# Creating a new Part Type using the PCB & CAE Decal Wizards V9.1

Note: This tutorial is modified from the original tutorial to bring it up to date for version 9.1 and to expand on some additional features that may be useful for ECE477 senior design students. The original tutorials are located at:

# http://www.people.vcu.edu/~rhklenke/tutorials/PADS/PADS\_Tutorial\_3.3V Reg/SimpleReg.html http://www.people.vcu.edu/~rhklenke/tutorials/PADS/PADS\_Tutorial\_New\_Part/NewPart Type.html

A part is an object that can be represented in a schematic and as a physical item on a printed circuit board, or PCB. In order to be used in a design the part must be set up for use in the schematic and layout portions of the PADS application. Three items are represented by the part:

- 1. CAE Decal, which is used as the schematic symbol;
- 2. PCB Decal, which represents the physical part that will be placed on the circuit board;
- 3. Electrical data, which is specific information for that part such as resistor and capacitor values, power ratings, manufacturer and other information.

Many parts may use the same CAE and PCB decal symbols; the symbols are contained in the libraries that come with PADS. Those items not included in the libraries must be manually generated by the schematic Part Editor and the layout PCB Decal Editor. Two methods demonstrated here are the use of the PCB decal wizard and modification of an existing footprint.

### What you will learn:

- Creating and Managing User Libraries
- Creating a PCB Decal
- Creating a CAE Decal
- Creating a Complete Part

### Section 1 - Create a schematic symbol using the Part Editor & CAE Decal Wizard.

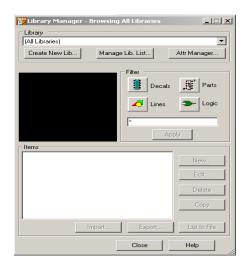
We will now create a schematic symbol for the National Semiconductor LM2675M-5 voltage regulator chip.

1.1 Open PADS Logic. Select Start > Programs > ECE Software > Mentor Graphics SDD > PADS 9.1 > Design Entry >



**PADS Logic**. The PADS Logic welcome screen should appear.

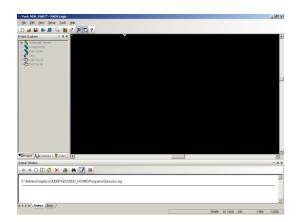
1.2 Create a user library. Select File > Library...



Then press the **Create New Lib**... button. Navigate to your N:\(**Pads Library**) and create a parts library, for this case **tutorial.pt9**. Click on the **Manage Lib. List**... and verify that your newly created parts file is located in the Library drop-down list. Press OK and then close the library manager.

We will be creating a schematic part for the National Semiconductor LM2675M-5 switch mode voltage regulator chip. The relevant datasheet can be found <u>here</u>. (Page 24 of the data sheet contains information on the part's physical dimensions).

1.3 Open the Part Editor. Select **Tools > Part Editor**. The PADS Logic window should now look similar to the image below.



1.4 Select File > New. On the Select type of editing item dialog, choose Part Type and press OK.



1.5 Click the **Edit Electrical** icon, *i*, and the **Part Information for Part** window pops up as shown below.

Part Information fo	or Part - NEW_PART		X
General PCB Decals Part Statistics Pin Count: Decal:	Gates Pins Attributes Connector Pin Mapp	Logic Family	
Gate Count: Signal Pin Count:	0	Ref Prefix: U Families	
Options ☐ Define mapping ( ☑ ECO Registered Prefix List:	of Part Type pin numbers to PCB Decal Part	C Connector C Connector C Die Part C Filip Chip	
Check Part	ОК	Cancel Help	

1.6 To specify the schematic decal to be created select the **Gates** tab. Click **Add**, doubleclick in the CAE Decal 1 field, type **LM2675** and press Enter. Then press the OK button.

			rt - LM2585			1
ieneral	PCB Dec	als Gat	es   Pins   Attribute	es Connector Pin N	(apping	
						Reset
NO	DECA	a				
			Edit	Add	Delete	
Gate	Pins	Swap	CAE Decal 1	CAE Decal 2	CAE Decal 3	CAE Decal 4
A	5	0				
Check P	art			OK	Cancel	Help

1.7 To open the CAE Decal Editor press the Edit Graphics <sup>D</sup> button on the toolbar to open the CAE Decal Editor. The Select Gate Decal dialog will pop up.

