



Our Mission

Our goal at Quickroute Systems is to deliver the ultimate in integrated, intuitive electronics design systems.

We started over 10 years ago with Quickroute, and now with Electronic Design Studio 3 we are setting new standards in total integration by combining not only schematic and PCB design into one package, but also powerful mixed mode simulation and desktop manufacture.

You can literally design, simulate and manufacture your PCB on the desktop - something that we believe is a world first!

For more information on any of our products see our new website on www.dotqr.com.



Electronic Design Studio 3



Be The First

Introducing Electronic Design Studio 3 (EDS), the most powerful integrated EDA system we have ever produced! EDS combines schematic and PCB design, simulation and desktop manufacture into one package.

"EDS 3 is not just a version upgrade but a quantum leap by the British company offering engineers, designers and educationalists considerable benefits in terms of convenience, cost and time savings, by a trio of seamless integrated software systems providing schematic design, simulation, autorouting, CAD/CAM and desktop manufacture - all in one box" **Elektor Electronics Dec 2001**

EDS integrates all elements of the design process so that you will spend less time switching between different software packages and worrying about netlists and more time designing! Our built in realtime CADCheck system also makes sure that your PCB design stays synchronised with your schematic(s), so you can say goodbye to 'update' or 'engineering change' files.

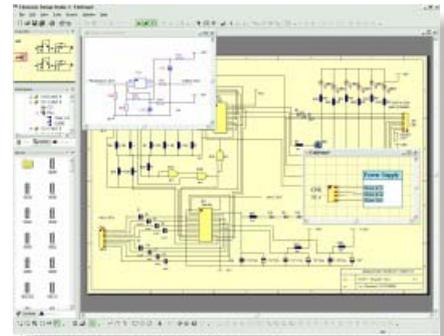
Netlist and project navigators also make moving around large designs much easier. With a click of the mouse you can jump to any symbol, gate or pin on your design instantly. Cross probing can also help in fault finding and circuit testing.

New hierarchical design and 'follow me' wiring tools have made EDS more intuitive to use and the new Viper II autorouter cuts through complex multi-layer designs fast.

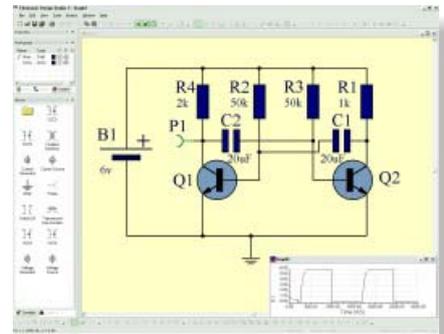
State of the art copper pour, support for TrueType fonts (even on the PCB) and our new polyblend module also ensure you get professional PCB results.

EDS 3 is available in 3 versions; Personal, Advance and Ultra. A feature matrix is included on the back of this brochure.

Schematic



Simulation



PCB



Desktop Manufacture



Professional Schematic Capture

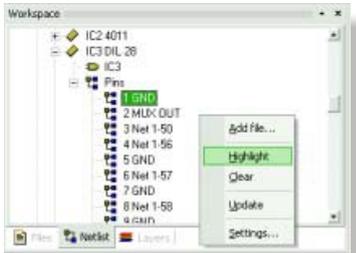
Creating schematics with EDS could not be easier. Simply drag and drop components from the docking library browser into your documents and wire them up using our new automatic "follow me" wiring tools. Just click on the start position, and move your mouse to the destination. EDS automatically places wires and junctions as it follows your mouse and intelligent pins provide instant feedback when connections are made. Of course you can also use traditional manual wire placement for fine control.

Editing is easy too with powerful node punch insertion, segment editing, click and drag indent repositioning, rubber banding, and comprehensive clipboard support. There is also unlimited undo/redo so you can be confident any mistakes can be instantly corrected.

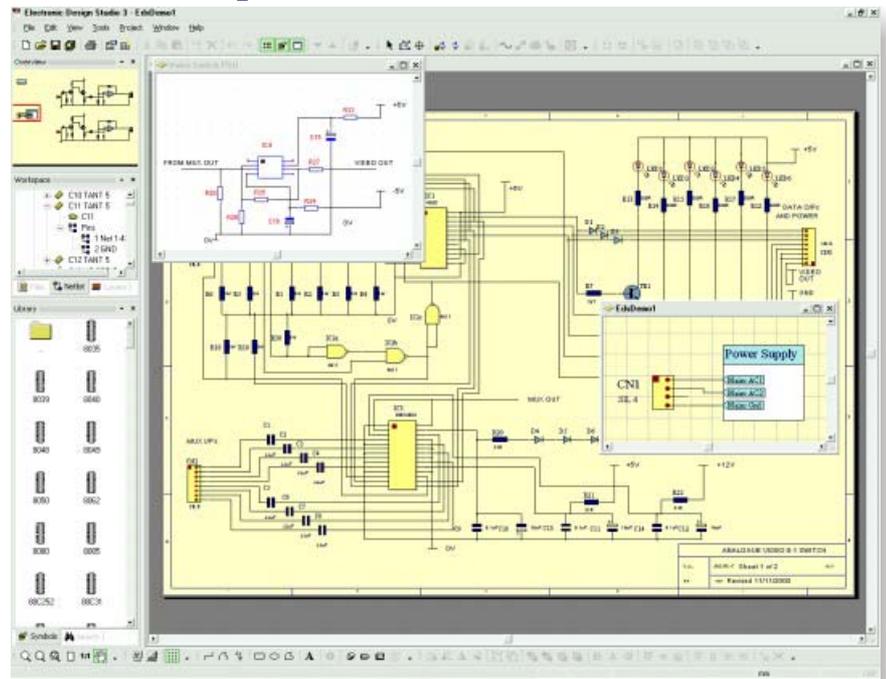
As your project expands you can add an unlimited number of schematic sheets. Top down design is also supported using hierarchical schematics which allows you to group parts of your circuit into modules, or sub-circuits. Linking between sheets or modules is handled using our new tags or more traditional global nets.

The fast Netlist and Project navigators also let you instantly jump to any module, symbol, gate or even terminal with the click of a mouse button. You can also cross-probe to the PCB, great for fault finding.

EDS 3 also introduces Document Schemes which let you change the look and feel of your schematics from serious technical documents, to high colour presentation styles.



Fast Navigation Tools



Features

- Superb DTP quality output
- Drag and drop symbol placement
- Automatic "follow me" wiring
- Powerful segment/node editing
- Rotate components to any angle
- In-situ symbol editing
- Drag and drop symbol creation
- Intelligent symbol pins indicate when connections are made.
- Intellisnap guides cursor to terminals
- Unlimited schematic sheets
- Automatic power rail support
- Global nets/bus support
- New tag objects enable easy off page linking with document, project or module scope
- New unlimited hierarchical design with full module support
- Full clipboard support with automatic re-annotation.
- Separate annotation layers
- Group and lock objects and layers
- Fast zoom and drag pan
- Overview window gives birdseye view
- Unlimited Undo/Redo
- Duplicate and repeat functions
- Document styles instantly change your document look and feel.
- Print preview with zoom
- Netlist Import/Export
- Netlist navigator lets you jump to any symbol, terminal or gate in your project instantly
- Bill of Materials generation with database.

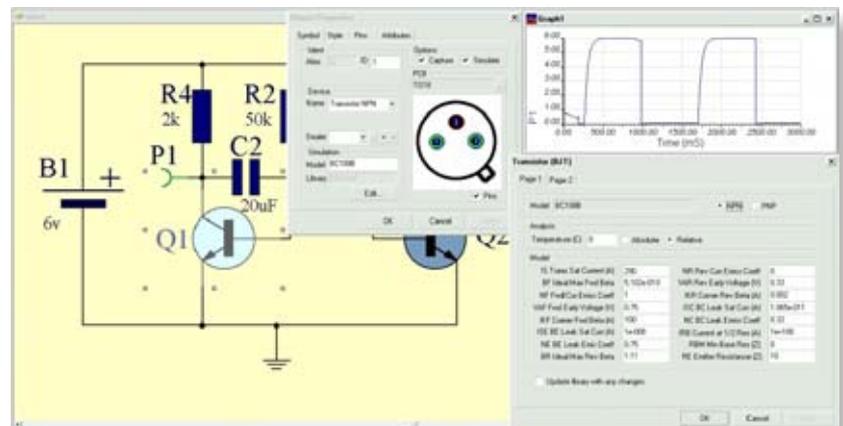
Simulation

Our new simulator is based on a SPICE 3f5 and XSPICE 1.0 engine. It is fully integrated with EDS so that you can use both internal and external models.

Internal models are created inside EDS and cover most of the fundamental simulator models (resistors, BJT, voltage generators, etc). You can edit all the model parameters using comprehensive dialog boxes, or create your own new models. Parameters are saved with your designs. External models are SPICE/XSPICE subcircuits, or models which are automatically imported into EDS.

