

Easy-PC for Windows

Version 11.0 Update

Design Calculators

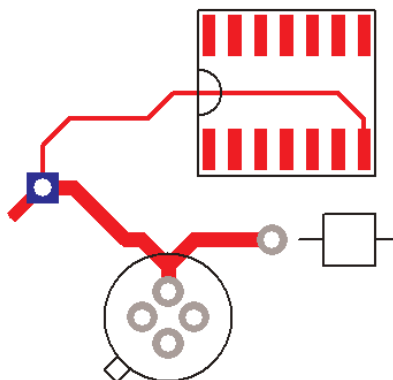
You can select tracks or vias in your design and perform basic electrical calculations on them such as:

- Width of a track required for a given current
- Impedance of a selected track
- Optimum copper density to use
- Approximate temperature rise on a track
- Resistance of a via given its physical characteristics

These useful calculators produce approximations of actual circuit performance to aid your decision process in completing a successful layout. Also built into this dialog is a standard scientific calculator - no more searching the desk when you really need one.

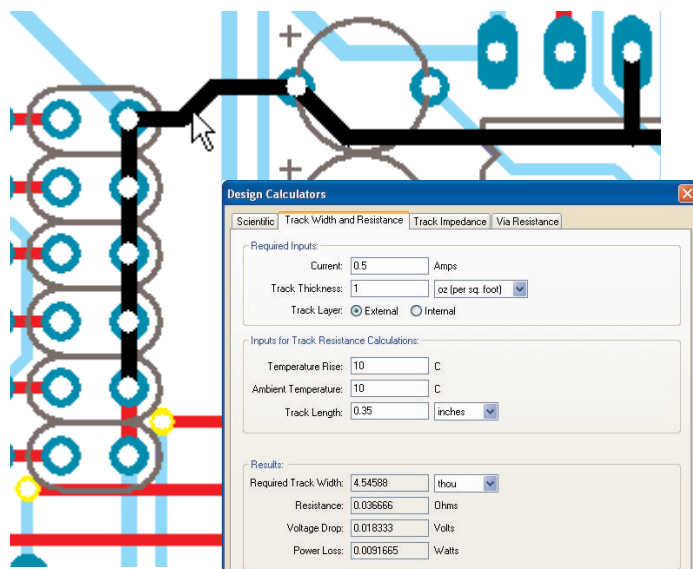
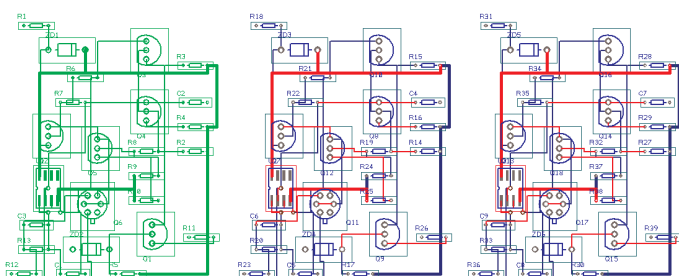
Star / Delta Points

This allows you to connect two or more related nets together, such as Analogue and Digital ground at a single point, but have them act independently in the netlist and with no DRC errors.



Apply Layout Pattern

The new feature is used for replicating design 'channels' or repetitive circuitry. For example, imagine an RGB circuit with three channels. Use the Group option in Schematics to bind the channels, then in the PCB design, place and route one channel. Using the new Apply Layout Pattern option, copy the placement and routing of the first channel to produce a fully laid out PCB.



Design Calculators help your day-to-day work process

Design Revision Analyser

Automatically identify any alterations made between design revisions and generate a report of the differences. The design revision analysis feature eliminates the time-consuming and potentially error prone task of manually finding all the changes in your designs as they progress through new revisions.

Schematic Design Rules Checking

Perform electrical rules checking on your Schematic design to ensure that:

- Components with 2-pins aren't bridged
- Bus terminals are not single nets
- Coincident items on different nets
- Net pins have a connection
- Unconnected pins are reported
- Split nets are reported
- Unfinished nets and connections are reported
- Unlabelled nets are reported

Auto Rename in Schematics

Now you can automatically rename nets and components in your Schematics design. Gaps in name sequences can also be collapsed to make them sequential, saving you time.

Auto Rename Nets in PCB

Also available is Auto Rename Nets in the PCB design editor. It only had Rename Components previously.

Groups

Groups items together in either the Schematic and PCB design editors so they act as one design item. Groups made in Schematics are automatically passed through to the PCB editor; these can also be used in the Apply Layout Pattern feature.

Shape Editing Panel

Create and edit any shapes in the design using the numeric based shape editing panel. Type in the lengths of sides to easily and accurately create the shape. A huge time-saving feature.

Shape Information

Points

Rel

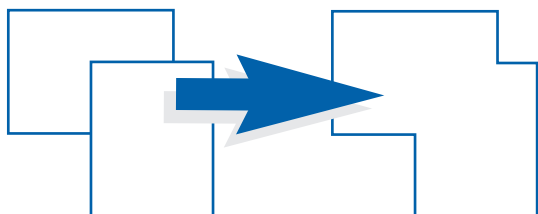
Auto

Closed

Type	X	Y
Start Point	149.86	6.35
Line To	149.86	-93.65
Line To	-0.14	-93.65
Line To	-0.14	6.99
Line To	52.07	6.99
Line To	52.07	-25.40
Anti Clockwise To	80.96	-25.40
With Centre At	66.52	-25.40
Line To	80.96	6.35
Line To	149.86	6.35

Merge Shapes

Merge two or more shapes together to produce a single entity. More complex copper shapes can also be produced using shape subtraction.



Dotted and Dashed line styles

This will also appeal to users as an general all-round enhancement to improve documentation.

Track Analysis

Accurately report track information such as the track length, the number of vias for each net and routing path efficiency. This report is especially important when track topology is critical and precise data about it is required for analysis.

BRANCHING MULTI-DROP

Net Name	Pins	Vias	Track	Tree	Efficiency
A2	5	5	8980.3268	3187.3898	0.35
A13	4	5	8201.1299	4046.9685	0.49
A15	5	2	9089.9528	5145.5197	0.57
A5	4	4	3199.2717	1882.8425	0.59
+12V	31	7	24571.0709	14498.4291	0.59
A9	4	3	5142.4803	3129.7047	0.61
A14	8	3	8534.9291	5721.1102	0.67
N_WR	4	4	7782.1063	5307.4567	0.68
D6	10	5	13457.4252	9422.5906	0.70
VCC	79	11	47347.6929	33325.8701	0.70

Additional Features

- Auto-numbered sequential back-up files
- Lock/Unlock command
- Routing and Keep-out areas for Pro-Router
- Improved DRC checks
- Layer toggling on keys
- Design based Properties values - Title, Author, Layer Name
- Auto-mirror text when changing layers
- Plated/Non-plated status on board cutouts
- Plus lots of other small enhancements