User Manual
TopoR 6.1

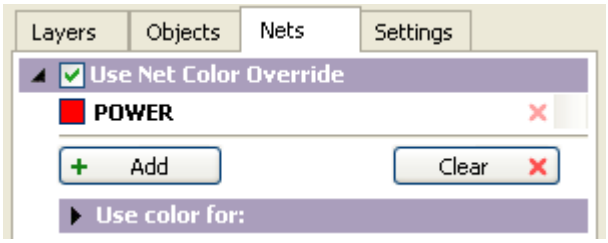
July 2015

What's New in TopoR (6.0 and 6.1)

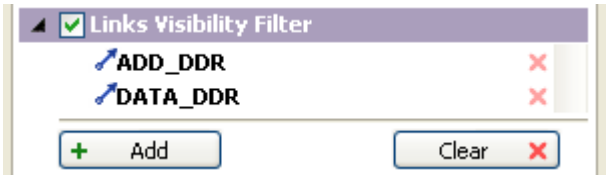
Version 6.0 was a major overhaul of how the system looked and worked. The current version 6.1 continues and builds upon the changes introduced in the previous version.

New in Version 6.1

1. The quality of multilayer routing has improved significantly.
2. Automatic placement has been implemented.
3. An option to display selected nets (signals, net groups) in a specified color has been added (**Nets** tab in the visibility control panel).



4. An option to selectively display net lines has been added (**Nets** tab in the display control panel).



For details, see Display control panel.

5. "Staggering" of close-together DRC violations has been implemented.
6. The version of the TopoR PCB format has been updated. For details, see the TopoR PCB 1.1.3 format specification on www.eremex.com.

Designs developed in version 6.0 will open with slight data omissions in the current version; scrolling and scale parameters, color and visibility settings will be lost. To make sure all information is preserved, first export from version 6.0 to TopoR PCB format, and then import the result into version 6.1.

User Interface

1.The program's look has been updated. There have been some changes to the toolbars and main menu.
2.The project control panel has been updated. The **Project History** tab has been added. Projects are sorted by opening date.
3.The **Open Containing Folder** command in Windows Explorer opens the folder with the selected file. The **Copy Project** command copies all project files to a folder you specify.
4.The autorouting control panel has been added, where you can select nets for routing and change autorouting settings. The autorouting variant table is shown when you start autorouting.
5.A control panel for the following automatic procedures has been added: calculation of wire shapes, wire route optimization, shifting of vias and components.
6. The Options dialog box now includes the **Hotkeys** tab.
7. Shortcut keys can be defined or redefined for most operations.

Topology Editor

1. The editor toolbar has been updated. Related tools have been grouped. **FreeStyle** mode is now one of the editor tools. A new tool has been added for measuring the distance between objects
2. Arc-shaped wire segments are supported in all editor modes. In **FreeStyle** mode, multi-level undo is supported.
3. There is a new context menu, where all possible operations on selected objects are available.
4. In manual wire routing mode, automatic searching and highlighting of wires in the current layer has been implemented. You can accept the suggested route (this routes the wire automatically) or ignore it and continue manual routing.
5. The object selection filter has been updated and extended.
6. You now have the choice to automatically remove vias that have become redundant due to automatic procedures.
7. **Online DRC** checks do not display bogus violations any more.

Design Properties Editor

1. Automatic procedure settings have been moved from the editor to the corresponding toolbars. Editing settings have also been moved and are now available in the **Editor Settings** dialog box.
2. Recently changed sections are now marked in the editor.
3. The undo histories of the topology editor and the properties editor are now independent.

Autorouting

1. Piecemeal routing without fixing the wires' geometric shape has been implemented. Previously laid out wires remain flexible while the optimal wire shape is calculated.
2. Differential signals can now be split automatically.
3. BGA routing has been enhanced in that it is now aware of impedors located under the BGA component.

Other

1. Data recovery after a crash has been implemented. If the crash occurred during autorouting, then autorouting results are also recovered.
2. The TopoR PCB format has been updated. For details about the changes, see the specification for version 1.1.2 of the format.
3. All designs developed in **TopoR** versions prior to 5.2 can be transferred to the current version. For that, export your design from the old version to the TopoR PCB format, and then import the resulting file into **TopoR** 6.0.

About the User Manual

This User Manual is made up of four major sections, A to D: introductory, main, auxiliary and reference. The introductory section contains only basic information about the **TopoR** system; all users who have worked with previous versions of the software can safely skip it. The main section details the use of **TopoR** and is intended for both beginners and experienced users. The Additional Features section describes functionality that only experienced users can take full advantage of. Finally, the Reference is a collection of miscellaneous information that generally needs to be looked up.

Much of the manual's content has seen considerable changes compared to the manual for version 5.4. Some sections were redone (Creating a Project, Preferences, Editor Toolbar, Action Bar, Context Menus, Selection Filter, Display Control, Data Recovery, Autorouting, Finding Variants, Conditional Routing, Via Moving (F7)), because the functionality in question was either unavailable in prior versions or implemented in a markedly different way.

A. Before You Begin

Part A of the manual is intended for users who are new to the **TopoR** CAD system. Here you will find general information about the system and instructions on installing it and getting started.

General Information about TopoR

The **TopoR** (**Topo**logical **R**outer) CAD system is a unique high-performance topological router for printed circuit boards. It has the following distinctive features:

High routing speed and excellent routing quality

A robust toolset that considerably reduces design times

This is achieved by the use of unique algorithms and an unconventional approach to solving complex problems.

The PCB editor (topology editor) in **TopoR** features unique automatic procedures, including:

Calculating an efficient geometric shape for wires using topological paths

Moving components about a laid-out board without breaking the existing layout.

TopoR is compatible with a variety of CAD systems.

File import and export formats: ASCII (**.pcb** – PCAD ASCII PCB, **.asc** – PADS), DSN (**.ses** – Specctra and Electra), BRD (**.brd** – Eagle), HKP (Mentor Graphics Expedition)

Export-only formats: DXF, Gerber and Excellon (drill file).

Technical Details and Limitations

The maximum number of routing layers varies from version to version.

Number of routing layers: 2 to 32

Total number of copper layers: up to 32

Precision: 0.1µm

Licensing

The primary (unlimited) **TopoR UN** license supports boards that contain up to 32 signal layers. In addition, several limited licenses are available:

TopoR 2L	Boards with up to 2 signal layers
TopoR 4L	Boards with up to 4 signal layers
TopoR 8L	Boards with up to 8 signal layers
TopoR 16L	Boards with up to 16 signal layers

For information about purchasing, see the company website: http://eda.eremex.com/howtobuy/.

Contact Us

OOO "Eremex" Moscow, Profsoyuznaya ulitsa, 108. Phone.: +7 (495) 232-1864, Fax: +7 (495) 234-0640

E-mail: info@eremex.com, ICQ 464808130, Skype: supporteremex, Web: www.eremex.com

For information about purchasing the TopoR system, see http://eda.eremex.com/howtobuy/. To purchase other software, write to sales@eremex.com.

We welcome any questions regarding the installation and use of our software, comments, bug reports and enhancement suggestions at info@eremex.com. Also note that you can ask a question or search for an answer on the forum at http://forum.eremex.com/index.php?/forum/2-topor/

During one year following a license purchase, all customers receive free technical support that includes assistance at any design stage and software updates. To request technical support, contact support@eremex.com

In addition to this manual, you can find more information about TopoR in the FAQ section of our website: http://eda.eremex.com/support/faq.html

Installation

System Requirements

The following is the minimum hardware and software configuration.

PC-compatible computer with an Intel® Pentium® III-1000 MHz processor or higher

Operating system: Microsoft® Windows® 2000 (SP3), XP (SP2 or SP3), Vista, Windows 7, Windows 8

Microsoft® Internet Explorer 5.0

Windows Installer 3.0 or higher

RAM: 512 MB

Free hard disk space: 100 MB

SVGA monitor and a graphics adapter, 256 colors, 1024x768.

Mouse with a scrolling wheel

Installing the Software on the Hard Drive

Do not connect the dongle before you have installed the dongle drivers.

1. Run the installer
2. Follow the instructions.
3. After the files have been copied, the dongle driver installer will start automatically.

After the dongle drivers have been installed, connect the dongle. In the folder you specify, the installer creates the **BIN** subfolder where the executable file are copied. The **HELP** subfolder contains documentation files. The **EXAMPLES** subfolder contains more subfolders with complete PCB project files, one project per subfolder. Make sure you have write access to the newly-added folders when you work with the program.

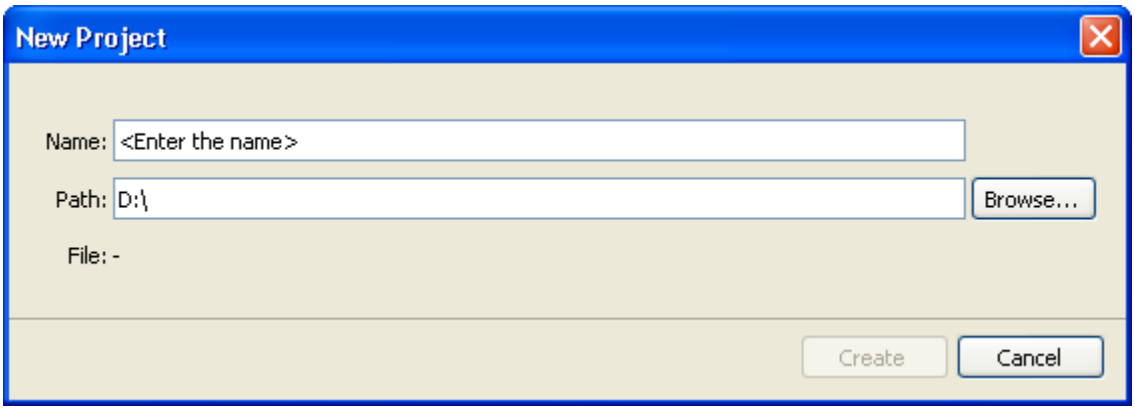
The uninstaller is launched the conventional way (for example, using the **Programs and Features** facility in the Control Panel). Uninstalling the program removes the **BIN** folder and all default example files. If you want to keep them, copy the **EXAMPLES** folder elsewhere.

Getting Started

Working with PCBs in **TopoR** consists of the following steps:

1. Edit parameters
2. Perform autorouting
3. Edit topology manually
4. Check for compliance with design rules
5. Result output

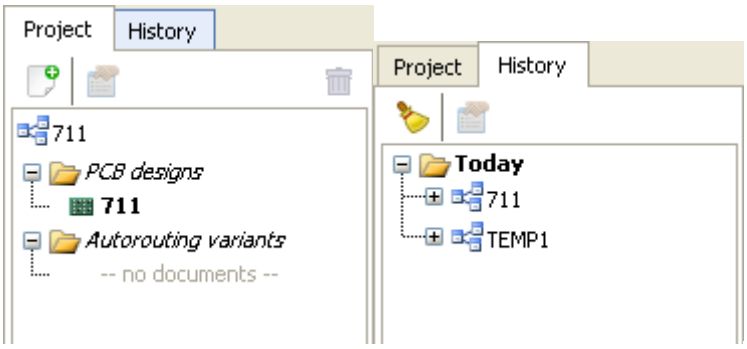
The suggested typical scenario is to perform step 1 once and then repeat steps 2 through 4 until a suitable result is achieved; then proceed to step 5.
To start working in the TopoR CAD system, open an existing project or create a new one. To create a project, click the **Create Project** button located on the toolbar, or click **File ► Create Project** in the main menu. In the **New Project** dialog box that pops up, specify the project name and the path to the project file.



The name you specify is automatically appended to the entry in the **Path** box. You can edit the resulting path manually. You can also create a project during the import of a design.

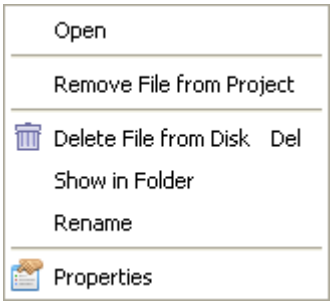
Creating a Project

The panel has two tabs: **Project** and **History**.



The **Project** tab shows the contents of the current project. The **History** tab lists recently opened projects. When you open a project from the **History** tab by clicking **Open** in the context menu, the specified project becomes the current project.

The context menu for files on the **Project** tab looks like this:



Open — Open **`.fsx`** and **`.fsb`** files in the editor (**`.fsb`** store autorouting variants).

Continue Autorouting (available only for **`.fsb`** autorouting variants) — Optimize the selected variant (see Autorouting). All autorouting parameters are reused from when the variant was produced.

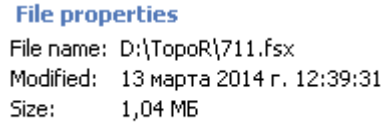
Remove from Project (available only for **`.fsx`** designs) — Exclude the file from the project but keep it on disk.

Remove from Disk — Remove the file from the project and delete it from disk. After deletion, the file cannot be restored.

Open Containing Folder — Open the folder with the file in Windows Explorer.

Rename — Available for file names and the project name. The standard file system restrictions apply to file names (for example, you cannot use the following characters: **`> < | ? * / \ : " )`**.

Properties — A dedicated **File Properties** panel opens.



.

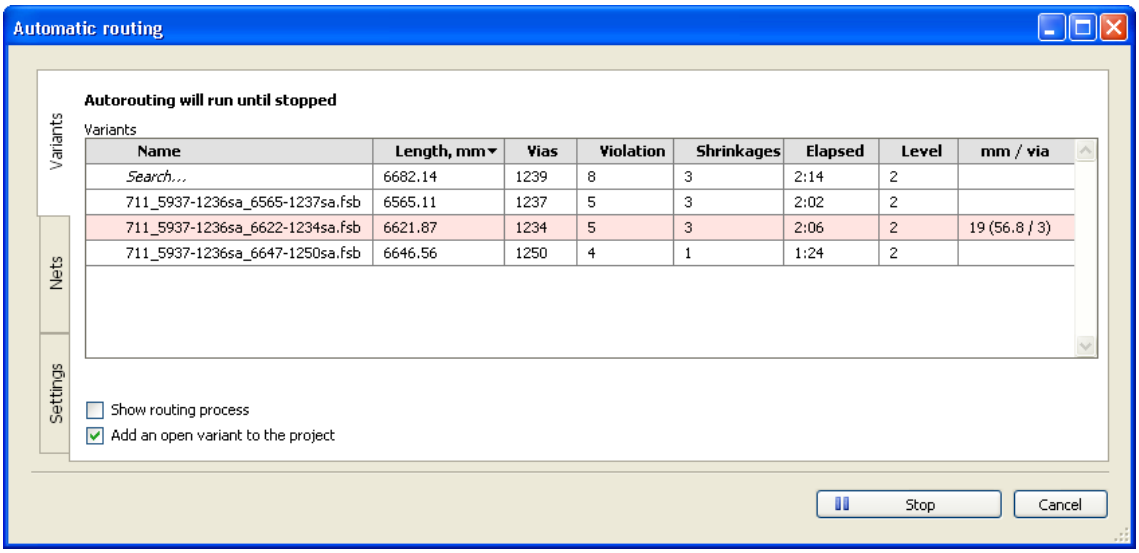
If the file format version is not supported by **TopoR**, such a file is marked with this icon:  .

Files that are missing are marked with this icon: 

Adding Files to the Project

Clicking the **Add File to Project**  icon opens the file selection dialog box. Only **`.fsx`** files can be added this way.

Another way to populate the project is to add autorouting variants (after optimization has been stopped). This lets you include any selected variants in the project. When you open a variant in the editor, it is added to the project automatically.



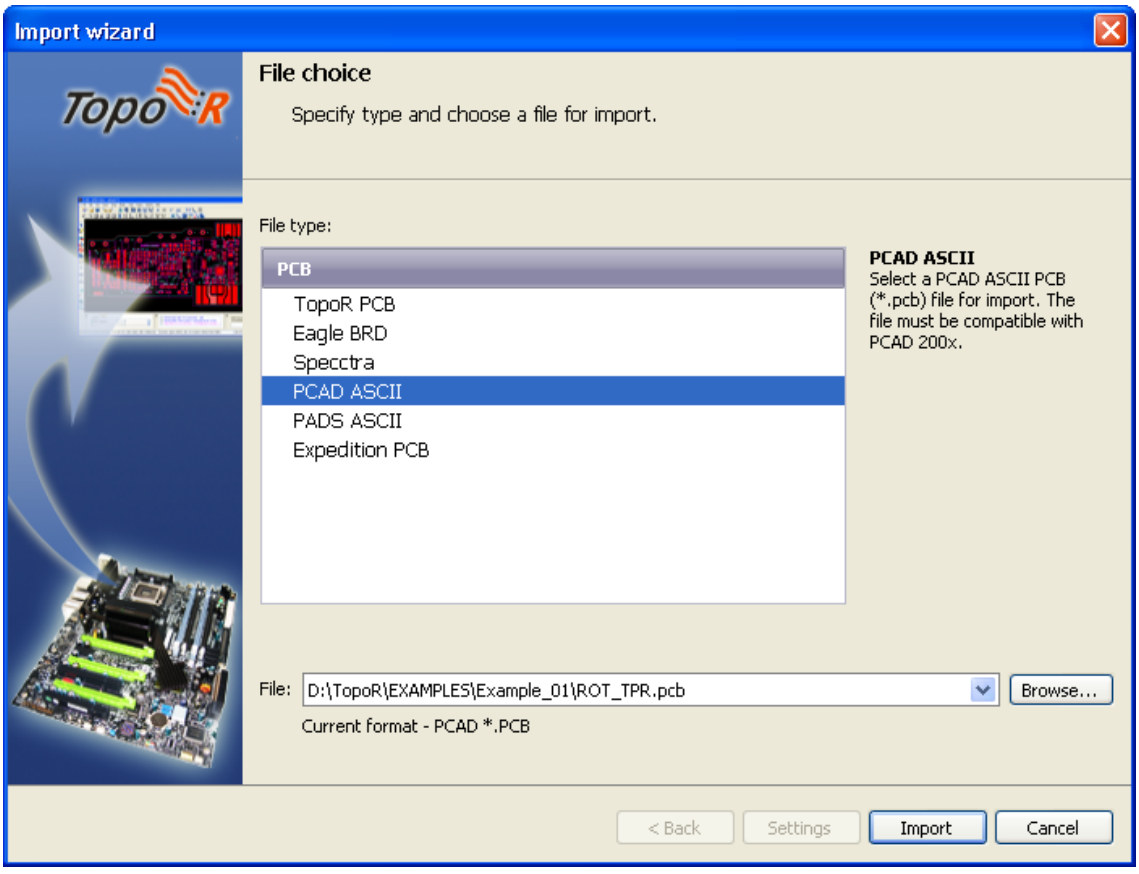
To add all current variants, click the **Add all variants to the project** link.

Importing Designs

To start working on PCB design in **TopoR**, first import a file produced by a different CAD system. Files written by the following CAD software can be imported:

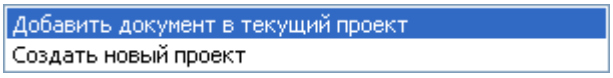
Specctra (**.dsn**) Specctra/Electra;
PCAD ASCII PCB (**.pcb**) PCAD 2000 – 2006;
PADS ASCII PCB (**.asc**) PADS 3.5 – 2005.1;
Expedition PCB (**.hkp**) Expedition 2005 – 2007
Eagle (**.brd**)
TopoR PCB (**.fst**) An open plain text format for defining PCB designs made in **TopoR**.

The **Import** button is located on the toolbar. Clicking this button (or **File ► Import** in the main menu) opens the **Import Wizard**. On the first step, select the format of the file you want to import and the file itself. If the format you selected supports import configuration, then the **Settings** button becomes available. If the current folder has no files of that format, then the **Import** button in the wizard is not available either, and the path is bright red.

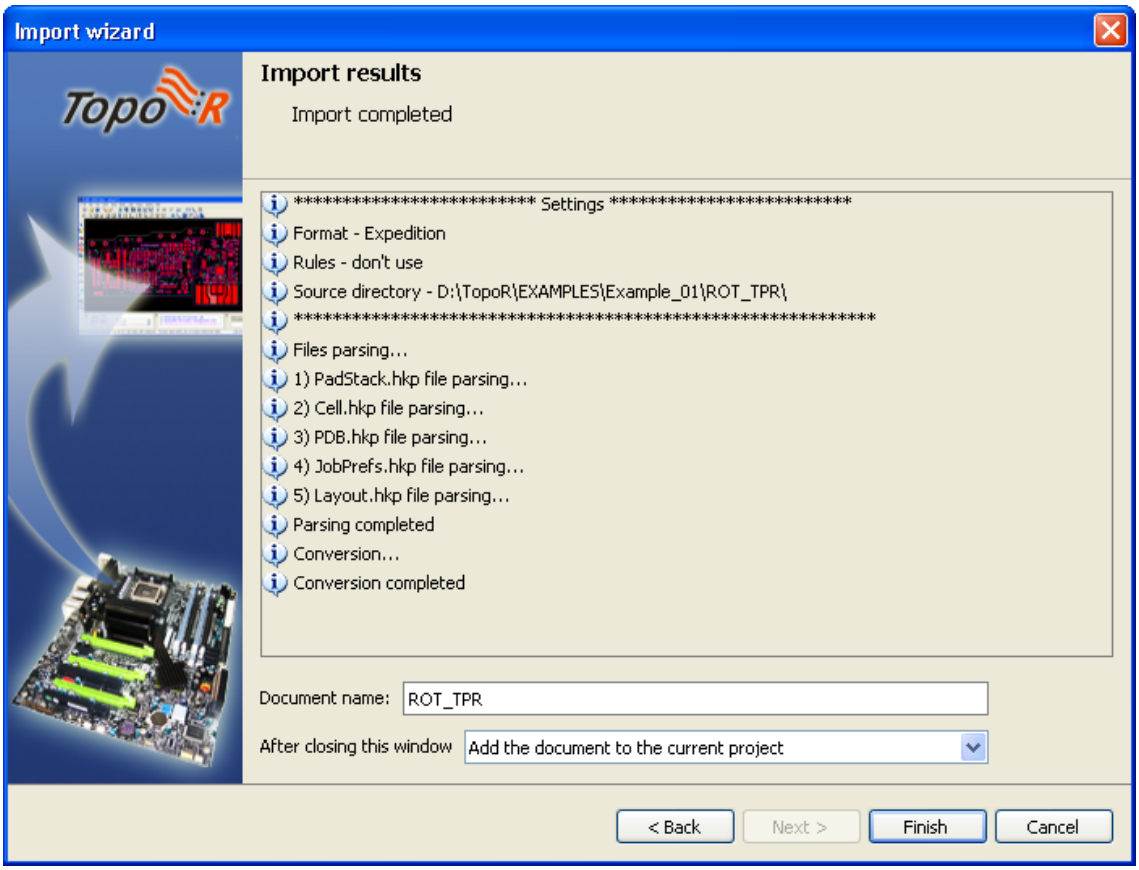


After a successful import, the last step of the wizard shows details about the progress of the import. At the bottom of the page, specify a name for the newly created document.

If a project is already open in **TopoR**, you can add the new document to the project. For that, in the list under the document name box select **Add document to current project**.



