



Introduction

Many Component Manufacturer's share their component data in native formats for use within CAD / CAE tools. This document describes some of the manufacturer's that supply schematic symbol and PCB Footprint information suitable for use with the Cadence Allegro / OrCAD tools. It also has locations for 3D Step models.

Available library data

1. UltraLibrarian offer free OrCAD Capture schematic symbols, PCB Editor footprints and 3d step models direct from their website <http://UltraLibrarian.com>. Login required but you can register for free.
2. SnapEDA offer free OrCAD Capture schematic symbols, PCB Editor footprints and 3d step models direct from their website <http://snapEDA.com>. Login required but you can register for free.
3. Samacsys offer free OrCAD Capture schematic symbols, PCB Editor footprints and 3d step models direct from their website <http://Samacsys.com>. Login required but you can register for free.
4. From within OrCAD Capture choose Place > Search Providers and login using your Cadence account (and your UltraLibrarian account (from point 1 above) and you can search both the Samacsys and UltraLibrarian search engines. This will allow you to place the symbols directly onto the schematic page. The PCB Footprint (and 3d Step model if available) are placed in local directories that are automatically added to the relevant paths (psmpath, padpath and steppath) so you can use the items immediately.
5. Texas Instruments <http://ti.com> have Allegro PCB Footprints available. They are normally found under the specific component page then Quality & environmental data tab. Some parts have Allegro v13.6 footprints which may need to be dbdoctored before use. They also provide files suitable for UltraLibrarian (see point 13 below).
6. Altera <http://altera.com> have Allegro PCB Footprints available. They are normally found under the specific component page. Some parts have older Allegro footprints which may need to be dbdoctored before use.
7. Xilinx <http://xilinx.com/support/download/index.htm> have DE HDL symbols available. Login is required.
8. ST Microelectronics - http://st.com/stonline/stappl/resourceSelector/app?page=fullResourceSelector&doctype=ecad_models_and_symbols provide PSpice models and OrCAD Capture olb files. These are device specific but the link takes you to the search location on ST Micro's website.
9. NXP offer OrCAD Capture symbols and PCB Editor footprints for the LPC800 device range. <http://lpcware.com/content/nxpfile/lpc800-orcad-and-allegro-symbols>. Login is required.
10. Broadcom provide Cadence OrCAD Capture symbols, OrCAD/Allegro PCB Footprints, demo kit board files and module (*.mdd) files of their layouts. These are available from:- <http://community.broadcom.com/welcome>.
11. Analog devices offer Allegro and OrCAD PCB Footprints. Take a look at: - <https://www.analog.com/media/en/technical-documentation/application-notes/AN-1176.pdf>. They also provide files suitable for UltraLibrarian (see point 12 below).
12. Cadence has a starter library which is suitable for both DE HDL and DE CIS with associated PCB Footprints available from <http://cadence.com/products/orcad/pages/downloads.aspx>.

13. <https://www.ultralibrarian.com/products/free-reader> – Provide a software package called Ultra Librarian that the following vendors use which has data for both DE-HDL, DE-CIS and Footprint information for Allegro / OrCAD. Microchip <http://microchip.com> National Semiconductor <http://national.com>, Silicon Labs <http://silabs.com>, Linear Technologies <http://linear.com>, Analog Devices <http://analog.com>, Texas Instruments <http://ti.com>, Renesas <http://renesas.eu> and EXAR <http://exar.com>. The default install is free and allows you to open the source files from the vendors and create footprints and schematic symbols quickly. The full version (cost item) allows more functionality such as the ability to import schematic symbols and PCB Footprints and also unlimited access to their database. UltraLibrarian also offer a paid for service to create schematic symbols and PCB Footprints based on a supplied datasheet. Their website contains full details.
14. Samtec Connectors offer both schematic symbols and PCB Footprints. These are now all available via <http://snapEDA.com>
15. OrCAD Library Builder is a utility for use directly with PCB Editor. This tool designs IPC 7351B standard footprints, 3d Step Models and OrCAD Capture symbols directly from PDF datasheets. You can extract the datasheet table to build your schematic symbol. For more details go to <https://www.parallel-systems.co.uk/orcadlibrarybuilder> or contact sales@parallel-systems.co.uk for further details.
16. Many vendors also supply evaluation boards. The component footprints can be found as part of the evaluation board file (*.brd). To extract the footprints / padstacks from an evaluation board use the File – Export – Libraries command from within PCB Editor, select all items, browse to a directory name then click Export. The relevant *.dra, *.psm, *.bsm, *.osm, *.fsm, *.ssm and *.pad files are exported to the named directory.
17. 3D Step models can be found from most components manufacturers but there is a free website:- <http://3dcontentcentral.com> that has most STEP models.

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.