Both internal and external models can be selected and associated with schematic symbols and PCB footprints using our comprehensive Component Database management system. You can also mark symbols as being excluded from simulation or capture.

To simulate your circuit, just attach probes anywhere on your schematic and press the simulate button. Graphs are shown in their own window, ready to paste into your documents.



Features

- SPICE 3f5/XSPICE 1.0 compatible
- Wide range of internal models supplied with powerful editing/creation facilities.
- Import external SPICE/XSPICE models.
- Component Database links schematic, simulation and PCB footprint information.
- Graphs shown in their own window ready to cut & paste into your reports.

Advanced PCB Design

EDS features our most powerful PCB design system ever! Available with unlimited layers, 1nm resolution, unlimited copper pour zones, our new Viper II rip up and retry autorouter, shape based connectivity and design rule checking, TrueType font support, flexible CAD/CAM output and our new Desktop Manufacturing (DTM)- it's a hard combination to beat at any price!

Automatic Capture & Update

EDS automatically generates a PCB rats nest from your schematic or netlist documents. The appropriate footprints are placed on the PCB board, and you can reposition them as required ready for routing. At all times EDS can keep your PCB in sync with your schematic(s) or netlist. You don't need to worry about netlists or run separate programs, updating is fully automatic.

Viper II

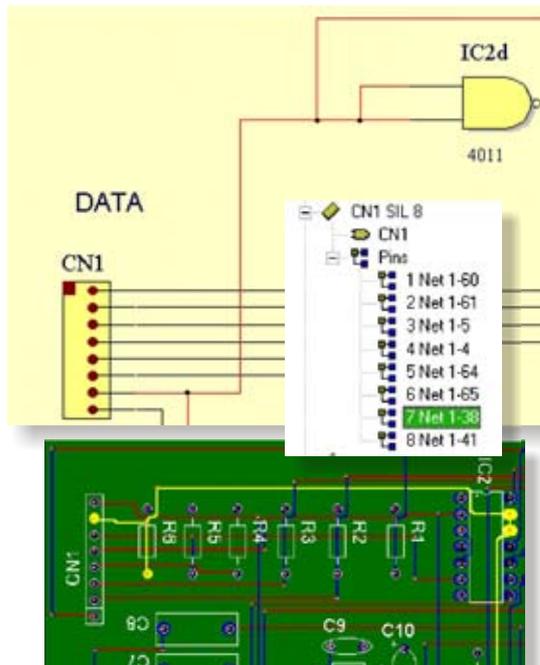
Viper II is our latest generation of hybrid multi-layer autorouter. Fast and flexible, Viper II takes account of your design rules, supports simultaneous routing of multiple track widths, and has a flexible range of via support options. Available with rip up and retry, Viper works hard to maximise success on even complex or dense designs. Viper also supports routing on 1 layer, and can also route without using vias - useful for in-house prototyping.

Shape Based Checking

EDS checks all your designs for connectivity violations. All elements are tested, not just track ends and pads! EDS can spot overlaps anywhere in your design and highlight them for editing. Full online design rule checking is also available.

