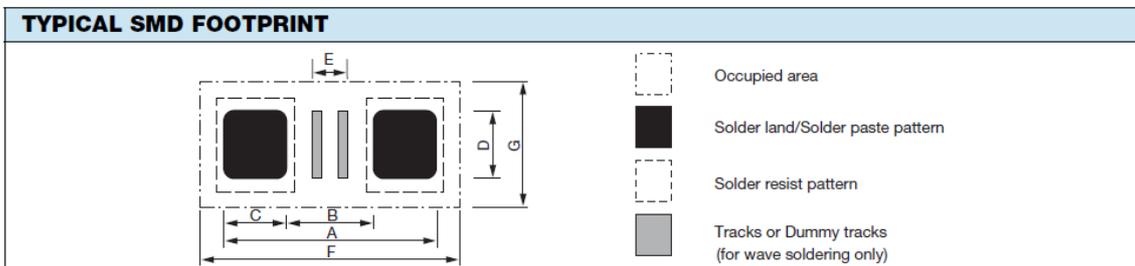


Definition og anvendelse af alternative footprints

How to Define and use Alternative Footprints.

Many PCB designs today have the need for alternative footprints for the same part. There may be a different manufacturing process that requires you to place a different footprint onto the PCB. The part number, footprint size will be identical but you may need a different pad size, soldermask opening or larger amount of solder paste to be applied depending on the process. Cadence OrCAD and Allegro PCB Editor use a property called ALT_SYMBOLS to control what are acceptable alternative footprints. The App note describes the setup and usage model for this property.

In this example we have an 0805 resistor that requires two different footprint definitions. The first uses a standard IPC-7351-A nominal 0805 footprint which uses a reflow process. The alternative is when this part is placed on the bottom side of the PCB. This is because the PCB is a mixed technology board, it uses both through hole and surface mount parts, so for surface mount parts on the bottom side of the PCB we need an 0805 footprint that is suitable for a wave flow manufacturing process. You can see from the screenshot below the pitch between the pads is slightly different but there is also a requirement for tracks or dummy tracks when the parts are wave soldered. The difference between the two footprints is pin pitch and I have added a route keepout area between the pads to allow 0.36mm area for route to go through. I have also added a copper area (which can be deleted if actual tracks need to route through) to cater for the dummy track items.



REFLOW SOLDERING									
SIZE	FOOTPRINT DIMENSIONS IN mm							PROCESSING REMARKS	PLACEMENT ACCURACY
	A	B	C	D	E	F	G		
0201	0.65	0.23	0.21	0.30	n/a	0.90	0.60	Reflow or hot plate soldering	± 0.05
0402	1.50	0.50	0.50	0.50	0.10	1.75	0.95		± 0.15
0603	2.30	0.70	0.80	0.80	0.20	2.55	1.40		± 0.25
0805	2.80	1.00	0.90	1.30	0.40	3.05	1.85		± 0.25
1206	4.00	2.20	0.90	1.60	1.60	4.25	2.25		± 0.25
1210	4.00	2.20	0.90	2.50	1.60	4.25	3.15		± 0.25

WAVE SOLDERING									
SIZE	FOOTPRINT DIMENSIONS IN mm							PROPOSED NUMBER AND DIMENSIONS OF DUMMY TRACKS	PLACEMENT ACCURACY
	A	B	C	D	E	F	G		
0603	2.40	1.00	0.70	0.80	0.20	3.10	1.90	1 x (0.20 x 0.80)	± 0.10
0805	3.20	1.40	0.90	1.30	0.36	4.10	2.50	1 x (0.30 x 1.30)	± 0.15
1206	4.80	2.30	1.25	1.70	1.25	5.90	3.20	3 x (0.25 x 1.70)	± 0.25
1210	5.30	2.30	1.50	2.60	1.25	6.30	4.20	3 x (0.25 x 2.60)	± 0.25

Applying the ALT_SYMBOLS property in OrCAD Capture.

In OrCAD Capture locate the part that requires the alternative footprint and then either double click the part or single click the part and the RMB (right mouse button) Edit Properties to invoke the Property Editor. For this example the property is applied to R1-R8.

The default property editor window is shown. You can see the Current properties like PCB Footprint and Value. Change the Filter by from Current properties to Cadence-Allegro. There is now a visible property called ALT_SYMBOLS. This is the property we populate with the alternative footprint value. In this case the value is 0805_new_ws.

