

Title: Connection Symbol Properties

Product: OrCAD Capture and Allegro DE CIS

Summary: Explains how to short some pins with identical nets on a symbol and how to define not connected footprint pins on schematic symbols.

Author/Date: Waller Marco/ 25.2.2010

Table of Contents

1	Introduction	2
2	Shorten same net pins.....	2
2.1	Edit Part.....	3
2.2	Add the Pack Short property	4
2.3	Making Pin invisible	5
3	Define pins which on the Footprint exist but are not connected	7
3.1	Edit Part.....	7
3.2	Add the NC property	8

1 Introduction

The Pin Number on a schematic symbol has to match with the pin number on the footprint.

Otherwise you get an error when you create the netlist.

If you have several pins on a symbol with the same net, you may want to display the pin only once on the schematic symbol.

The other pins you can shorten with the PACK_SHORT property. Then you can ignore some pins, so that they're not visible on the schematic –symbol.

2 Shorten same net pins

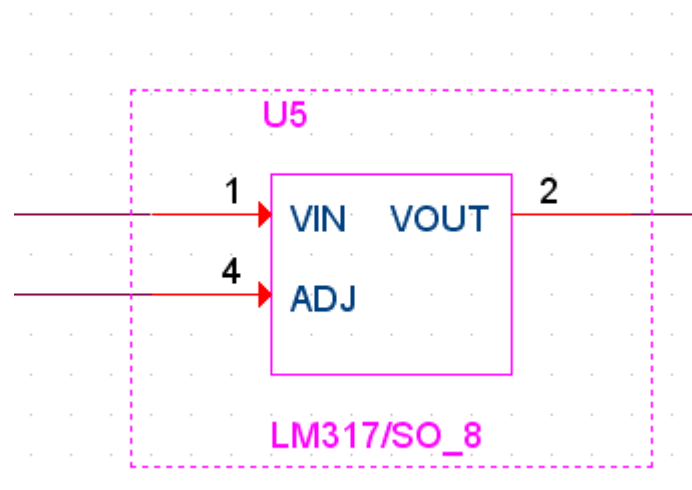
A LM317 with several outputs is used as an example. The component has 4 V_{OUT} pins 2,3,6 and 7. These pins are connected together.



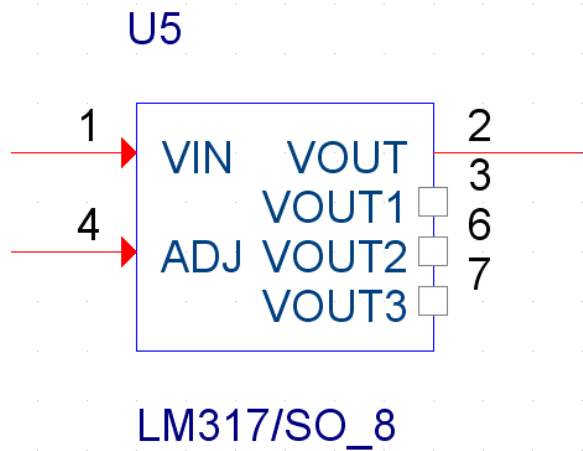
SOIC-8
D SUFFIX
CASE 751

Pin	
1.	V_{in}
2.	V_{out}
3.	V_{out}
4.	Adjust
5.	N.C.
6.	V_{out}
7.	V_{out}
8.	N.C.

At the end you like to have a schematic-symbol like following:

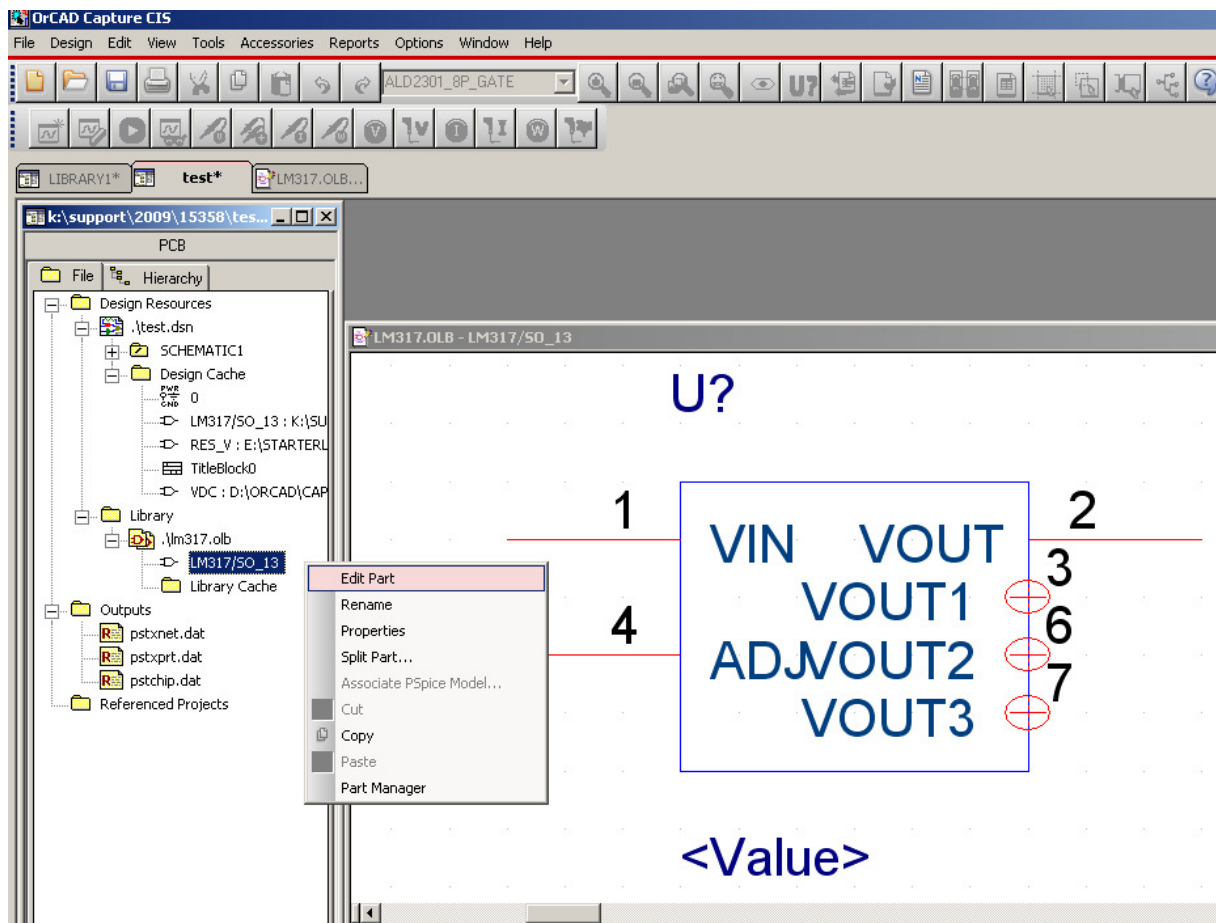


First, on the symbol you have to place every pin:



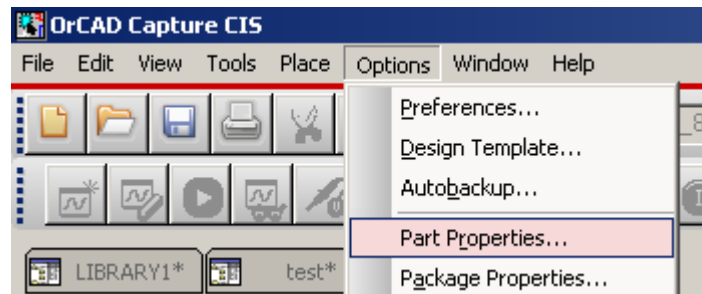
2.1 Edit Part

Select the symbol on the library, click edit part:



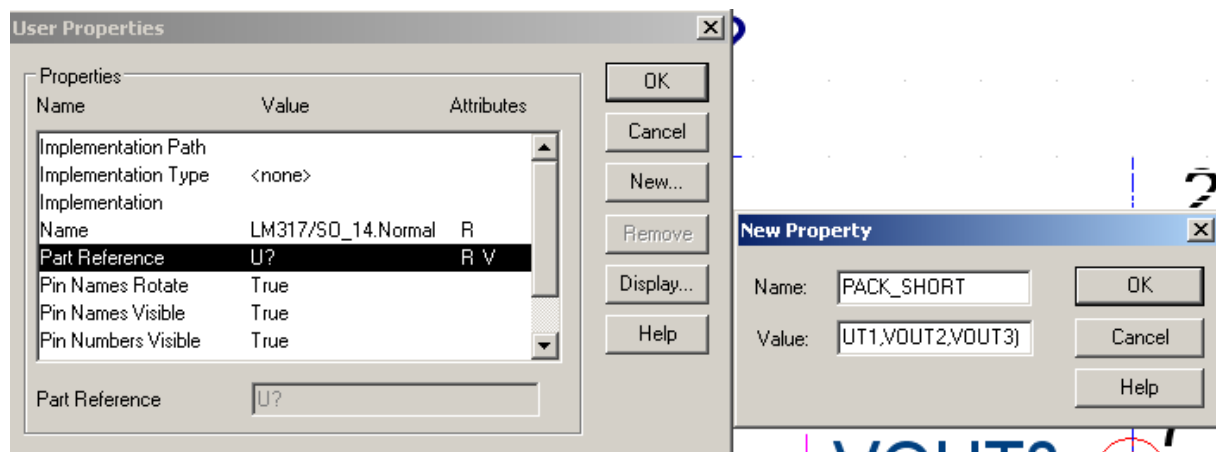
2.2 Add the Pack Short property

Then on the menu-item Options->Part Properties you can add properties to the symbol.



Add the property PACK_SHORT and add the Pin's which you want to shorten.

The Value which you have to enter in this case is: (VOUT,VOUT1,VOUT2,VOUT3)



The syntax is following:

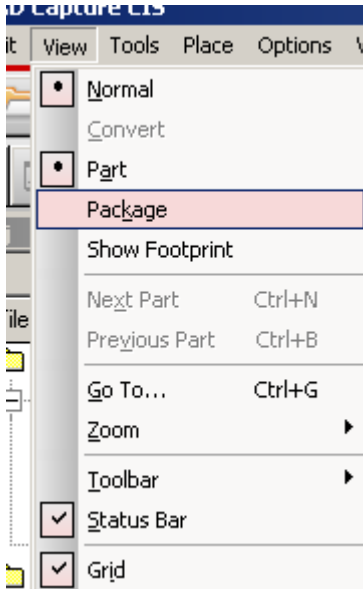
Syntax:

`PACK_SHORT=(<group1>)(<group2>)[<group3>]`

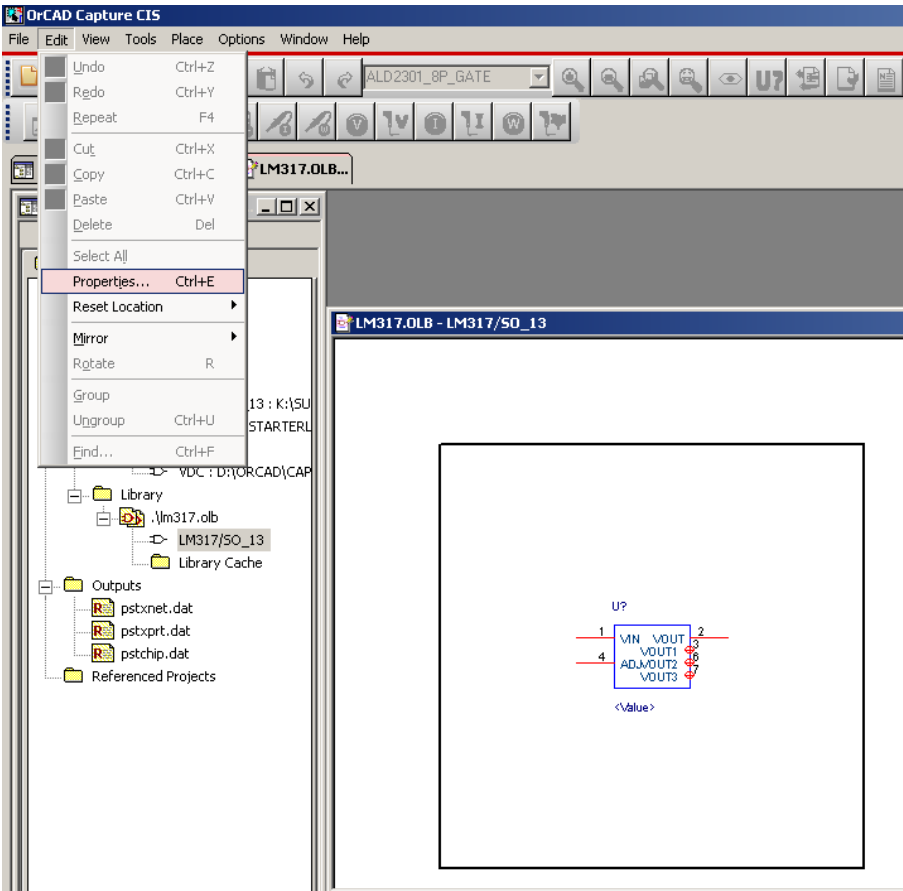
Where: <group> indicates (logicPin1, logicPin2 ... [logicPinN])

