**Parallel Systems**

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.

This is controlled using the Capture.ini file. The easiest way to confirm this is to start OrCAD Capture and look at the opening entry in the Session Log. You will see the actual location of the ini file.

```
Session Log
INI File Location:D:\Working\cdssetup\OrCAD_Capture/17.4.0/Capture.ini
```
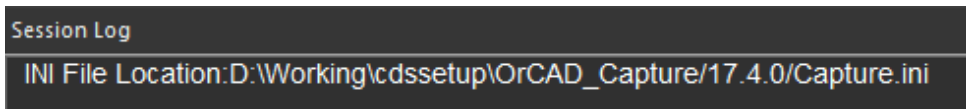
Open the Capture.ini file using a text editor. Look for the following entry, if it doesn't exist then add at the top of the capture.ini file.

```
[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 psmpath 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.
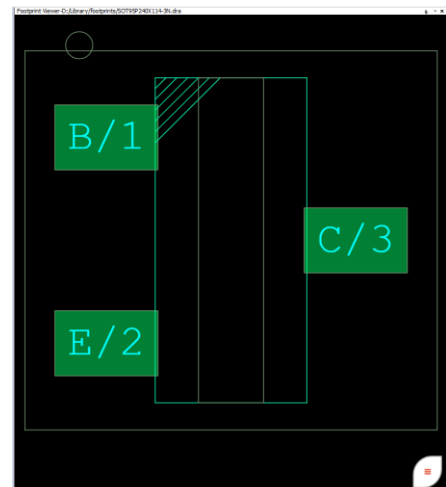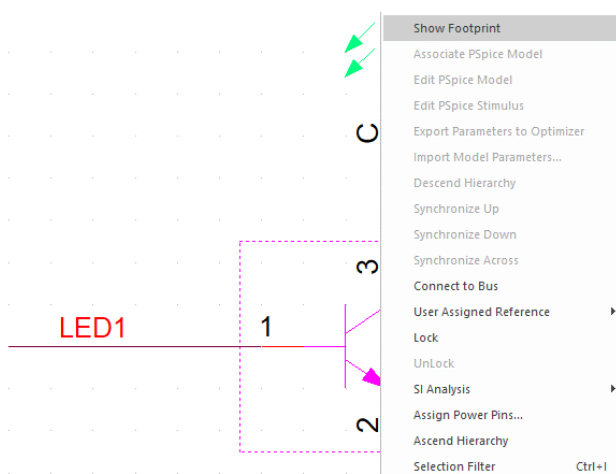

You can select pins in the schematic view and the corresponding pin will be highlighted in the footprint view.

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.

This is controlled using the Capture.ini file. The easiest way to confirm this is to start OrCAD Capture and look at the opening entry in the Session Log. You will see the actual location of the ini file.

Session Log
INI File Location:D:\Working\cdssetup\OrCAD_Capture/17.4.0/Capture.ini
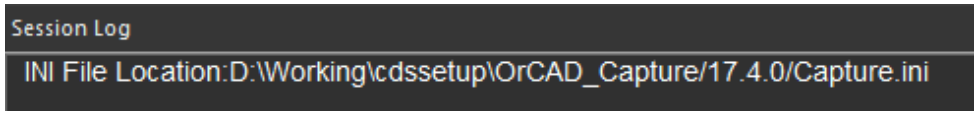
Open the Capture.ini file using a text editor. Look for the following entry, if it doesn't exist then add at the top of the capture.ini file.

```
[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.