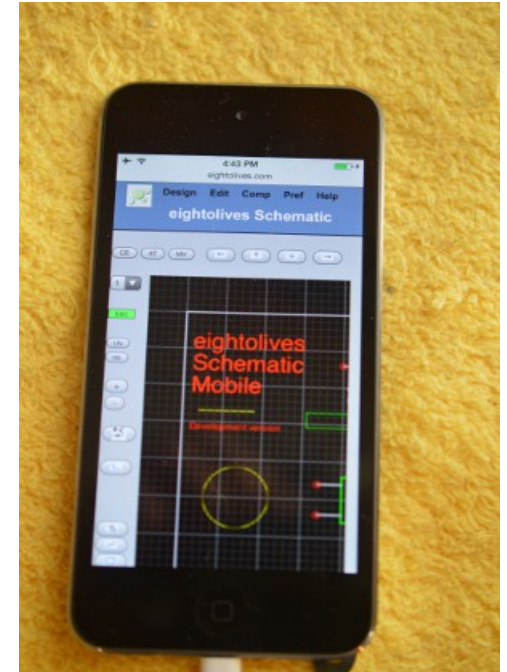




Using Vendor Symbols and Footprints in Schematic Mobile



William_Kaupinis@eightolives.com
June 16, 2015

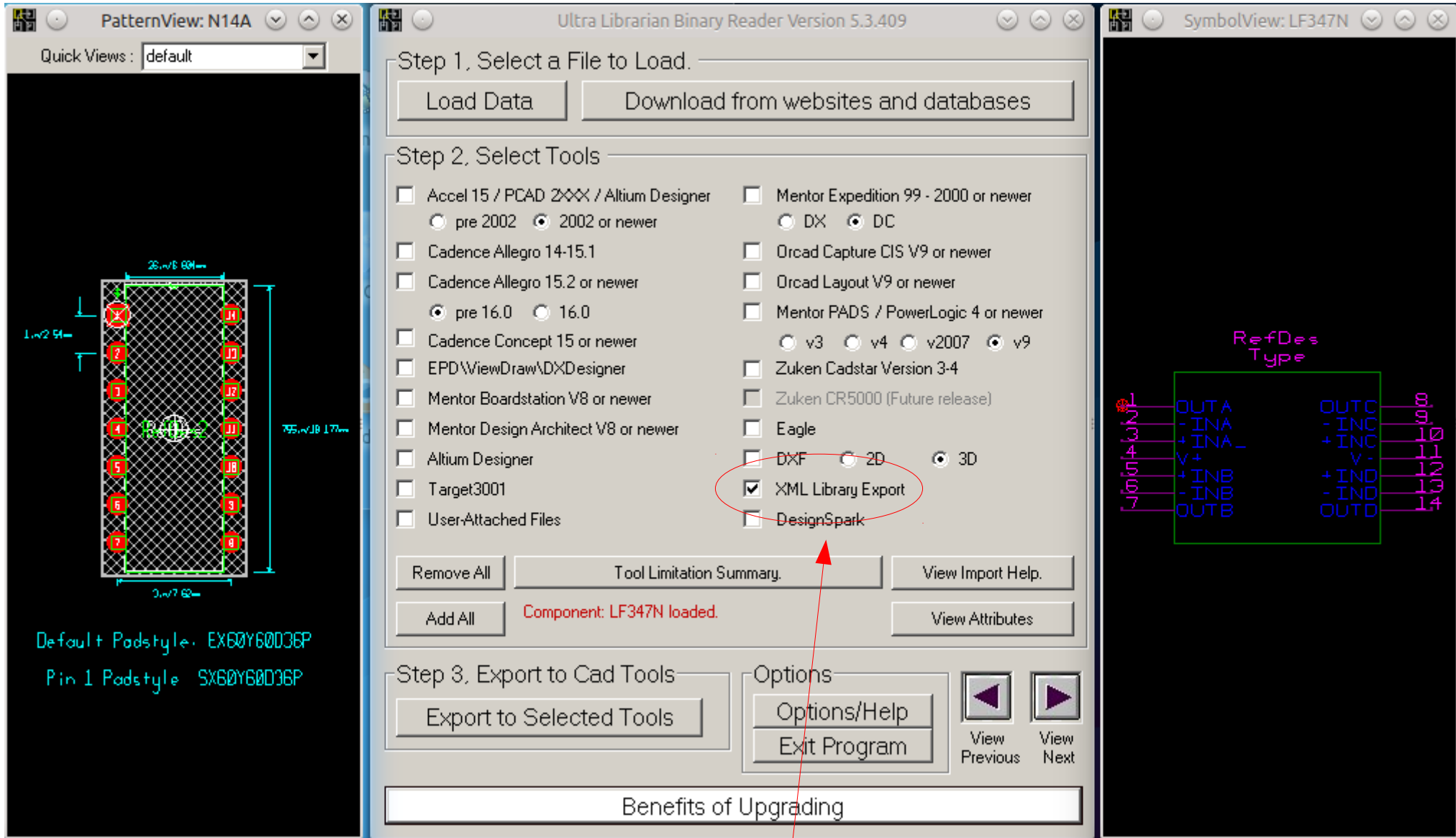
About Symbols

- Symbols represent components in a schematic.
- Generic components like resistors, capacitors and inductors have generic symbols available in the generic symbol library.
- Multi-pin connector symbols can be easily generated by the Connector Wizard
- Integrated circuits are often modeled as Black Box type symbols. They can be created with the aid of the Block Symbol Wizard with some editing.
 - **Some part vendors provide symbols and footprints for their parts**

Vendor Symbols

- Vendor symbols are often provided in the proprietary, binary “.bxl” format
- The commercial tool, UltraLibrarian by Accelerated Designs, is available to translate the “.bxl” format to the symbol format used by your CAD tools.
 - A free, UltraLibrarian Binary Reader is available.
- Schematic Mobile can input the vendor symbol data once it has been translated to an “XML Library Format” “.xml” file

UltraLibrarian Screen Shot



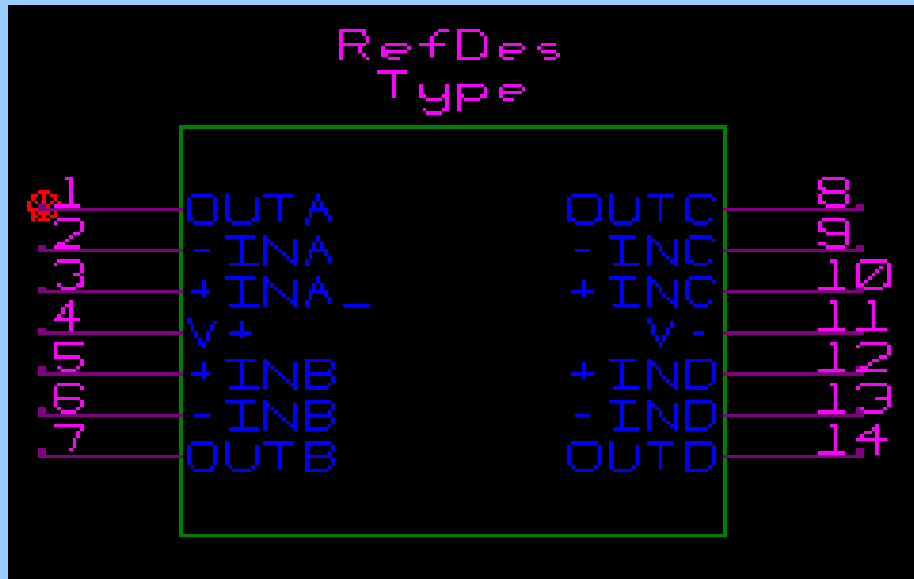
Select "XML Library Export" to create .xml file.

