PROTEUS PCB DESIGN

Proteus PCB Design combines Schematic Capture and PCB Layout modules to provide a powerful, integrated and easy to use suite of tools for professional PCB design.

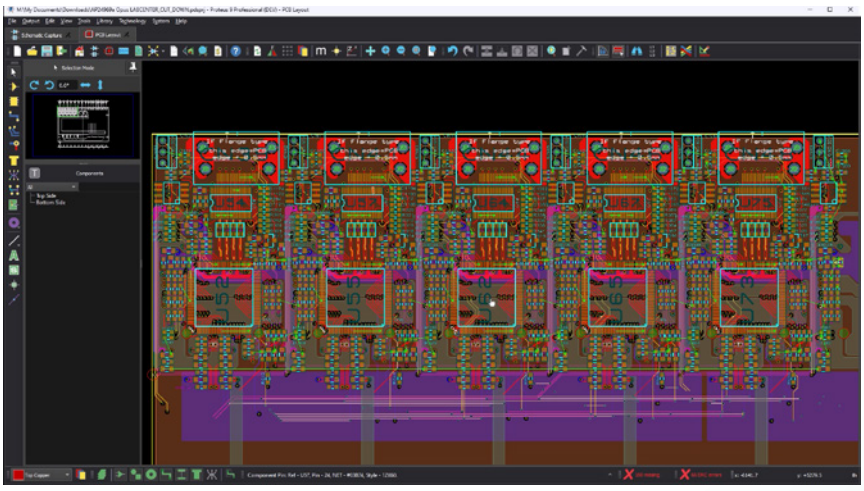
Productive PCB Design Tools

The Proteus PCB Design product range is a professional's choice for modern PCB Layout. With over 30 years of continuous development and innovation, our focus remains on adding functionality while maintaining a simple, clean user interface and tight integration with the schematic design.

Proteus consists of a single 64-bit application framework with several modules (Schematic, BOM, Layout, 3D Viewer, etc.). Each module opens in a tab within the application window but can then be dragged and dropped to create additional windows and enable side-by-side viewing. This allows you to not only work with the traditional schematic and layout windows but also to combine or split off additional modules according to the work you are doing at a particular time. For example, Schematic and BOM for report generation, Layout and 3D Viewer for verification and so on.

The main goal of the unified application framework is to improve and better support the user workflow. Changes made on the schematic during design will reflect visually in the layout and you can review and confirm pending changes or go back and edit further on the schematic. Similarly, annotation changes made on the PCB can also feed back to the schematic. For example, the PCB can be automatically re-annotated after placement (e.g. top left to bottom right) and the schematic module will update accordingly. This puts control over the design process firmly in the hands of the engineer.

Labcenter's focus is on functionality and value for our customers. We are continually adding new features and tools to the Proteus design Suite to ensure we have the features demanded by modern design techniques. To ensure our customers receive the best value, we equip all Proteus PCB products with a rich set of core functionality including a world class shape based autorouter as standard. More advanced features such as diff pair routing and route length matching can then be added in the higher product levels as required. Upgrading your Proteus design Suite is just a matter of paying the difference as we provide a 100% credit for your previous purchase.



"PCB design is our business. We review PCB layout software on an ongoing basis and Labcenter has topped the list for the last 10 years. Certainly the most productive and very, very affordable. We have licences for other very expensive products but they don't get much use."

Donald Kay - Don Alan Pty

PCB LAYOUT

3D Visualization with import and export of STEP/ IGES models.





Easily route differential pairs and length match critical high speed signals.



LIBRARY PARTS

Creating library parts for schematic and layout can be tedious and error prone and diverts the engineer from the task of actually making the product. Fortunately, there are now a variety of third party websites with catalogues of literally millions of library parts available for import. We've integrated these web results directly into the library pick form where a double click on a result will seamlessly download and import the part. There is also a complete managed libraries workflow with optional GIT/SVN versioning. This enables librarian control over library parts and restricts usage in project design to the approved managed library set.



BOARD SETUP

The layer stackup manager lets you specify the number of cores in the board, the thickness of each layer and the order in which they are assembled to make the final PCB. Proteus can then work out the legal, manufacturable set of default drill passes for the layout. You can add or remove drill passes over various layer ranges if you need to. When you are then routing a track and you place a via, the software can automatically choose the smallest via range from source layer to destination layer. It can also use the via depth in length matching calculations for high speed signals. Both the layer stackup information and the drilling table are then exported as part of the the Gerber X2 fileset for manufacture.



BOARD CONSTRAINTS

Design rules in Proteus can be as simple or as detailed as you need. You start with a board global rule which defines a base set of clearances. You can then add additional rules for specific layers or for specific sets of signals which will override the global clearances where appropriate. Finally, you can draw out a design rule room on the PCB and set specific clearances in that area. This is particularly useful for things like BGA escape areas or fine pitch SMD fanout regions. Once your board constraints are set up the DRC system will notify you of any violations via a simple click to zoom list viewer, making it easy to find and fix problems during board layout.



ROUTE PLACEMENT

Proteus is equipped with both push and shove manual routing and follow-me-routing (hug mode) and the user can toggle between modes via keyboard shortcut at any time. All route placement is fully design rule aware and the Visual DRC feature enlarges nearby objects by their clearances to help guide optimal route placement. Corners can be switched between curved or mitred and route widths changed dynamically in the Route Command Centre. High speed signals can be length matched either to each other or to a specified distance. A dedicated routing mode is available for routing differential pair which includes automatic skew correction. Finally, a shape based autorouter is included as standard to quickly achieve routing completion on non-critical nets.

Re-use previously designed schematic & layout circuitry with Project Clips.





Use Assembly Variants to manage different product configurations.

Labcenter Electronics Ltd.
The Stables, Hartlington Hall Estate, Near Burnsall, N. Yorks, BD23 6BY, UK
Tel: (+44) 0 1756 753440
Web: www.labcenter.com, Email: info@labcenter.com
Labcenter YouTube Channel: http://www.youtube.com/c/LabcenterElectronicsLtd