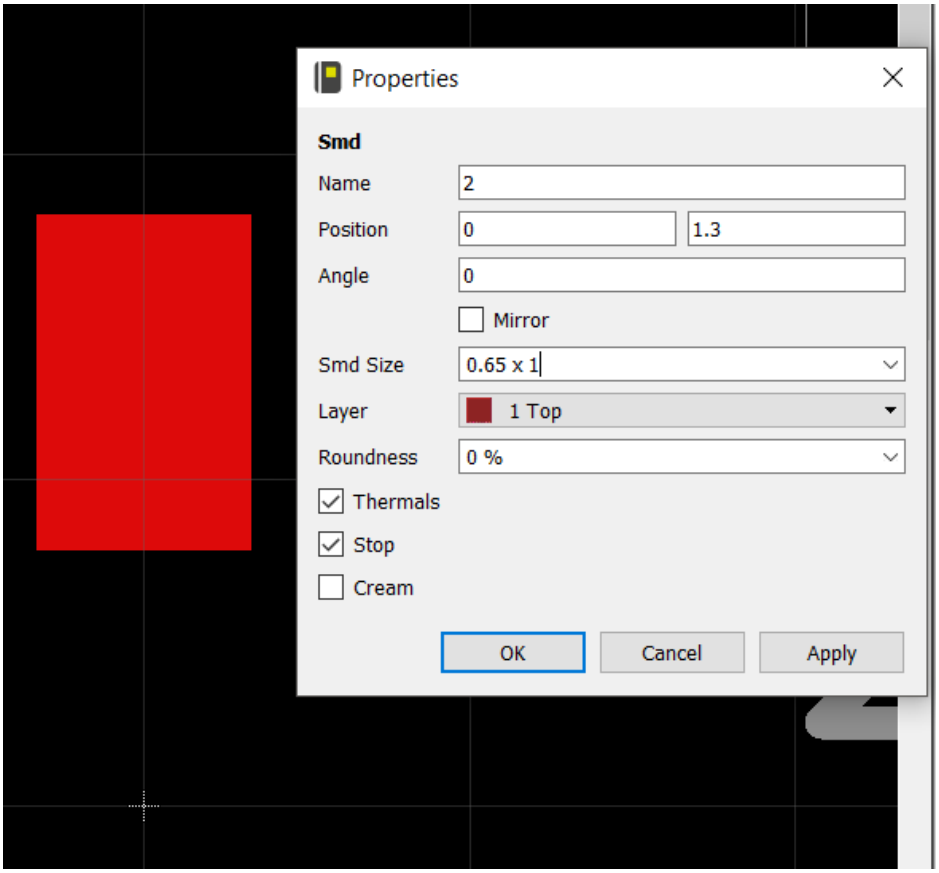
5) Looking at the datasheet, the origin shown should represent the origin on the package grid. The units are in mm so make sure to change your grid to mm.



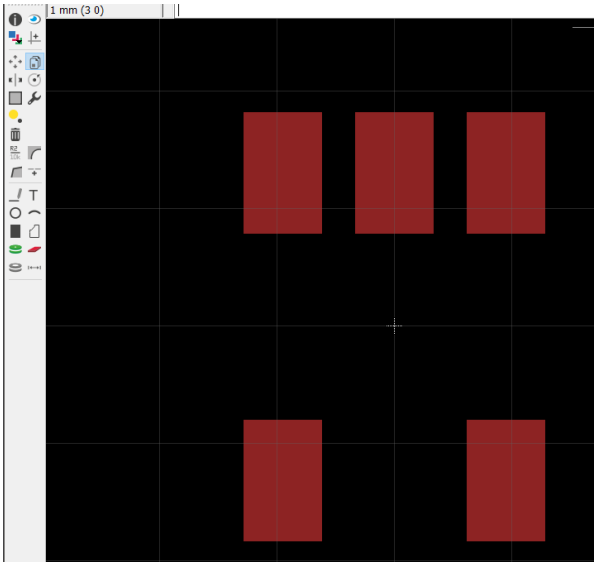
6) Pad 2 looks to be in line with the y-axis, so we will make that pad first. Using the smd pad tool, place a pad on (0, 1) for now. You can calculate the pad sizing based off the datasheet. The width is given to be 0.25mm (min) to 0.5mm (max). We will use the max pad width and give it some extra width (0.65 mm). The length can be calculated by taking the difference of 2.8 – 1.6 and dividing it by 2. (2.8 -1.6)/2 = 0.6 mm. Again, we will give it some overage to make it easier to solder (1 mm). The final pad sizing should be 0.65 x 1. Enable thermals and stop.