2.3 Making Pin invisible

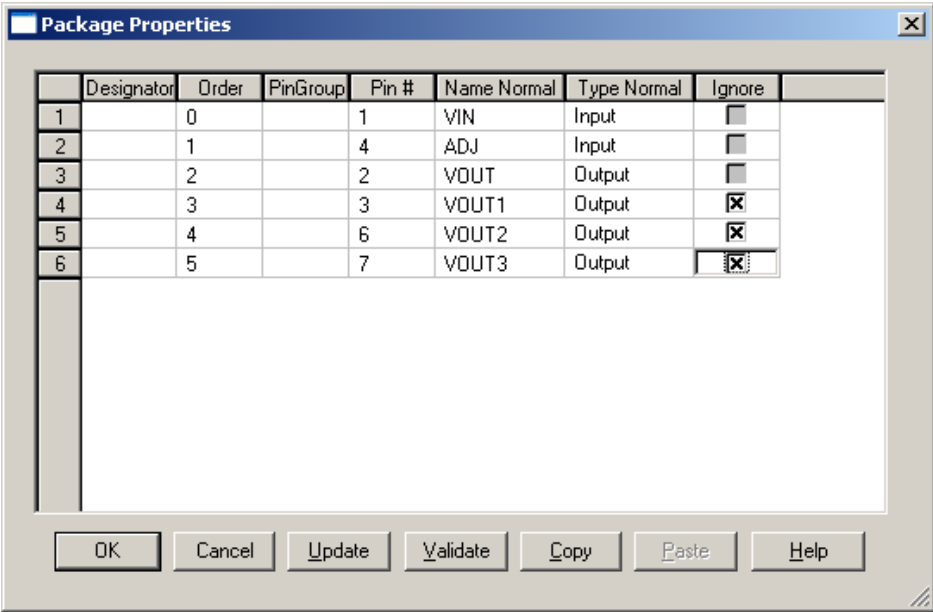
Change The View Type to Package:



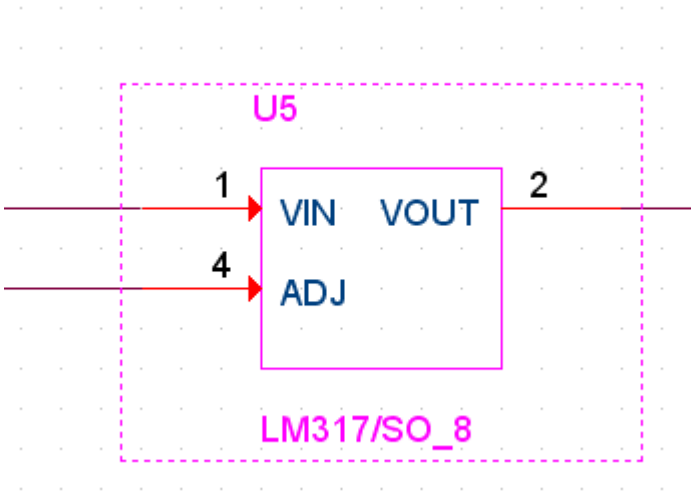
Select Edit->Properties:



Change the Pins which you don't want to see on the symbol to Ignore:



And at the end your symbol looks like following:

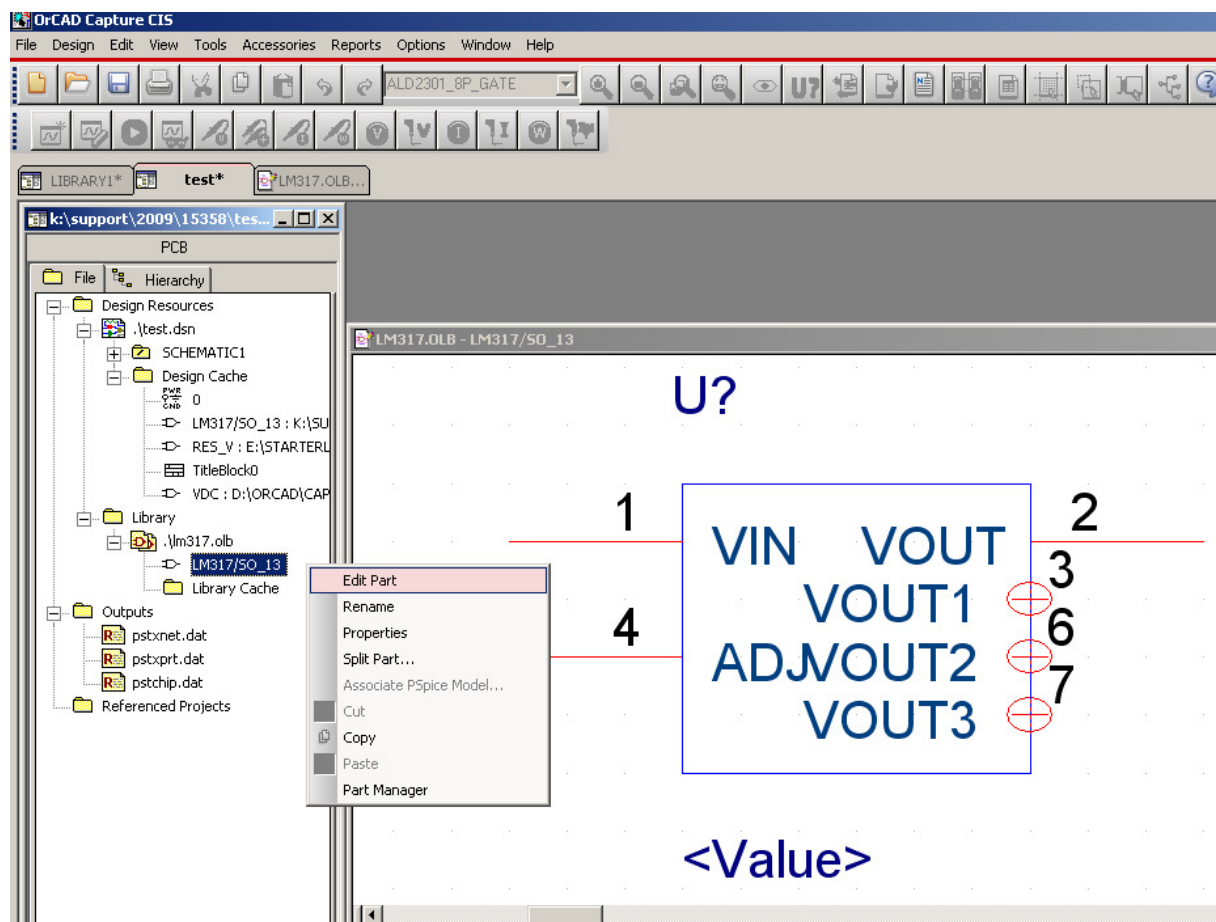


3 Define pins which on the Footprint exist but are not connected

If you have some existent pins on a footprint which are not connected, you have to define them on the schematic symbol. Otherwise you get error when you create a netlist. You can define these pins as not connected on the symbol.

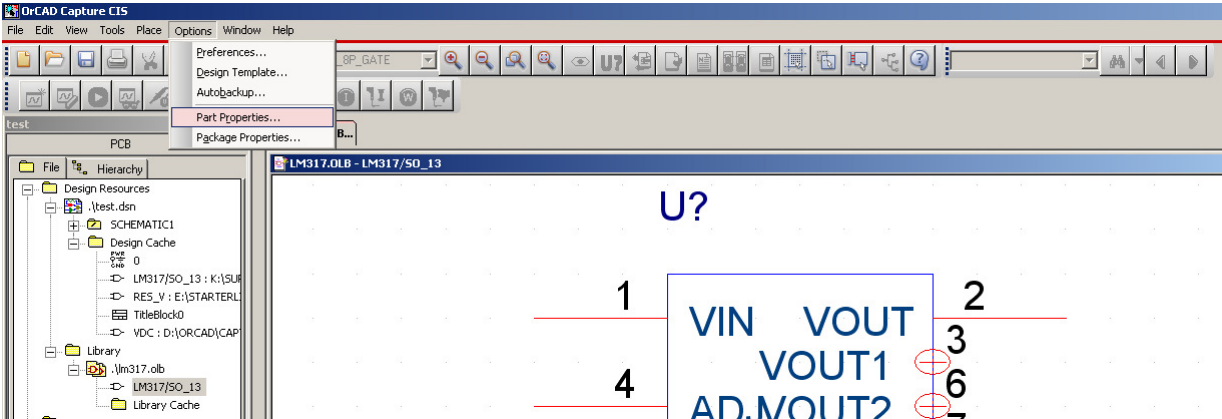
3.1 Edit Part

Select the symbol on the library, click edit part:



3.2 Add the NC property

Then on the menu item Options->Part Properties you can add properties to the symbol.



Add the property NC and add the Pin's which you want to define as not connected:

