



## Introduction

Many users want the ability to see the actual PCB Footprint that is defined for the parts in OrCAD Capture. There are two methods to see the actual PCB Footprint. This app note describes both methods.

### How to setup OrCAD Capture to enable Footprint view.

Close OrCAD Capture down and locate a file called Capture.ini. Depending on the version of software this file can be found in the following directories on your computer: -

Pre 16.6 - <your\_install\_dir\tools\Capture where <your\_install\_dir> is normally c:\Cadence\SPB\_16.x if the default installation is followed.

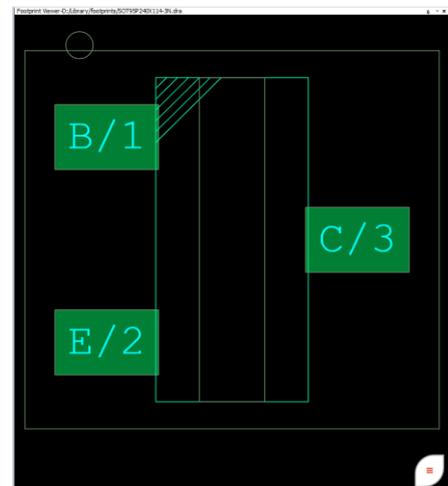
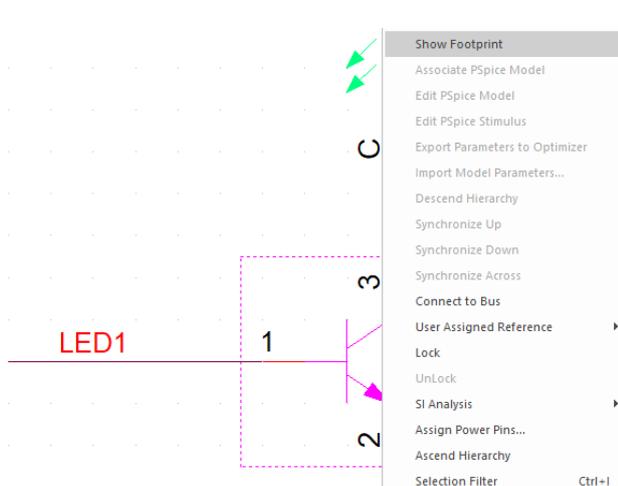
16.6 or later - %HOME%\cdssetup\OrCAD\_Capture\XX.X.X (version release) where %HOME% is your HOME directory. To locate this type %HOME% in the address bar of a Windows Explorer window.

Open the Capture.ini file using a text editor. Look for the following entry, if it doesn't exist then add at the top of the capture.ini file.

```
[Footprint Viewer Type]
Type=Allegro
[Allegro Footprints]
Dir0=Path to PCB Editor Footprints
Dir1=Path to PCB Editor Footprints
```

The entries for Dir0 and Dir1 are the full pathnames to the location where all the PCB Editor Footprints are. This is usually the same as the psmath and padpath settings from PCB Editor. It should be noted that only Cadence PCB Editor footprints are supported for this feature. You can have multiple path entries (Dir2 etc.) but the directory MUST be a mapped network or local drive. You must also have access to both filename.dra and filename.?sm (psm, bsm).

Start OrCAD Capture, select a part and right click > Show Footprint, you will now see the PCB footprint viewer.



You can select pins in the schematic view and the corresponding pin will be highlighted in the footprint view.

## How to make schematic symbols visible in Capture CIS

When using Capture CIS (or DE CIS) you have the ability to see a quick view of your PCB Footprint and schematic symbol before choosing to place the part in the design. The PCB Footprints are defined as per the previous section of this document. For the schematic symbol you need to specify the path for the library either directly using the Part Selector Configured Libraries or the recommended method is using the Part Library Directories setting. This is the preferred method because you only need to specify a directory location rather than adding all the libraries individually.

Close OrCAD Capture down and locate a file called Capture.ini. Depending on the version of software this file can be found in the following directories on your computer: -

Pre 16.6 - <your\_install\_dir>\tools\Capture where <your\_install\_dir> is normally c:\Cadence\SPB\_16.x if the default installation is followed.

16.6 or later - %HOME%\cdssetup\OrCAD\_Capture\XX.X.X (version release) where %HOME% is your HOME directory. To locate this type %HOME% in the address bar of a Windows Explorer window.

Open the Capture.ini file using a text editor. Look for the following entry, if it doesn't exist then add at the top of the capture.ini file.

```
[Part Library Directories]
Dir0=Path to schematic symbols
Dir1=Path to schematic symbols
```

It is important to remember that you only need to specify the path to the directory. The library name itself is not required. Once done restart the Capture CIS, use Place > Database Part. Locate the part required, left click to select it. If the path names defined above point to a valid library location, you will see a quick view of the schematic symbol and the PCB Footprint.

The screenshot displays the OrCAD Capture CIS interface. On the left is a hierarchical tree of component libraries including Flash Memory, MicroController, Op Amp, Power Management, Regulator, TH, Inductors, Misc, Resistors, Switches, Transformers, and more. The main workspace is divided into three panes: a schematic symbol, a PCB footprint, and a parts table. The parts table below shows a list of components with their properties.

Table	PART_NUMBER	Part Type	Description	Value	PCB Footprint	Schematic Part	Number of Pins	Operating Temperature Maximum	Operating Temperature Minimum	Package Size	Package Height	Package Type	Company Part Status	Rated Power	Propagation Delay	Supply Current	Number of Channels	Function	Supply Voltage	Signal to Noise Ratio	Output Voltage	Number of Gates	Output Low Current	Voltage Input High	Number of IO
7	ICs	EIA-000871	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8
8	ICs	PSL-000000	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8
9	ICs	PSL-000000	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8
10	ICs	PSL-000000	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8
11	ICs	PSL-000000	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8
12	ICs	PSL-000000	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8
13	ICs	PSL-000000	EMAGSM04le	C. Memory D	1M47F03Y	SOC127P121	1M47F03V_N3	28	70 C	0 C	SOC-28	3.05 mm	SMD	obsolete	1.0 W		4.5 - 5.5 V		SRAM		2.4 V		2.1 mA	2.2 V	8

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®, SPECTRA®, Verilog®

**Other Trademarks**

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

**NOTICE OF DISCLAIMER:** Parallel Systems 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, Parallel Systems 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. Parallel Systems 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.