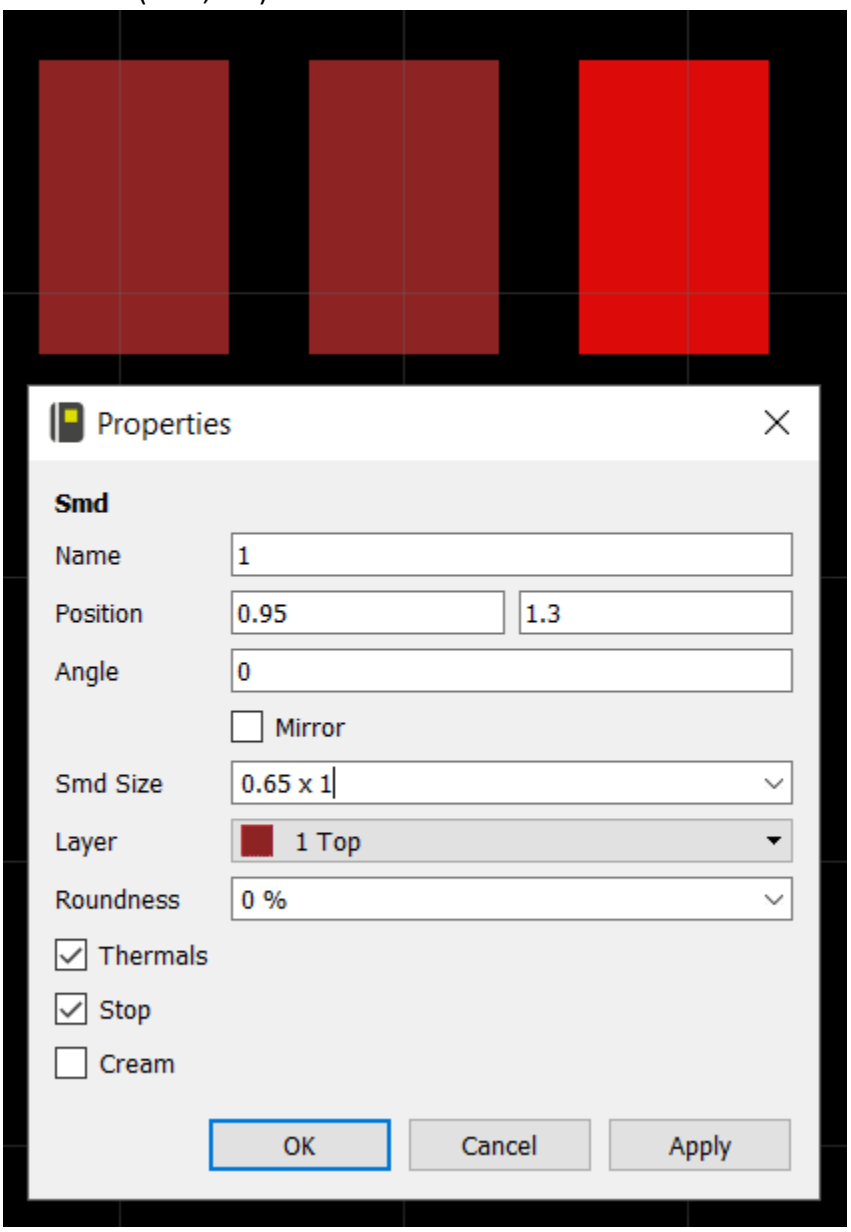
7) Finalize the pad 2's position. Using the datasheet, find the distance to the center of the pad from the origin. The equation should be half of the IC body (1.6) + half of the pad length (1) = 0.8 + 0.5 = 1.3. The final position for pad 2 should be (0, 1.3).

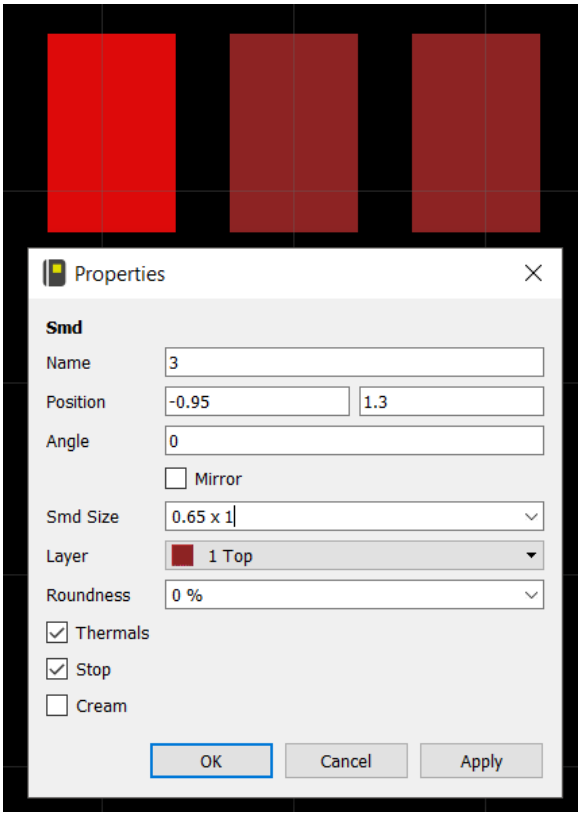


8) You can now use the copy tool to create 5 pads total. Place them apart from each other. They will have the same dimensions but will need to be moved to the right positions.

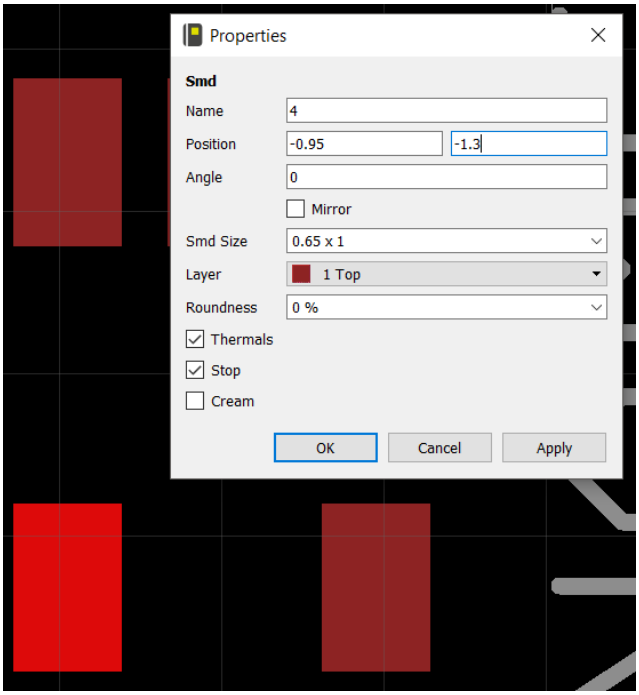


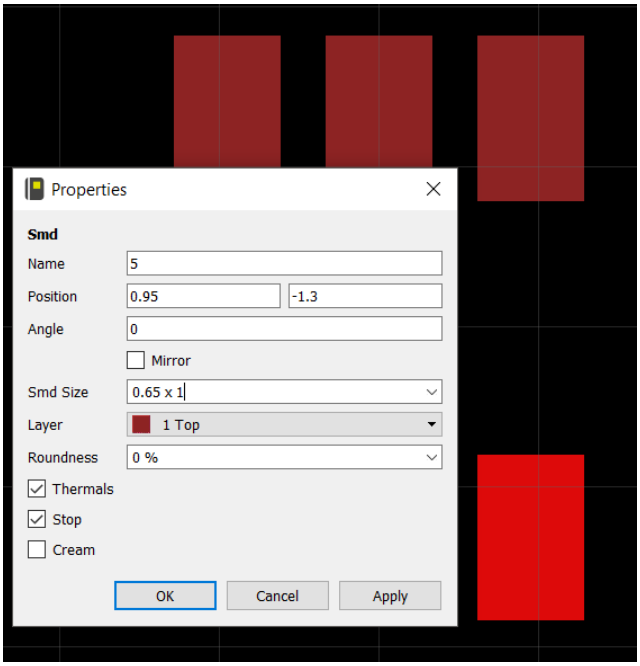
9) Next, we will place pads 1 and 3. According to the datasheet, the pitch between pads is 0.95mm apart. Thus, the position for pad 3 should be (-0.95, 1.3) and the position for pad 1 should be (0.95, 1.3).