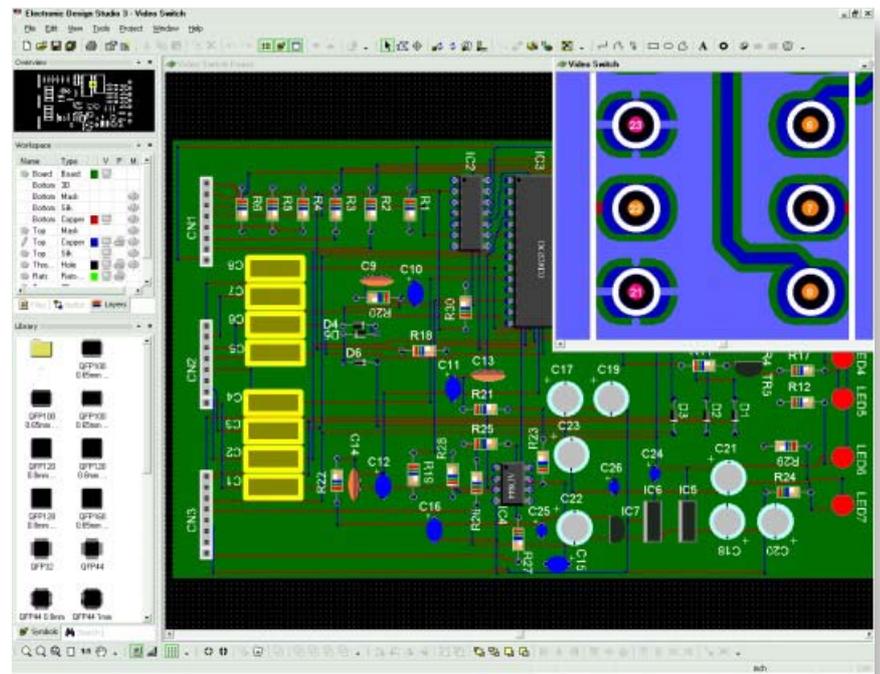
Copper pour

EDS uses a zone based copper pour system which



Cross Probing & DRC Support

"excellent product" A.J.



gives you maximum flexibility. Zones are areas of any shape on your PCB. Any object inside the zone (pad, track, etc) will be automatically surrounded with a gap of the desired size. As you update your designs, zones are updated automatically. You can also associate zones with any net (e.g. GND or VCC) to produce power planes, and objects belonging to the net will be allowed to link to the zone.

Zones support a wide range of fill styles, thermal relief, island removal, etc. You can also convert zones to polygons for detailed editing and vice versa. All together our zone system gives you one of the most powerful copper pour facilities available.

Polyblend

Polyblend is a new module which lets you manipulate copper areas in

Features

- Fully automatic capture and updating with online CADCheck
- Available with unlimited layers and resolutions down to 1nm
- Rotate components to any angle
- In-situ symbol editing
- In-situ symbol pad shape/style editing
- Intelligent pads indicate pin number and show when connections are correctly made
- Intellisnap guides cursor to pads
- Viper II autorouter available with rip up and retry and net-style support
- Manual tracking support with 'follow me' tracking, auto via and auto necking
- Copper pour module includes support for unlimited zones of any shape

new ways. Tools allow you to weld or combine shapes to form new shapes, or cut holes or shapes out of existing copper areas. Using polyblend you can create complex pad, or copper areas using simple primitive shapes.

CAD/CAM

EDS is still one of the few PCB design systems in the world that offers support for TrueType fonts on PCB designs, including export to Gerber. We support both R-274D and R-274X Gerber standards with a wide range of options to suit most Gerber plotters. Drill files can also be created with new embedded tool files.

Desktop manufacture (DTM)

New with EDS 3, Desktop Manufacture (DTM) allows you to machine and drill a PCB directly on your desktop using CAM technology. Linking to a range of CAM machines, EDS can turn your single or double sided PCB designs into machined PCB's, quickly and effectively using new 'isolation engraving' technology.

- Zones can have a range of fill styles, thermal relief, and map to any net
- Viper II autorouter works together with zones to minimise rat lines
- Split power plane support
- '3D look' PCB symbols, great for presentations or reports
- Overview window gives birdseye view
- Unlimited Undo/Redo
- CAD/CAM export supports R-274D and R-274X Gerber formats and NC-Drill with embedded tools table. Additional support for older photoplotters
- Netlist Export
- Automatic rat line optimisation
- Desktop manufacturing for PCB production on the desktop. Links to a range of CAM hardware
- Extensive libraries

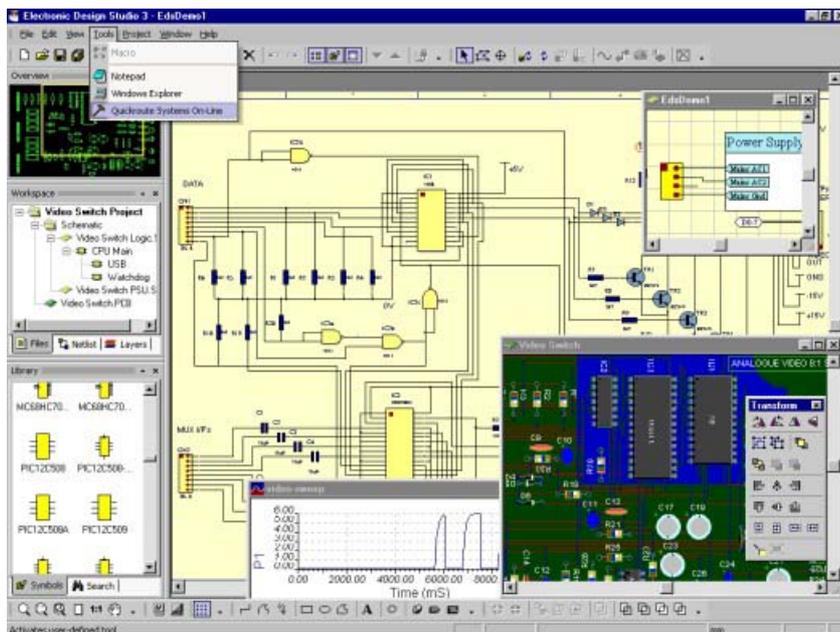
Project Management

One of the major features of EDS is its ability to manage projects. Because EDS can open schematic, PCB and netlist documents at the same time, you can manage the entire project from within EDS.

File, module and netlist navigators let you instantly jump to pins, pads, gates and symbols with the click of a mouse button. You can also cross-probe effortlessly between schematic and PCB, and vice versa.

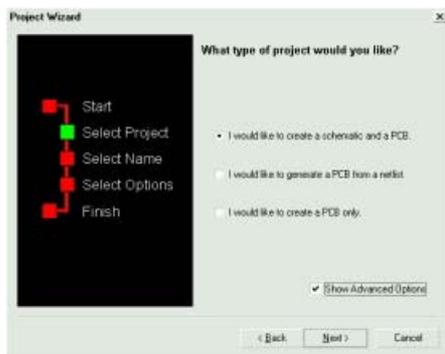
Another advantage of the project based architecture of EDS 3 is that the PCB document can also be kept up to date with your schematic/netlist at all times, our built in CADCheck system ensures that.

EDS also includes a Project Wizard which in a few simple steps creates all the necessary documents for your new design. The Project Wizard can also be used to automatically generate a PCB from a netlist created by another EDA package.



Features

- Project Wizard sets up new projects in seconds.
- Project types include schematic/PCB, netlist/PCB and PCB only.
- Docking workspace window provides access to all the documents in your project. Click on a document (or module) to instantly open and activate the document.
- Symbol annotation is managed on a project wide basis.
- Netlist manager lets you jump to any symbol, gate or terminal on your schematic or PCB in an instant.
- Cross-probe between netlist/schematic/PCB
- Layer manager lets you set up layers for printing, CAD/CAM output, and also lock layers if required.
- Fully customisable toolbars including the facility to create your own toolbars with your own button designs!
- Docking windows can be repositioned or floated as required.
- Create your own accelerator mappings
- Separate settings are stored for each user
- User defined tools menu



Project Wizard

Feature Matrix

EDS 3 is available in 3 main versions; Personal, Advance and Ultra. This feature matrix shows the major differences between the versions.

	Personal	Advance	Ultra
Schematic Design & Capture	✓	✓	✓
Multiple Sheet Schematics	✓	✓	✓
Power rail and bus support	✓	✓	✓
SPICE/XSPICE Simulation	✓	✓	✓
Viper II Autorouter	✓	✓	✓
CAD/CAM R-247D/X and Drill	✓	✓	✓
Realtime Connectivity Checking	✓	✓	✓
In Situ Symbol/Pad Editing	✓	✓	✓
Copper Pour with unlimited Zones		✓	✓
Online Design Rule Check		✓	✓
Viper II Rip up and Retry Module		✓	✓

	Personal	Advance	Ultra
Support for Autoroute Net Styles		✓	✓
Cross Probing		✓	✓
Tags for Offsheet Linking		✓	✓
Polyblend Tools			✓
Heirarchical Schematics			✓
Auto Track Mitre/Smooth			✓
Unlimited Layers			✓
Copper (polygon) optimisation			✓
Layer Removal			✓
Desk Top Manufacture (DTM)			✓
Resolution	1um	1um	1nm

Minimum Requirements

IBM® PC compatible with a Pentium® or compatible processor, Windows® 95/98/ME/NT 4/2000/XP, 32M ram, 100M hard disk space, SVGA (256 colour or more) graphics and a mouse or other pointing device. More memory may be required when editing larger PCB designs.

Copyright © 2002 Quickroute Systems Ltd. All rights reserved. E. & O. E. Specifications subject to change without notice. The Quickroute Systems logo is a registered trademark and "Quickroute" is a trademark of Quickroute Systems Ltd. All other trademarks are acknowledged and respected.

Quickroute Systems Ltd
 Dean Clough Halifax HX3 5AX UK
 Tel 01422 255010 Fax 01422 255014
 Email sales@dotqr.com
 Web www.dotqr.com