

# LIBRARY FACILITIES

- ✓ Extensive Pre-Supplied Libraries
- ✓ Automated Part Import (Footprint)

- ✓ Automated Part Import (Schematic)
- ✓ Assisted Manual Part Creation

## Introduction

Device libraries in the schematic and the layout are the building blocks of any project and are a fundamental piece of CAD software. However, with millions of electronic parts available and multiple packagings available for a given part it is impossible to supply a complete listing directly in the software package.

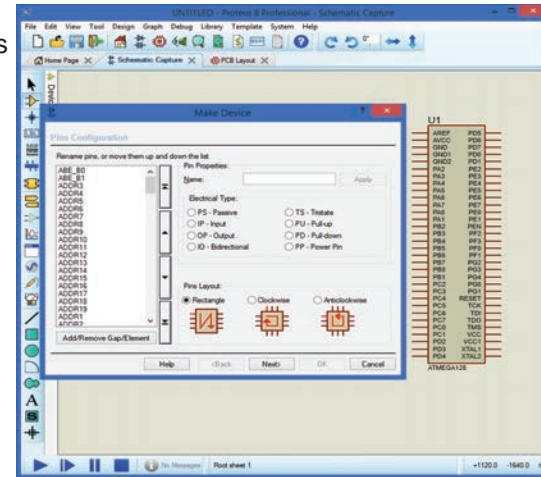
Proteus includes several methods to help you get designing with the parts you want. First, we aim to provide a comprehensive set of ready-made schematic components and layout footprints. Second, we aim to take advantage of manufacturer information and emerging standards in dedicated tools to enable automated import of parts in both schematic and layout. Finally, we aim to make it as quick and easy as possible for users to create and manage their own library parts in the Proteus software.

## Automated Part Import (Schematic)

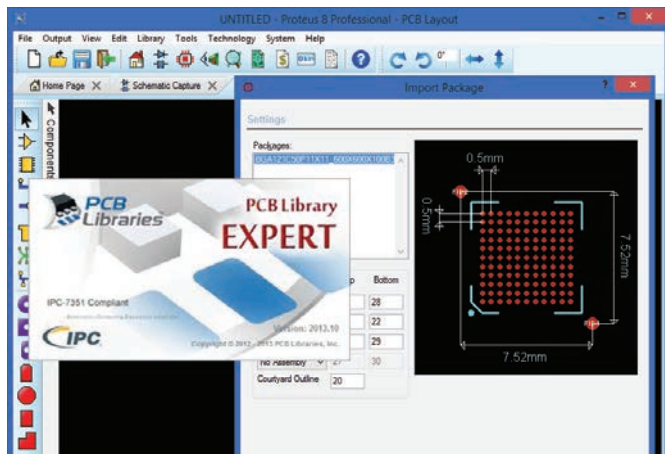
When a schematic component doesn't exist in the pre-supplied libraries it is almost always a IC, BGA or microcontroller chip. Fortunately, the vast majority of silicon vendors provide BSDL files for such parts and Proteus can use this file to automate the creation of the part.

The boundary scan description language (BSDL) is a description language for electronics testing using JTAG. From a schematic component perspective there are two main benefits in this file format:

1. The part name, number of pins, electrical type and pin names are all part of the file. This means we can automatically create a basic schematic component for the part.
2. The binding between each pin name on the schematic and the pad name of the footprint is included. This means that we can provide an assisted/automatic packaging for the schematic component.



## Automated Part Import (Footprint)



Proteus provides a lot of flexibility and functionality for creating different types of footprints and putting them into libraries for re-use. However, for complex parts or where a lot of parts are needed, the use of third party tools may get the job done faster. Also, if libraries of parts exist from a different product, it is helpful to translate and import them into Proteus.

The PADS ASCII Library Part importer is the mechanism by which we support this. PADS ASCII is a widely recognised file format for describing a PCB footprint and offers a semi-automatic way to translate a footprint into a Proteus library part.

PCB Libraries ([www.pcblibraries.com](http://www.pcblibraries.com)) is one example of a tool which contains multiple templates for creating library parts and then lets you build and export them for import into your CAD system. Ultra-Librarian ([www.accelerated-designs.com/ultra-librarian/](http://www.accelerated-designs.com/ultra-librarian/)) is another popular tool that exports PADS ASCII footprints. In either case, from the import side Proteus simply reads a PADS ASCII file and maps the layer names to form a Proteus PCB footprint.

# Pre-Supplied Libraries

Proteus comes as standard with a large and varied library set for both schematic capture (component) and PCB Layout (footprint). Each schematic capture component is packaged with one or more PCB footprints for seamless transition to the PCB layout module. Additional packagings can easily be added or edited at design time if required.

The schematic capture includes libraries for TTL, CMOS, ECL, Microprocessor, Memory and Analogue ICs plus libraries with hundreds of named Bipolar, FET and Diode discrete semiconductors. Also included are manufacturer specific libraries from National Semiconductor, Philips, Motorola, Teccor, Texas Instruments, Dallas and Zetex amongst others - an expanding total, currently well over 35,000 parts. Both European and American symbols for passives are supported and you can choose your preference globally via a single command.

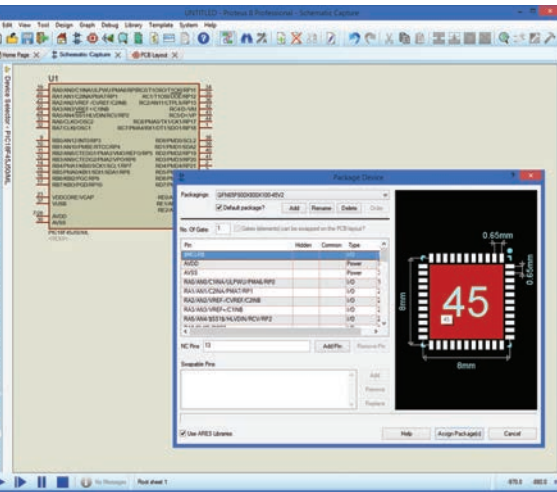
On the layout side, the supplied footprint libraries cover a large range of through hole components including all the most common IC, transistor, diode and connector packaging types. We also supply as standard the full IEC libraries and SMT footprints which include all the standard discrete and IC packaging styles. Full implementations of both the IPC-782 and the IPC-7351 standard surface mount standard libraries are included. Each footprint also contains either a dedicated STEP/IGES model, a detailed geometric model or a simple extrusion model for use in the 3D Viewer and export to MCAD software.



## Manual Part Creation

Regardless of how many library parts are supplied or how easy it is to import others there will be times where the best or only choice you have is to create your own parts. It's important therefore that the process in the software is both relatively easy and is as flexible as possible.

With the Proteus capture module, you simply draw on the schematic. The process is basically to define the component body (normally by drawing a rectangle), attaching the pins and then dragging a selection box and making the collection of pieces into a component. When you do this, you'll also package the part by mapping the pins on the component to the pads on the footprint you want to use in the layout module. For BGA's you can load in an Altera or Xilinx pin information file to package the part automatically.



On the PCB Layout, a footprint is typically a collection of pads and silkscreen graphics use to site the part on the board. However, it can also optionally contain a courtyard definition and have specific requirements in terms of solder mask and/or paste mask. As with the schematic capture the procedure in Proteus is to draw what you want, select it and then make it into a package. Sometimes, being able to draw gives you the necessary control as well as the flexibility to handle non-standard requirements. For example, a button contact pad is likely to want a custom soldermask definition around the part and the solder paste pattern on the thermal pad of a QFN is best considered with respect to the placement of any stitching vias.

3D Models can be included while creating a footprint, normally by specifying the name of the STEP file downloaded from the internet. However, you can create your own 3D model at this stage by defining the pin type, orientation, shape and extrusion of the physical part. This can then be used in the 3D Viewer module and form part of the STEP assembly should you wish to export to an MCAD package like Solidworks.

## Video Resources

There are several video resources to demonstrate the flexibility and ease of use of the Proteus PCB Libraries. These resources will show you how to make full use of the features mentioned above and utilise the libraries to their full extent:

 [Importing Schematic Components into Proteus](#)

 [Importing Footprints to Proteus Design Suite](#)

 [High Pin Count Device Handling](#)

 [Creating Advanced Footprints in Proteus](#)

 [Creating Button Pads in Proteus Design Suite](#)

 [Saving Proteus Footprints with Track Fan-out](#)