🞇 Select Gate Decal	×
Select Gate Decal to Edit Gate: Gate A Decal: LM2675	Alternative Decals:
OK Cancel	Help

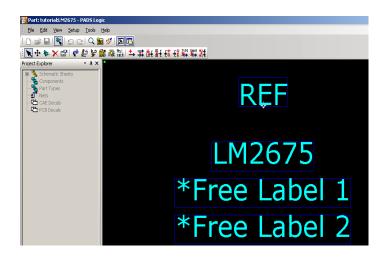
Press OK to continue. PADS Logic will display a warning stating that the selected decal doesn't exist and will be created. Press OK.

PADS		×
⚠	Selected gate decal does not exist - Creating decal LM2	675
	OK	

The CAE Decal Editor should now appear. Press the Decal Editing Toolbar icon, to bring up the toolbar as shown below.



Your display now looks like that below.



1.8 To start the CAE Decal Wizard, press the CAE Decal Wizard button, 3, on the Decal Editing toolbar to launch the Decal Wizard.

💕 CAE Decal Wizard			X
Preview:	Pin Length Horizontal: Vertical: 200 • 200 •	Left Pins Pin Decal: Pin Count: PIN <b>2</b>	4 PN 🔻 4 📖
	Pin Spacing       Horizontal:     Vertical:       100     100	Pin Decat     Pin Count       PIN     2	Lower Pins Pin Decal: Pin Count: PIN  2
	Box Parameters Min Width: Min Height 200 • 200 •	OK	Cancel Help

1.9 Make the following changes so that your Decal Wizard is filled in as shown below: In the Pin Spacing area, type 300 in the Horizontal box and 200 in the Vertical box. In the Left Pins area and Upper Pins area, set the Pin Count to 1. In the Right Pins area and Lower Pins area, set the Pin Count to 2.

CAE Decal Wizard			X
Preview:	Pin Length Horizontal: Vertical: 200 • 200 • 200 •	Left Pins Pin Decal: Pin Count PIN I I	Upper Pins Pin Decat Pin Count PIN 1
	Pin Spacing Horizontal: Vertical: 300 × 200 ×	Right Pins       Pin Decal:     Pin Count       PIN     2	Lower Pins Pin Decat Pin Count: PIN 2 .
	Box Parameters Min Width: Min Height: 200	OK	Cancel Help

When the specified values have been entered, press OK. The CAE Decal window should now look similar to the one below.

Tool Note: You may and out on the decal changes to the pins. may be used to zoor 1. To enter zoom mo W. Use the left mou

1	💕 Part: tutorial:LM2675* - PADS Lo	gic						
	Eile Edit View Setup Tools	Help						
Ì	i D 📽 🖩 💽 හු හු ව 🎗 皆	ダ 🖪 🖪						
	i 📐 🕂 🍬 🗙 😭 🔮 🖆 🖌 🕯	🛿 🎘 🕌 ե	\$\$ 11 11 11 11 11 11 11 11 11 11 11 11 1	O The smar and				
ĺ	Project Explorer 🔹 🕈 🗙	•.				2		
	E- Schematic Sheets					SWP		
	Components							
	- Nets					<sup>4</sup> → INETNAME III 4: TYP=U SWP=0		
	CAE Decals					JE III		
	PCB Decals					WWW		
						<u>H</u>		
						₹ T		
				V1:TYP=U SWP=D NE	NAME 1			V 2:TYP=U SWP=D
				RECEIPTED SWPED	X			
								V J:TYP=U SWP=D
		•			÷	n	LH2675	
					<u>*</u>	*	Free Label 1 Free Label 2	
					NAM	MMN		
					MEL	Ę		
					0=d	0d(		
					Serren swee vertwas	#S:TYP=U SWP=0 NETNAME →		
						5:		
1	Brên : .					-		

1.10 Name and Number the pins. Right-click in an empty area and select the **Select Terminals** menu item. Double-click on the leftmost pin. The Terminal properties dialog will open.

Terminal	Properties	×
Decal:	PIN	Change Decal
Number:	0	
Name:		
Swap class:	0	
Type:	Load	•
<u>OK</u>	Cancel	Help

Type 7 in the **Number** box, **Vin** in the **Name** box, and select **Power** in the **Type:** dropdown box. Then press **OK**. Repeat this process for the remaining 5 pins. Proceeding clockwise, name and number the pins as follows:

Number: 4, Name: FB Type: Undefined

Number: 1, Name: CB Type: Undefined

Number: 8, Name: VSW Type: Undefined

Number: 6, Name: GND, Type: Ground

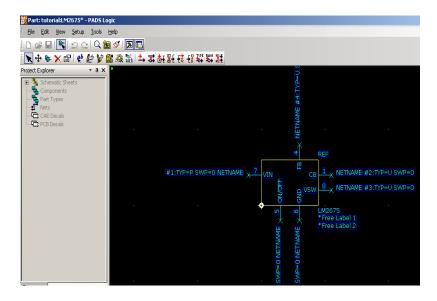
Number: 5, Name: ON/\OFF

Note: For pin 5, the name will appear as ON/OFF. The back-slash is used to specify a logical not and will make a bar appear over the word immediately following it.

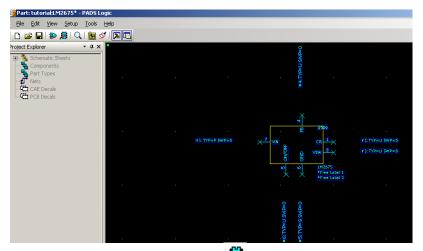
When you are finished naming and numbering the pins, the CAE Decal Editor should look similar to the image below.

The outline size may be modified by selecting the Modify 2D Line icon,  $\mathbf{k}$ , and adjusting the box as needed, and moving the pins to create a more readable symbol. The pins are not connected to the box outline so you will need to adjust the box size then drag the pins to the periphery. Press **Esc** when you are done re-sizing the outline then click

and drag the terminals so that the ends lie on the outline.



1.11 To save the schematic symbol select File > Return to Part. When prompted to keep changes, press Yes. The Part Editor should now look like this:



Click on the Edit Electrical icon,  $\overset{\textcircled{}}{\mu}$ , and select the Pins tab as shown below.

fip: Multiple value	es in one colum	n can be edited at	once by using the E	dit button.		
Pin Group	Number	Name	Туре	Swap	Seq.	Reset
Gate-A	7	] VIN	Power	0	1	<b></b>
Gate-A	1	СВ	Source	0	2	Edit
Gate-A	8	VSW	Source	0	3	
Gate-A	4	FB	Load	0	4	Add Pin
Gate-A	6	GND	Ground	0	5	
Gate-A	5	ONAOFF	Source	0	6	Add Pins
						Renumber
						Copy Paste

You will note from the data sheet that the device actually has eight pins but that pins 2 and 3 are unused. Click the Add Pin button to add pins 2 and 3. Enter 2 and 3 under the **Number** column. Click on both in the **Pin Group** column and change the attribute to **Unused Pin** as shown below.

Pin Group	Number	Name	Туре	Swap	Seq.	Reset
Gate-A	7	VIN	Power	0	1	
Gate-A	1	CB	Source	0	2	Edit
Gate-A	8	VSW	Source	0	3	
Gate-A	4	FB	Load	0	4	Add Pin
Gate-A	6	GND	Ground	0	5	
Gate-A	5	ONAOFF	Source	0	6	Add Pins
Unused Pin 🔄	2					Delete Pins
Unused Pin	3					Deleter IIIs
						Renumber
						Сору
						Paste
						Import CSV

Click OK.

Minimize PADS Logic. At this point you will either create a PCB footprint using the **PCB Decal Wizard** as detailed below in **Section 2**, or create a custom footprint as shown in **Section 3**.

### Section 2 - Create a PCB footprint using the PCB Decal Wizard

Start PADS Layout. Select Start > Programs > ECE Software > Mentor Graphics SDD > PADS 9.1 > Design Layout and Routing > PADS Layout. The PADS Layout welcome screen should appear.