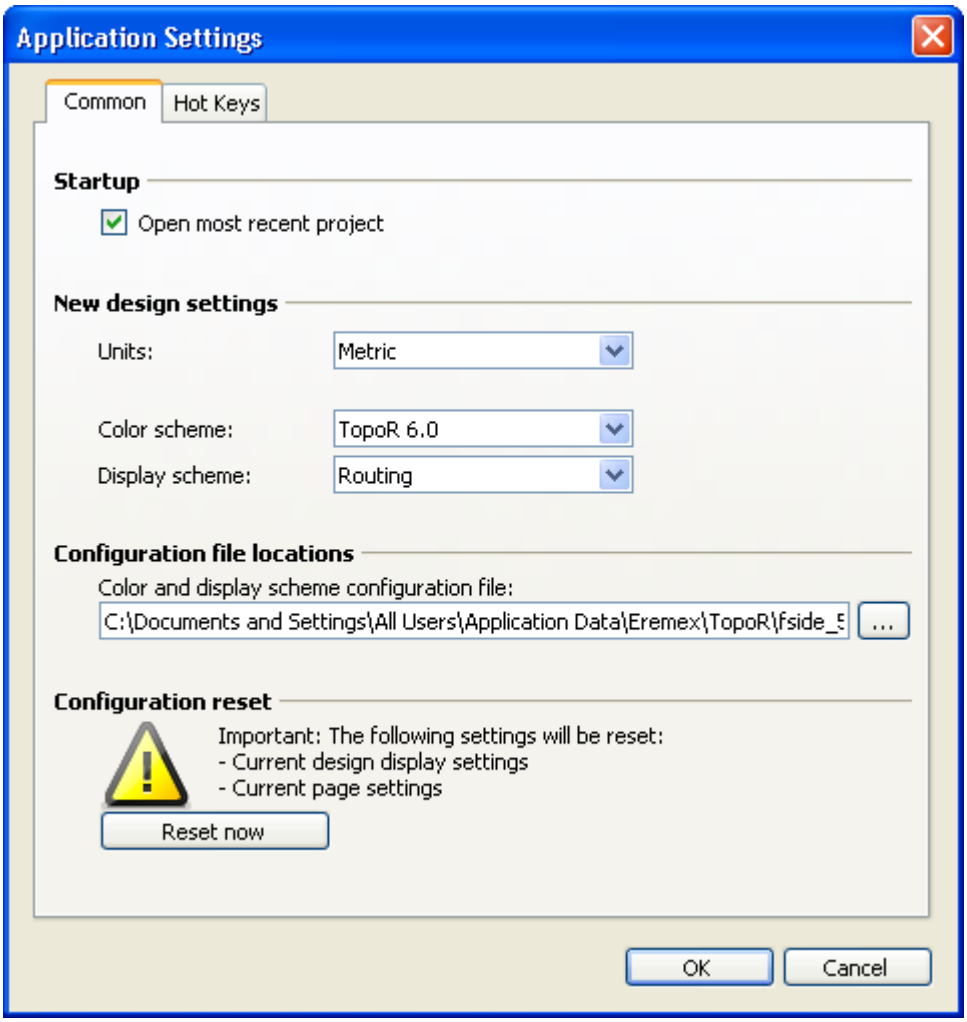
Selecting **Create a new project** opens the **New Project** dialog box (but closes the current project first). After you have created a project, the document will be included in the new project and opened in the topology editor.



Preferences

Selecting **Tools ► Application Settings** in the main menu opens the **Application Settings** dialog box. These settings are global, meaning that they apply to the program in general, not just the current document.

General



Use the **Open most recent project** in the **Startup** group of options to automatically reopen the project that was opened last.

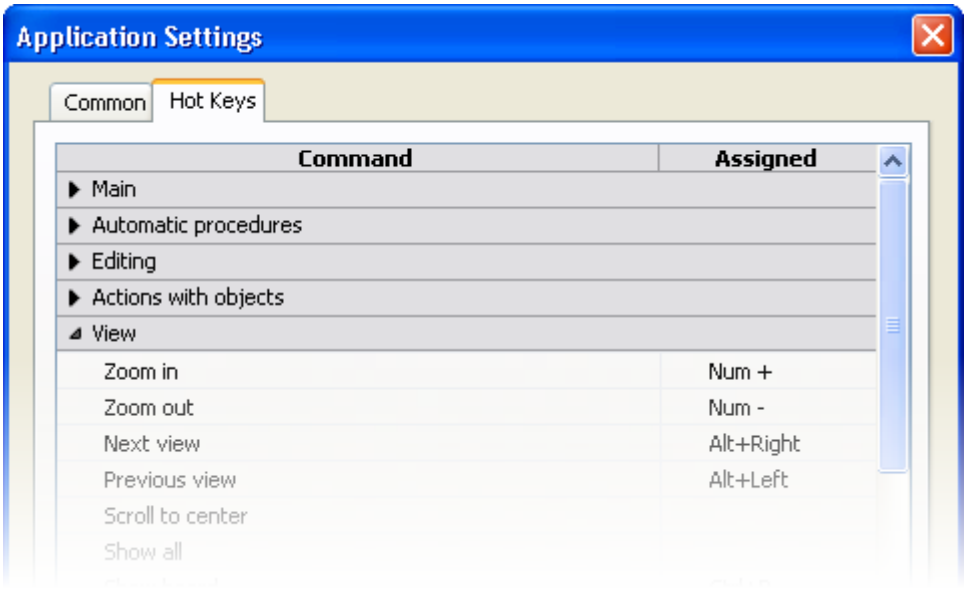
The **New design options** group of settings lets you choose the units (metric or imperial), color scheme (P-CAD 2006, Pulsonix, Topor 4.3, Print, Inverse, Mask) and display scheme (by default, the scheme is selected from the following list: Routing, All On, All Off, Top, Bottom). All of these settings are applied to the design when a new project is created or when an existing project is imported.

The **Color and display schemes** group of settings contains a reference to a file with option schemes. If you specify a file that doesn't exist, the file will be created and made the current configuration file. All current schemes will be saved in that file.

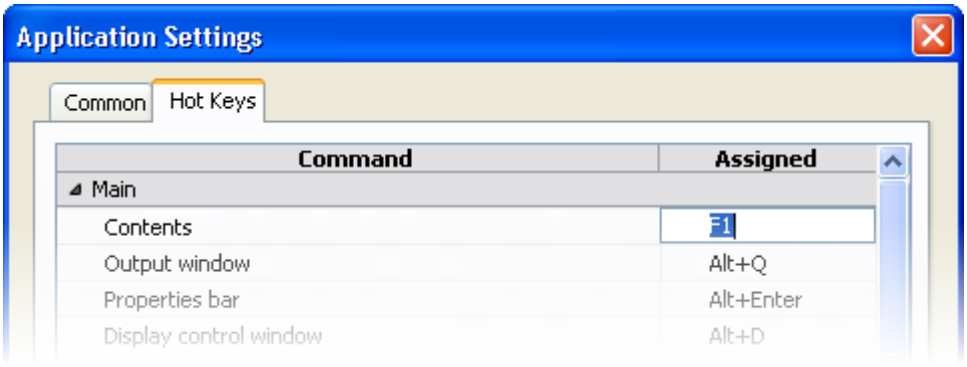
The **Revert settings** group lets you restore the defaults for all current options.

Hotkeys

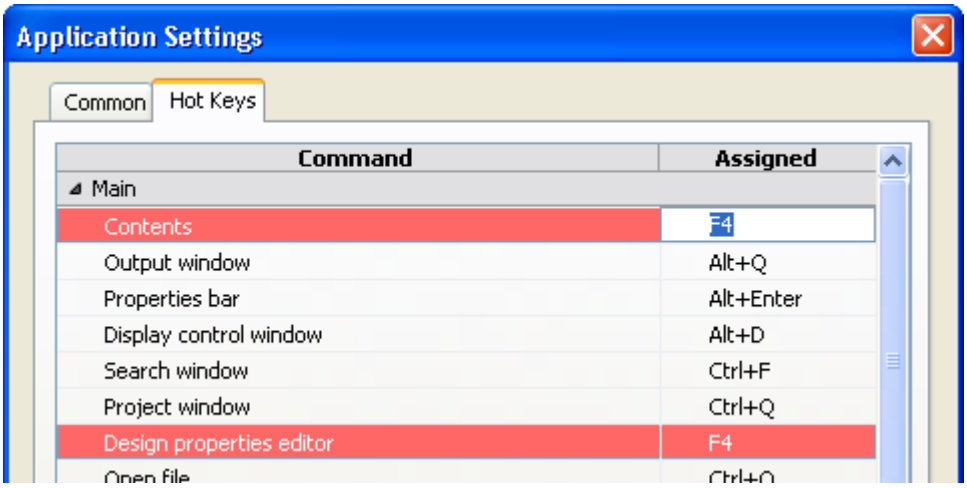
The **Hotkeys** tab shows all current hotkey assignments grouped into seven categories: **General, Automatic procedures, Editing, Object operations, View, Select, Manual wire routing**.



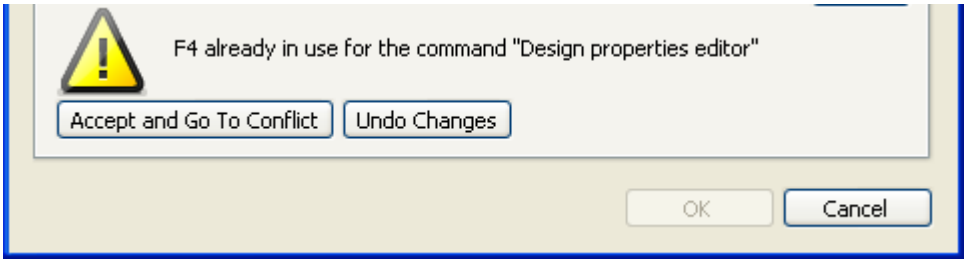
For a full list of hotkeys, see the last section of this manual. (D. Reference, Hotkey List)
Keys that cannot be redefined are marked gray. To redefine a key for an operation, click its entry. This highlights the current hotkey assignment, and a key press is expected.



You can assign almost any key on the standard keyboard (exceptions are listed below). Hotkeys with or without modifiers (**Ctrl, Shift, Alt**) are supported. When you press the modifier key alone, this is indicated as long as the key is held. To exit key assignment mode, press a valid key that is not yet defined. If it is defined, the conflicting pair of assignments is highlighted red:



At the bottom of the tab, a message shows details about the conflicting hotkey assignments:



Pressing **Del** cancels the current assignment and exits assignment mode.

Entries that are different from default assignments have a pale yellow background. To revert all changes, click the **Defaults** button.

List of exception keys that cannot be assigned

Esc
Tab
CapsLock
PrintScreen
ScrollLock
NumLock
Pause/Break
Context_Menu
Win

Other operations that support hotkey assignment

In addition to the list of hotkeys in the Reference, some operations can have hotkeys but by default they don't.
Automatic procedures / DRC
Editing / Create label
Object operations / Convert to serpent
Object operations / Zip wires
Object operations / Unzip wires
Object operations / Pour all copper areas
Object operations / Unpour all copper areas

Object operations / Repour all copper areas
View / Center
View / Show all
View / Metal layers on/off
View / Copper areas on/off
View / Grid on/off
View / Wires on/off
View / Vias on/off
View / Only active layer on/off
View / Keepouts on/off
Select / Invert selection
Display schemes / Routing
Display schemes / All on
Display schemes / All off
Display schemes / Top
Display schemes / Bottom
Color schemes / P-CAD 2006
Color schemes / Pulsonix
Color schemes / Topor 4.3
Color schemes / Print
Color schemes / Inverse
Color schemes / Mask

Note that a hotkey can be assigned for any display scheme.

B. Getting Started with Designing in TopoR

This part of the manual is the most important one. The sections contained in it are intended for engineers and other specialists working on PCB design and using TopoR. The information included is sufficient for becoming productive with the system.

Editing Parameters

Design parameters include:

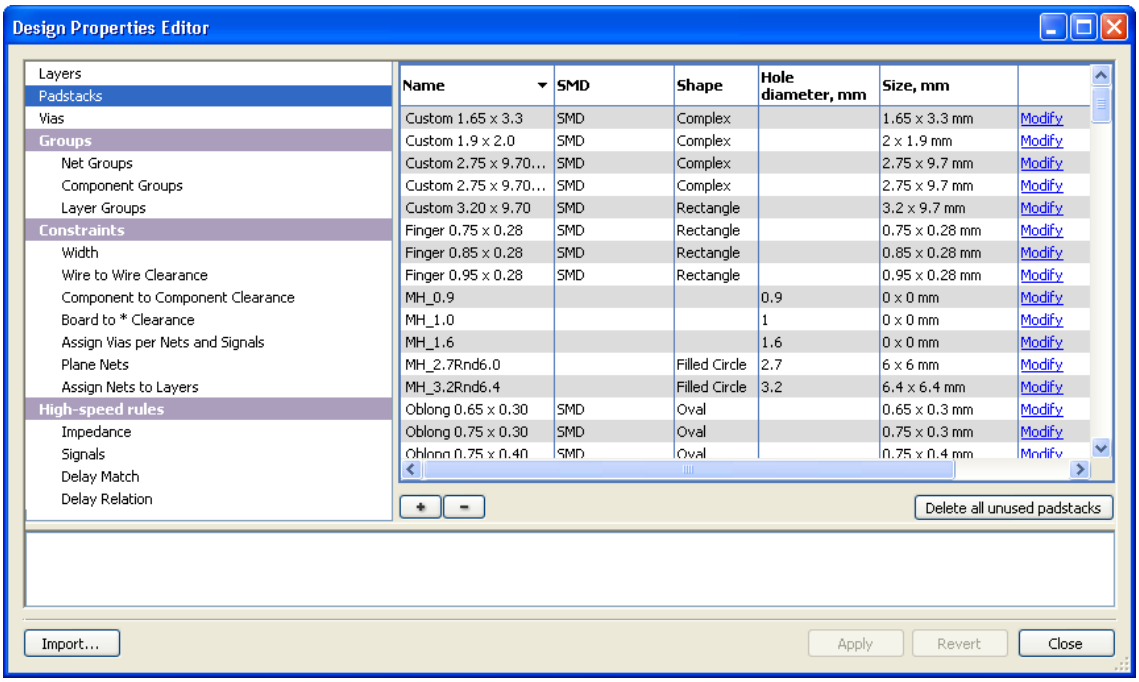
The layer stack
Padstacks
Types of vias
Rules for automatic and manual routing

In addition, this section of the manual describes object grouping and configuration editing.

The High-speed rules panel of the properties editor is described in part C, in the section about high-speed appliances.

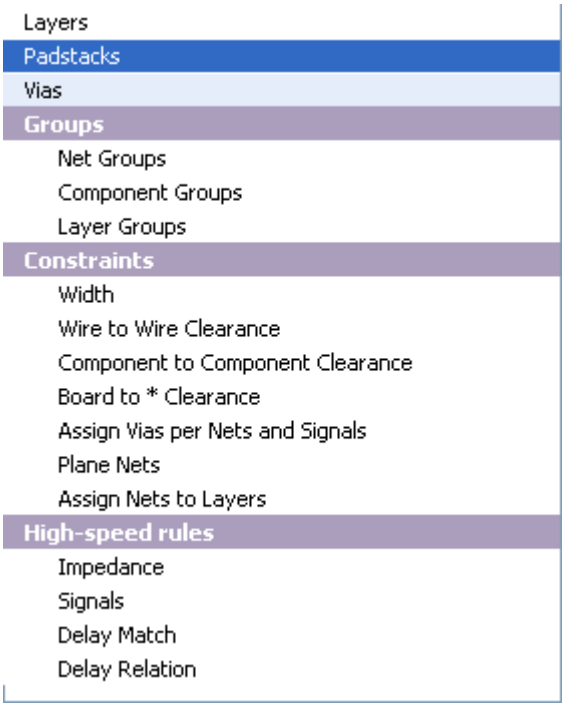
General

To open the **design properties editor**, click **Design ► Properties Editor** in the main menu or press **F4**, or click the corresponding icon  in the toolbar.



1. Navigation Pane

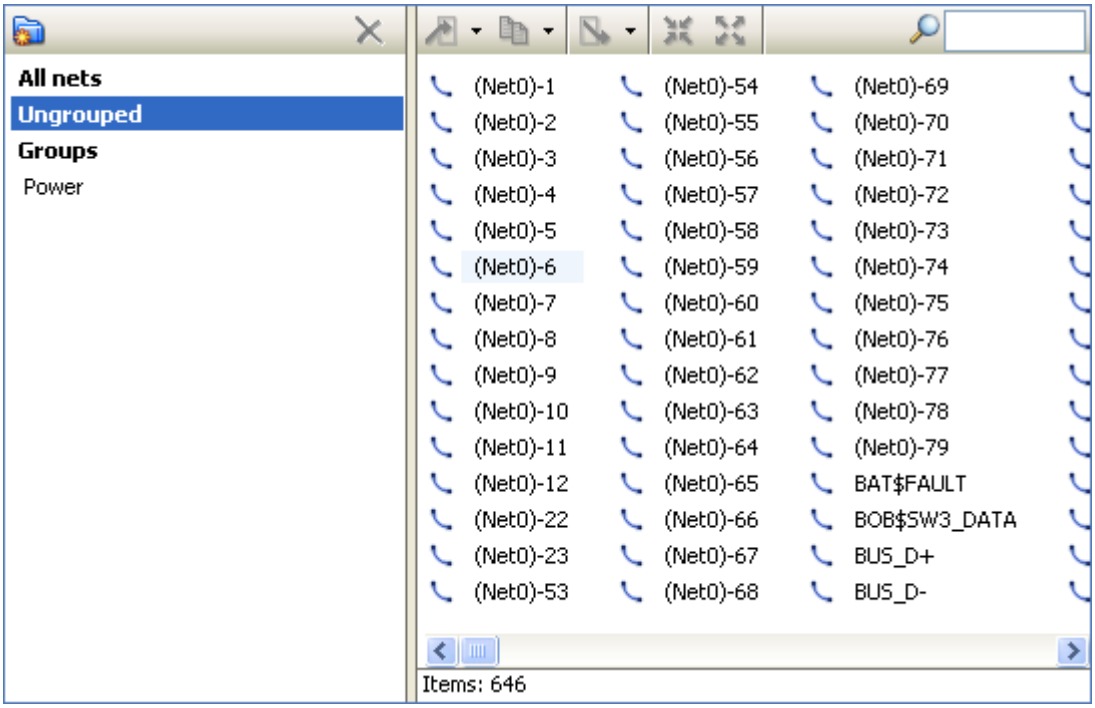
The navigation pane contains entries that you can click to change the contents of the display pane.



2. Display Pane

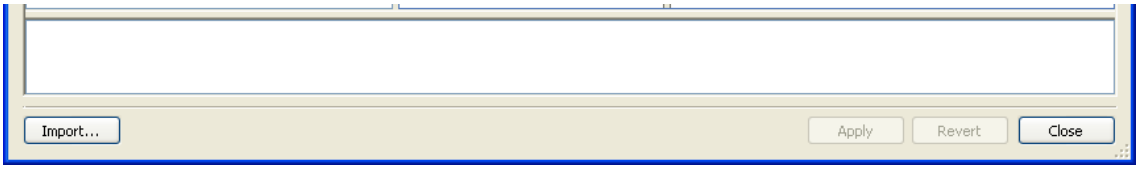
The contents of the display pane depend on what is selected in the **navigation pane**. Whenever design properties are changed, the **Apply** and **Reset** buttons become available. Clicking **Apply** puts all the changes into effect and keeps the editor window open. Clicking **Reset** rolls back all changes that have not been applied. Clicking **Close** either closes the editor immediately or, if there are changes, prompts

whether to apply them. Click the **Import** button to copy properties from another design. While the **Design Properties Editor** dialog box is open, you can continue editing the board topology.



3. Information Bar

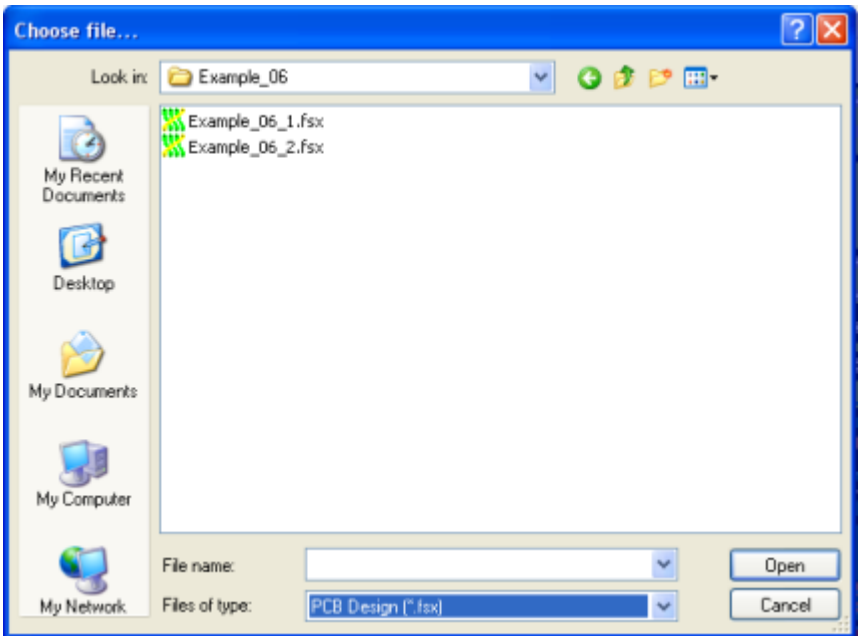
The **information bar** is under the navigation pane and display pane. It shows messages about errors, design rule violations, etc.



Importing Properties

As mentioned previously in the section about importing, **TopoR** is not a standalone development tool but should be used in conjunction with other CAD systems. Therefore, situations might occur where you need to make changes that are not possible in **TopoR** but can be done in external CAD software. For example, you might require adding or removing a net or component. Such changes should be made in an external CAD system, and then the design should be reimported into **TopoR**. Quick property copying (import) is available for easily restoring rules and settings.

Clicking the **Import** button in the lower left part of the **design properties editor** opens a file selection dialog box so you can specify the file that stores the properties you need. This can be a **`.fsx`** or **`.fst`** (TopoR PCB) file.



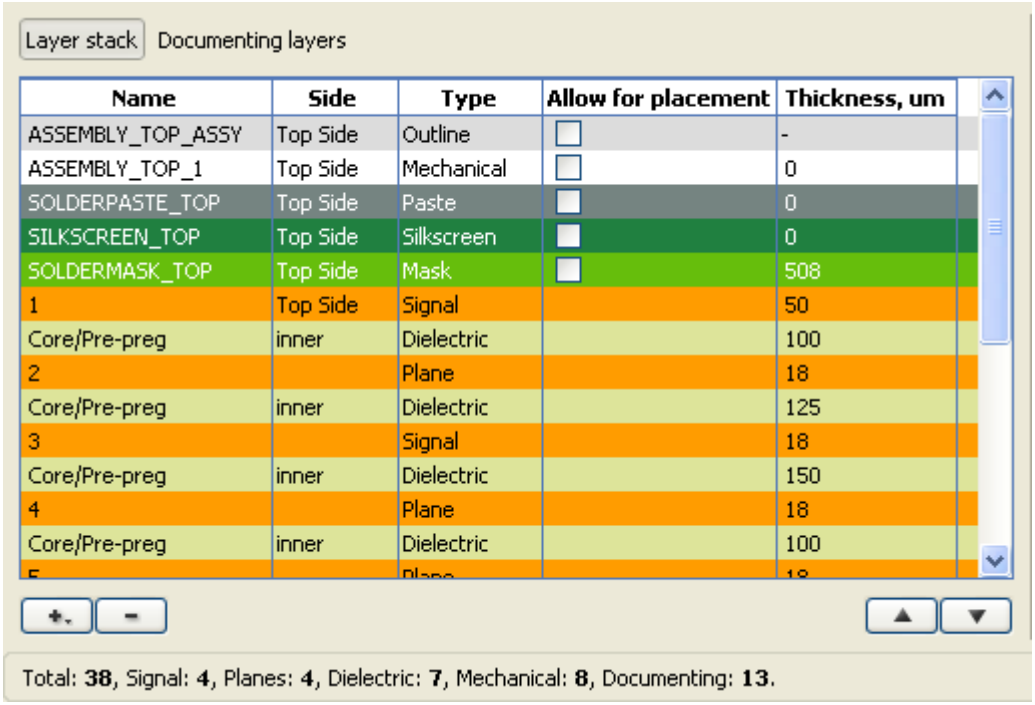
The following properties will be copied over:

- Net groups
- Component groups
- Layer groups
- Signal groups
- Wire width rules
- Wire-to-wire clearance rules
- Component-to-component clearance rules
- Via type assignments
- Backup layer nets
- Per-layer net assignment
- Net properties
- Board edge clearance rules
- Signals (including assigned links)
- Impedances
- Per-group equalization rules
- Mutual equalization rules

All current properties are removed. If the design defines width rules that are not present in the specified file, they are also removed.

Layers

The **Layers** section has two tabs: **Layer stack** and **Documenting layers**. The **Layer stack** tab shows the layers in a table.



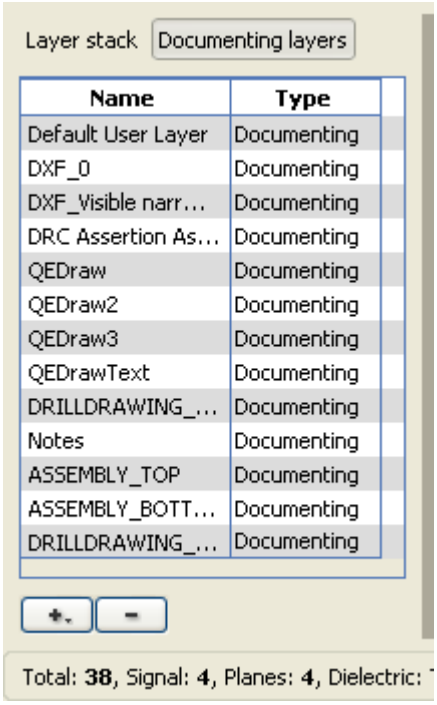
1. **New Layer** button
2. **Delete Selected** button
3. **Move Selected Up** button
4. **Move Selected Down** button

The table lets you rename layers, change their type and set their thickness. Miscellaneous mechanical layers support the **Consider during placement** option. If it is selected for a layer, then the outlines of the components on the layer will be taken into account during autoplacement procedures and when components are moved automatically or manually.

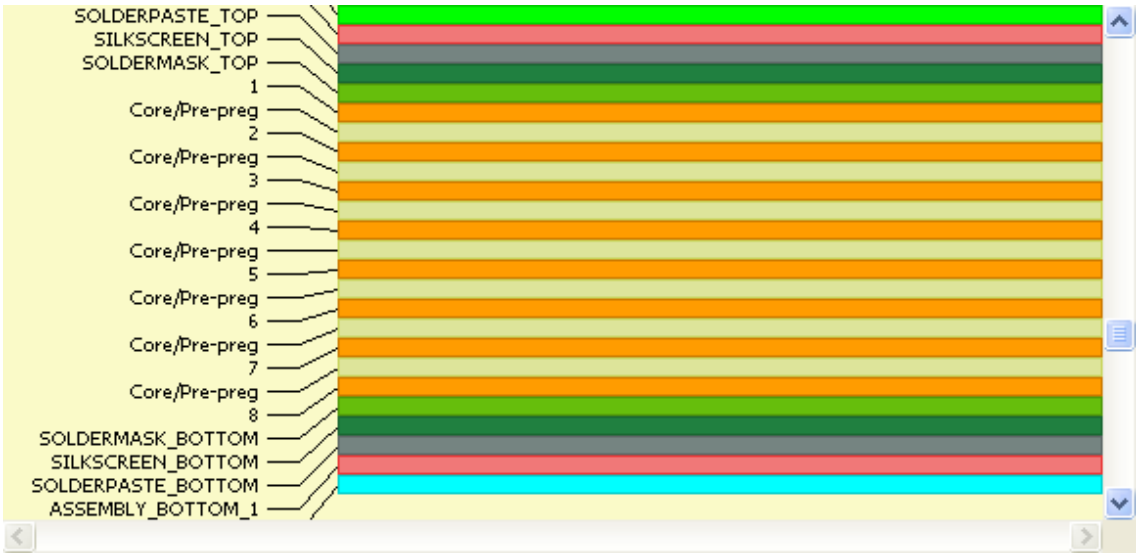
To change the order of layers in the stack, use the **Move Selected Up** and **Move Selected Down** buttons. To the right of the table is the layer stack diagram. It has two display modes: uniformly-thick and proportionately-thick (according to the thickness values in the table) layers.

To create and delete layers, use the following buttons: and . When you create a layer, select its type in a drop-down list.

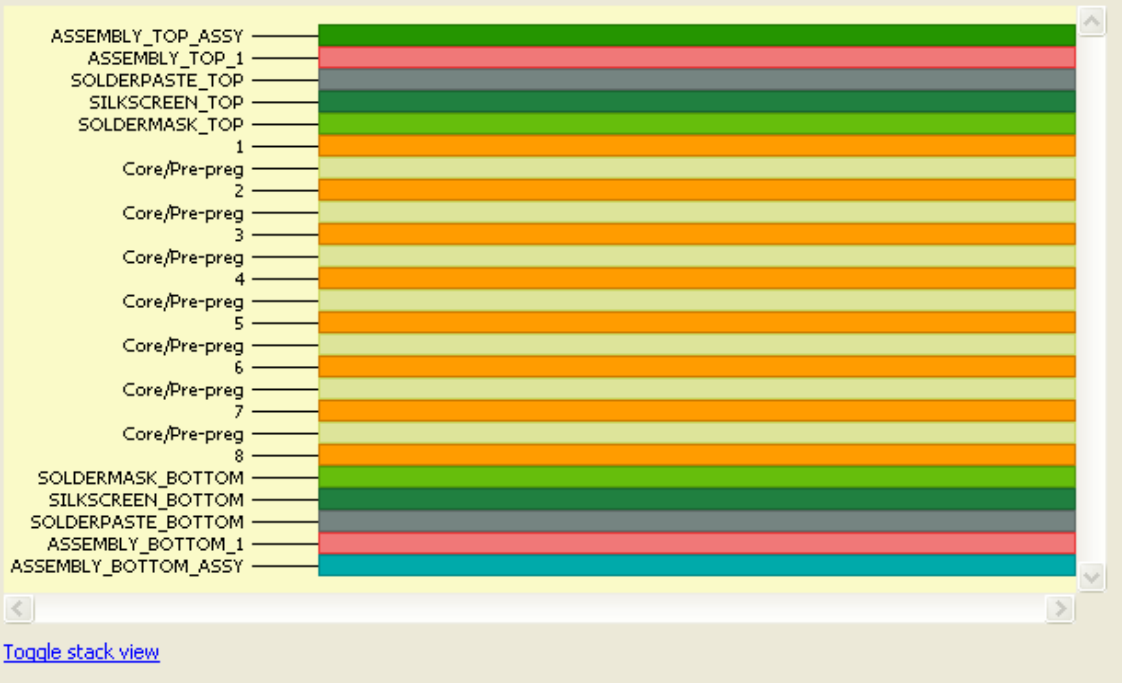
The **Documenting layers** tab shows layers that have the Documenting type.



On the right, the **Layers** pane shows the layer stack diagram.



You can use the “Toggle stack view” link under the diagram if necessary:


ASSEMBLY_TOP_ASSY
ASSEMBLY_TOP_1
SOLDERPASTE_TOP
SILKSCREEN_TOP
SOLDERMASK_TOP
1
Core/Pre-preg
2
Core/Pre-preg
3
Core/Pre-preg
4
Core/Pre-preg
5
Core/Pre-preg
6
Core/Pre-preg
7
Core/Pre-preg
8
SOLDERMASK_BOTTOM
SILKSCREEN_BOTTOM
SOLDERPASTE_BOTTOM
ASSEMBLY_BOTTOM_1
ASSEMBLY_BOTTOM_ASSY
Toggle stack view

Padstacks

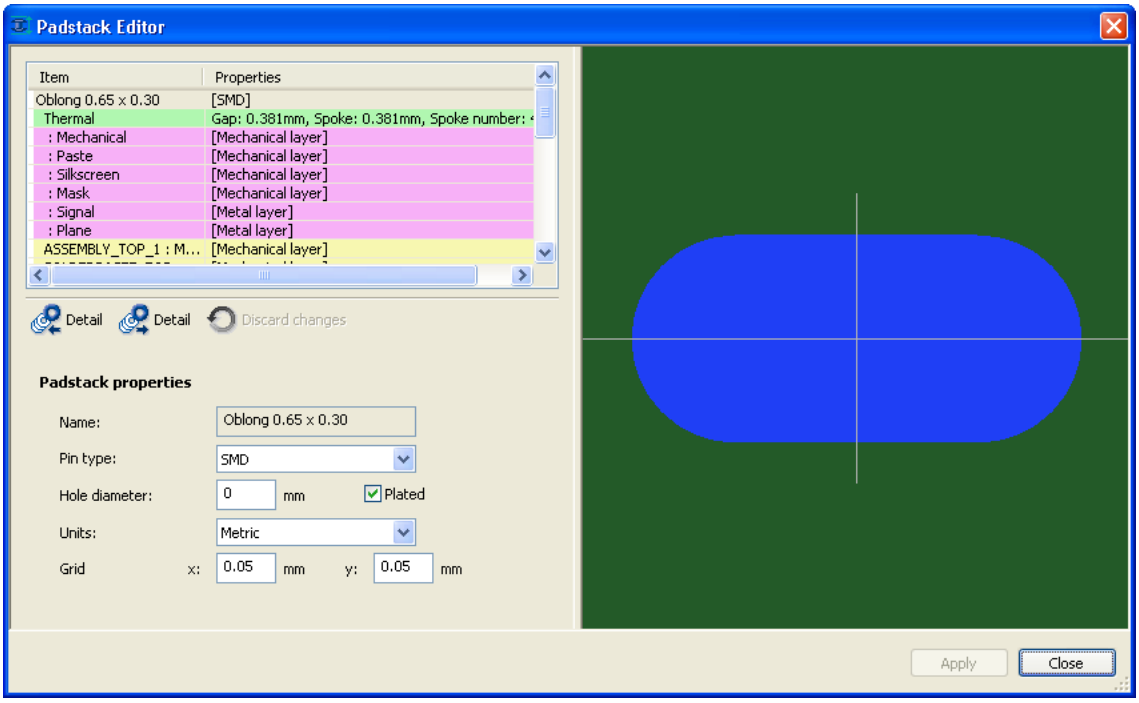
The **Padstacks** section that contains the following for each listed padstack:

- Name
- Type (through, SMD)
- Shape (on the component placement layer)
- Hole diameter (for through padstacks)
- Pad size on the component placement layer (or bounding box size for polygonal pads)
- **Modify**, **Add Padstack** and **Remove Padstack** commands

Name	SMD	Shape	Hole diameter, mm	Size, mm	
Custom 1.65 x 3.3	SMD	Complex		1.65 x 3.3 mm	Modify
Custom 1.9 x 2.0	SMD	Complex		2 x 1.9 mm	Modify
Custom 2.75 x 9.70...	SMD	Complex		2.75 x 9.7 mm	Modify
Custom 2.75 x 9.70...	SMD	Complex		2.75 x 9.7 mm	Modify
Custom 3.20 x 9.70	SMD	Rectangle		3.2 x 9.7 mm	Modify
Finger 0.75 x 0.28	SMD	Rectangle		0.75 x 0.28 mm	Modify
Finger 0.85 x 0.28	SMD	Rectangle		0.85 x 0.28 mm	Modify
Finger 0.95 x 0.28	SMD	Rectangle		0.95 x 0.28 mm	Modify
MH_0.9			0.9	0 x 0 mm	Modify
MH_1.0			1	0 x 0 mm	Modify
MH_1.6			1.6	0 x 0 mm	Modify
MH_2.7Rnd6.0		Filled Circle	2.7	6 x 6 mm	Modify
MH_3.2Rnd6.4		Filled Circle	3.2	6.4 x 6.4 mm	Modify
Oblong 0.65 x 0.30	SMD	Oval		0.65 x 0.3 mm	Modify
Oblong 0.75 x 0.30	SMD	Oval		0.75 x 0.3 mm	Modify
Oblong 0.75 x 0.40	SMD	Oval		0.75 x 0.4 mm	Modify
Oblong 0.85 x 0.30	SMD	Oval		0.85 x 0.3 mm	Modify
Oblong 0.90 x 0.55	SMD	Oval		0.9 x 0.55 mm	Modify
Oblong 0.95 x 0.30	SMD	Oval		0.95 x 0.3 mm	Modify

Delete all unused padstacks

Clicking **Modify** or **Add Padstack** opens the **Padstack Editor**.



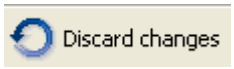
The editor shows the following:

- Padstack name and properties
- Thermal properties
- Names and properties of layer classes
- Names and properties of PCB layers
- Pad shape for the layer group and each of the layers where the component is placed, if the pad is present on the specified layer or group of layers; table rows with this information are not color-coded

The pads for each layer and group of layers are single-detail pads. To add a pad to a layer or group of layers, select the table row that corresponds to the layer or group of layers, and click the **Add Detail** 

button under the table. The adjacent **Delete Detail**

button deletes the pad.

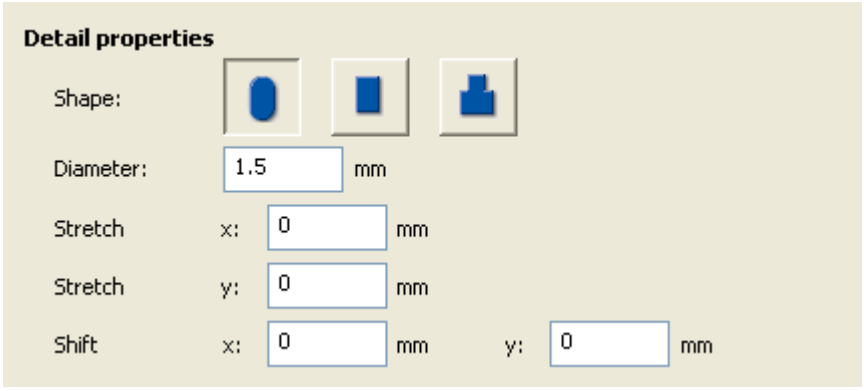
Clicking the **Discard Changes** 

button cancels the changes you have made in the editor.

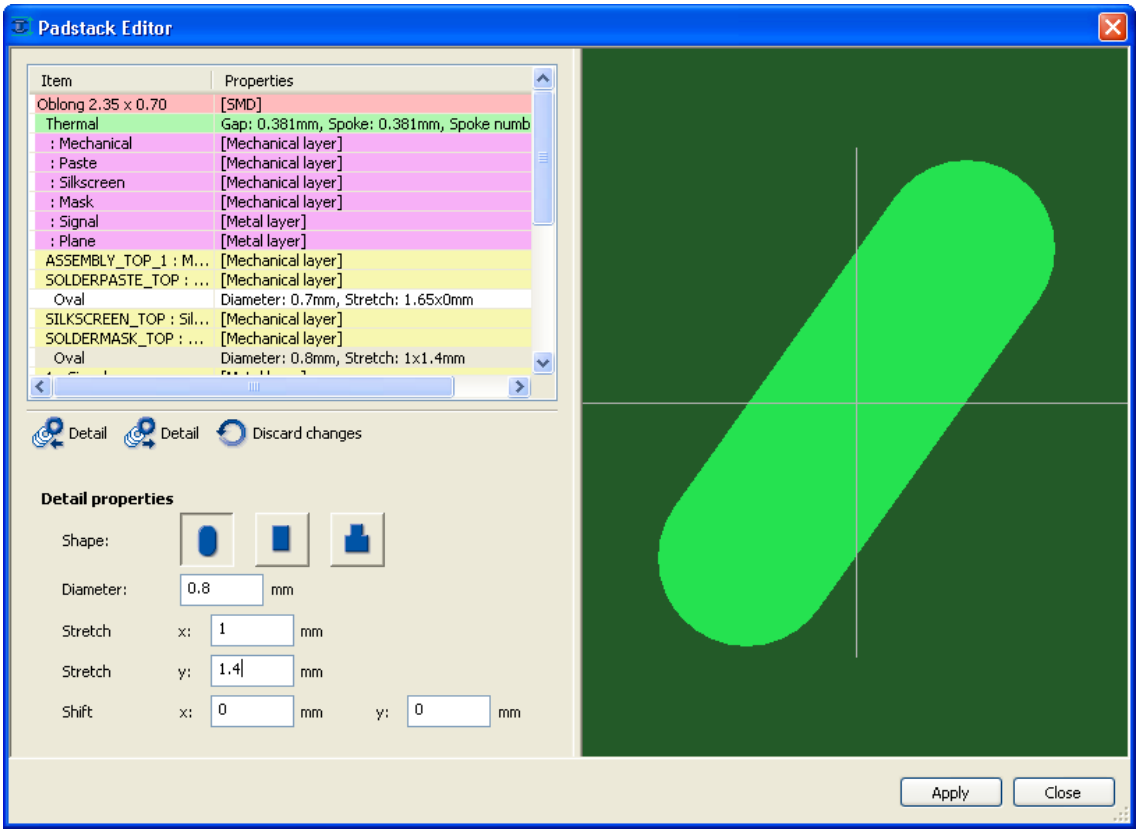
Pad Shapes

TopoR supports three pad shapes: oval (circle), rectangle and polygon.

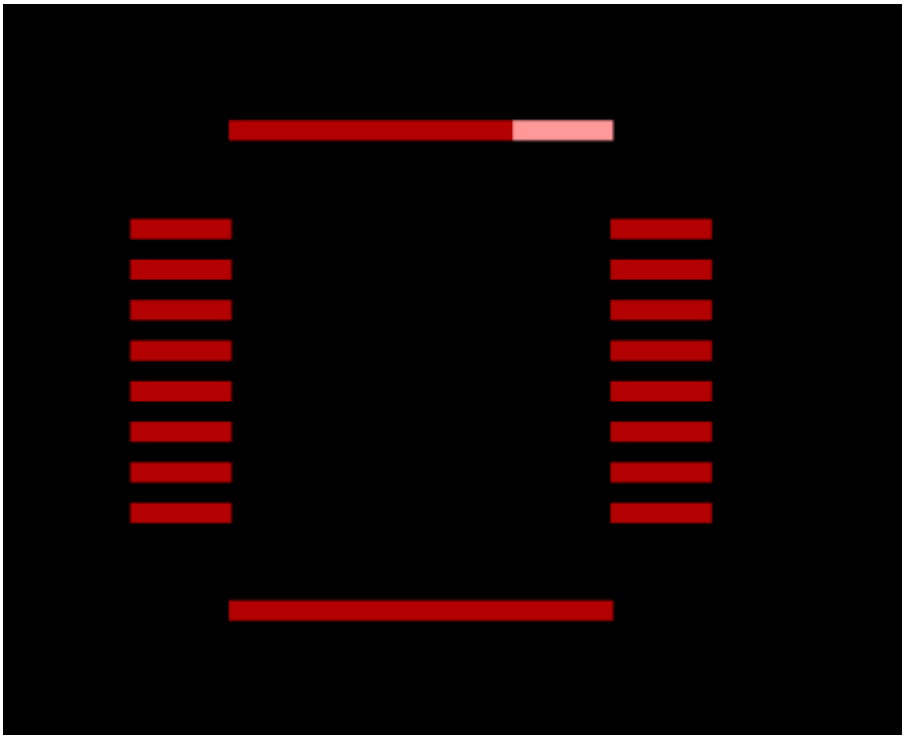
Oval—this is a shape derived by displacing a circle in a linear direction. The parameters are diameter (pad width), stretch in X and Y, and shift in X and Y relative to the origin.
Circle—this is an oval where the stretch in X and Y is zero.



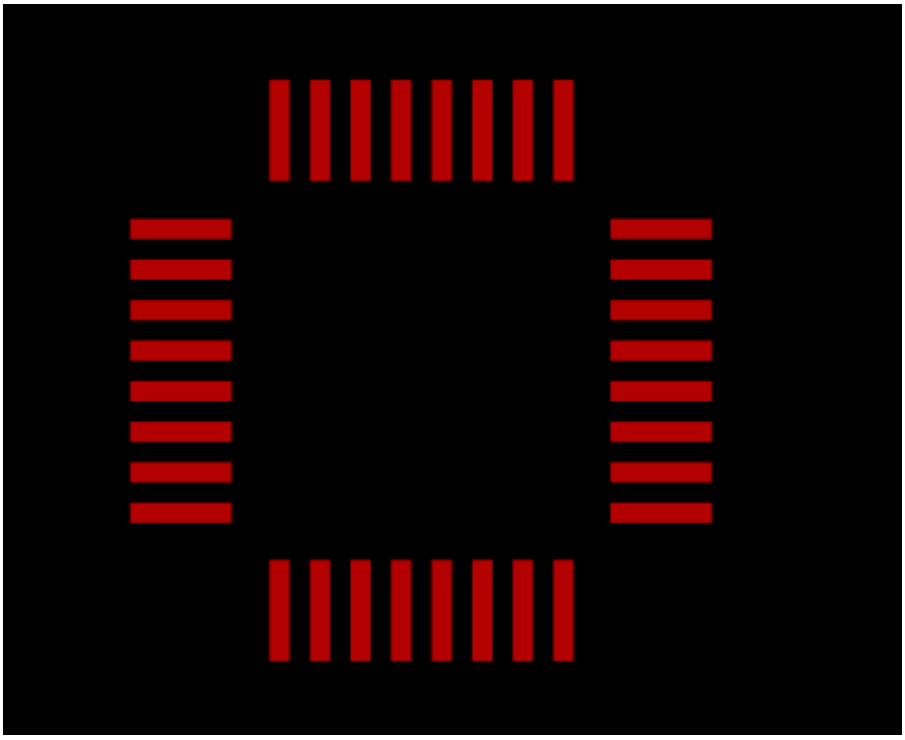
To set the angle and length of an oval, use the **Stretch** parameter. An oval's orientation is not necessarily horizontal or vertical, but can be arbitrary (defined by the ratio of stretch in X to stretch in Y). The pad center must sit on the oval's axis.



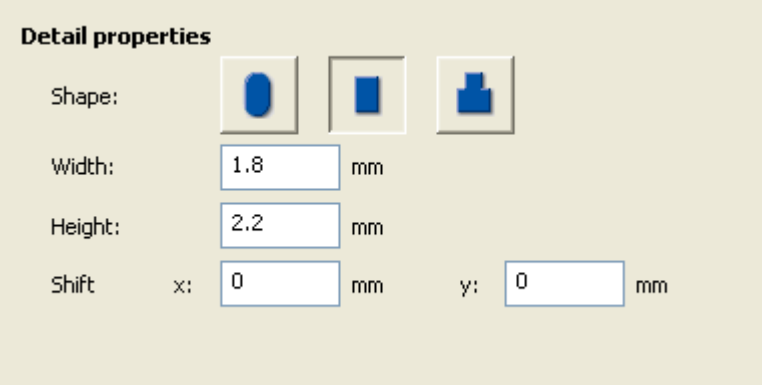
Sometimes you cannot choose the correct orientation for a pad due to conflicting use of this pad by one or more components (with different orientations).



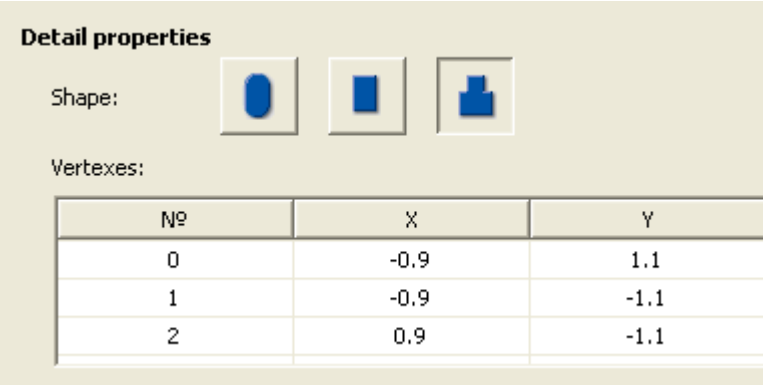
In this case, change the pad type in the component or in the library asset. That is, set different pad types for the different orientations.

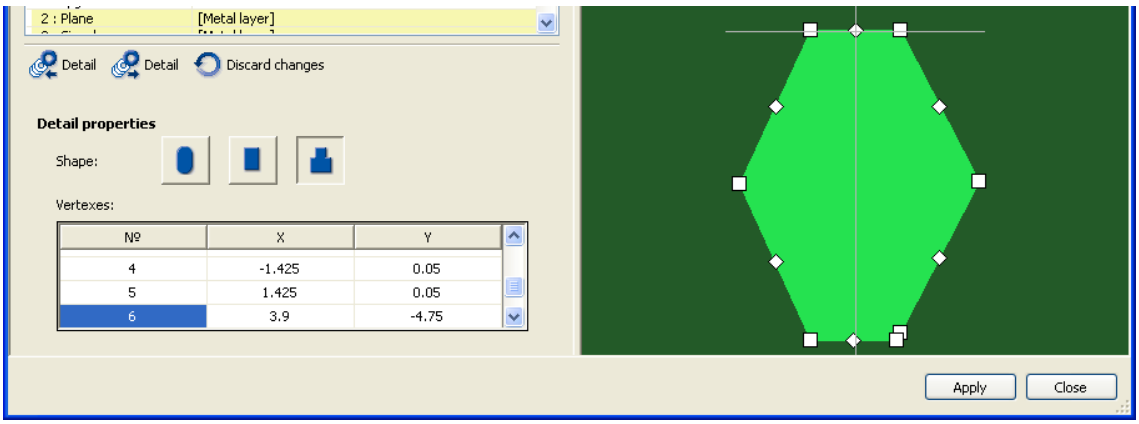


Rectangle—this shape is defined by width, height, and shift in X and Y relative to the origin.



Polygon—this shape is defined by using the table to specify point coordinates or in the graphical editor. Changes made in the editor modify the table automatically, and the other way around.

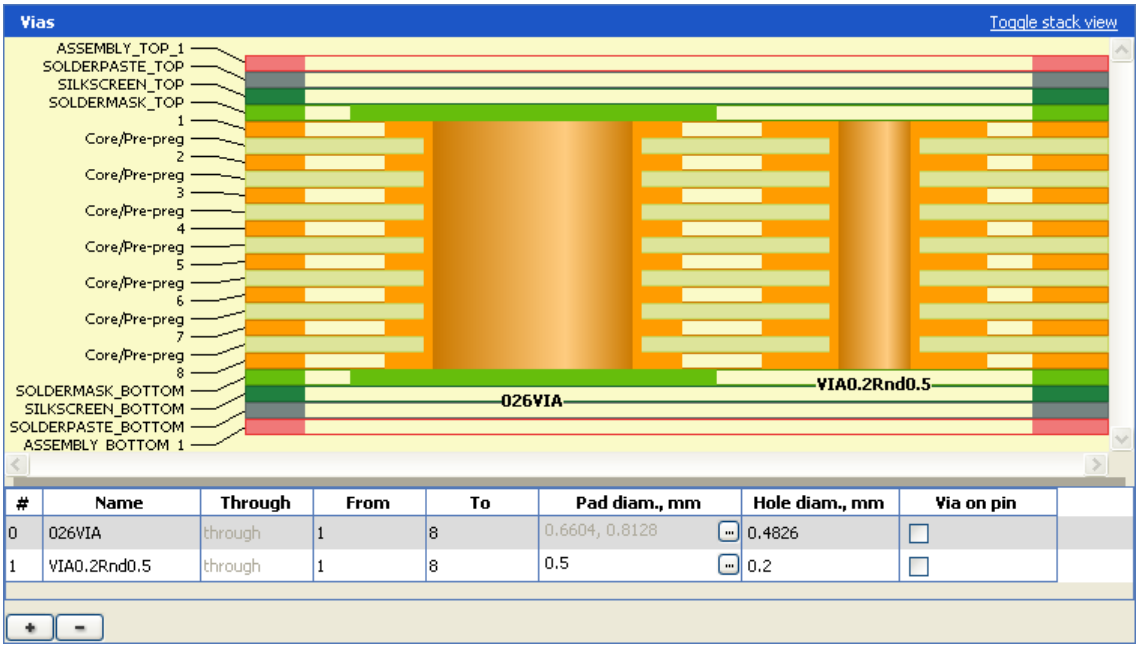




When you rotate components, pads are also rotated. Each type of pad must define either a pin or a planar pad. A component can have both pins and planar pads at once.

Vias

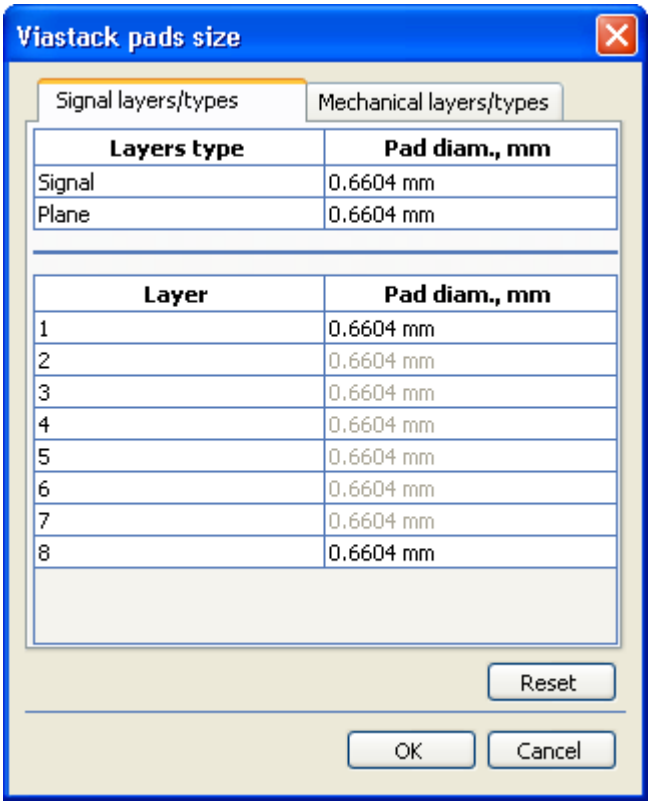
The **Vias** section contains a table of via parameters, a visualization canvas, and the **Add Via** and **Remove Via** buttons.



When you add a new via (by clicking **Add Via**), the selected table row is duplicated, and you can modify the parameters if necessary.
The table shows the following information about each listed via:

Name
Whether or not it is a through via (check box)
Range of layers
Pad diameter
Hole diameter
Whether the via is on a pin (check box)

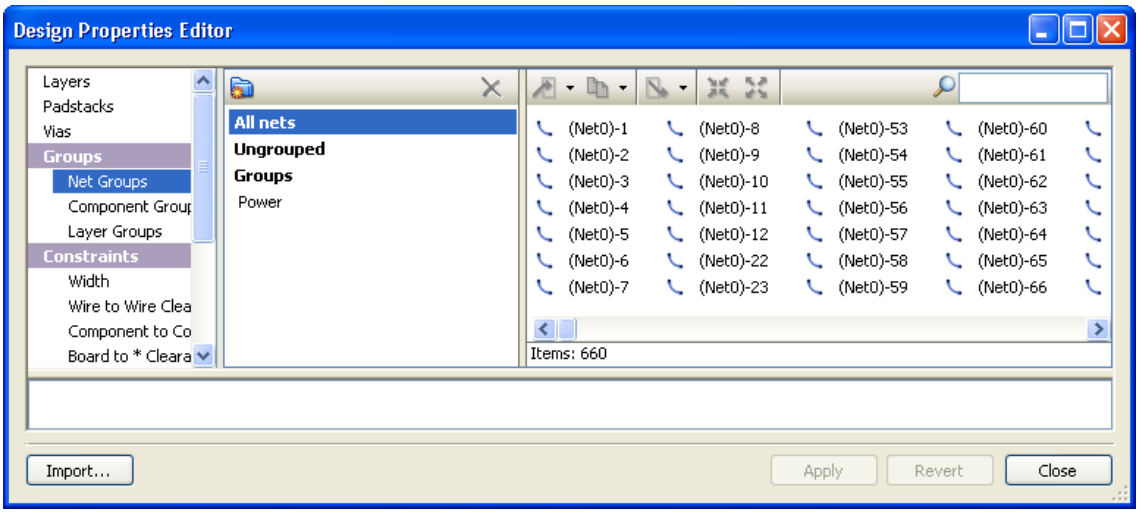
Clicking the button opens the **Viastack Pads Size** dialog box, where you can change pad sizes on any signal or mechanical layer.



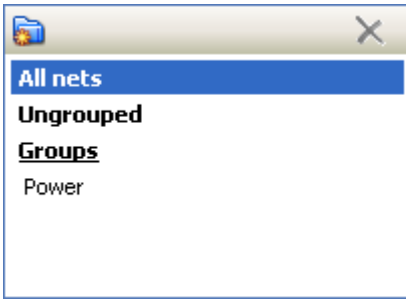
If pad diameters vary across layers, then the **Pad diam** column lists the values and makes them non-editable.

Object Grouping

The net grouping panel is shown as an example of working with the grouping tools. The **grouping panel** is made up of several parts.



1. Navigation Pane



The navigation pane contains all available object groups. Groups can form a hierarchical tree. In addition to the groups, the navigation pane provides the following special items:

All nets

Displays in the grid all objects of a particular type (nets in this case) that exist in the design.

Ungrouped

Displays in the grid all objects of a particular type (nets in this case) that are not members of any groups.

Groups

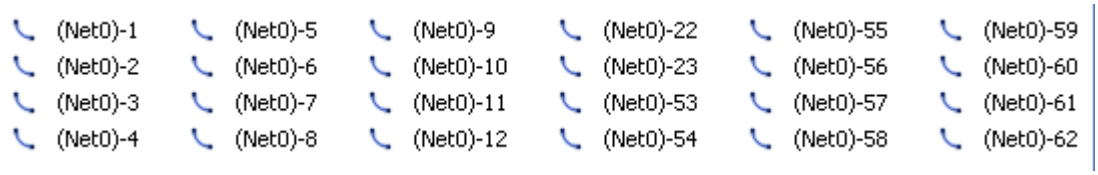
Displays in the grid all top-level groups.

Only one node can be selected in the navigation pane at a time.

2. Toolbar

The toolbar has two sections. The left section contains buttons for creating a group and removing the group selected in the navigation pane. The right section lets you perform the actions described above on selected objects.

3. Display Pane



The display pane shows a grid of objects for the node selected in the navigation pane. Using the objects' context menu, you can do the following:

Move to group

Objects are removed from their current group and added to the specified group. This action is available only for grouped objects. You can also drag objects into groups to for the same effect.

Copy to group

Objects remain in their current group, and are also copied to the specified group. You can **Ctrl**-drag objects into groups to for the same effect.

Remove from group

Objects are removed from their current group. This action is available only for grouped objects.

Group

Objects are organized into a new group.

Ungroup

This action is available only for groups. Groups are deleted, and their members are moved up one level in the hierarchy (where applicable).

Rename

This action is available only for groups.

If the selected object is a member of a group, that group is highlighted light gray in the navigation pane.

4. Filter Box

If a string is entered in the filter box, then the display pane shows only those objects whose names contain this substring. The filter box is cleared when you select another node in the navigation pane.

5. Information Bar

The information bar shows the number of objects. If no objects are selected in the display pane, then the total is shown. Otherwise, the number of selected objects is shown.

Creating Hierarchical Group Structures

Groups are primarily used for specifying different routing rules (widths, clearances and so on).
A hierarchical group structure is useful for setting common rules for multiple object groups. In this case, you should create a higher-level group and add to it the groups you need.
In the navigation pane, you can drag groups around to change the hierarchical structure. To put the group at the topmost level, drag it into the **Groups** item.

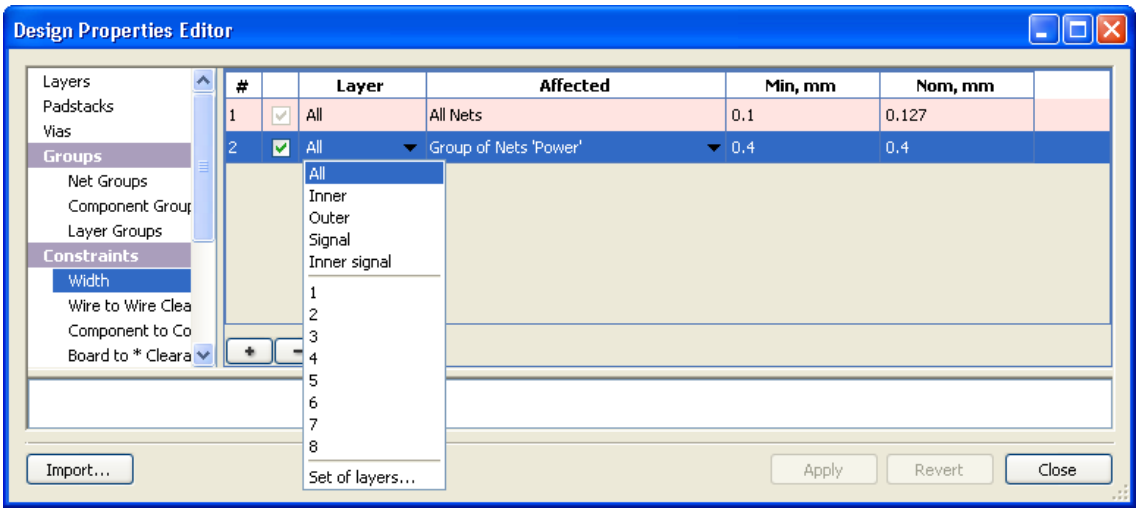
The hierarchical structure has the following limitations:

1. A group cannot be a member of more than one other group.
2. Groups at the topmost level cannot be members of other groups.

Constraints

The Constraints section of the **design properties editor** lets you set limits for wire widths, clearances (between wires, between components, from the edge of the board to components and wires) and per-net via assignments.

Wire Width



Specify the minimum and nominal width.

The minimum value will be used in bottlenecks unless a different path is available or near pads that are narrower than the wire.

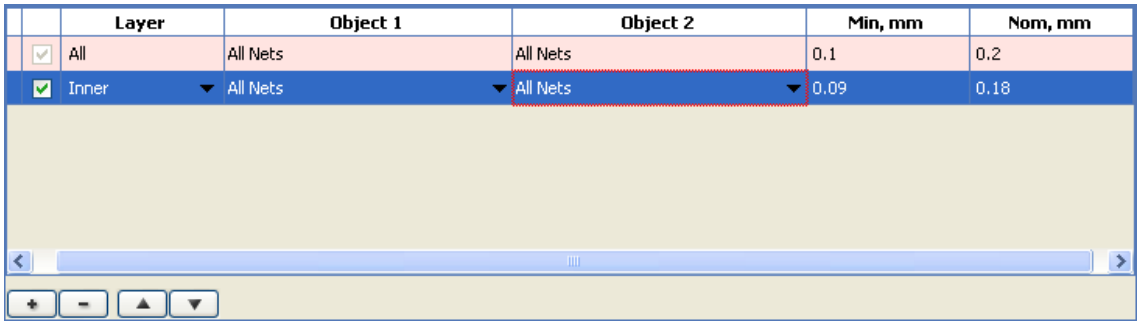
Select the layer or layer group where the constraint is active.

Specify what the constraint is applied to: net, group of nets, all nets.

The order of rules in the table defines their priority, meaning which rules will be applied sooner to overlapping group members. The higher up the rule in the table, the higher its priority.

During autorouting, width constraints for individual wires set during editing are ignored. Constraints from the design properties are used instead.

Wire Clearance



Specify the following:

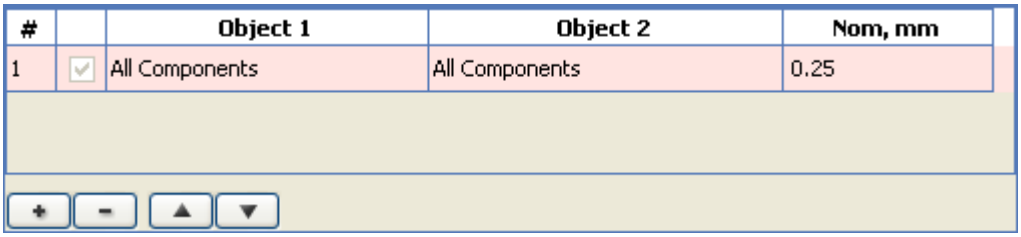
Minimal and nominal clearance

Object 1 and object 2 that the constraint applies to

Layer or layer group that the constraint applies to

Object 1 and object 2 are nets, net groups, signals or signal groups.

Component Clearance



Specify the following:

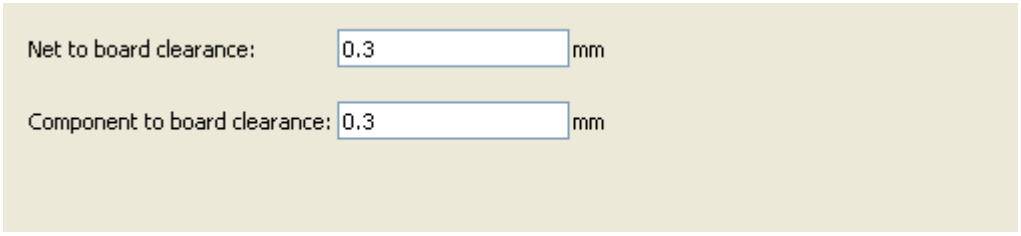
Nominal clearance

Object 1 and object 2 that the constraint applies to

Layer or layer group that the constraint applies to

Object 1 and object 2 are components, component groups or all components.

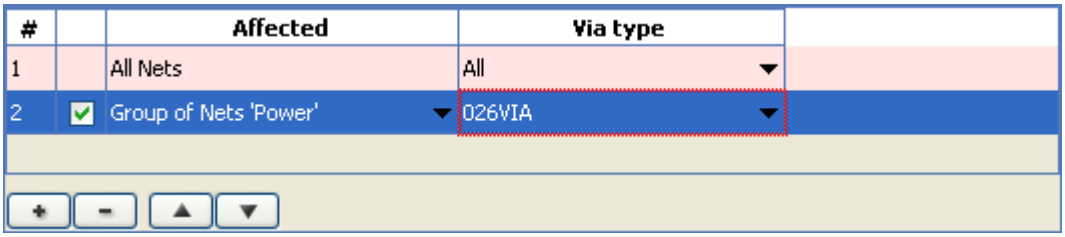
Board Edge Clearance



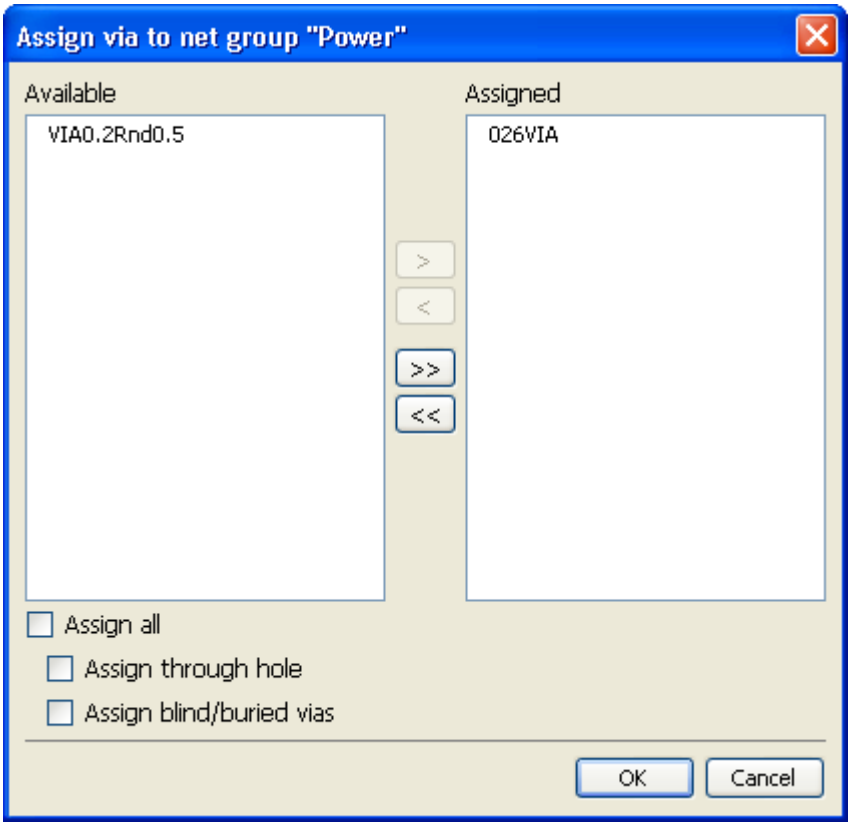
Specify the clearance between components and the board edge, and between wires and the board edge.

Assign Via to Net

By default, vias are created with the minimal possible diameter. However, this is not suitable for power nets, for example. Therefore, **TopoR** lets you customize via types for individual nets or net groups



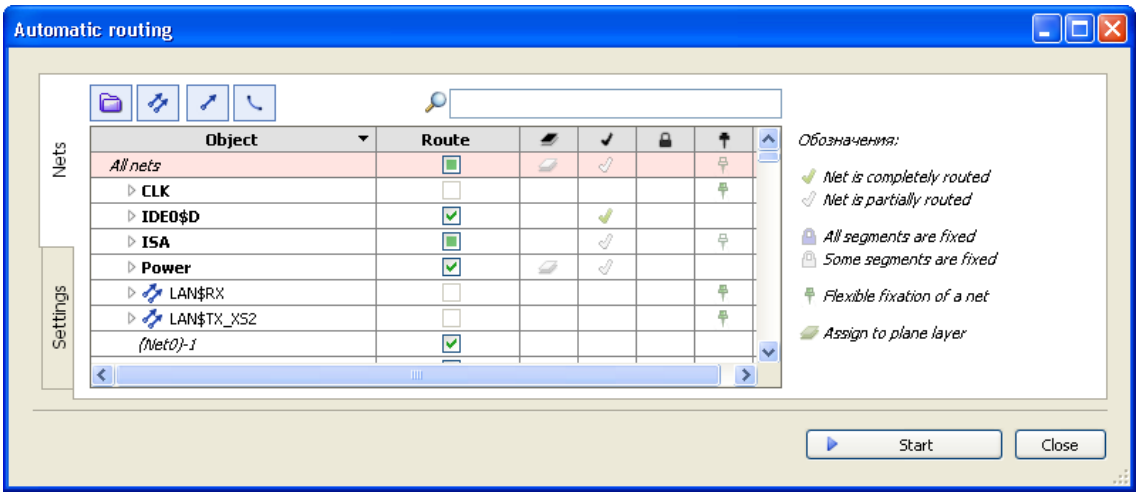
Use the + and − buttons to manage the list of nets to assign vias to.
Clicking in the **Via type** column opens a dialog box where you can select a single via type or multiple types at once.



Autorouting

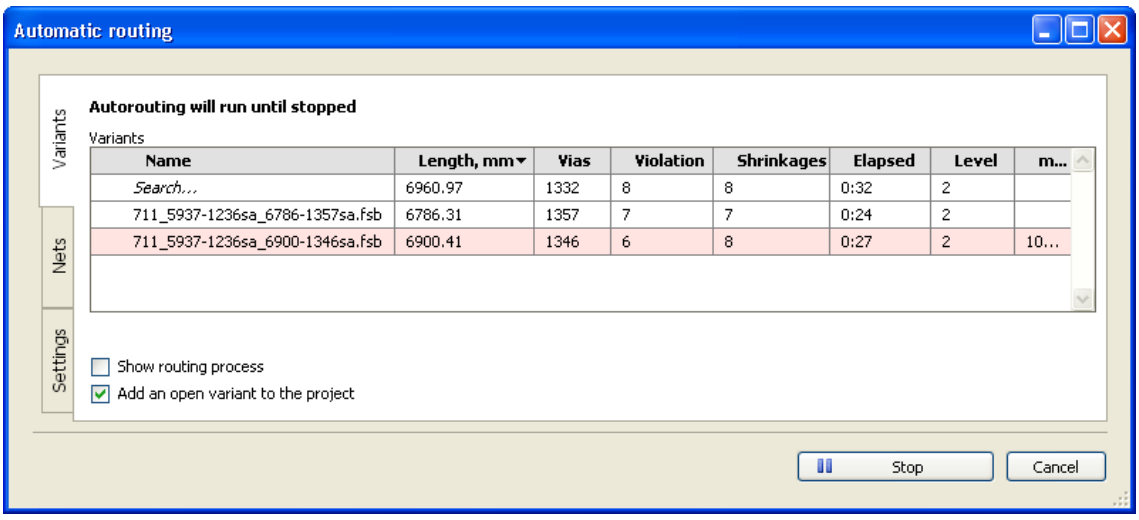
100-percent complete net routing is done almost instantaneously. All connections are routed, even where this might entail violations of technological constraints. Later, these violations are eliminated automatically.

TopoR 6.1 provides a dedicated autorouting control panel. To open it, click **Routing** ►**Autorouting** in the main menu or click the **Autorouting** button.



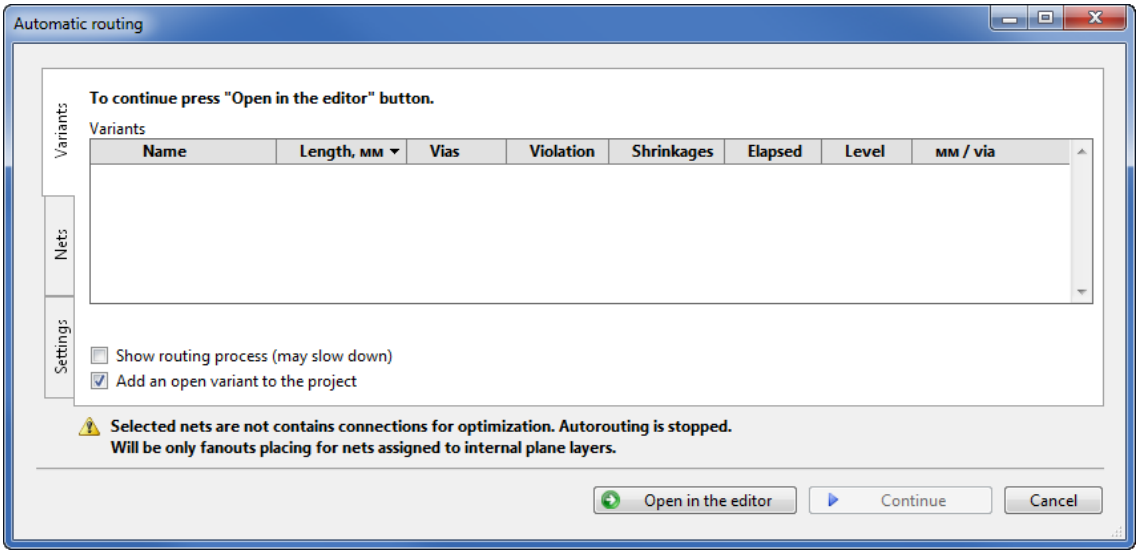
Before routing is first started, the autorouting control panel has two tabs: Nets and Settings. After routing has been started at least once, the Variants tab is added.

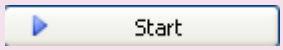
This tab contains a table with the parameters of eligible saved variants.



During wire rerouting, combined topology is untangled, resulting in a global minimum of vias for the current topology. You can stop optimization at any time by clicking **Stop** and continue by clicking **Resume**. To proceed with the design, click the **Open for editing** button.

Autorouting isn't started when selected nets are not contains connections for optimization. Only fanouts will be placing for nets to internal plane layers.

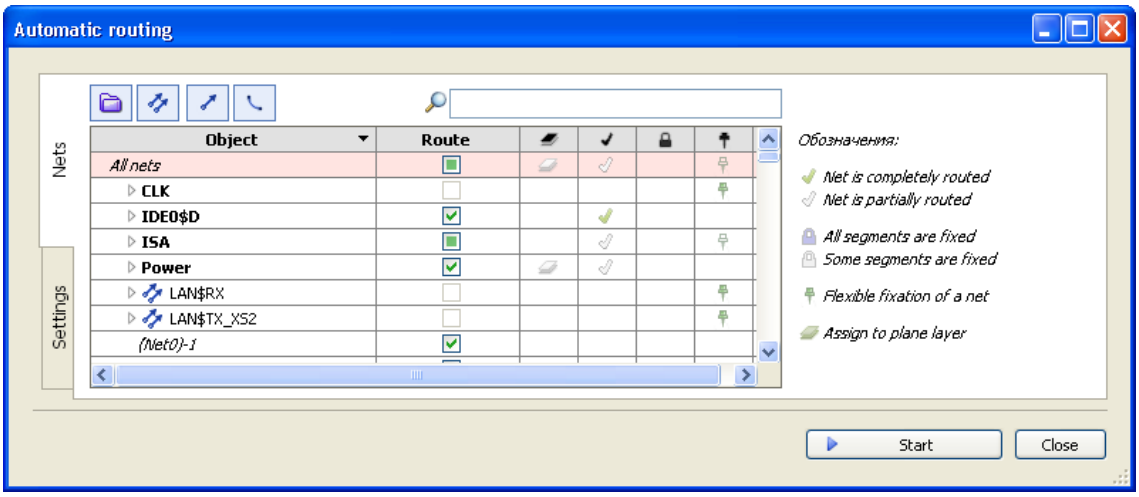




button is inactive in following cases: **1)** no nets for autoroute; **2)** exceeded number of tracing layers (for version of TopoR); **3)** there are errors crossing contacts; **4)** there are errors crossing vias at non-routing nets.

If there are some mistakes from this list, the system informs about first of them.

Nets Tab



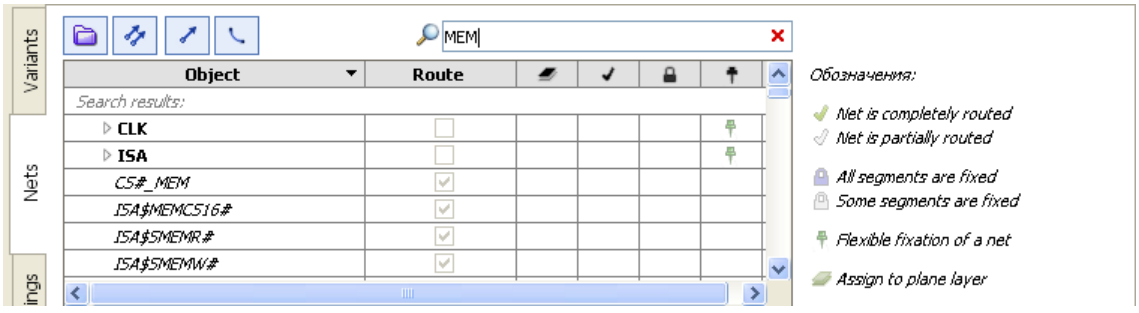
The **Nets** tab shows all objects (nets and net groups) in the current project. Group names are in **bold print**, and net names are in *italics*.

The four buttons at the top let you show or hide the following: net groups, differential signals, signals and nets. For all displayed objects, you can enable or disable routing (check boxes in the **Route** column).

You can use any column of the net table for sorting by clicking the column's header. Clicking repeatedly toggles between ascending and descending sort order.

When you sort by **Object**, net groups are sorted independently of standalone nets.

The text box next to the magnifying glass icon is for specifying a search filter. Entering a string lets you view only those entries whose names contain the specified substring. In the screenshot below, the "MEM" filter string was applied.



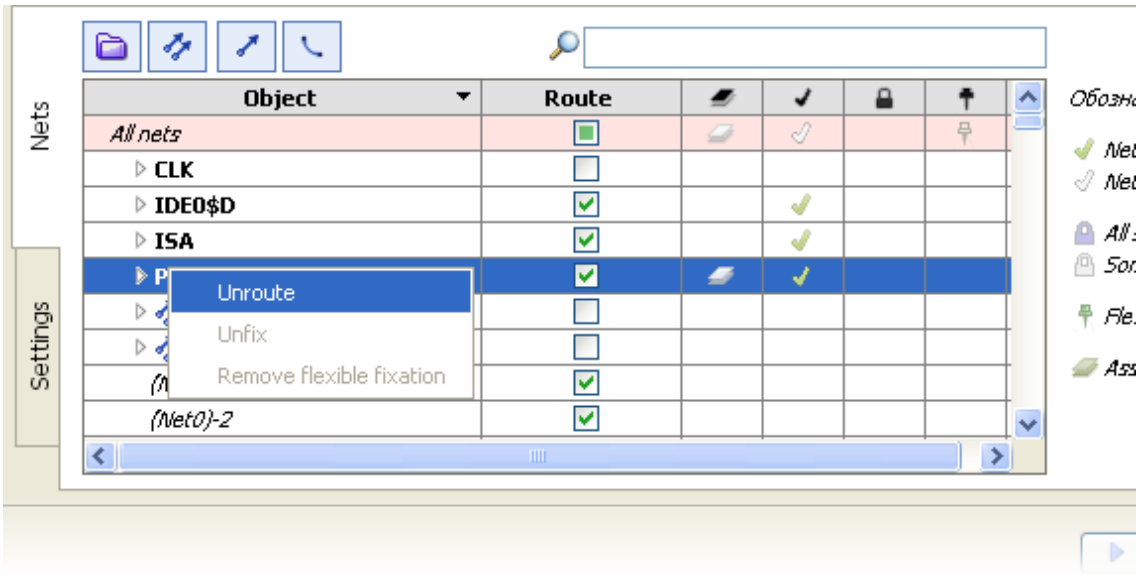
Selecting and clearing the check box in the **Route** column for a net group enables and disables routing for all nets in that group.

To the right of the table is its legend, where you can look up the meanings of the icons used in net table. The columns marked ✓, 🔒 and 📌 in the autorouting panel indicate the state of the layout. The column marked ▰ shows nets and net groups assigned to a backup layer.

The ✓ column shows whether a layout is available for the net or net group. The mark in this column can be dashed, meaning that there is a partial layout.

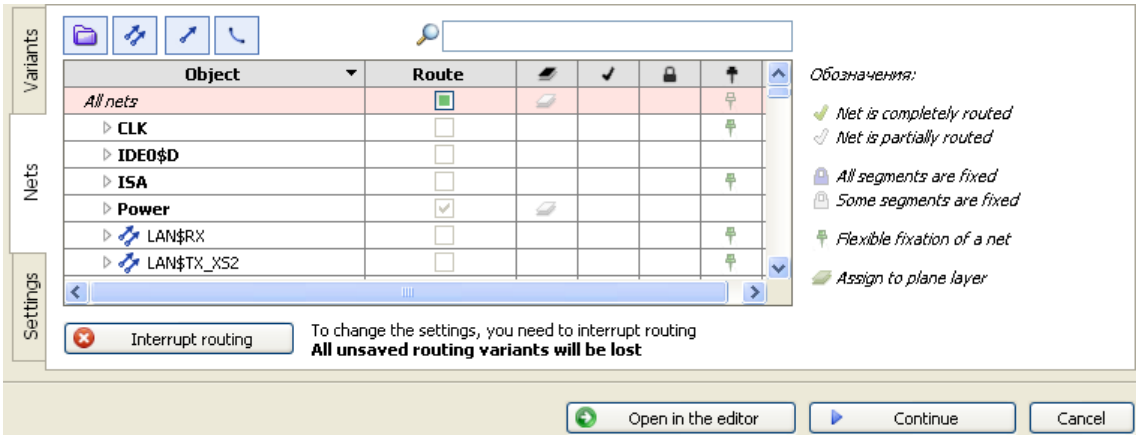
The 🔒 and 📌 columns show whether nets are fixed. The 🔒 icon means a fixed net or net group; the 📌 icon means flexibly fixed nets and groups.

To change routing and fixedness settings for an entry, right-click it and use the shortcut menu.



For example, all nets in the **ISA** net group are flexibly fixed, so the routing option has been disabled for it. Right-clicking the group lets you delete its layout, unfix it or enable routing for it.

Changes to net routing settings are blocked during autorouting, so all options to that effect will not be editable.

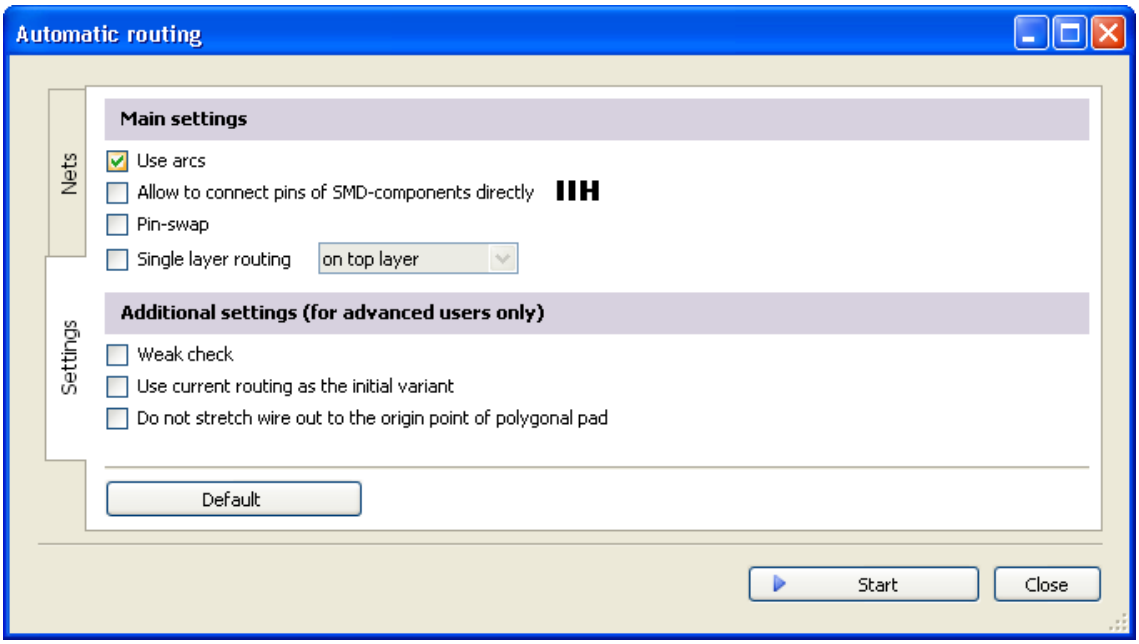


To change the settings, first click the 

button. Then select the objects to route and click the **Run** button to start net routing. For details about the operation of the router, see Finding Variants.

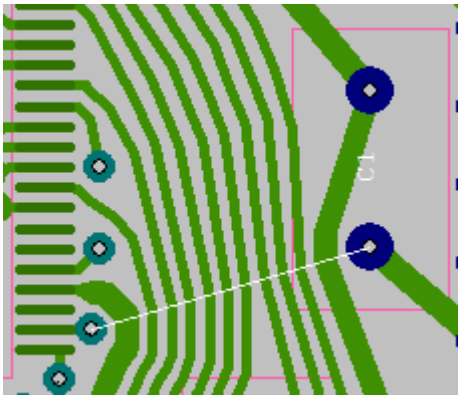
Settings Tab

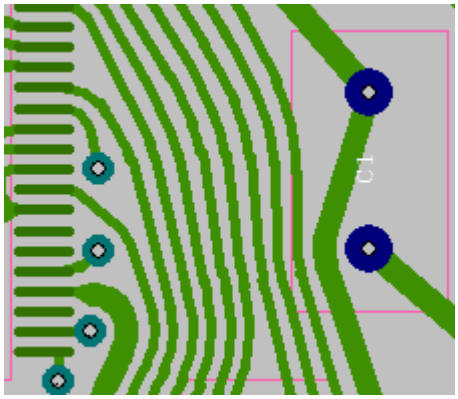
The **Settings** tab contains information about the current autorouting configuration.



General Options

The **Use arcs** option lets you include arc-shaped wires. Enabling and disabling this option changes which wire shape calculation tool is active (**F5**).





The **Connect planar pads directly** option is self-explanatory.

By default, using same-level equivalent pads is disabled. If you select the **Reassign functionally equivalent component pads** option, this helps switch nets that approach functionally equivalent pins. For details, see Reassigning Functionally Equivalent Pads.

Routing runs in multi-layer mode by default. The **Single layer routing** option enables single-layer routing for boards with pin components where one of the layers uses metal and the wire links are located on the component side. Selecting the top layer or bottom layer indicates to the router which metal layer to route. When doing single-layer routing, consider enabling the **Weak clearance check** option and using automatic component shifting.

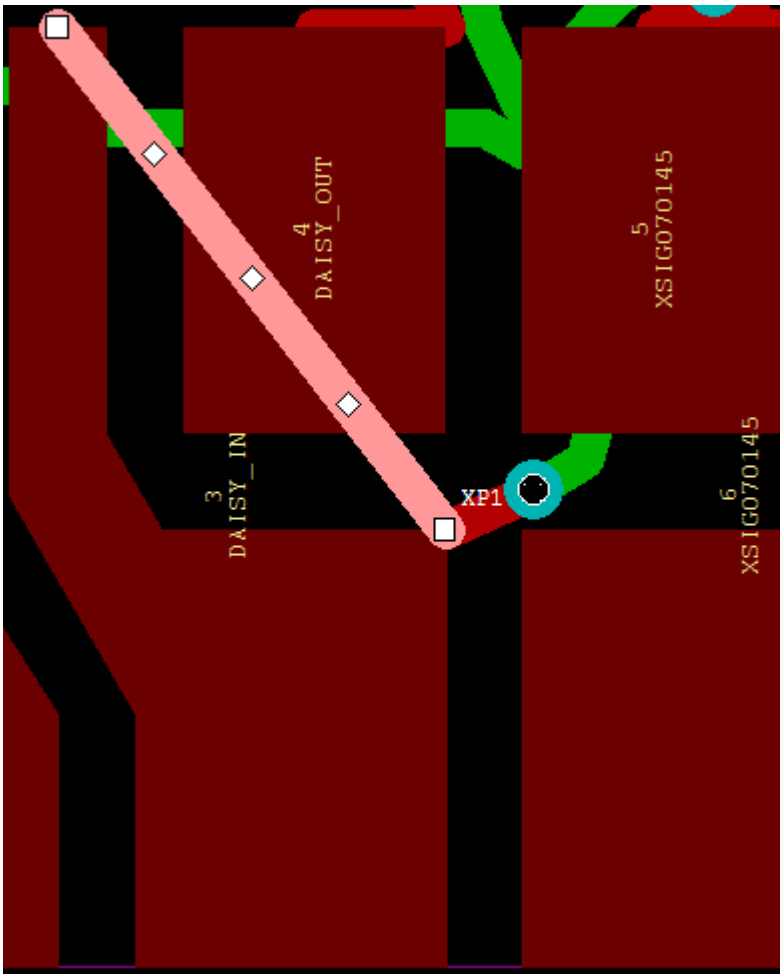
Advanced Options

When the **Weak clearance check** option is enabled, the router is allowed to ignore violations that may occur between the pads of different (non-fixed) components. These violations are supposed to be subsequently eliminated using the automatic component shifting procedure (**F7**).

This procedure must be started manually after a routing variant has been opened in the editor.

The **Reuse existing layout as a starting point** option lets you take advantage of the current layout of the nets you are routing.

The **Don't stretch wire to polygonal pad origin** option is required for connecting wires to polygonal pads with complex polygonal shapes. If this option is cleared, violations like the following can occur around such pads:



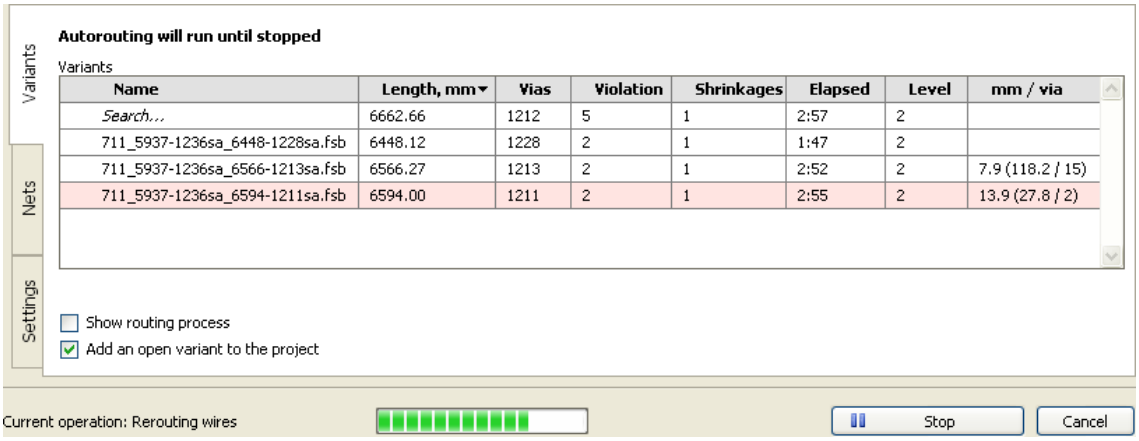
Here, the selected wire touches another pad and is not even needed at all (its only purpose is to stretch the wire to the origin); to avoid creating it, select the option above.

The **Defaults** button resets all options to **TopoR** defaults (in particular, all check boxes are cleared).

Finding Variants

Click **Routing ► Delete Layout** in the main menu to remove all wires before routing, including fixed wires. This is a way to perform routing from scratch.

After routing has been started, the **Variants** tab appears in the routing control panel.



TopoR automatically selects the best routing variants and saves them in the table using the following naming convention: **`name_LLL-VVV.fsb`**, where **`name`** is the board name, **`LLL`** is the total wire length in millimeters, and **`VVV`** is the number of vias.

In addition to the estimated total wire length and number of vias, the **`.fsb`** file name contains information about the routing mode. Routing mode "**`w`**" means weak check, and "**`s`**" means strict check. If the file name contains the letter "**`a`**", this means that arc-shaped wires were allowed during routing. The digits at the end of a file name (if any) do not have any meaning and are used just to make the file name unique.

The variant name is shown in the table's **Name** column, and each table row also includes the following parameters of the variant:

Length is the total length of unsmoothed connections (usually exceeds the length of smoothed-out connections by 10 to 15 percent).

Vias is the number of vias.

Errors is the number of design constraint violations.

Shrinkages is the number of width reductions in nominally-wide wires.

Elapsed is the time it took to get the result.

Round is the number of the net shifting iteration that produced the variant (round 1 means the very first iteration).

mm/via is the comparison between this variant and the previous best. By default, the current variant list is sorted by number of violations, then by ascending length. For the shortest variant, the **mm/via** is usually empty, and for subsequent variants with as many violations and fewer vias the column shows the length increase (in previously specified units) per removed via.

All of the table rows can be sorted by any parameter in ascending and descending order even as the routing is in progress.

During routing, variants can be added to the table and removed from it. The algorithm that controls this is quite complex. In most situations, the following rules apply:

A variant is removed (or rather, superseded by a new one) if the new variant is better on all parameters considered by the router (length, number of vias, number of violations, number of shrinkages).

A variant is added if the new variant is at least as good as the best existing one on one parameter or a combination of parameters.

When there have not been any improvements to the existing variants in a long time, the system shows the **Autorouting may be stopped** message in the variant view.

Optimization Criteria in TopoR

The major quality criteria in the **TopoR** system are as follows:

Number of errors

Number of vias

Total length of connections

Number of shrinkages (wire width reductions in bottlenecks)

The system automatically stores variants with the best parameters. The first saved variant appears only after the first optimization cycle has been completed or 20 seconds after the start of the optimization. Subsequently, saving occurs as the layout quality improves. The variants in which all parameters are worst will be removed automatically.

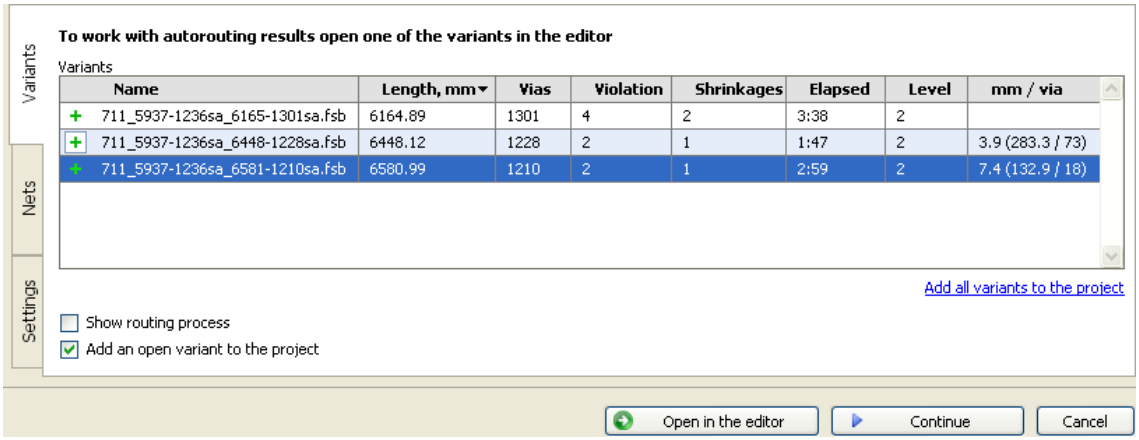
The optimization process can continue indefinitely until it is stopped manually using the **Stop** button. Do not stop the optimization process before the value in the **Round** column reaches 5–10. If **TopoR** keeps saving new versions, then it's best not to stop it. When there have not been any improvements to the existing variants in a long time, the system shows a message that suggests stopping the optimization.

