



Cadence SPB Free Viewers

Cadence SPB Free Viewers

There are a couple of methods to access the free viewers for Cadence SPB OrCAD and Allegro tools. These will open filename.opj, filename.dsn, filename.brd, filename.dra, filename.mdd. The two options are driven by whether you have a full Cadence SPB installation. This app note describes the two methods.

Option 1 - No Cadence Product Installation.

Go to [Download OrCAD Viewer | OrCAD](#), fill in the registration form:-

OrCAD Viewer Free Download

Complete this form* to download the Cadence® OrCAD® FREE Physical Viewer that allows you to view and plot databases from OrCAD Capture and OrCAD PCB Editor.

*If you do not see a form on this page the content may be blocked by your browser's private browsing.

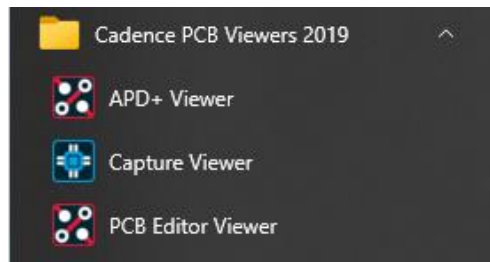
First Name*	Last Name*
Company	
Email*	Phone Number*
Location ▼	

I have read/accept the **Cadence Terms of Use and Privacy Policy.**
*

Get up-to-date information on products and solutions by opting in to receive emails from us. You can unsubscribe at any time.

Submit

You will receive an email with a link to Download the software. Please check any junk/spam folders if you do not see the email. Once downloaded, run the executable which installs the software. Once complete there will be a new entry in the Windows start menu



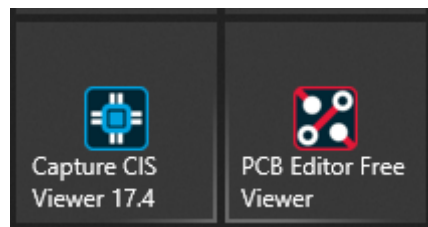
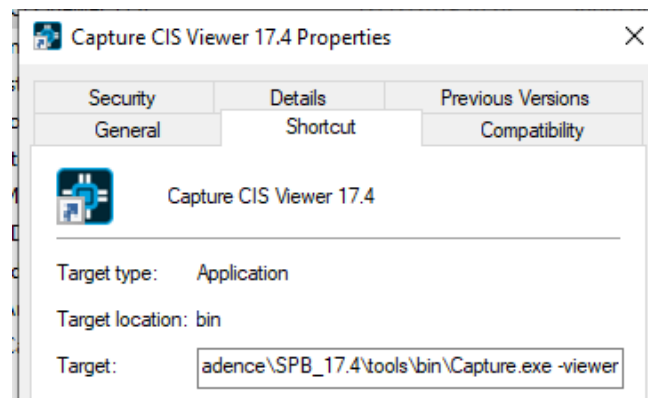
Use Capture Viewer to open filename.opj and filename.dsn, Use PCB Editor Viewer to open filename.brd, filename.dra and filename.mdd.

Option 2 – Full Cadence Product Installation

If you already have the suite of tools installed, you cannot install the free viewers as in option 1. You can however make custom shortcuts using the steps below.

PCB Editor Viewer – Go to C:\Cadence\SPB_17.4\tools\bin and locate the allegro_free_viewer.exe, select and right click – Create Shortcut. You can then put this on the Desktop, Start Menu or Taskbar. Suggest renaming this to be PCB Editor Free Viewer 17.4.

Capture Viewer – Go to Start – Cadence PCB 17.4-2019 – Capture CIS 17.4 and right click – More – Open File Location then use CTRL+C and CTRL+V to paste a new copy of the Capture icon. Rename this to Capture CIS Viewer 17.4. Now right click on the icon and choose Properties and change the Target value so that it has -viewer at the end so C:\Cadence\SPB_17.4\tools\bin\Capture.exe -viewer. Ok this.



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®, SPECCTRA®, 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.