NXP Semiconductors and EMA Design Automation have collaborated to provide NXP customers with schematic symbols, PCB layout footprints and 3D Step models for certain NXP products.

Please follow the instructions below to extract CAD files in desired output format:

Step 1: Download and install the [Ultra Librarian free reader](https://www.ultralibrarian.com/products/free-reader).

Step 2: Download the BXL file(s) from NXP Tools/Software page.

Step 3: Open the .bxl file with the Ultra Librarian software.

PCB footprints, schematic symbols and 3D step models are available for download in a vendor neutral BXL format, which can then be exported to most EDA CAD/CAE design tools of your choice using the Ultra Librarian Reader. The reader is available as a free download.

The Ultra Librarian free reader is a subset of the Ultra Librarian toolset that can generate, import, and export components and their attributes in virtually any EDA CAD/CAE format.