To decide on the right moment to stop, you can also consider the changes in the ratio of total wire length to the number of vias (the **mm/via** column). Practice shows that good results are achieved at the ratio of 25mm/via (1000mil/via).

Selecting the Best Variant

Unlike other systems, **TopoR** simultaneously optimizes multiple alternative variants of the layout, with different parameters optimized differently. You can select the layout configuration that suits you best. Select those of the automatically saved variants that you find optimal, and add them to the project.

To add a variant from the table to the project, either right-click it and select **Add to project**, or click the + icon in variant entry.



The following operations are available from the context menu for score table items:

Add to project adds the file to the project and writes it to disk.

Delete removes a file from the table. If there is no such file in the project, then it is removed from disk as well.

Start up resumes optimization of the selected variant.

Open for editing opens the file in the editor.

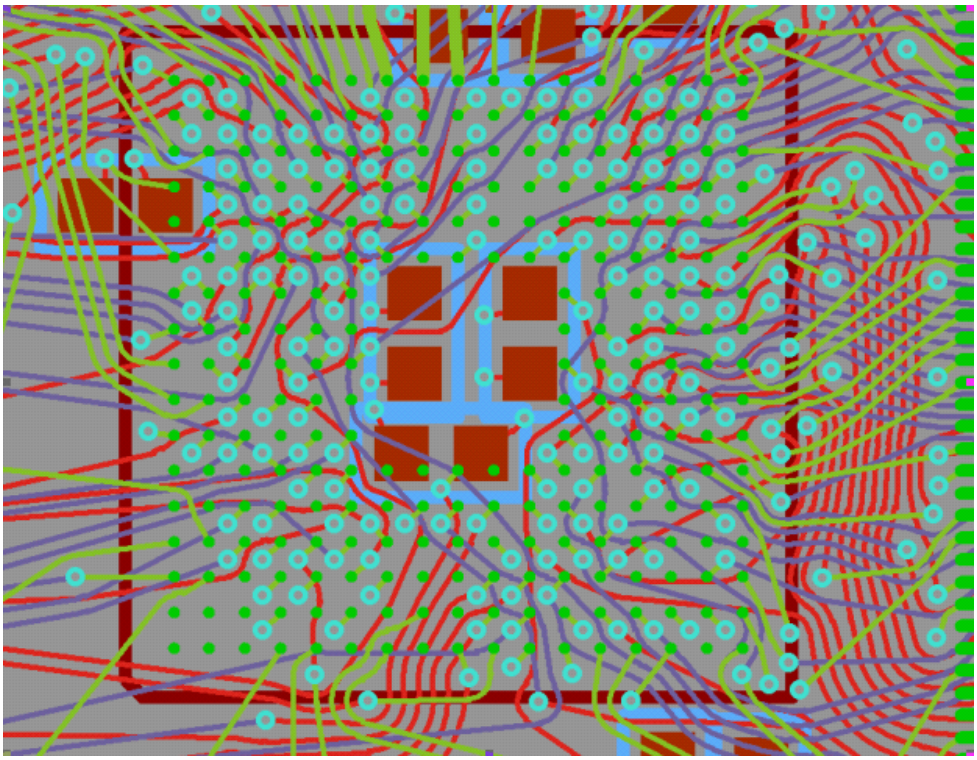
You can also add all of the discovered variants to the project at once. For that, click the Add all variants to the project link under the bottom right corner of the score table.

Using Saved Variants

Intermediary variants are saved to **`.fsb`** files. To continue improving layout quality, select a saved variant and resume optimization. In this case, the process starts from the state in the saved **`.fsb`** file rather than from the beginning. You can delete the variants that you are not interested in. For that, right-click the variant in the score table or in the project tree and select **Delete**.

TopoR does not provide any way to restore deleted variants

Routing BGA Components



BGA components are processed in autorouting mode. When routing BGA components, the smallest interlayer via type is selected automatically. This via must be assigned to a routing rule. Two routing rules are also selected automatically: one rule for wires on the outer layer where the pads are located, and the other for wires on other layers.

TopoR 6.1 supports special routing for BGA components whose pads are at the intersections of a rectangular grid. Routing of components with pads in a checker pattern or components at a non-right angle is not supported.

Conditional Routing

TopoR 6.1 provides *conditional routing* of wires, meaning a special autorouting mode where the optimal route is found only for a subset of the wires. To avoid deletion of the previous layout but keep it out of the way, consider *flexible fixing* of wires.

The procedure is generally as follows:
1) Select the nets to route.
2) Start routing.
3) After routing has stopped, edit the result manually.
4) If the resulting routes of the selected nets are acceptable, enable flexible fixing for these nets.
5) If necessary, go to step 1.

Flexible fixing is intended for preserving the topology of a net without necessarily keeping its geometry (for example, a wire between two pads can be placed higher, lower or further to the right or left than the original).

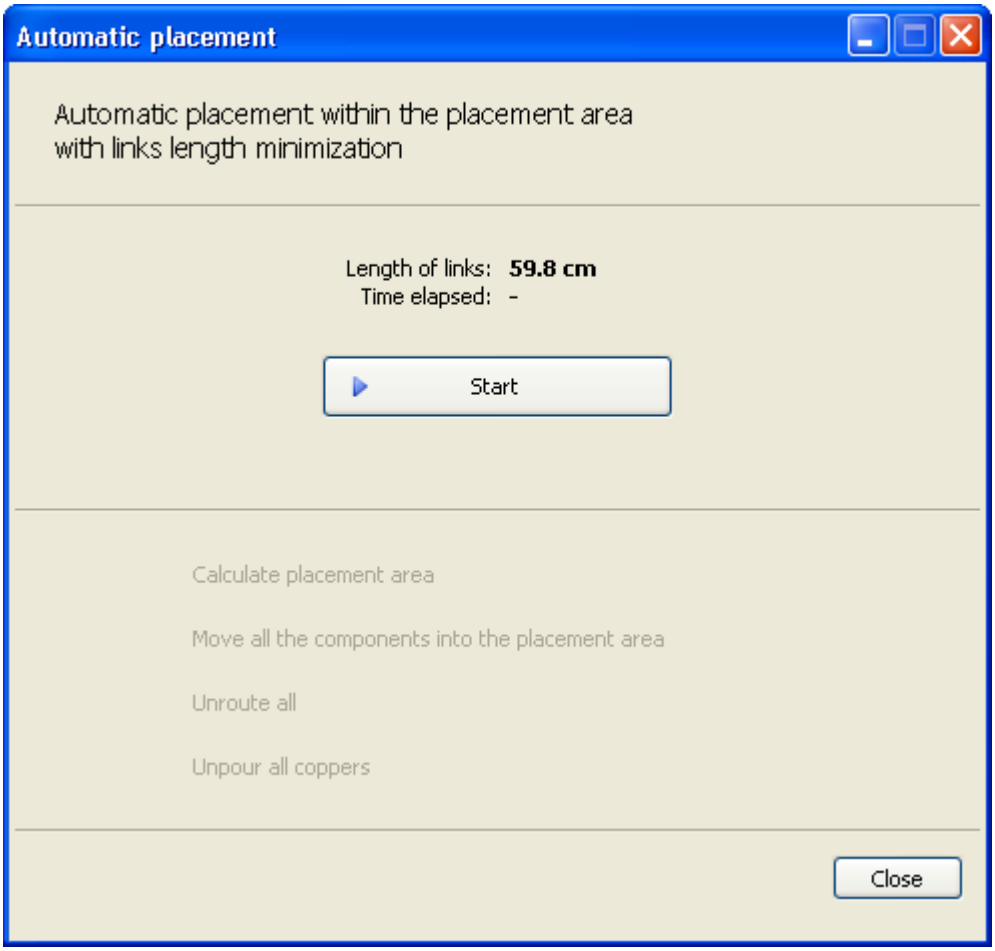

When the flexible fixing option is enabled, the routing control option is disabled automatically.


The paths of flexibly fixed wires are not automatically optimized during automatic component shifting (see Refine Tool).

If the results of autorouting are not quite acceptable, the routed nets should not be fixed (flexibly or otherwise). In that case, you can enable the **Resume routing from current configuration** option on the autorouting settings tab. As long as the option is disabled, all non-fixed routable nets will be automatically deleted when autorouting starts.

Automatic placement

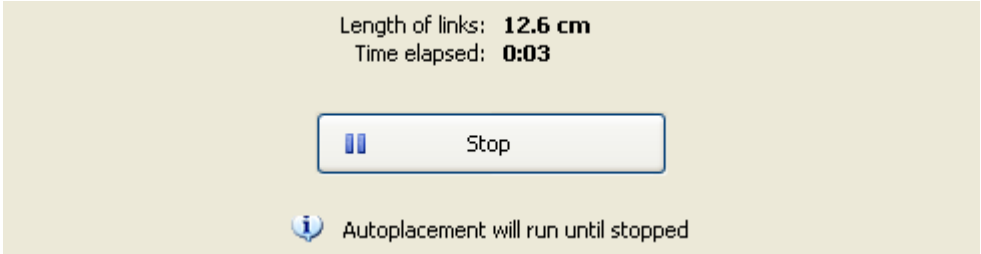
The Automatic Placement dialog box is a part of the **TopoR** system starting with version 6.1. To open it, use the main menu (**Tools ► Automatic Placement**), or the **Automatic Placement** button.



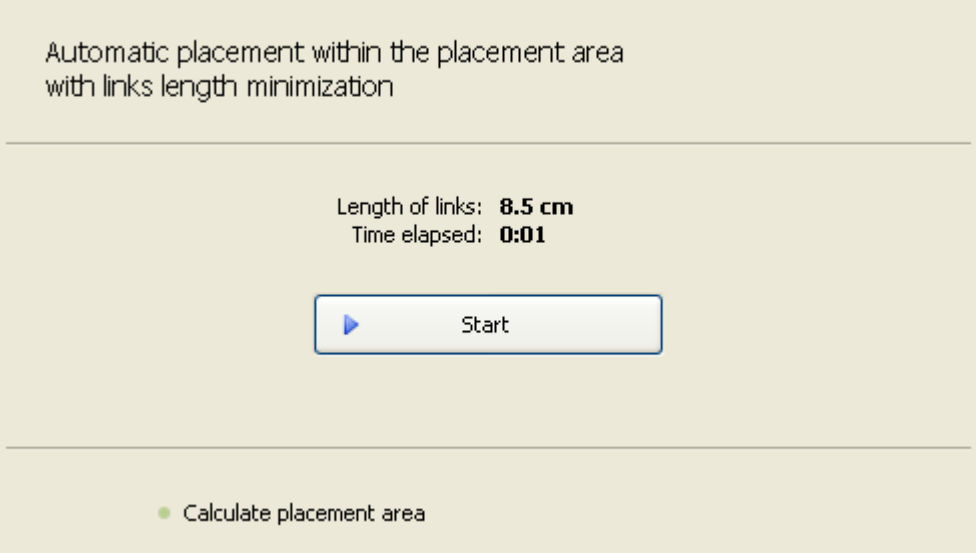
The automatic placement procedure (automatic positioning of components in the placement area while keeping the length of links to a minimum) can run only if there is at least one unfixed component; otherwise, you get a warning that the procedure cannot start (instead of a warning that the existing routes are about to be removed).

If automatic placement is allowed, then the entire existing layout will be deleted before it starts.

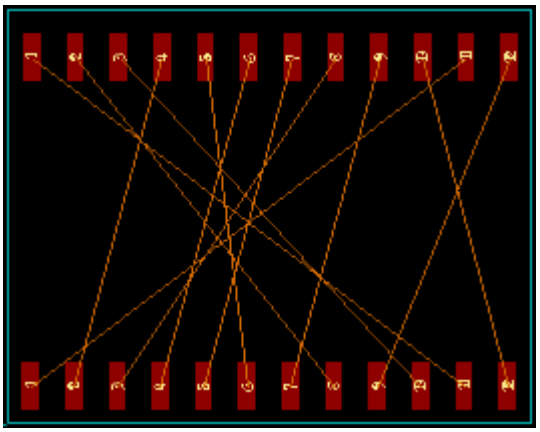
After you start the procedure, the dialog box shows the current length of links, the time elapsed since the start and a reminder about the need to stop the procedure manually.



If you stop the procedure, the dialog box remains open in a paused state. Later you can either continue automatic placement where you left off or perform some auxiliary operations (calculate the placement area, move all components into the placement area, and so on) first.

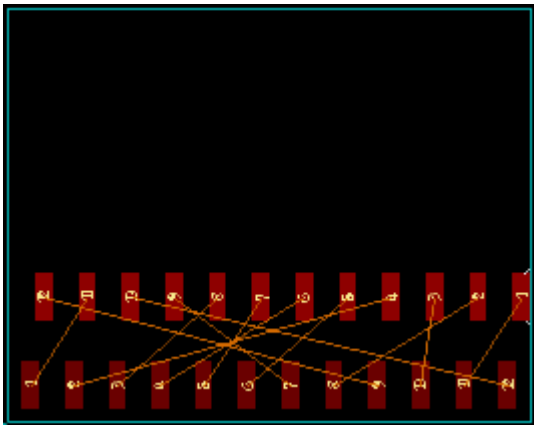


Example of the automatic placement procedure at work

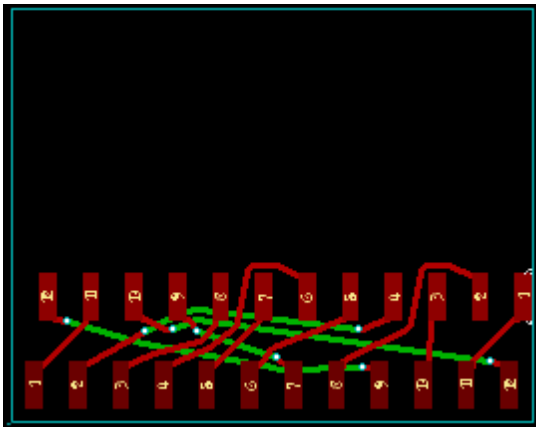


Pin positions before the procedure - length of links are 272.666 mm.

What the automatic placement dialog box shows.



Pin positions after automatic placement (length of links reduced to 127.705 mm).



The same after 30 seconds' worth of autorouting (the route length is 144.501mm, 8 vias, 9 DRC violations).

Locks

When the automatic placement dialog box is open (before the procedure is started and during a pause), all editor and automatic procedure panel tools are disabled except the ruler. In addition, the **Insert** menu is fully disabled.

While the procedure is in progress, all of the following are unavailable: functionality of the automatic placement dialog box, editor pane, automatic procedures pane and all buttons of the main panel except the **View** group (meaning, only the following buttons are available: **Previous View, Next View, Show All, Show Board**):

All of the main menu is also disabled, except the following: **View** and **Help** items, **Close Project** and **Exit** in the **File** item and **Summary** in the **Design** item.

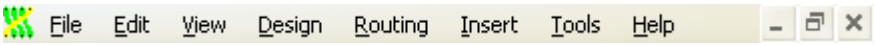
If you click **Close** while the procedure is in progress, you are prompted to confirm that you want to interrupt automatic placement. If you click **Close** during a pause or before starting the procedure, no prompt is displayed.

An additional feature of the automatic placement dialog box is the option to unpour coppers. This action is available if there is at least one poured copper. It unpours all existing coppers, including those that are fixed.

TopoR Editor Tools

This section describes the main menu of the program, the toolbar, the action bar, the selection filter, the visibility pane, the search pane and the context menus.

The main menu has the following sections:

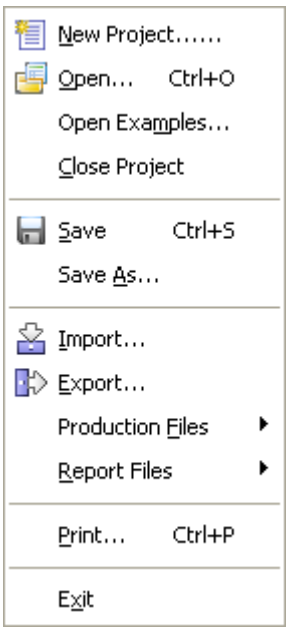


To open a section, click it or use the key combination **Alt** + underlined letter (F, E, V, D, R, I, T, H).

Main Menu Sections

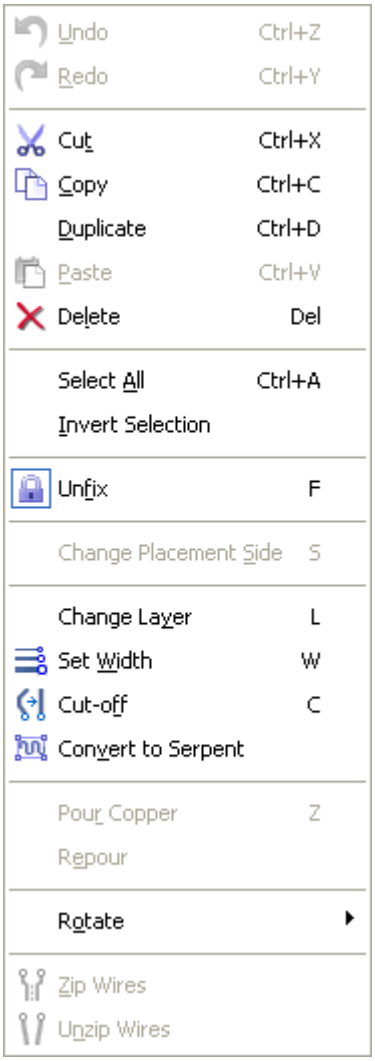
The contents of all menu sections are always the same, but the items that are currently unavailable are grayed out.

File

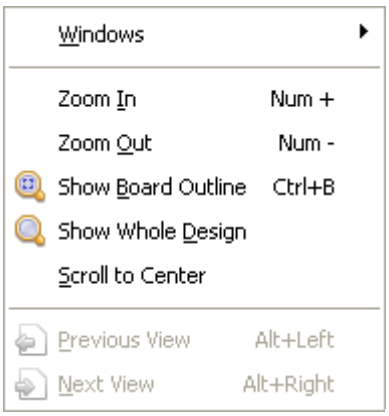


The **Open Examples** item takes you to the **EXAMPLES** folder, which contains the projects shipped with **TopoR**.

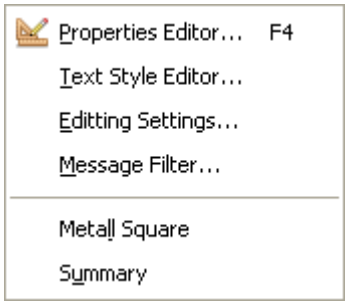
Editing


Undo Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Duplicate Ctrl+D
Paste Ctrl+V
Delete Del
Select All Ctrl+A
Invert Selection
Unfix F
Change Placement Side S
Change Layer L
Set Width W
Cut-off C
Convert to Serpent
Pour Copper Z
Repour
Rotate
Zip Wires
Unzip Wires

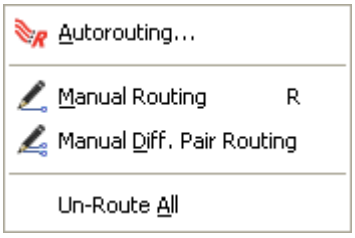
View


Windows
Zoom In Num +
Zoom Out Num -
Show Board Outline Ctrl+B
Show Whole Design
Scroll to Center
Previous View Alt+Left
Next View Alt+Right

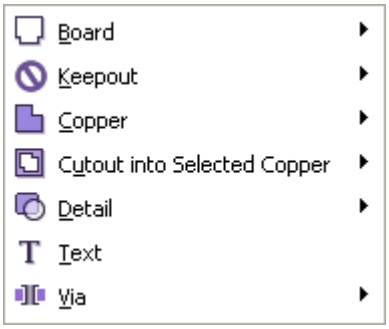
Design



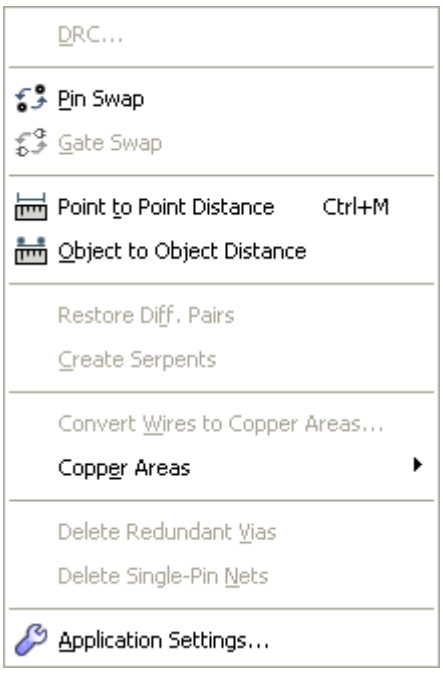
Route



Insert



Tools



Help

Using particular menu items is described in related sections of the manual.

Editor Toolbar

The toolbar is used for switching editor modes. It is a vertical bar that is located to the left of the main editor window by default.

At any time, only one of the modes is active. Which mode is active is indicated by a highlighted button in the toolbar. Most modes have variations (submodes). To switch submodes, right-click the icon of the mode you want. Switching the submode changes the icon of a button.

The selected submode for each mode is persistent during the current program session, but it is not saved to any configuration files and is not restored when you next start the program (including after a crash).

Modes and Submodes

– Default editor mode (used for selecting and moving objects)

: , , – Move objects in **Freestyle** mode

Submodes: don't push and don't consider clearances, push vias, push components and vias

, – Route

Submodes: route wires, route differential pairs

Board object creation modes:

Object to create/ submodes	Polygon	Rectangle	Circle	Arc from center	Arc from points	Circle	Line
Board outline			-				
Keepout							
Copper area							
Cutout in copper							
Detail on mechanical layer							

T – Create text labels

, , , – Create vias

Submodes: , single via, , series of vias, , matrix of vias

, , – Reassign

Submodes: , reassign pins, , reassign gates

, , – Measure distance (ruler)

Submodes: , distance between points, , distance between objects

Using the Toolbar

Click a mode button to make the mode active. If the mode has submodes, then the current submode becomes active.

Right-click a mode button to open the submode menu. The submode you select becomes current and active.

Pressing **Esc** exits the selected mode and switches the editor to default mode.

Exception to this rule: exiting the "ruler" takes the editor back to the previous mode.

Single-Action and Toggle Tools

Moving components in **FreeStyle** mode, wire routing, pin reassignment, via creation and distance measurements are all *toggle tools*, meaning that after you perform an action with the tool, the mode is not exited.

All other modes (all object creation modes and text label creation mode) are single-action tools, meaning that after you perform an action with the tool, the editor returns to default mode.

To use any of these modes as a toggle tool, hold down **Shift** when you activate the mode.

Action Pane

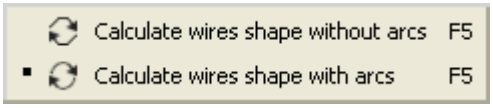
The topology editor action pane is used for the most common automatic procedures.

This is a vertical panel located to the left or right of the main editor workspace. By default, it is on the left of the main window, under the editor toolbar.

The top three buttons are auto-procedure buttons; the last button ("droplets") is a toggle.

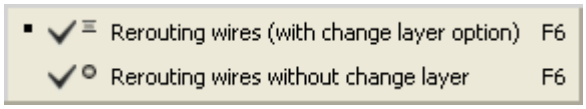
Auto-Procedure Buttons

Calculate wire shapes; there are two submodes:

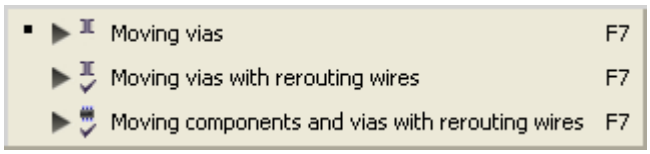


You can select one of two ways to calculate wire shapes: with and without the use of arcs. Using arcs means that wires take smooth paths around obstacles while observing the required clearance and go in straight lines from bend to bend. Not using arcs means wires consist only of straight lines.

Reroute wires; there are two submodes:



Move vias; there are three submodes:



To run an auto-procedure in the current mode, click its button. To change the submode, open the submode menu by right-clicking the button. In the menu, click the submode you want.

The submode set for the "Calculate wire shapes" auto-procedure is saved in the design file. The submodes set for the other two auto-procedures are persistent only during the current editing session.

If you are editing an autorouting variant, then the current submode for the "Calculate wire shapes" auto-procedure depends on the state of the **Use arcs** option in autorouting settings.

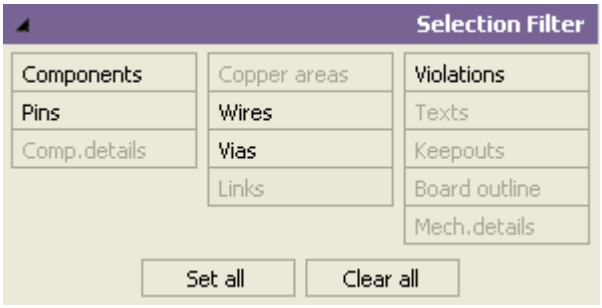
Droplets

The **Droplets** button is a toggle that controls the corresponding wire shape option. When the button is highlighted, the option is on. Clicking the button also changes the shapes of all wires on the board. The state of the toggle is saved with the current design.

Selection Filter

To conveniently select objects of various types, use the selection filter.

The filter has 12 independent flags. If a flag is set, then objects of the corresponding type can be selected in the editor (including the **Select All** command). Clicking **Select All** in the filter sets all 12 flags, and clicking **Clear All** unsets them.



To set or unset a flag, click it. The other flags are not affected.

To set a single flag and unset all the rest, right-click the flag.

Whether specific flags are available depends on the current mode (if the object type is not editable in the current mode, the corresponding button is grayed out).

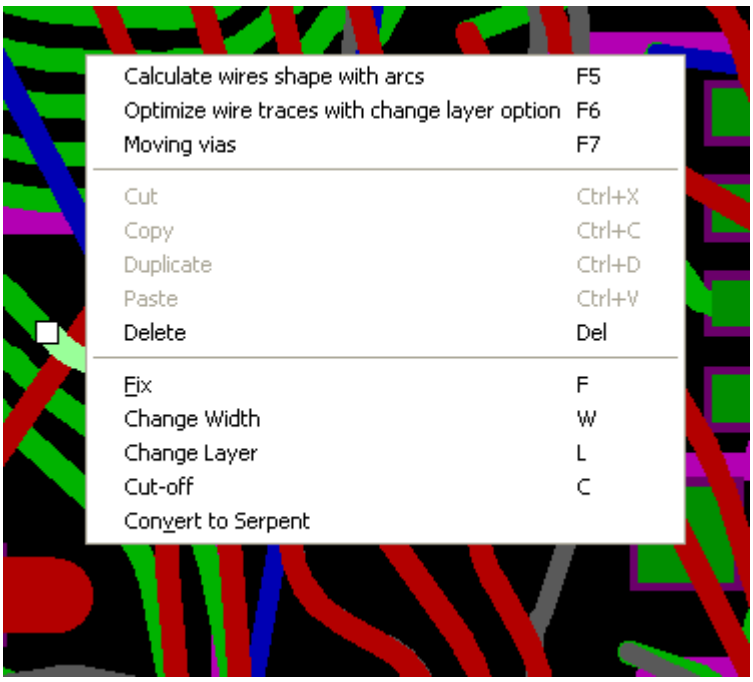
.

If you want to select an object but cannot, make sure selecting it is not disabled in the selection filter

Clicking **Ctrl+A** selects all objects for which selection is enabled in the filter.

Context Menus

Right-clicking anywhere in the workspace brings up a context menu. Alternatively, press the **`Windows Context Menu`** key on an extended keyboard. These menus have a generic part and a specific part.



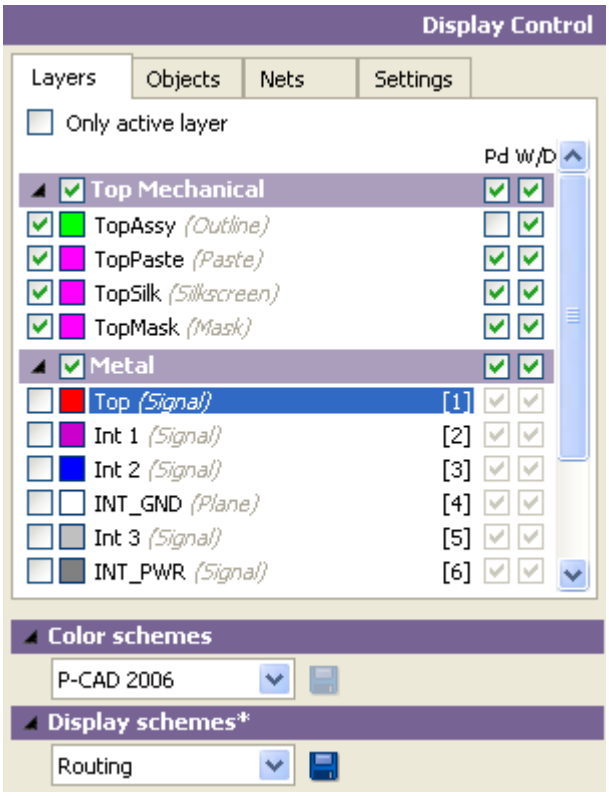
The generic part of the context menus contains the following items: **Cut, Copy, Duplicate, Paste** and **Delete**. Items that cannot be used at the moment are grayed out. Whether items are available depends on which objects are selected and what is in the clipboard.

The specific part of a context menu lists actions depending on what kinds of objects are selected. For example, the screenshot above shows the specific part for when a wire segment is selected. For each available action, the corresponding hotkey is indicated.

Which Actions Are for Which Object Types

Open Padstack Editor — component pad, mount hole, freepad
Open Impedance Editor — wire with adjustable impedance, serpent
Open Via Editor — via
Open Style Editor — text label
Fix, Unfix — component, mount hole, freepad, wire segment (including differential pair segments), serpent, via
Rotate — component, component label, copper area, cutout in copper, keepout, board outline, text label
Change Placement Side — component
Re, Un — copper area
Zip, Unzip, Change Width, Change Layer, Straighten — wire, wire segment, serpent
In addition, the context menu provides the **Convert to Serpent** action for all wires and the **Convert to Wire** action for all serpents.

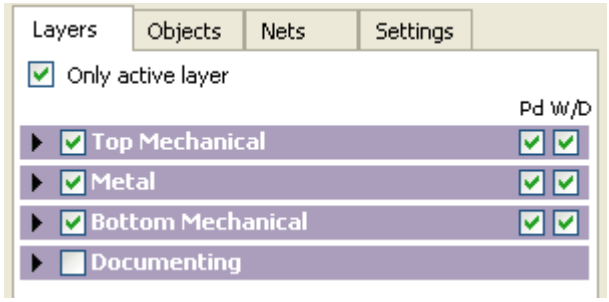
Display Control



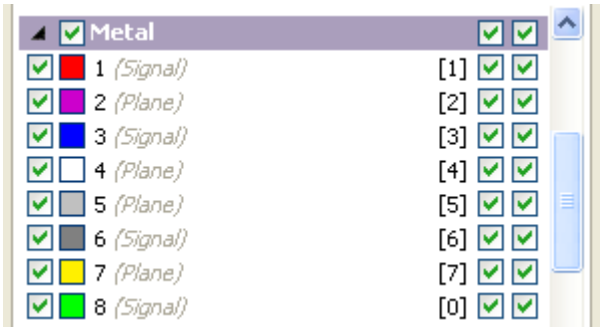
Layers

The **Layers** tab lets you show and hide layers (top/bottom mechanical, metal and documenting), and also show and hide pads, wires and details on each layer.

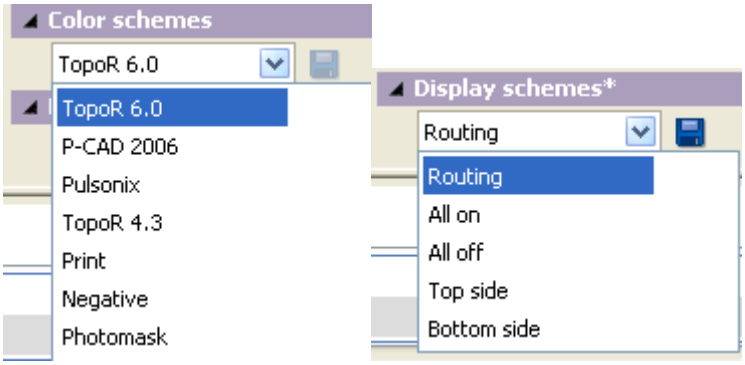
TopoR has the notion of an *active layer*, meaning the layer where searching for objects starts under the cursor. The active layer is drawn on top of all others. When you create a new design, the top metal layer becomes active, but any layer can be made active instead. New objects (keepouts, copper areas, text labels) are placed on the active layer if possible. If the **Only active** option is selected, then all other layers are hidden.



The name of a metal layer is followed by a hotkey in square brackets. This hotkey makes the layer active.



At the bottom of the pane are tools for managing schemes. **TopoR** makes a distinction between color schemes (for configuring object colors) and display schemes (for configuring object visibility).



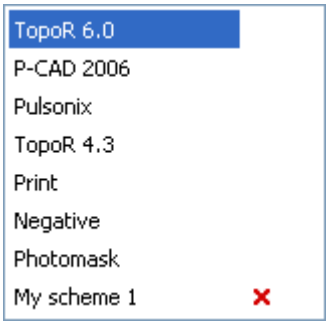
The following operations on schemes are available:

Set the current scheme from a drop-down list.

Change the current scheme. After you have made configuration changes, the **Save** button becomes available next to the scheme drop-down list. Predefined schemes are not editable.

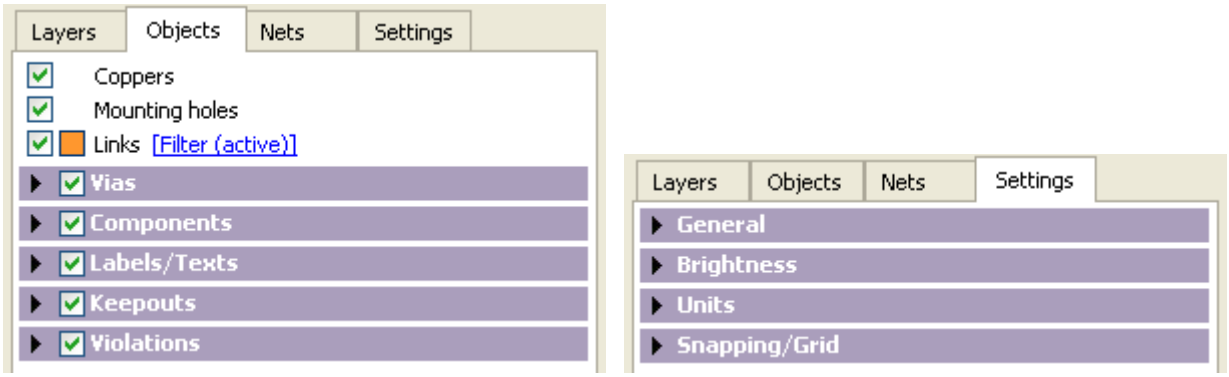
Create a new scheme. For that, type the name of the new scheme in the drop-down list's edit box and click **Save**.

Delete a scheme.



For that, click the **Delete** × icon in the drop-down list. Predefined schemes cannot be deleted.

Objects and Settings tabs



On the **Objects** tab you can hide, unhide and change the color of the following:

Copper areas
Mount holes
Links
Vias
Components
Text labels
Keepouts
Violations

On the **Settings** tab you can:

Select the workspace background
Select the board outline color
Select whether to show wires as thin lines

If this option is enabled, the wires on the board are displayed as thin lines; otherwise their actual width is shown. Thin lines can be useful when you edit areas densely packed with wires where no more room can be made due to design constraints. In this case, it can be difficult to work out the topology of wide wires that run close by or even over one another.

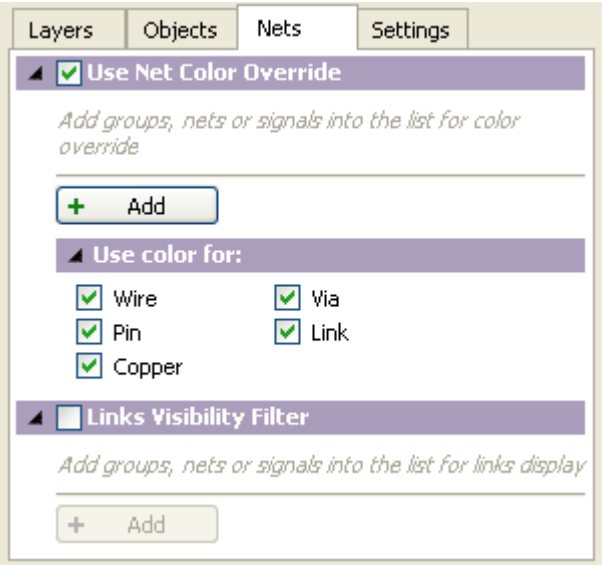
Hide and unhide serpent shapes
Configure brightness for selected and dimming for unselected objects
Select whether to use imperial or metric units

By default, this option is set to metric, so the total wire length is shown in centimeters and wire width in millimeters. These are the customary units that are most commonly used for representing those characteristics.

Enable or disable angle snapping

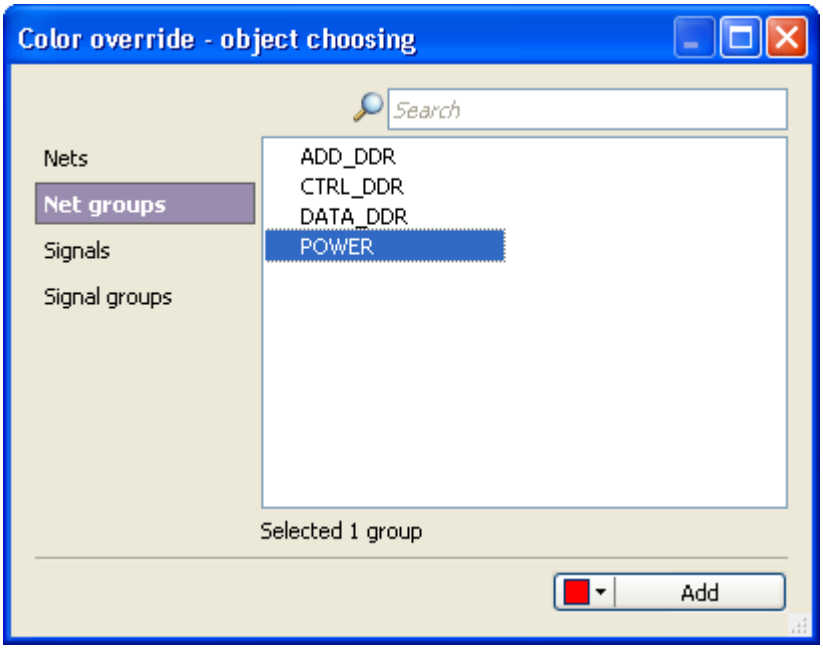
Specify the type (points or lines), color and horizontal and vertical step for the grid

Nets

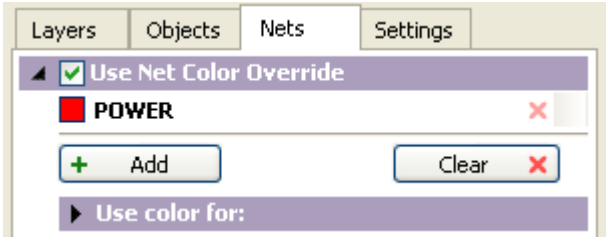


The **Nets** tab controls net display modes. The **Use Net Color Override** option helps group signals, nets or net groups by setting a custom color for them. If this option is enabled, you can select the objects to assign the custom color to wire, pin, copper, via or link.

Click the Add button to open an object selector where you can specify the object type (net, group, signal or signal group), select objects of this type and add them.

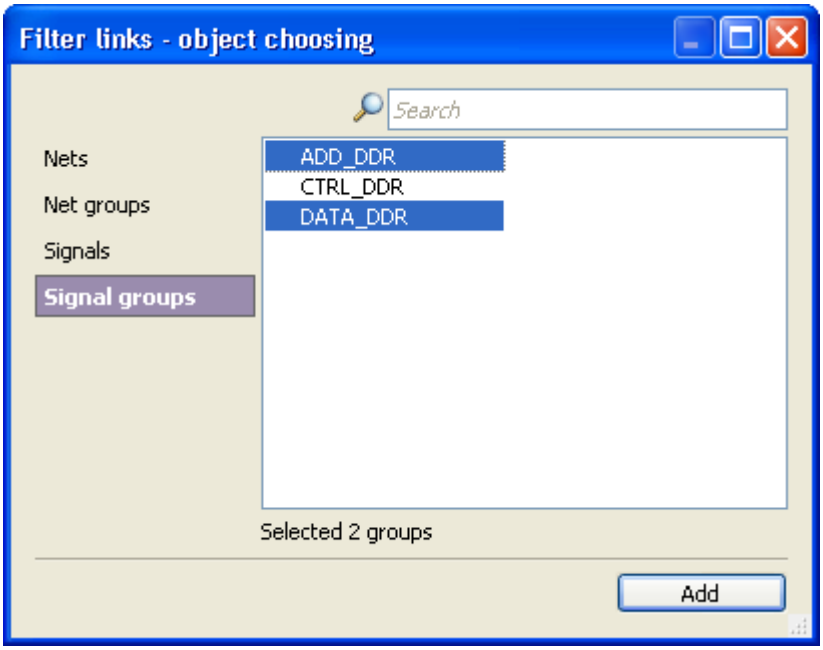


The objects you add are shown in the list:

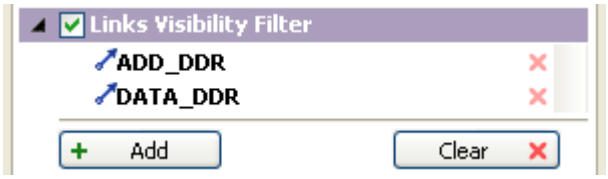


The **Net line visibility filter** lets you select signals, nets and net groups where you want unrouted net lines to be displayed. Click the 

button to open a filter dialog box similar to the one used for specifying override colors.

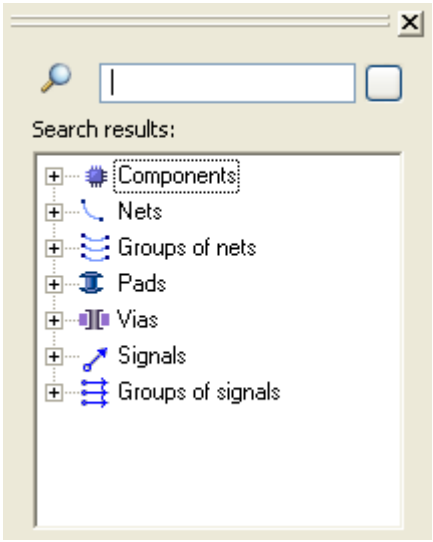


The nets, signals and signal groups you add are shown in the display filter list:



You can clear this list and the color override list at any time.

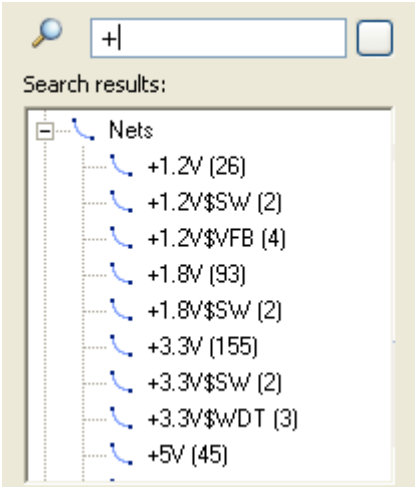
Search Panel



To open the Search panel, click **View ► Windows ► Search panel** or press **Ctrl+F**. This panel helps you find and select components, nets, pads and vias of specific types, and signals by name.

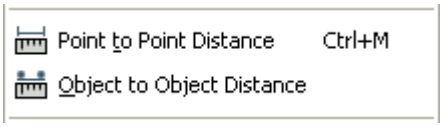
Clicking a name selects the corresponding object. Double-clicking it centers the workspace view on the object.

The filter box makes searching more convenient: type a string to narrow the list down to only those names that contain the specified substring.

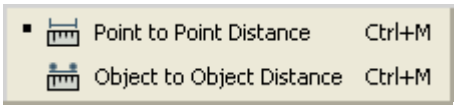


Ruler Tool

The ruler is used for measuring distances on the board. You can activate it from the menu (mode 1: **Tools ► Point to Point Distance** or **Ctrl+M**, mode 2: **Tools ► Object to Object Distance**).

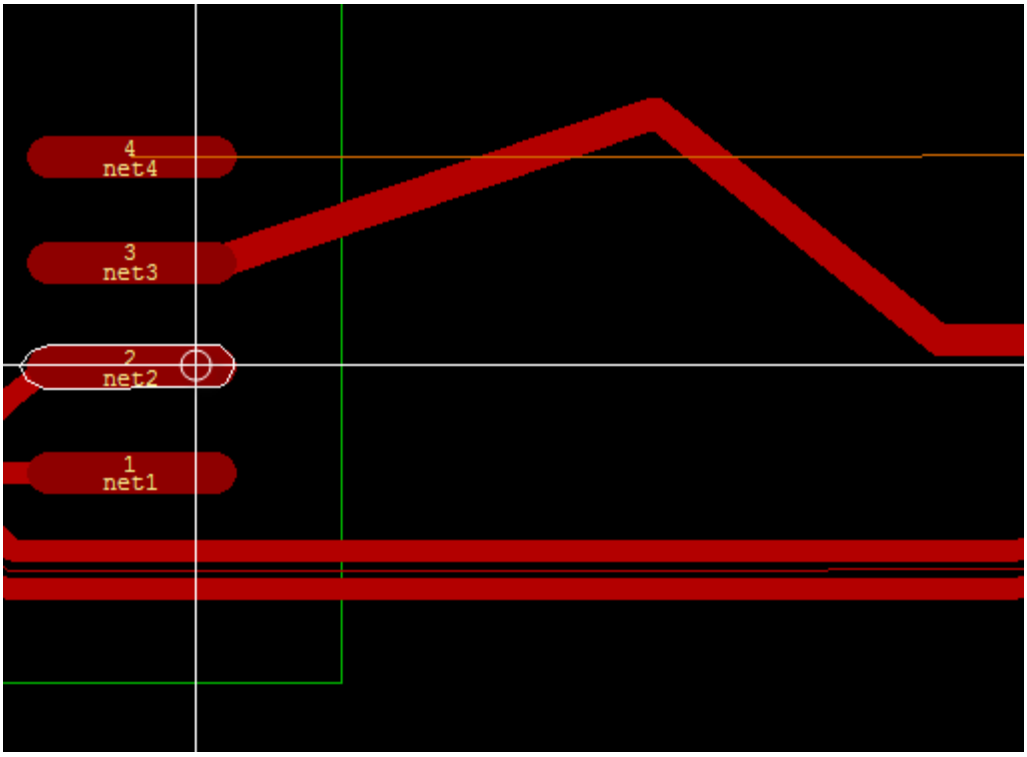


You can also activate it from the editor toolbar (one of the two submodes is enabled, and you can change the submode by right-clicking the ruler icon (or):



The ruler is also enabled with the **Ctrl+M** hotkey. This activates the ruler in the submode that is currently selected on the toolbar.

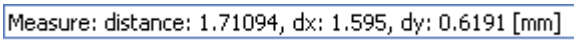
When you hover the cursor over an object's area, its outline is highlighted, and a small circle appears at the cursor's hotspot.



Clicking at this time selects the first object and marks the current cursor position as point A. After that, the tool is ready for selecting the second object. Until you select it, the ruler draws a line from point A to the current cursor position. After the second object is selected, the tool calculates the distance and draws a corresponding line.

