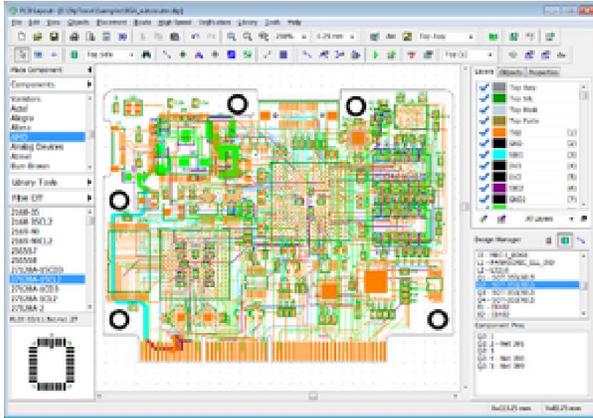


# DipTrace

DipTrace



DipTrace PCB Layout

**Developer(s)** [Novarm Ltd.](#)

**Initial release** August 2004

**Stable release** [3.3.1.3 / 5 March 2019; 10 months ago](#)

**Written in** [Delphi<sup>\[1\]</sup>](#)

**Operating system** [Windows](#)

**Available in** 22 languages

**Type** [Electronic design automation](#)

**License** [Trialware](#)

**Website** [diptrace.com](#)

**DipTrace** is an EDA/CAD software for creating [schematic](#) diagrams and [printed circuit boards](#). The developers provide a multi-lingual interface and tutorials (currently available in English and 21 other languages). DipTrace has 4 modules: schematic capture editor, PCB layout editor with built-in shape-based [autorouter](#) and 3D-preview & export, component editor, and pattern editor.



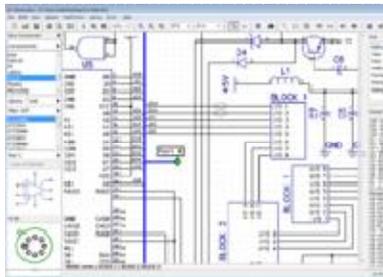
# Contents

- [1 Basic features](#)
- [2 Schematic capture](#)
- [3 PCB layout](#)
- [4 3D-preview and export](#)
- [5 Component editor](#)
- [6 Pattern editor](#)
- [7 Pricing](#)
  - [7.1 Freeware and hobbyist versions](#)
- [8 Version history](#)
  - [8.1 Version 3.0](#)
  - [8.2 Version 3.2](#)
- [9 Community](#)
- [10 See also](#)
- [11 References](#)
- [12 External links](#)

## Basic features

- Simple user interface
- Multi-sheet and hierarchical schematics
- High-speed and [differential](#) signal routing
- Smart manual routing modes
- Wide import/export capabilities
- High-speed shape-based [autorouter](#)
- Advanced verifications with real-time DRC
- Real-time 3D PCB preview
- Export of PCB to [STEP](#) 3D file format
- [ODB++](#) and [Gerber](#) (including Gerber X2) manufacturing outputs

## Schematic capture



Screenshot of DipTrace schematic capture module

Advanced circuit design tool with support of multi-sheet and multi-level hierarchical schematics that delivers a number of features for visual and logical pin connections. Cross-module

management ensures that principal circuits can be easily converted into a PCB, back-annotated, or imported/exported from/to other EDA software, CAD formats and net-lists. DipTrace Schematic has ERC verification and Spice export for external simulation.

## PCB layout

Engineering tool for board design with smart manual routing, differential pairs, length-matching tools, shape-based autorouter, advanced verification, layer stackup manager, and wide import/export capabilities. Design requirements are defined by net classes, class-to-class rules, and detailed settings by object types for each class or layer. When routing with real-time DRC, the program reports errors on the fly before actually making them. DRC also checks length and phase tolerances for differential pairs and controls signal synchronization for nets and buses (including layer stackup and bonding wire induced signal delays). The board can be previewed in 3D and exported to STEP format for mechanical CAD modeling. Design rule check with in-depth detailing and net connectivity verification procedures are available.

## 3D-preview and export



screenshot of the DipTrace's 3D module

This module includes real-time 3D preview & export feature. It shows the model of the manufactured printed circuit board with all components installed. Rotate board in three axes, zoom in and out in real time, change colors of the board, copper areas, solder mask, silkscreen, and background. 3D preview works on all stages of the design. Board can be exported to STEP or VRML 2.0 formats for mechanical CAD modeling. More than 7500 3D models of PCB packages are supplied for free. Externally designed 3D models in \*.wrl, \*.step, \*.iges, and \*.3ds formats can be uploaded and attached to patterns in Pattern Editor or PCB Layout.

## Component editor

Manage component libraries and create single- or multi-part components by selecting a template and its dimensions, defining visual and electrical pin parameters, setting up a Spice model, and attaching pattern with a 3D model to finalize component creation. BSDL import, bulk pin naming, and pin manager tools for pins and buses. Importing libraries from different EDA formats. More than 140000 components in standard libraries.

## Pattern editor

Draw patterns with various types of shapes, pads, holes, and dimensions. Circle, lines (headers, DIP), square (QFP), matrix (BGA), rectangle (RQFP), and zig-zag standard templates. Creation of pattern is basically selecting a template, entering a couple of vital parameters, drawing the silkscreen, and launching automatic pad renumbering. Custom templates can be created for non-standard patterns. DXF import makes creating complex layouts easier.

## Pricing

Edition	Maximum pins and signal layers	Price
Full	Unlimited pins, unlimited signal layers	\$ 995, € 905
Extended	2000 pins, 6 signal layers	\$ 695, € 630
Standard	1000 pins, 4 signal layers	\$ 395, € 355
Lite	500 pins, 2 signal layers	\$ 145, € 130
Starter	300 pins, 2 signal layers	\$ 75, € 65

### Freeware and hobbyist versions

A version of DipTrace is freely available with all the functionality of the full package except that it is limited to 300 pins and non-commercial use<sup>[2]</sup> or 500 pins (non-commercial use, for a moderate charge)<sup>[3]</sup> and two signal layers. Power and ground plane layers do not count as signal layers, so the free versions can create four-layer boards with full power and ground planes. No board size restrictions.

## Version history

### Version 3.0

- Differential pairs: define differential pair and its rules; automatic or manual defining of paired pads; paired routing and editing of differential pair; single-track differential pair routing and editing; phase tune tool (place custom / regular size meanders); real-time control of phase and length tolerance; differential pair manager; support of differential pairs for external autorouters, recognition of paired traces.
- Custom user-defined keyboard shortcuts for tools and dialogs.
- ODB++ (version 7.0) manufacturing output.
- [Gerber X2](#) manufacturing output.
- DRC rule details (easy editing of routing constraints).
- Tree view of 3D models in All Models list, sorted by categories (folders).

- Overall speed and memory optimization for large designs.
- Optimized UI fonts.
- 8143 new components.
- 5694 new STEP models for 3D.

### Version 3.2

- Length matching rules.
- Real-time length comparison table.
- Layer stackup table.
- Using layer stackup and pad signal delay for trace length and differential pair phase calculation.
- Meander tool for any trace, easy resizing and moving of meanders.
- DRC same net clearance check (Trace to Trace, SMD to Pad, SMD to Via, SMD to SMD).
- Altium ASCII import (Schematic, PCB, libraries).
- Eagle XML import (Schematic, PCB, libraries).

## Community

Some hobby and educational groups such as the PICAXE forum members have developed libraries specific to the PICAXE range of microcontroller as produced by Revolution Education including many of the frequently used associated integrated circuits. PICAXE related libraries can be found in the net.<sup>[4]</sup>

In January 2011, [Parallax](#) switched from [EAGLE](#) to DipTrace for developing its printed circuit boards.<sup>[5]</sup>

## See also

-  [Electronics portal](#)
- [Comparison of EDA software](#)
- [List of free electronics circuit simulators](#)