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.

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: