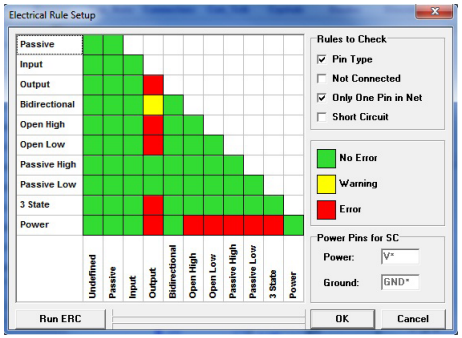
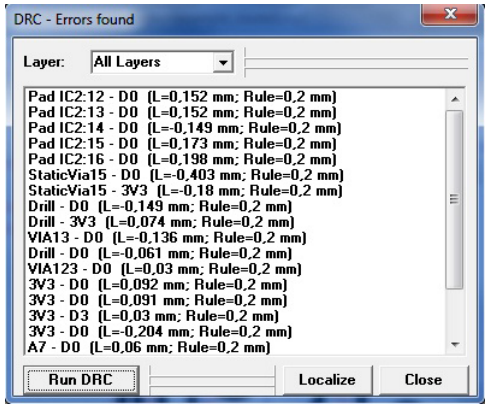
DipTrace environment consists of 4 modules: Schematic Capture, PCB Layout with autorouter and 3D, Component editor and Pattern editor.

DRC (Design Rules Check)

We are convinced, that precise verification of design should be one of the main features of quality PCB Design environment. DipTrace won't give you a chance to make an error, if you will follow all verification procedures. In Schematic Capture module Electric Rule Check (ERC) is





availableand has detailed settings of what will be considered as an error. Hierarchy is checked by special verification. In PCB Layout DRC (Design Rules Check) is conducted. It can use Class-to-Class Rules. User can define sizes of holes and clearances between different objects on the different layers. List of errors is convenient and understandable.

Check Net Connectivity

It's another key verification procedure, checking if all nets are connected and reports all isolated copper pour areas.

Compare to Schematic

This verification checks if the PCB project corresponds to source schematic. It shows all net structure errors and unknown components.

Manufacturing output

Post processing is quick. DipTrace supports number of professional manufacturing formats. You can produce RS-274X Gerber and N/C Drill files, which are accepted by manufacturers. DipTrace supports export of true-type fonts and raster images. DXF output is also available.

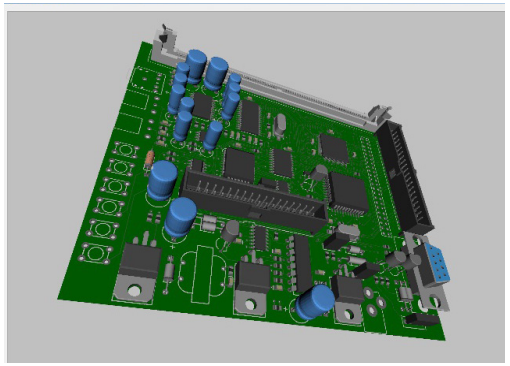
Production with milling method is supported

Import/Export

DipTrace allows you to exchange schematics, layouts and libraries with other EDA and CAD packages (DXF, Eagle, P-CAD, PADS, OrCAD). You can import and export Accel, Allegro, Mentor, PADS, P-CAD, Protel and Tango netlist formats.

3D Preview

DipTrace gives wonderful opportunity to see in 3D, how your board is going to look like after manufacturing, with all the patterns installed. You can rotate, zoom in and out, change colors in real time. 3D Preview module with hardware acceleration supports .vrml and .3ds models. More then 2,500 3D models are included for free in 2.2 version of DipTrace.



DipTrace

Professional PCB Design Tool

DipTrace – Advanced PCB Design Environment Easy and Professional

You need to design a complex multilayer printed circuit board with thousands of pins or just a simple single-sided PCB? You don't want to spend hours, learning new software, or pay sky-high prices for products with functions you won't ever use? All you need is a correct PCB design tool – DipTrace.

Friendly and Reliable

DipTrace is an advanced, state of the art PCB Design environment, satisfying the most strict requirements of nowadays. It's easy-to-use, reliable, but has lots of features, never seen in medium-level products before. After switching to DipTrace you won't need any extensive trainings or reading tons of books. Usually customers become productive within hours and enjoy simplicity and functionality of DipTrace. PCB tutorial, included in the package, helps to get familiar with basic opportunities of the software and contains detailed instructions to create a simple board. The design process can be speeded up with number of hot keys for all cases.

- User friendly, intuitive interface
- Wide range of capabilities
- Product is for both, professionals and casual users
- Commitment to the future

Expanding your opportunities

Intuitive user interface and simplicity of design environment doesn't mean it lacks functionality. Multi-sheet and Hierarchical schematics, cross-probing between Schematic and PCB, high-speed autorouter, advanced manual routing, blind/buried vias, multiple verification features, real time 3D PCB preview, wide import/export possibilities, 100,000+ packages in libraries and others will help both professionals and hobbyists to achive even most complicated tasks in short terms with this software tool. DipTrace is able to offer everything you need to create the PCB from idea to the state of ready manufacturing files. It is packed with features, you will enjoy. DipTrace delivers wonderfull performance you need, for affordable price.

Since 2004, when the first version of DipTrace was released, developers are following key principles: DipTrace Software package consists of Schematic capture module, PCB Layout, Component editor, Pattern editor, Shape-based autorouter, 3D PCB Preview. Developers continue to expand capabilities of the product and continuously work to make the environment even more friendly to you.

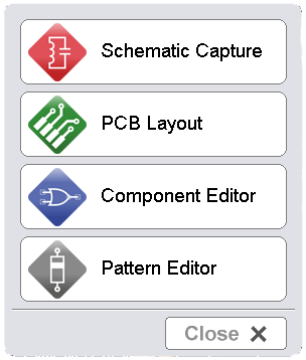
DipTrace is always up-to-date and changes with the improvements in production technologies. Hence, you can be sure, that your investment in the product will work on you.

DipTrace – PCB Design with Pleasure

DipTrace supports Windows 98, ME, XP, Vista, Windows 7 (32/64 bit) and Linux (Wine)

Easy-to-use

New customers usually became productive, literally, within hours, doesn't matter if you're a professional engineer or new to PCB design. DipTrace has intuitive interface and supports simple Drag&Drop methodology. All menus are designed, following "Look&Feel" philosophy. Instrument panels can be fully customized. You can develop your own color scheme, this can really speed up working process. Lots of functions in DipTrace have hot keys. This is really helpful, as it gives you the speed and ability to accomplish even the most complex projects much faster.

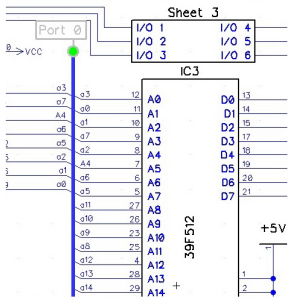


Integrated Environment

Cross-probing is fully developed in DipTrace. All modules of the design environment are tightly connected to each other. It's easy to convert schematic to PCB, renew PCB from the Schematic, or Back Annotate Schematic from PCB. All modules share similar key principles, so you won't be confused.

Smart Project Structure

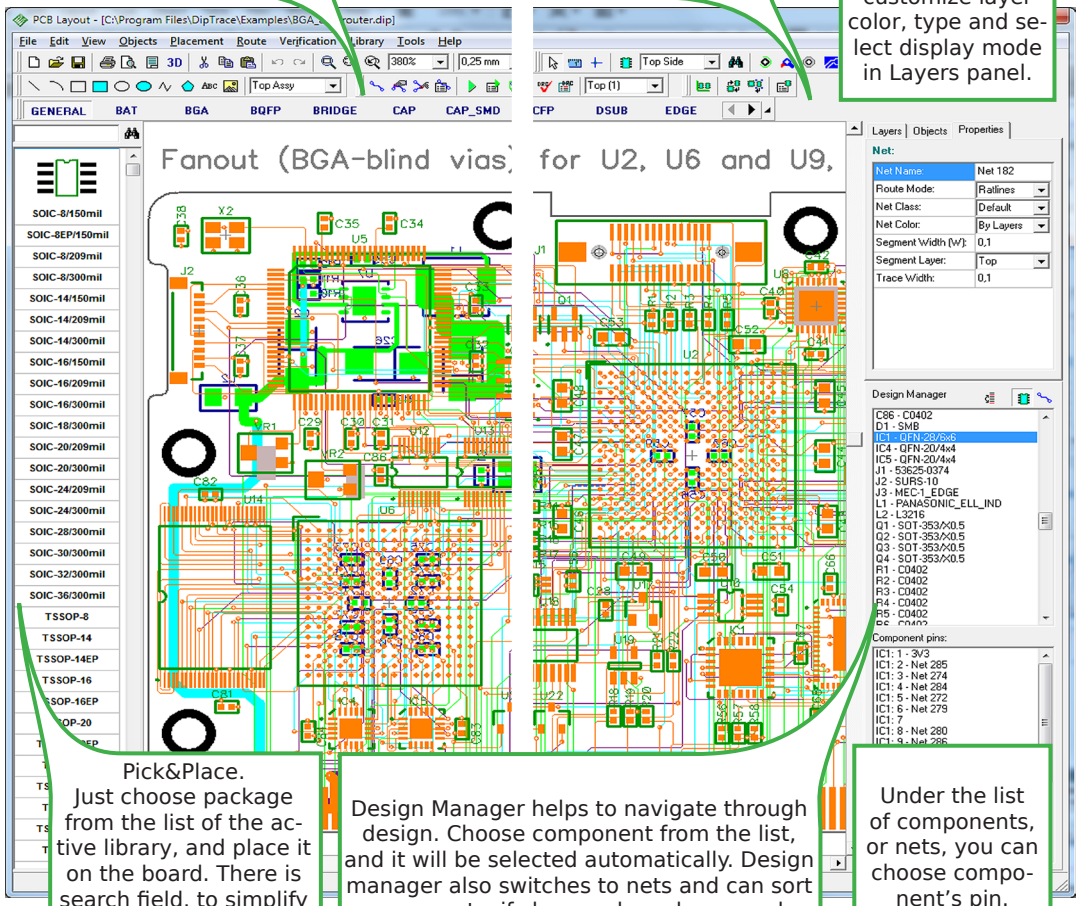
DipTrace is convenient not only for designing simple, but also very complex projects. Nets in DipTrace can be divided by net classes. User can specify different parameters for different net classes: trace width, clearance, trace length limits e.t.c. You can set Class-to-class rules, which have priority over other class clearances. Net classes from Schematic are automatically transferred to PCB Layout. “Via Styles” is the feature, organizing work with vias. Change in the parameters of via style automatically changes all vias of this style.



Libraries in DipTrace are easy-to-use and understand. Packages are arranged in corresponding libraries by type and manufacturer.

On the instrument panel of DipTrace there are lots of quick-access buttons. Panel itself is fully customizable. You can arrange panels in the way you want.

You can easily switch between layers, add new layers, customize layer color, type and select display mode in Layers panel.



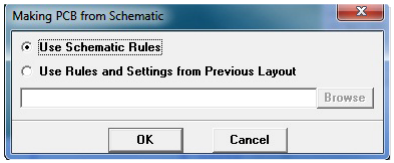
Pick&Place. Just choose package from the list of the active library, and place it on the board. There is search field, to simplify this process.

Design Manager helps to navigate through design. Choose component from the list, and it will be selected automatically. Design manager also switches to nets and can sort components, if changes have been made.

Under the list of components, or nets, you can choose component’s pin.

Save/Load Design Rules

In DipTrace you can save design rules from one project, to use it with another. So you don’t have to create all the Layers, Net Classes, Via Styles again. Design rules can be loaded from any PCB project, or from separate *.rul file.



Multi-level Hierarchy

DipTrace features multi-sheet, multi-level hierarchy in the schematic capture module. This allows you to draw even most complicated boards easily. Hierarchy in DipTrace is simple and logical, but gives wonderful abilities to user. Sheets in the Schematic are converted to hierarchy blocks and can be inserted into the main sheet, or into each other as much times, as needed. Nets on different sheets are connected with bus ports, net ports, or simply by name. To connect nets on different levels of hierarchy in DipTrace you can use global nets.

Multi-level Undo/Redo and "Recover Board" feature

Placement Features

You can arrange components inside or outside the board outline for convenience, use “Placement by list” feature, which becomes necessary, when you have hundreds of packages, but you need to find one and place it on the board manually. However, you can use Auto-placement and DipTrace will automatically place packages on the board with optimized distance of future traces between them.

High-speed Autorouter

DipTrace has a high-speed built-in Shape-based autorouter with advanced settings, capable of routing complex multi-layer boards with through-hole and Blind/Buried vias. Autorouter can create fanouts for SMD components. By default Design Rules Check is conducted automatically after autorouting.

DipTrace supports true-type fonts and raster images for PCB silkscreen

Manual Routing

For complex projects lots of PCB designers require good manual routing tools, which are available in DipTrace PCB Layout. Routing nets manually is fast and convenient with 45, 90 degrees angles, arcs and curves modes. DipTrace can highlight only the current net, this makes manual routing much more easier. Hot keys for each action helps to work very fast.