

Symbols and their definitions in PCB Editor

Introduction

PCB Editor uses multiple symbol files for various functions within the PCB Editor tools. This technical note describes the different types and when to use them in PCB Editor. It also describes how symbol files are created, used and updated within the Cadence PCB Editor hierarchy.

Type of Symbol: Format Symbol

File extension: OSM (fOrmat SyMbol)

Used for: Drawing blanks, notes

Description: Format Symbols have no electrical or mechanical significance and have no electrical or mechanical pins. Format symbols are comprised of lines and text and have no reference designator.

Type of Symbol: Mechanical Symbol

File extension: BSM (B? SyMbol)

Used for: Mounting holes, fiducials

Description: Mechanical Symbols have none, or more, pins but any pins will have no pin numbers and the symbol will not have a reference designator.

Type of Symbol: Package Symbol

File extension: PSM (Package SyMbol)

Used for: Component Footprints, anything that needs to be connected

Description: Package Symbols have one or more connection pins, pins that have Pin Numbers, and may have none, or more, mechanical pins, that have no pin numbers for mounting. Package Symbols have a reference designator.

Type of Symbol: Flash Symbol

File extension: FSM (Flash SyMbol)

Used for: Drawing blanks, notes

Description: Format Symbols have no electrical or mechanical significance and have no electrical or mechanical pins. Format symbols are comprised of lines and text.

Type of Symbol: Shape Symbol

File extension: SSM (Shape SyMbol)

Used for: Drawing blanks, notes

Description: Format Symbols have no electrical or mechanical significance and have no electrical or mechanical pins. Format symbols are comprised of lines and text.



Symbol Usage

Below is an overview of how symbols are used within the PCB Editor environment.



For Symbol creation when the symbol type is created the relevant PAD file is stores as a cached copy inside the symbol. The Save command also creates the relevant compiled symbol (that PCB Editor uses).

Symbol Creation



On Import into PCB Editor (from Capture, DEHDL or via File – Import – Logic) the PAD and *SM files are cached into the Board file. Import takes these files directly from the library defined by PSMPATH and PADPATH.



If the Padstack is modified locally in the design users can either Update to Design which only modifies the cached copy of the pad or Save to.... which updates the library defined padstack.





Local cached copies of the *SM and PAD files can be replaced / refreshed directly from the library sources using the following:-

Place>Update Symbols



Finally locally cached PAD and *SM files can be exported from the board files to create a new library. The DRA file is also created ready for user editing.

File>Export>Libraries



nation as one possible om any claims of ims any warranty this implementation is free