LIBRARY PART IMPORTER

Proteus supports automatic library import of both schematic parts and PCB footprints. Over 14 million parts now available!

Universal Library Import

Library Import:

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 it is impossible to supply a complete listing directly in the installed libraries of the software package.

The solution in the Proteus Design Suite is twofold. First, we provide an integrated web search directly in the library picker dialogue. This provides free access to millions of parts and a double click will import the schematic component and footprint directly into the project.

Where a part has not yet been created you can request it be made for you. This is a free service offered by our partner with an average turnaround of 24-48 hours per part.

Secondly, we provide an import dialogue form that can import parts from third party vendors such as Ultra-Librarian, SnapEDA and PCB Library expert. During this manual import process large unwieldly schematic bodies can be split into smaller elements and the pins then re-arranged around the various component bodies.

The Import Workflow:

Most often you will work directly from the library picker in the Proteus schematic and simply double click on a web result to import the part into your project.

For larger parts you may decide to use the import part dialogue form which gives you greater control over the number of elements and pins for the schematic component. In either case you'll get both the schematic part and the associated footprint along with the pin mapping between them.

For most parts, the 3D STEP file will also be imported for use in the 3D Viewer and subsequent export to the MCAD tool of your choice.



How It Works:

- Library Part Import from Integrated Web Search
- Library Part Import from SnapEDA
- Library Part Import from Ultra Librarian
- Library Part Import from BSDL Files

LIBRARY PARTS

Fully automatic part import with **Samacsys Library Loader**.

Full support for **Ultra-Librarian** and **Digikey** part downloads.

Schematic Part Import

When bringing in parts from a supported vendor the import of the schematic part takes place alongside the import of its associated PCB footprint(s). This allows the packaging information (the pin to pad associations) to also be brought in. The import wizard includes a body editor screen for large pin count parts allowing you to both split them into multi-element parts and also to re-arrange the pins between elements or around the component bodies. You can also import standalone schematic parts using the BSDL file format which is normally supplied by the part manufacturer for large IC's.

PCB Footprint Import

The PCB footprint will be imported alongside its associated schematic component when you download the part from a supported vendor website. Solder mask, solder paste, stitching vias and padstacks are all supported along with courtyard and silkscreen definitions. Additionally, the import log in Proteus will verify the incoming data and warn you if any problems are identified. We also support PCB footprint import from the popular PCB Library Expert tool which allows you to easily design your own part by entering datasheet numbers into a grid and then exporting the resulting footprint to Proteus.

3D Model Import

In many cases the downloaded part will also contain a STEP model representing the 3D shape of the silicon device. If so, Proteus will bring this in as well and you will see a preview during the import of the PCB footprint. Several edit boxes allow you to change the orientation of the 3D model in the event it is rotated around an axis relative to the 3D PCB. If no part is included and you can't find one online you can easily create a basic 3D model inside Proteus by entering simple script and dimension information. This will also be translated and exported as part of the STEP assembly for the PCB should you later want to send the board to an MCAD tool.

Manual Part Creation

In the unlikely event that you need to or want to create your own parts manually you can do so simply by drawing on the schematic or layout using primitive pieces. For a schematic part the process is basically to define the component body, attaching the pins and then 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. 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.

Full support for **SnapEDA** part downloads.

Full support for **PCB Library Expert** part creation.

Labcenter Electronics Ltd.
The Stables, Hartlington Hall, Burnsall,
N. Yorkshire, England, BD23 6BY
Tell: +44 (0) 1756 753440, Fax: +44 (0) 1756 752857
Web: www.labcenter.com, Email: info@labcenter.com
Labcenter YouTube Channel: http://www.youtube.com/c/LabcenterElectronicsLtd

Labcenter Electronics www.labcenter.com