	A	B	C
	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1
Color	Default	Default	Default
Designator			
Graphic	RES.Normal	RES.Normal	RES.Normal
ID			
Implementation			
Implementation Path			
Implementation Type	<none>	<none>	<none>
Location X-Coordinate	570	570	570
Location Y-Coordinate	140	180	220
Name	INS2976	INS3042	INS3076
Part Reference	R1	R2	R3
PCB Footprint	0805_new	0805_new	0805_new
Power Pins Visible	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Primitive	DEFAULT	DEFAULT	DEFAULT
Reference	R1	R2	R3
Source Library	C:\EMA\CIP-EISCHEM	C:\EMA\CIP-EISCHEM	C:\EMA\CIP-EISCHEM
Source Package	RES	RES	RES
Source Part	RES.Normal	RES.Normal	RES.Normal
Value	10K	10K	10K

New Row... Apply Display... Delete Property Filter by: Cadence-Allegro

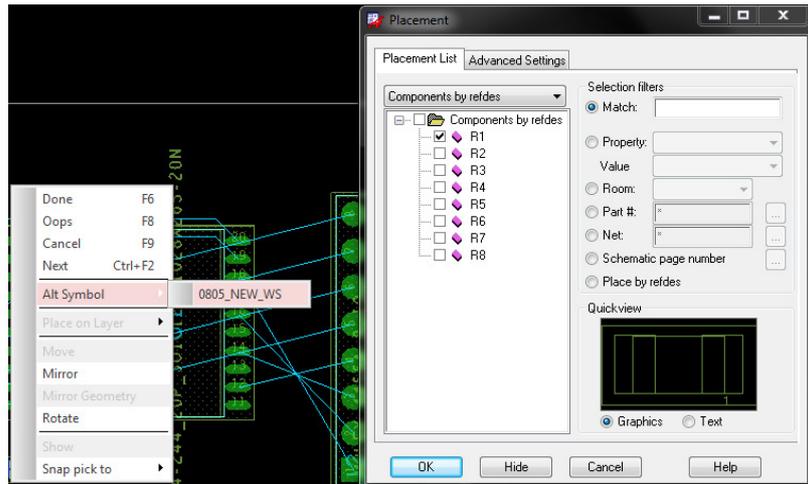
	A	B	C	D
	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1	SCHEMATIC1
ALT_SYMBOLS	0805_new_ws	0805_new_ws	0805_new_ws	0805_new
BOM_IGNORE				

Once the property is there save the design and netlist to PCB Editor as you would normally. The only point to note is that the ALT_SYMBOL is a default property that is transferred to PCB Editor automatically because it exists in the allegro.cfg file (located <your_install_dir>\tools\capture folder) as a Component Definition Property. If it is not you would need to add the ALT_SYMBOLS=YES to the ComponentDefinitionProps section in the allegro.cfg file as shown below.

```
[ComponentDefinitionProps]
ALT_SYMBOLS=YES
```

Changing components to use the Alternative Footprint.

Once the netlist has been imported you place components as you would normally. The default footprint used for the resistors is the 0805_new because that is the value for the property PCB Footprint. If you want to use the alternative footprint you can change this whilst placing the components using the RMB – Alt Symbols function.

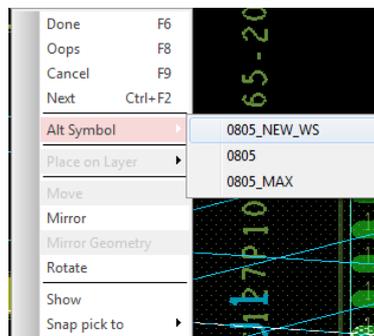


You can also change which footprint you use after placement by either using Edit – Move then select the part and using RMB – Alt Symbol or using the Placement Application Mode and selecting the part with the LMB (left mouse button) and then RMB – Alt Symbol.

The Alt_Symbol property allows users to have multiple alternatives in the list. To do this use the property editor to add the property value as you did earlier but this time add the alternatives as a comma separated list, for example:-

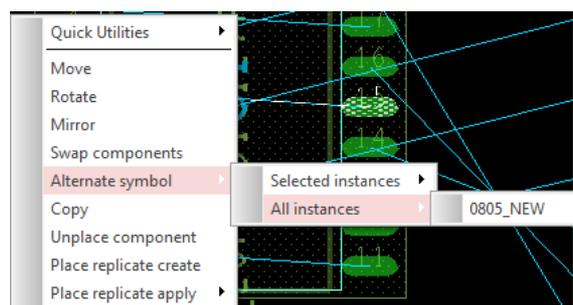
	A
	+ SCHEMATIC1 : PAGE1 : R1
ALT_SYMBOLS	0805_new_ws,0805,0805_max

Once the design has been netlisted you can place the part as described in the previous section of this App note but this time when you use the RMB – Alt Symbol there is a selectable list to choose from that matches the list in the property editor in Capture.



Updating all instances in one step.

There is a feature in the Placement Application Mode that allows users to change all the footprints of a certain type to a specific alternate. To do this hover over the relevant symbol in PCB Editor and then use RMB – Alternate Symbol – All instances and select the relevant alternate footprint.



Using Capture CIS and ALT_SYMBOLS.

The ALT_SYMBOLS can be used in a CIS Database. You just need to add the property and relevant alternative footprints to each part much in the same way you would add any other property to a CIS Database. Again a comma separated list is used. The database property needs to be mapped to Alt_Symbols and transferred to the design.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Nordcad Systems A/S is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Nordcad Systems A/S makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Nordcad Systems A/S expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.