



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. Texas Instruments <http://ti.com> have Allegro PCB Footprints available. They are normally found under the specific component page. Some parts have Allegro v13.6 footprints which may need to be dbdoctored before use. They also provide files suitable for UltraLibrarian (see point 12 below).
2. Altera <http://altera.com> have Allegro PCB Footprints available. They are normally found under the specific component page.
3. Intersil <http://Intersil.com> have Allegro PCB Footprints available. They are normally found under the specific component page.
4. Xilinx <http://xilinx.com/support/download/index.htm> have DE HDL symbols available.
5. 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.
6. NXP offer OrCAD Capture symbols and PCB Editor footprints for the LPC800 device range. <http://lpcware.com/content/nxpfile/lpc800-orcad-and-allegro-symbols>
7. 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>.
8. Analog devices offer Allegro and OrCAD PCB Footprints. Take a look at: - <http://search.analog.com/search/default.aspx?query=allegro&local=en>. They also provide files suitable for UltraLibrarian (see point 12 below).
9. <http://snapEDA.com> is a new service that offers free CAD Libraries for all the Cadence tools including schematic symbols, PCB footprints and simulation models. They provide *.scr files for the PCB Footprints that can be run from within PCB Editor or they also provide a batch file that can be run externally. The schematic symbols they provide an EDIF file that can be imported from within OrCAD Capture.
10. 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>.
11. Parallel Systems have a starter library of PCB Footprints available from:- http://mediafire.com/download/i1re9htt65tab8m/PCB_Editor_Footprints.zip
12. <http://accelerated-designs.com> – 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. You can also access the Accelerated Designs Database and have a daily download limit of 5 per day free of charge. 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. Accelerated Designs also offer a paid for service to create schematic symbols and PCB Footprints based on a supplied datasheet. Their website contains full details. You can also just go to <http://ultralibrarian.com> and search for parts, there are PCB footprints, OrCAD Capture schematic symbols and 3D Step models that can be downloaded for free.

13. Samtec Connectors offer both schematic symbols and PCB Footprints. Select the library you require and download the zip file from :-
http://samtec.com/productinformation/technicalspecifications/pad_schematics.aspx
14. PCB Libraries <http://parallel-systems.co.uk/products/pcb-library-expert> provide a paid for tool that will generate IPC 7351B approved PCB Footprints and 3d Step Models. Contact sales@parallel-systems.co.uk for further details.
15. OrCAD Library Builder is a new utility for use directly with PCB Editor. This tool designs IPC 7351B standard footprints, 3d Step Models and OrCAD Capture directly from PDF datasheets. You can extract the datasheet table to build your schematic symbol. For more details go to <http://parallel-systems.co.uk/products/orcad-library-builder> or contact sales@parallel-systems.co.uk for further details.
16. Footprint Maker is a skill program written by a user. It contains approx 3,000 PCB Footprints. You need to run the Performance License or higher because you need the ability to run skill programs to create the footprints. The executable is available via the edaboard.com forum. You need to be a registered user (free) to access downloads from the forum. - <http://edaboard.com/thread64711-5.html>
17. The OrCAD Capture Marketplace has a Component search area that also has schematic symbols and PCB Footprints supplied by the vendors. For more details go to :- <http://orcadmarketplace.ema-ed.com/ComponentSearch.aspx>.
18. 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.
19. 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.