Create the .xml file

- Three steps to convert in UltraLibrarian:
 - Step 1 – Load the vendor's “.bxi” file
 - Step 2 - Select “XML Library Export”
 - Step 3 - Press the Export to Selected Tools button
- The resulting “.xml” file is stored in the “UltraLibrarian\Library\Exported” directory
 - The filename is of a format *date_stamp.xml*
 - You should copy it to your project directory and rename it to be recognizable

Open the .xml file with Schematic Mobile

- Vendor's UltraLibrarian symbol

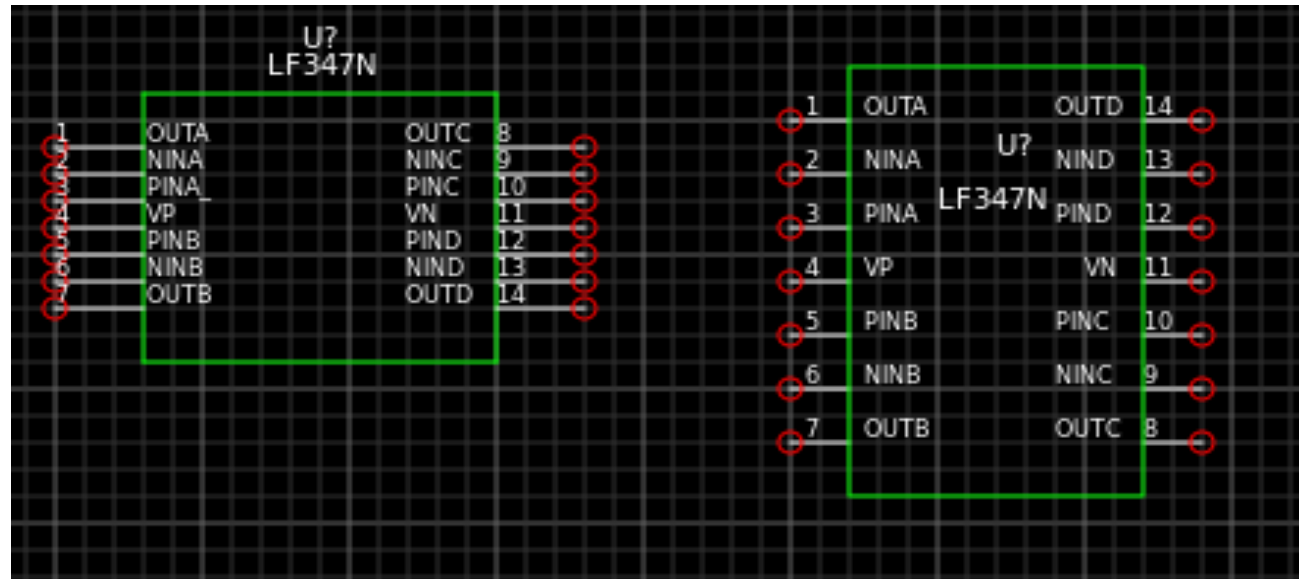


- Schematic Mobile import



Schematic Mobile modifies pin names to be "VHDL" compatible.
 "+" is changed to "P" and "-" is changed to "N"