We will now create a PCB footprint, also called a PCB decal in PADS, for the National Semiconductor LM2675M-5 switch mode voltage regulator chip. The relevant datasheet can be found <u>here</u>. (Page 24 contains information on the part's physical dimensions).

2.1 To open the PCB Decal Editor in PADS Layout, select **Tools > PCB Decal** Editor. The PADS Layout window should now look similar to the image below.

Eile	Edit	⊻iew	Setup	Tools	Hel	P																	
🛩 🕻	<b>3</b>   (+	l) Top			•	P	S	4	9 🔛	1	$\Omega$	2	Q	-	2 😒	] ا 🕈	A	-					
roject E	Explore	r		• # ×		,								1	1	1							Ī
∎ €	Layer	s .																					
	Comp PCB D	onents ecals																					
	Nets																						
																					duo	ne	
																					- Y	36	
CProj	ant [				-																		L
	Vindov				•	1																	

2.2 Start the PCB Decal Wizard. Enable the Drafting Toolbar by pressing the state button. From the Drafting Toolbar press the Wizard button: State Control Con

🖉 Decal Wizard		×
Dual Quad Polar BGA/PGA		
Decial     Device type     Orientation     Orientation     Overtical     Overtical     Overtical     Overtical     Overtical     Overtical     Overtical	Default	Preview
Height (H): 50 Center C Pin 1		
Pins	Placement outline	
Pin count: 8 Clockwise © CCW	Width: 400 •	2
	Height: 450	
Diameter: 60  Drill diameter: 35	Mask over(under)size	3 6
Pin pitch (P): 100 💉 🔽 Plated	Solder: 0 ·	
Row pitch: 300		<b>4</b>
Pin 1 shape © Square © Circle © Square © Circle		View from bottom side
Square pins Corner type: 90 Degrees Radius:		Display Colors Active layer: <all layers=""></all>
C Decal Calculator		Show dimensions
Package type: Ceramic Flat Package	Protrusion variation:	Nominal  All C Used in calculations
W Dimensions Min	Max	
	Decal name:	Celculate
	the stan Origin, W and Mas	ulate decal pins will get measurements and locations according to dard. The following parameters may get updated: Drientation, //idth, Length, Row Pitch, Numbering direction, Placement outline k over(under)size. rd decal name will be generated

2.3 First, update the Units parameter in the bottom left corner of the window to **Inches** to match the datasheet; we can choose between Mils, Inches or Millimeters. A Mil is .001 inch and is a convenient dimension for layout when using inches as the measurement standard. **In this case we will use inches**, so use the parameter within the parentheses.

<u>CAUTION!</u> Data sheets must be carefully examined to make sure they are being correctly interprete dimensions are indicated as inches (millimeters), implying that the measurements shown are in inches However, if you look under the pictures you see a note which states:

# CONTROLLING DIMENSION IS MILLIMETE VALUES IN [] ARE INCHES

Be careful which parameter you use, and that you are consistent in use.

2.4 In the **Decal Calculator** box, change the Package type to **SOIC** which stands for Small **O**utline Integrated Circuit. This is an illustration of the type of device which would be used with the footprint you are creating. Note the picture and the abbreviations as they relate to the dimension table.

We will be creating a decal for a surface mount device, abbreviated SMD, 8-pin Small Outline Package. Experiment with the parameters and note how the **Preview** changes. For our device, update the device parameters as follows:

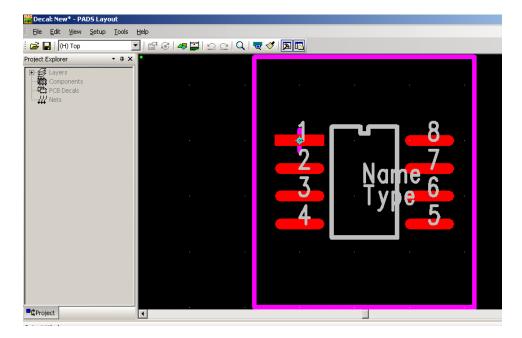
Device type – SMD Orientation – Vertical Height – 0.05 Origin – Pin 1 Pin count – 8 Numbering direction – CCW Width – 0.02 Pin pitch(P) – 0.05 (note: P refers to the picture in the Decal calculator window) Row pitch – Center to center - 0.236 (note: this should be based on the lead span, A2 in the Decal Calculator package, and should be approximately (A2 + 0.05 inches) to allow for solderability. If you designate the Center to Center to be the lead span this should automatically allow the required pad exposure.) Pin 1 shape – Rectangle (note: this pin differentiation aids in device orientation. ) Pin shape - Oval Rectangular pins – 90 degrees

Leave Placement outline and Mask over(under)size as default values.

PM1	Decal Wizard		
	Dual Quad Polar BGA/PGA		
	Decal Device type O Through hole • SMD • Vertical • Horizontal	Default	sw.
	Height (H): 0.05		
	Pins	Placement outline	
	Pin count: 8 Numbering direction	Width: 0.4	
		Height: 0.45	
	Width: 0.02  Length: 0.09	Mask over(under)size	
	Pin pitch (P): 0.05	Solder: 0	
	Row pitch	Paste: 0	
	Center to Center Value: 0.236		
	Pin 1 shape		
	Rectangle C Oval     O Rectangle      O Val		
	Rectangular pins	Dis	sola
	Corner type: 90 Degrees 💌 Radius: 0 👘		
	Decal Calculator		
	Package type: SOIC	Protrusion variation: Nominal	
		Protrusion variation: Nominal	

The Decal Wizard window should resemble the one below.

2.5 Press OK. You will now return to the PCB Decal Editor which should look like the image below.



2.6 Save the PCB footprint to your user library. Select **File > Save Decal**. In the Save PCB Decal to Library dialog, select N:\(**Pads Library**) for the library and type **LM2675** in the Name of PCB Decal field.

Save PCB Decal to	Library		×
Library:			
N:\Mentor\Libraries\tu	torial		▼
Name of PCB Decal:	NEW		
		ОК	Cancel

Press OK. If it asks if you want to create a new Part Type, click No.

To exit the PCB Decal Editor, select File > Exit Decal Editor to return to PADS Layout. Select File > Exit to close PADS Layout.

Return to **PADS Logic** and click on the Edit Electrical icon, *i*, click the PCB Decals tab and select the footprint you just created by highlighting it in the **Unassigned Decals** box and clicking the **Assign** button.

Click **Check Part** to make sure no errors or warnings are found. If there are none then click **OK** to close the **Edit Electrical** window. **Select File > Save to save the** 

part. Be sure to fill in the Name of Part with the part name as shown below.

M	Save	Part and Gate Decals As	×
	Library:		
	N:\Mer	ntor\Libraries\new_tutorial	
	Name o Names	of Part: LM2675R2	
	Gate	CAE Decal 1	_
	А	LM2675R2	
	E	dit OK Cancel Help	

**Select File > Exit Part Editor** to return to the schematic window.

You can now use the LM2675 part type in your PADS Logic and PADS Layout designs.

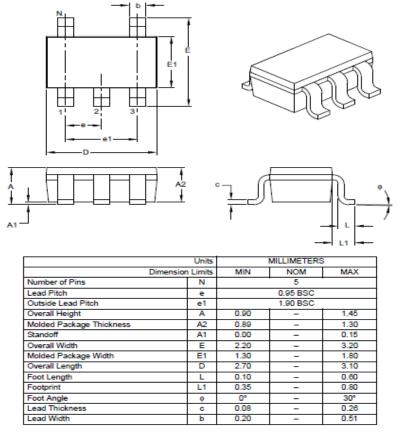
#### Section 3 - Custom footprint creation

Although the PADS libraries contain a large number of footprints, there may be instances when you need to create a unique footprint. First, create the schematic representation as outlined in Part 1.

- 3.1 Open PADS Layout Start > Programs > ECE Software > Mentor Graphics SDD > PADS 9.1 > Design Layout and Routing > PADS Layout.
- 3.2 Open the Library Manger File > Library and either create a library or make sure your desired library is accessible, then close the Library Manager window.
- 3.3 Open the PCB Decal Editor **Tools** > **PCB Decal Editor.** The window will open as shown below.