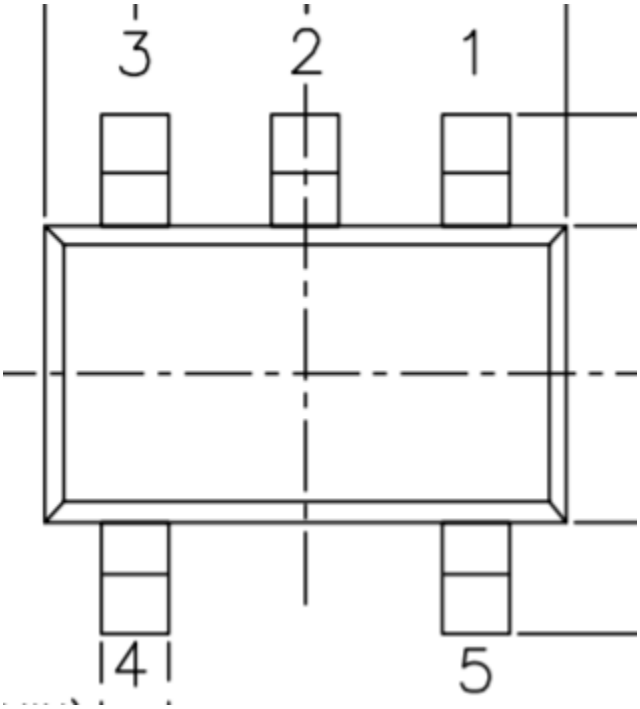


10) The chip is mirrored with respect to the x-axis (except for pad 2) so the positions of pads 4 and 5 should be mirrored. Pad 4's position should be (-0.95, -1.3) and pad 5's position should be (0.95, -1.3).

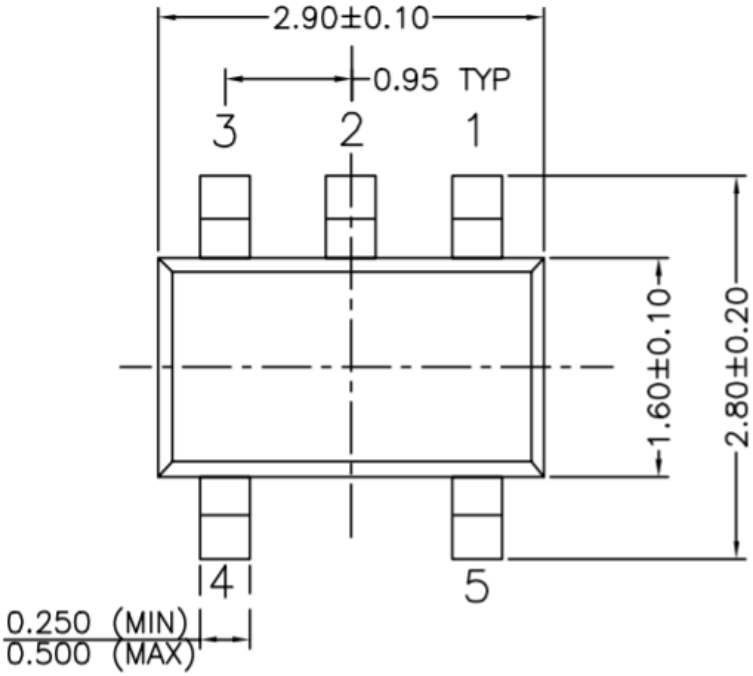




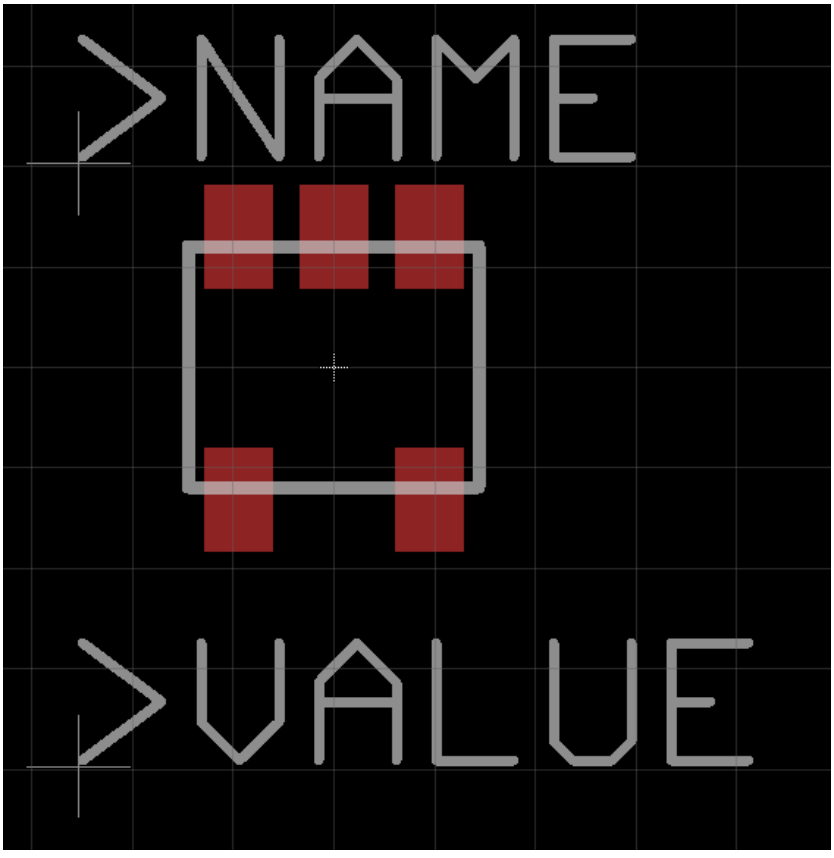
11) Name all pads based off the package in the datasheet.



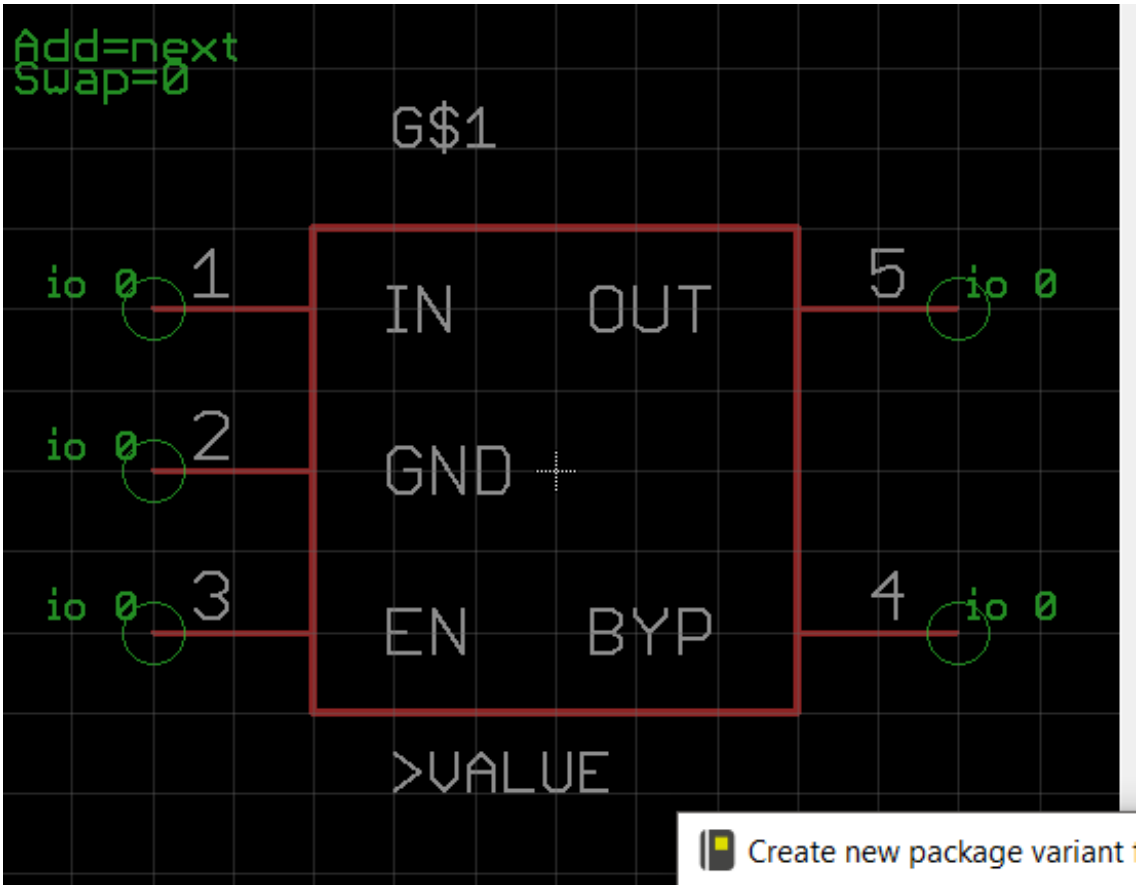
12) Use the datasheet to determine the dimensions of the chip body and use the wire tool to draw it (layer should be tPlace). Use math and determine the appropriate positions for the dimension wires. (Chip body is 2.9mm long and 1.6mm wide) Create the name and value labels as you've done before. When finished, your package should look like the package shown below.


2.90±0.10
0.95 TYP
3
2
1
1.60±0.10
2.80±0.20
4
5
0.250 (MIN)
0.500 (MAX)

TOP VIEW

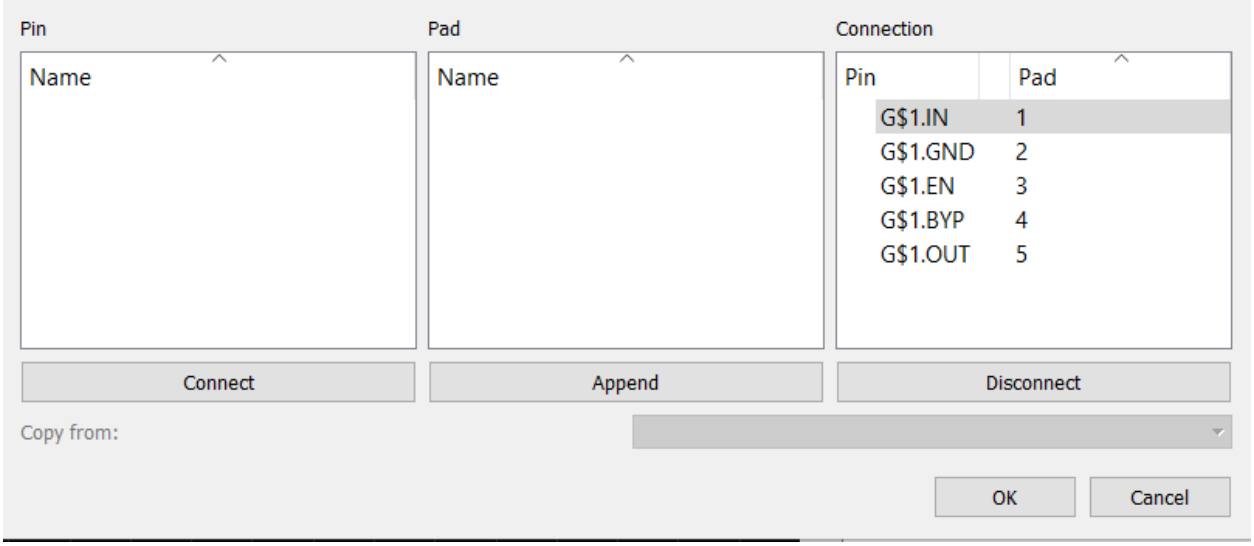

>NAME
>VALUE

13) Save and navigate back to the table of contents. Create a new device and name it "MIC5219-3.3". Add the schematic symbol and use the New option to add the package you've just made.



14) Use the Connect option to connect the pins and pads according to the table below (found in the datasheet).

Pin No. SOT-23-5	Pin Name	Pin Function
1	IN	Supply Input.
2	GND	Ground: MSOP-8 pins 5 through 8 are internally connected.
5	OUT	Regulator Output.
3	EN	Enable (Input): CMOS compatible control input. Logic high = enable; logic low or open = shutdown.
4 (fixed)	BYP	Reference Bypass: Connect external 470pF capacitor to GND to reduce output noise. May be left open.
4 (adj.)	ADJ	Adjust (Input): Feedback input. Connect to resistive voltage-divider network.
—	GND	Ground: Internally connected to the exposed pad. Connect externally to GND pin.



15) Save and exit to the table of contents. Congratulations! You have completed this tutorial.