Vendor Symbol vs Block Symbol Wizard



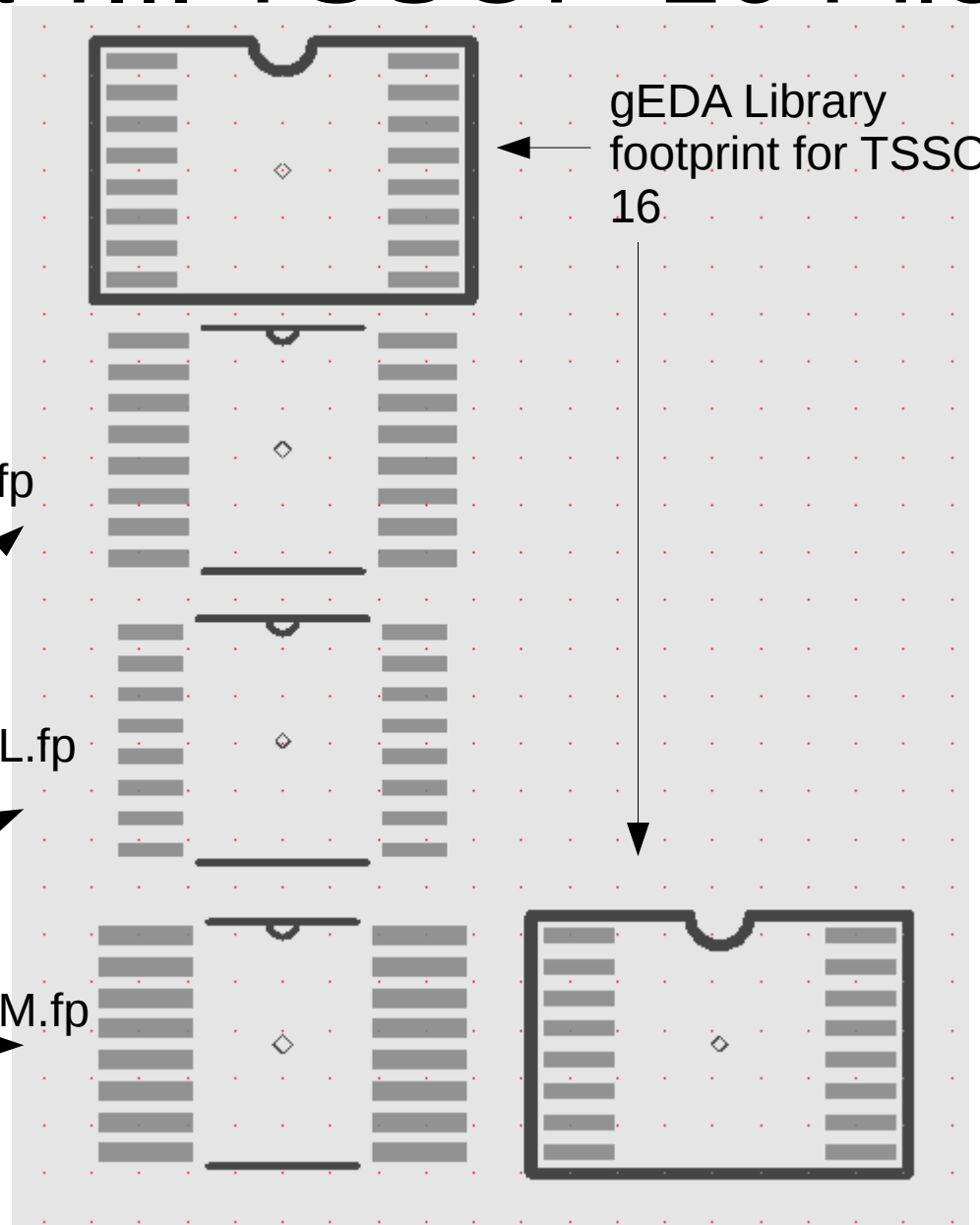
- Symbols can be created quite easily with Schematic Mobile's Block Symbol Wizard
 - Note: the Wizard's default pinout reflects the physical, counter-clockwise numbering of the part which may be preferred. Some vendors symbols also use this convention.

Vendor Footprint Data

- When you read the translated “.xml” file using Schematic Mobile, footprint files (.fp) usable by gEDA's PCB layout tool are created and temporarily stored in LOCAL project.
- Use eightolives Editor to open the text-based .fp file and save it to your project's footprint directory
 - Opening it in Schematic Mobile will prompt you to save it to your File System.
- Symbols in the schematic should have the **footprint** attribute set to the name of the .fp file to be used.

Example Using a T.I. TSSOP-16 File

Pin	Signal	Pin	Signal
1	*1EN	VP	16
2	IN2	*2EN	15
3	P154	IN1	14
4	P153	P254	13
5	P152	P253	12
6	P151	P252	11
7	P1D	P251	10
8	GND	P2D	9



Footprints created by Schematic Mobile from a T.I. provided .bxl file for a TSSOP-16 (PW) package.

The T.I. .bxl file specifies three different footprints.

PW16.fp

PW16-L.fp

PW16-M.fp

gEDA Library footprint for TSSOP-16

GEDA PCB Footprints

- The gEDA PCB layout tool normally retrieves symbol and footprint data from different shared directories. To access your project's footprints:
 - Either add your footprints to the shared directory
 - Or add a symbolic link from the shared directory to your project's footprint directory
- In Linux, the shared footprint directory is `/usr/share/pcb/pcbplib-newlib`

For more information

- Visit <http://www.eightolives.com>