🍰 🔜   (H) Top	-	-   🖆	18	4	<b>7</b>		<u>c</u> l	Q	-0	2 😒	] ا 🕈	<b>a</b> i	2								
oject Explorer	• å ×	•														·	•				F
Layers Components CB Decals		· ·																			
		· ·																			
		· ·																			
		· ·																			
		· ·																			
		· ·																			
		· ·																			
		· ·																			
		· ·																			
		· ·														¢.	те 30				
		· ·																			
		· ·																			
		· ·																			
		· ·																			
Project		•			_																
tput Window															_		_				

For this design we will use the footprint for a Microchip MCP73811 which is a charge management controller for lithium-ion and lithium-polymer batteries. Its datasheet is found <u>here</u>. The physical dimension is shown below.



Note that the dimensions are in millimeters. The default units for PADS are Mils, or 0.001 inch. To change this go to **Tools > Options**, select the **Global** tab and click **Metric**.

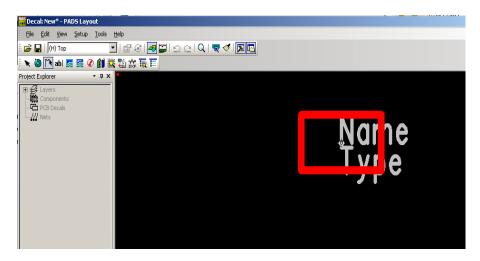
3.4 Select the **Grids** tab and change the Design and Display grid X: and Y: settings to 0.05mm as shown below. Click **OK** to exit the **Options** window

Options			_ 🗆 🗡					
Global De	sign   Routing	Dimensioning Drafting	g Grids					
Design g X: Y: IV Sna	rid 0.05 0.05 ap to grid	Via grid X: 0.635 Y: 0.635 ▼ 0.635	Fanout grid X: 0.635 Y: 0.635 ▼: 0.635 ▼: Snap to grid					
	F	Snap to test point grid	l i i i i i i i i i i i i i i i i i i i					
	Display grid	Hatch gri	id					
	× I	0.05 Copper:	0.254					
Y: 0.05 Keepout: 2.54								
		Radial Move Setup	]					

The outline for the body of the part is designated by dimensions D – overall length = 3.10 (max), and E1 – Molded package width = 1.80(max). To draw this rectangle select the

**Drafting Toolbar** and then select the **2D Line** button. Draw this box about the origin, by clicking on points (-1.55, -.9),

(1.55,-.9), (1.55,.9), (-1.55,.9), then right-clicking and selecting **Complete**. You should have a rectangle as shown below.



3.4 Note that the outline is currently on the Top layer as shown in the layer drop-down

box, (H) Top I in the upper left corner of the Layout window. This box should be on the silkscreen layer to show the part outline without being an unwanted conductive surface. Press the **Esc** key to leave any tool that may be selected, right-click in the Decal window and choose **Select Shapes**. Click the box and note that its color

changes from red to white to indicate that it has been selected. . Right-click and select Properties. Then window should appear as shown below.

👑 Drafting Pr	operties				
Туре:	2D Line		•		
Width: 0.254	Scale factor:	Arc approxima 0.0127	ation error:		5
Rotation:	Track clearanc	e:			
0.000	n/a	🔲 Solid cop	per		Net
Layer:		🗖 Bridge			
Тор					-
Assign Net by Net:	y Click and then s	elect a design (	object in the	workspace.	7
Restrictions Placemen Comp Compone Select All	oonent height		_	aint	ane area
ОК	Apply	Cancel	Help	Assign Ne	et by Click

We want this shape to serve as the body outline on the silkscreen and to also change the line thickness to a thinner outline, so change the Width to .15, the Layer to Silkscreen Top and click OK.

3.5 We will now add the terminals to the outline. Click the Terminal button. Click **OK** on the **Add Terminals** window to accept default. The window will close. Place the cursor somewhere close to the outline and click. Press the **Esc** key to deactivate the terminal mode. Right-click in the layout area and choose **Select Terminals**. Click on the terminal; it changes to solid white. You can move it by holding the left mouse button and dragging it to the desired location.

While it is selected, right-click the mouse and choose Pad Stacks. The window shown below will pop up. This window allows you to change the pad parameters to match those required by your device.

Pad Stack Propertie	s for Pin	×
Pin No: Plated: 1 (P)	Sh. Sz. Layer: CNN 1.524 < Mounted Side CNN 1.524 <inner layers=""> CNN 1.524 <opposite side<="" th=""><th>Applu</th></opposite></inner>	Applu
	Add Delete	Help
	Assign to all selected pins	Preview:
Parameters		+
Drill size: 0.889	Slot Paramet Slotted Length: Orientation:	

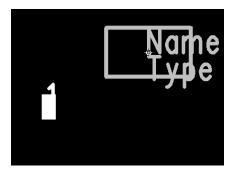
3.6 Examine the part diagram and note that the lead width, designated "**b**", is 0.51mm (max) and the Footprint, designated "**L1**", is 0.80mm (max). Remember that we need the pad to extend beyond the end of the lead for soldering purposes, so add an extra .12mm to the footprint length, so we end up with a pad dimension of 0.51mm x 0.92mm. First

click the rectangular pad,  $\square$ , in the **Parameters** box. Change the following so that the Pad Stack Properties for Pin is as shown below:

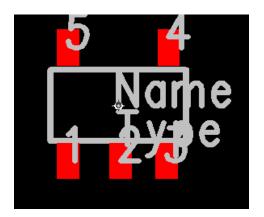
Drill size:	-	0 (no hole)
Width:	-	0.51
Length:	-	0.92
<b>Orientation</b> :	-	90

🚟 Pad Stack Properties	s for Pin	×
Pin No: Plated: 1 (P)	Sh. Sz. Layer: RNN 0.51 <mounted side=""> CNN 1.524 <inner layers=""> CNN 1.524 <opposite side<="" td=""><td>OK Apply Cancel</td></opposite></inner></mounted>	OK Apply Cancel
	Add Delete	Help
	Assign to all selected pins 🗖	Preview:
0.51 🗧 90	entation: 1.000 ÷ set: • Slot Parameter:	+
Corner type: Ra 90 Degrees <b>D</b> Drill size: 0	Orientation:	0 × v 0.000 × v 0 × v

Click OK. The layout area should resemble that shown below.



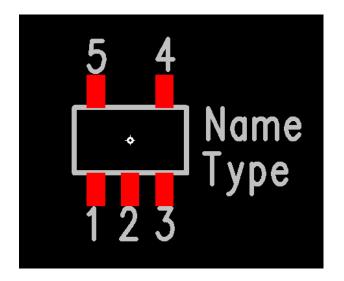
3.7 Copy the pad by typing CTRL + C. Paste four more pads in the general locations indicated by the part layout by typing CTRL + V, moving the outline as desired, and clicking the left mouse button to place. Your part layout should resemble that shown below.



3.8 The pads must now be precisely placed for proper alignment. Use the X-Y coordinate indicators shown in the bottom right corner of the display. Remember that we drew the chip outline to match the body of the device and centered about the origin. Move pin 2 so that the center of the pad lies on the X-coordinate of 0 and the top horizontal edge lies along the midpoint of the body outline, corresponding to the left side of the L1 dimension of the device layout.

The "e" dimension from the device layout is the center-to-center, or pin pitch measurement which is 0.95mm. Select pin 1 and drag it so that the X-coordinate is -0.95; select pin 3 and center it on 0.95, aligning the top horizontal edge on the midpoint of the body outline. Position pins 4 and 5 to the X-coordinates of 0.95 and -0.95 respectively, aligning the bottom horizontal edge of both with the midpoint of the body outline.

To move the text, change the selection mode to **Select Text/Drafting** and drag the text as desired . To move the pin numbers change the selection mode to **Select Terminal Name/Number**. Your device should resemble that below.



3.9 Save the part by clicking File > Save Decal and entering the library destination and name of the pcb decal as desired, similar to that shown below and click OK..

Save PCB Decal to	Library	×				
Library:						
N:\Mentor\Libraries\tutorial						
Name of PCB Decal:	SOT-23_MCP73811					
		ancel				

Click **Yes** to create new Part Type. Click **OK**, ignore the error message, re-enter the PCB decal name, and click **OK**.

Click **File > Exit Decal Editor** to exit. Exit Layout and Start PADS Logic. Refer to Section 3 to create a schematic picture for the part.