EAGLE Help

PIN

Function

Defines connection points for symbols.

Syntax

PIN 'name' options \*..

Mouse

Right button rotates the pin.

See also NAME, SHOW, CHANGE

Options

There are six possible options:

Direction

Function

Length

Orientation

Visible

Swaplevel

Direction

The logical direction of signal flow. It is essential for the Electrical Rule Check (ERC) and for the automatic wiring of the power supply pins. The following possibilities may be used:

NC not connected

In input

Out output (totem-pole)

I/O in/output (bidirectional)

OC open collector or open drain

Hiz high impedance output (e.g. 3-state)

Pas passive (for resistors, capacitors etc.)

Pwr power input pin (Vcc, Gnd, Vss, Vdd, etc.)

Sup general supply pin (e.g. for ground symbol)

Default: I/O

If Pwr pins are used on a symbol and a corresponding Sup pin exists on the schematic, nets are connected automatically. The Sup pin is not used for components.

Function

The graphic representation of the pin:

None no special function

Dot inverter symbol

Clk clock symbol

DotClk inverted clock symbol

Default: None

Length

Length of the pin symbol:

Point pin with no connection or name

Short 0.1 inch long connection

Middle 0.2 inch long connection

Long 0.3 inch long connection

Default: Long

Orientation

The orientation of the pin. When placing pins manually the right mouse button rotates the pin. The parameter "orientation" is mainly used in script files:

R0 connection point on the right

R90 connection point above

R180 connection point on the left

R270 connection point below

Default: R0

Visible

This parameter defines if pin and/or pad name are visible in the schematic:

Off pin and pad name not drawn

Pad pad name drawn, pin name not drawn

Pin pin name drawn, pad name not drawn

Both pin and pad name drawn

Default: Both

Swaplevel

A number between 0 and 255. Swaplevel = 0 indicates that a pin can not be swapped with another. The allocation of a number greater than 0 indicates that a pin may be swapped with any other in the same symbol with the same swaplevel number. For example: The inputs of a NAND gate could be allocated the same swaplevel number as they are all identical.

Default: 0

Using the PIN Command

The PIN command is used to define connection points on a symbol for nets. Pins are drawn onto the Symbols layer while additional information appears on the Pins layer. Individual pins may be assigned various options in the command line. The options can be listed in any order or omitted. In this case the default options are valid.

If a name is used in the PIN command, it must be enclosed in apostrophes. Pin names can be changed in the symbol edit mode using the NAME command.

Automatic Naming

Pins may be automatically numbered in the following way. In order to place the pins D0...D7 on a symbol, the first pin is placed with the following command:

PIN 'D0' \*

and the location for the other pins defined with a mouse click for each.

Predefine options with CHANGE

All options may be predefined with CHANGE commands. The options remain in use until edited by a new PIN or CHANGE command.

The SHOW command may be used to show pin options such as Direction and Swaplevel.

Pins with the same Name

If it is required to define several pins in a component with the same name, the following procedure can be used:

For example, suppose that three pins are required for GND. The pins are allocated the names GND@1, GND@2 and GND@3 during the symbol definition. Then only the characters before the "@" sign appear in the schematic.

It is not possible to add or delete pins in symbols which are already used by a device because this would change the pin/pad allocation defined with the CONNECT command.

Pin Lettering

The position of pin and pad names on a symbol relative to the pin connection point can not be changed, nor can the text size. When defining new symbols please ensure their size is consistent with existing symbols.