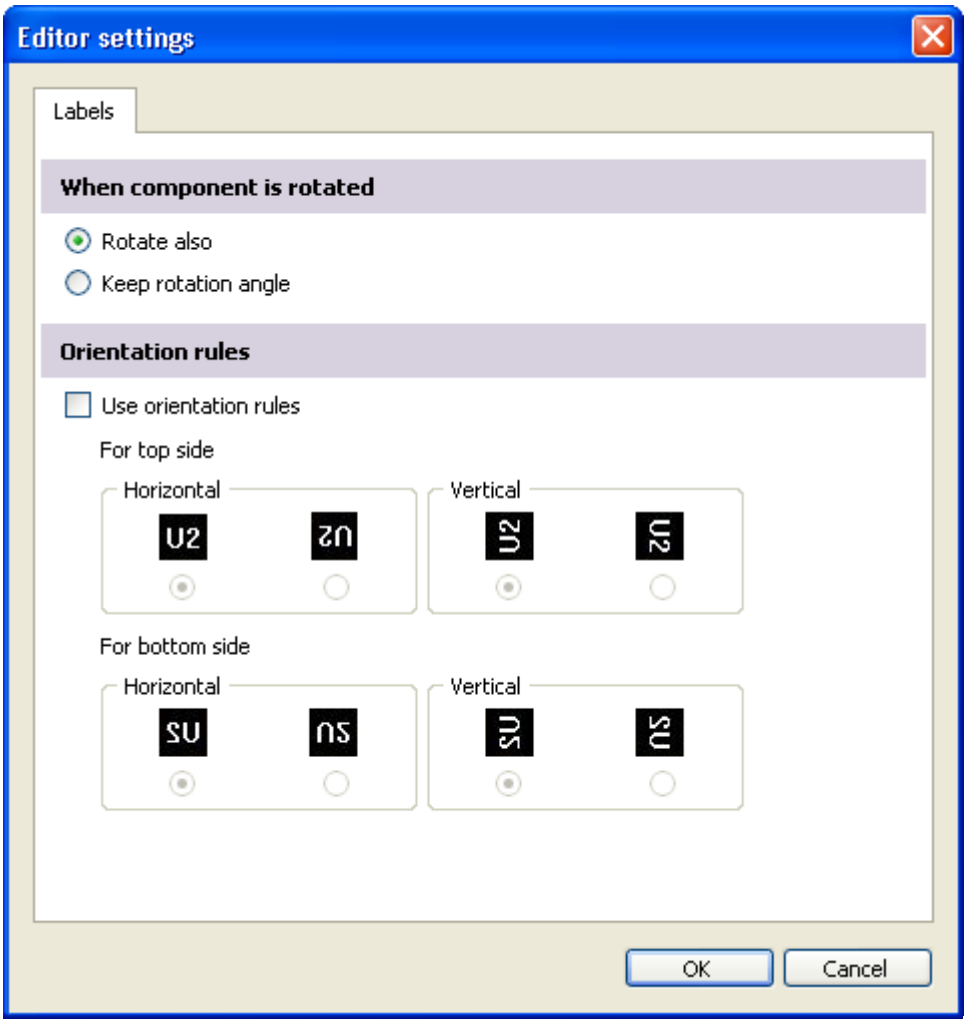
Distance between objects is the minimal possible distance between the points of the selected objects. If the object is a wire, it is the minimal distance for all its segments.

The measurement results (the value and the coordinate axis deltas dx and dy) are displayed in the status bar:


Measure: distance: 1.71094, dx: 1.595, dy: 0.6191 [mm]

Editor Settings

The **Design ► Editor Settings** menu item contains a single tab, **Labels**, where you can define the behavior of labels during component rotation and how labels are oriented in the design.



Behavior During Component Rotation

Rotate also — The label is rotated along with the component by 90°, 180° or 270°.

Keep rotation angle — Component rotation does not affect the label.

Orientation Rules for Labels

Selecting the **Use orientation rules** option lets you set orientation for labels on the top and bottom side.

Horizontal — Lets you set the orientation for all labels where the text is entered horizontally, so that the text can be viewed at the angles of 0° and 180°.

Vertical — If the text is entered vertically, it can be viewed at the angles of 90° and 270° degrees.

Label orientation rules do not apply to arbitrary-angle rotations.

Manual Editing

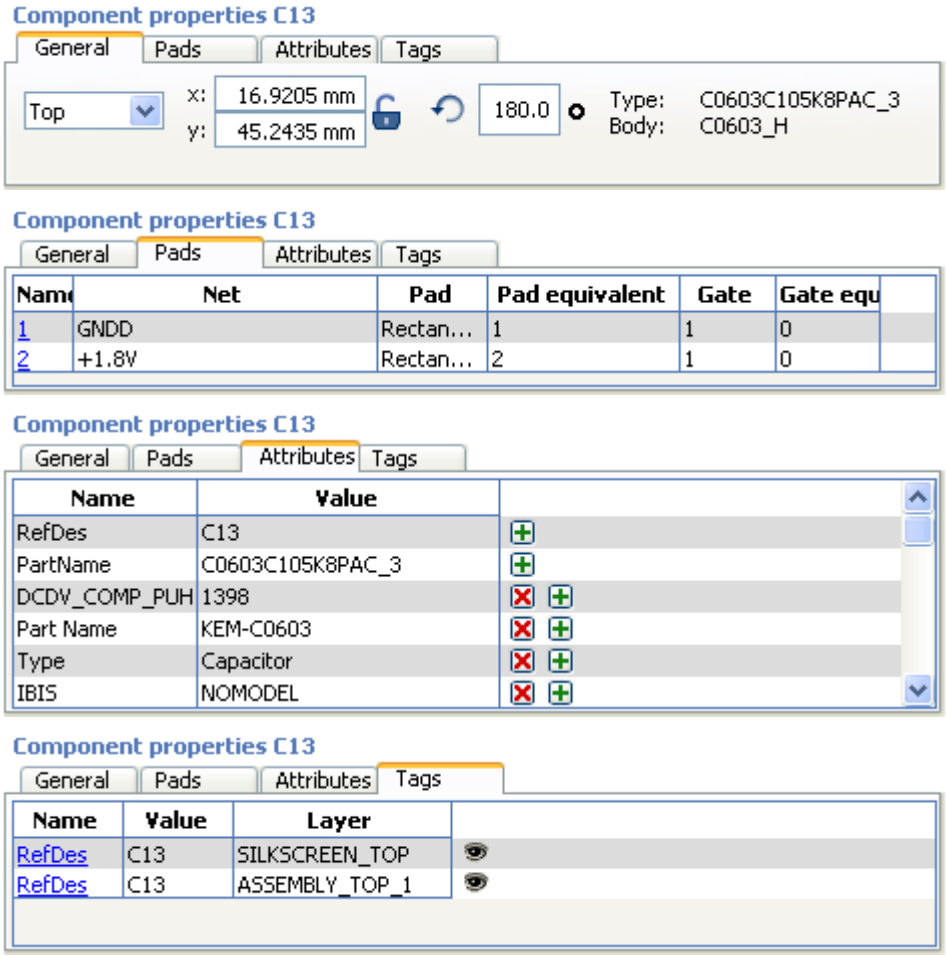
As noted above, **TopoR** is not only routing software but also an efficient topology editor. This section describes editing tools available in **TopoR**.

Component Operations

Selection

To select a component, click it. You can find brief information about the selected object in the center of the status bar (under the workspace). On the left of the status bar are the current project parameters (mode, wire length, number of vias and number of design constraint violations); on the right are the current cursor coordinates.

For more details about a component, see the **Properties panel**.



To select multiple components, hold down **Ctrl** while clicking them.

Alternatively, you can use box selection.

Fixing and Unfixing

To fix or unfix a selected component, click the **Fix** button or press the corresponding hotkey (**F** by default). As mentioned previously, during routing **TopoR** is capable of shifting (moving) components slightly. Some components need to be located in fixed positions. Such components include sockets, mount holes and so on. Unfixed components are considered approved for shifting. In particular, this means that unless strict checking is enabled, **TopoR** will not limit the number of wires between unfixed components; it is assumed that any resulting bottlenecks will be eliminated by manual component moving. Fixing components has a significant effect on the PCB topology. Therefore, you should fix only those components that really need it.

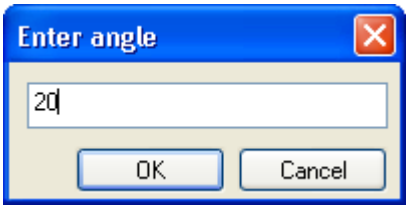
Changing Sides

To change the placement side for a component, click **Edit ► Change Component Side** in the main menu.

Rotations

To rotate components, use context menus, where the **Rotate** item has the following secondary items: **Rotate by 90°, Rotate by -90°** and **Rotate by Angle**. The first two commands have associated hotkeys. You can do the same by clicking **Edit ► Rotate** in the main menu.

To rotate by an arbitrary angle, click **Edit ► Rotate ► Rotate by Angle** and in the **Enter angle** dialog box that opens specify the angle you want in degrees:



All rotations are relative to the object's center in its current position.

For each object, **TopoR** is aware of a point that is assumed to be the object center. In components that have no discernible center, this point is the same as the center of the object's bounding box. To display bounding boxes, enable the **Bounds** option in the **Objects ► Components** section of the display control panel.

Working with Component Attributes and Labels

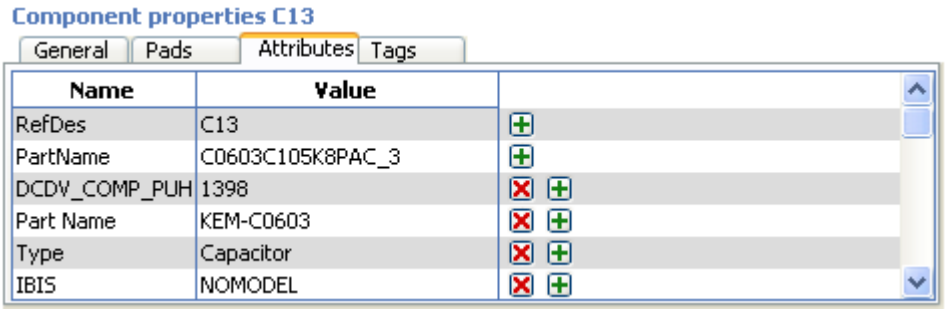
The list of the available attributes of a component is shown on the **Attributes** tab of the properties panel. The tools on the **Attributes** tab let you:

Add a new attribute. For that, specify the name and value of the attribute in the boxes on the bottom line of the attribute component list, and click the **Add Attribute** button to the right of the attribute value box.

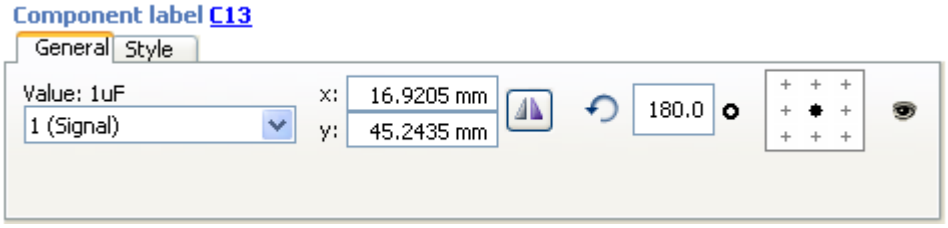
Delete an attribute. To delete an attribute, click the **Delete Attribute** button. The predefined **RefDes** (reference designator) and **PartName** (component name) attributes cannot be deleted.

Change the value of an attribute.

Add a label for an attribute. A label is what indicates the attribute on the board. To add a label, click the **Add Label for Attribute** button in the attribute row. An attribute can have multiple labels.



A newly-created label becomes the selected object. The properties panel shows the properties of the new label.



You can perform the following operations in the label's properties on the **General** tab:

- Change the layer for the label
- Move the label by specifying coordinates
- Rotate the label by an arbitrary angle
- Enable or disable mirroring
- Move the snap point
- Enable or disable the label

All style parameters can be edited on the **Style** tab. After a parameter has been changed, a label is assigned a “new style” that has no name and exists only for this particular label. This “new style” can be added to the style list by supplying a name for it and clicking **Save** to the right of the drop-down list.

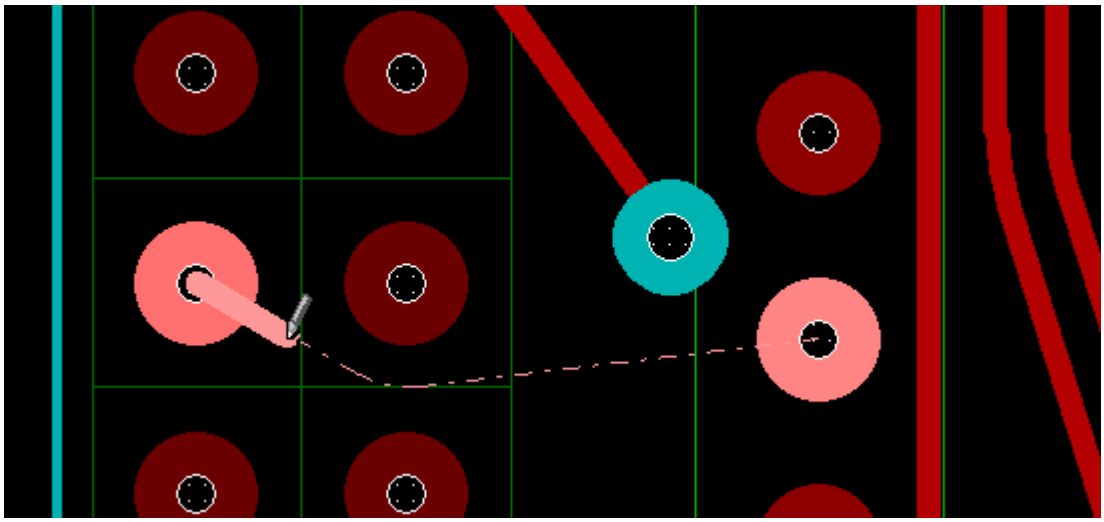
To disable a label, click the **On/Off**  button on the **General** tab. A disabled label is not displayed or written to any exported files. To enable a label, use the same button again in the label properties or on the **Labels** tab in the component properties.

Operations on Wires

Routing

The **Route Wires** mode lets you route wires on metal (signal, backup) layers from primitives (pads, polygons, vias, segments and T-points) that have nets assigned.

Unrouted nets or net segments are shown as straight line segments (link lines) that connect pairs of neighboring primitives that have two link slots. In manual routing mode, **TopoR** automatically searches for a violation-free wire path on the current layer. If such a path cannot be found, the wire is shown as dashed line.



You can accept the discovered path (default shortcut: **Shift+Q**) to route the wire automatically. Otherwise, automatic searching will be performed for each new point under the cursor. The target can be changed by pressing the **Q** key if the path to the new target is shorter.

If a path cannot be found on the current layer, then the link line is replaced by a “target designator” — a dashed line that runs to the nearest point on the non-connected net fragment.

To undo the previous action, press the **Backspace** key.

When you change the layer during while routing a wire, a via is automatically added for transition to the target layer.

TopoR does not force you to mind the precise geometry of the wire you are routing or to try and meet design constraints; setting its topological route is sufficient. During manual editing, overlap between the wire you are routing and other components is not considered an error. An even number of intersections with other wires in the absence of topological elements between them is not an error either. These violations are eliminated when the wire shape is calculated (default hotkey: **F5**).

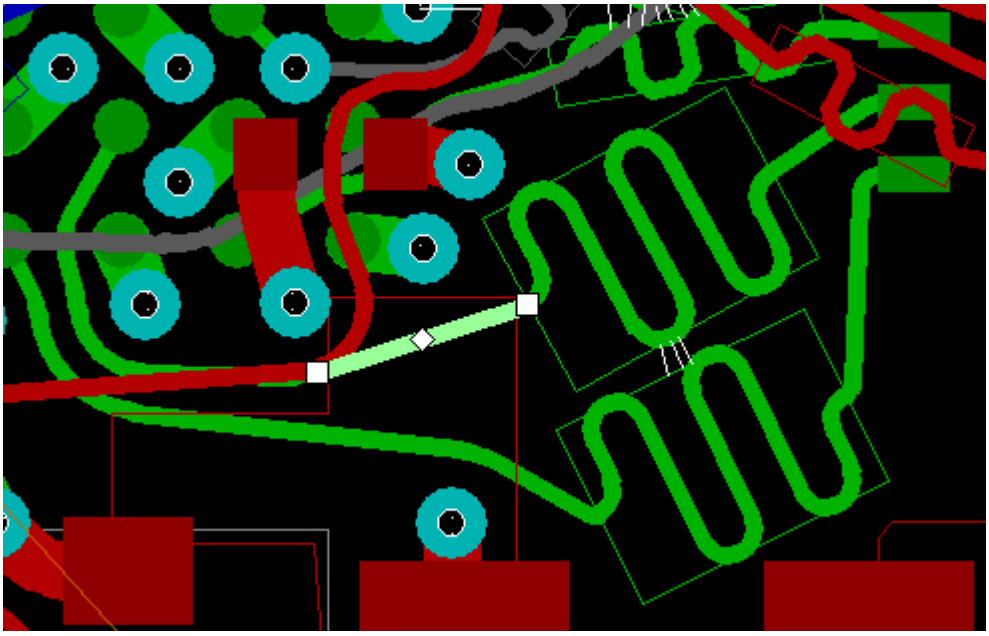
Deleting Wires

To remove a selected wire segment or via, click the **Delete** button or press **Delete**.

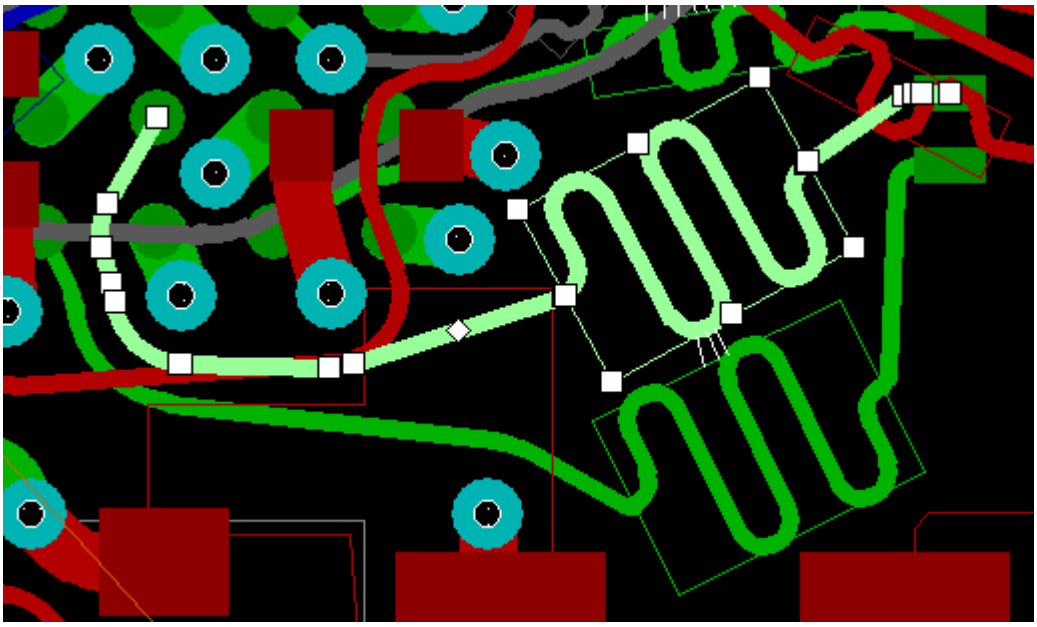
Selecting Wires and Segments

To select a wire segment (straight line segment), click it. The wire segment is highlighted with the color set in the options, and several markers appear on it (for short wires there are only two markers: at the start and end; the number of markers increases as the wire gets longer). You can edit the wire by moving these markers.

If you drag a wire by a spot that has no marker, the program performs a parallel displacement of the wire.

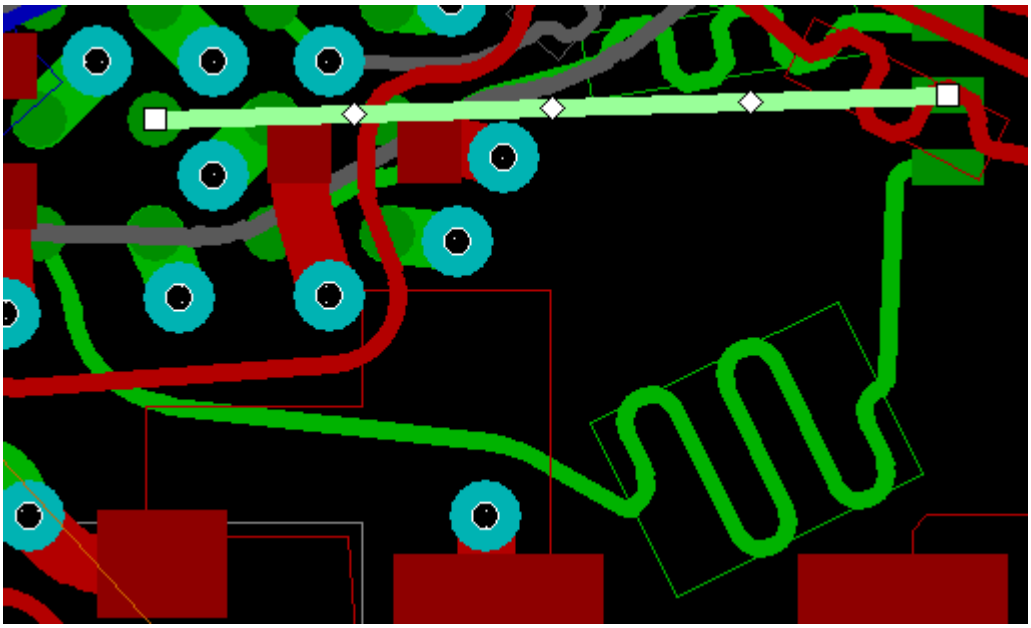


When one of the wire segments is selected, you can select additional segments by holding down **Shift** and clicking them. Likewise, **Ctrl**-clicking segments lets you select fragments of multiple wires.
To select the whole wire when one of its segments is selected, press **X** (cycle selection). Pressing the **X** key a second time selects the entire net that the wire belongs to (meaning all wires, pads and vias in the net), and pressing the **X** key a third time brings back the original segment selection.



Straightening Wires

If a multi-segment fragment of a wire is selected, clicking the **Straighten** button on the toolbar (or pressing the **C** key) replaces the selected part of the wire with a single segment; in other words, it straightens the selected part of the wire.



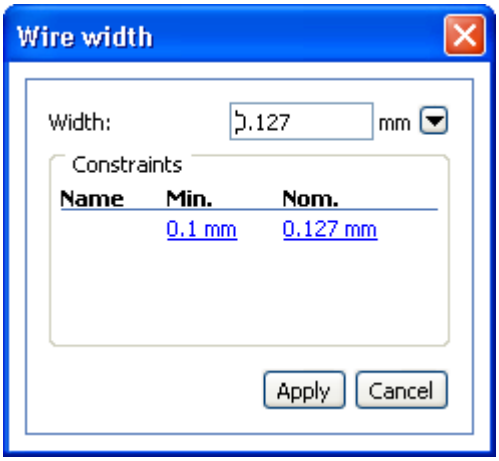
After wire straightening, confirm that the topological route of the modified fragment is correct and there are no topology violations: the route must not cross (in a topological sense) other wires on the same layer or pads belonging to other nets.

The following are considered topology violations:

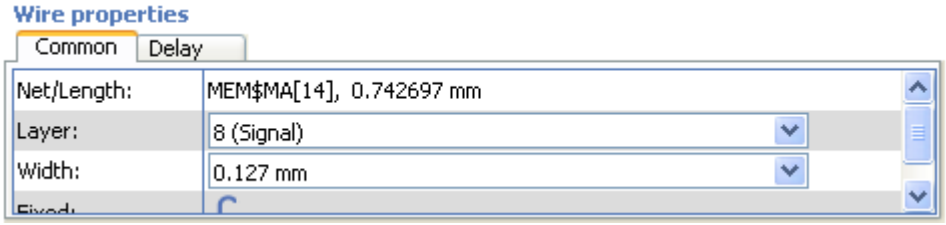
An even number of intersections of the wire with another wire on the same layer
Crossing the center of a pad that belongs to a different net

Changing Wire Width

To measure the width of a wire segment, do one of the following: use the wire context menu, click the **Change Wire Width** button or press the **W** key. In the **Wire width** dialog box that opens, specify the width value you want.

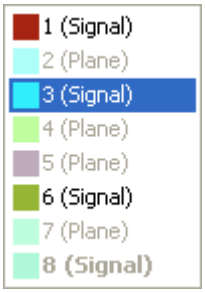


You can also change the width value in the **Wire properties** panel.



Changing the Wire Segment Layer

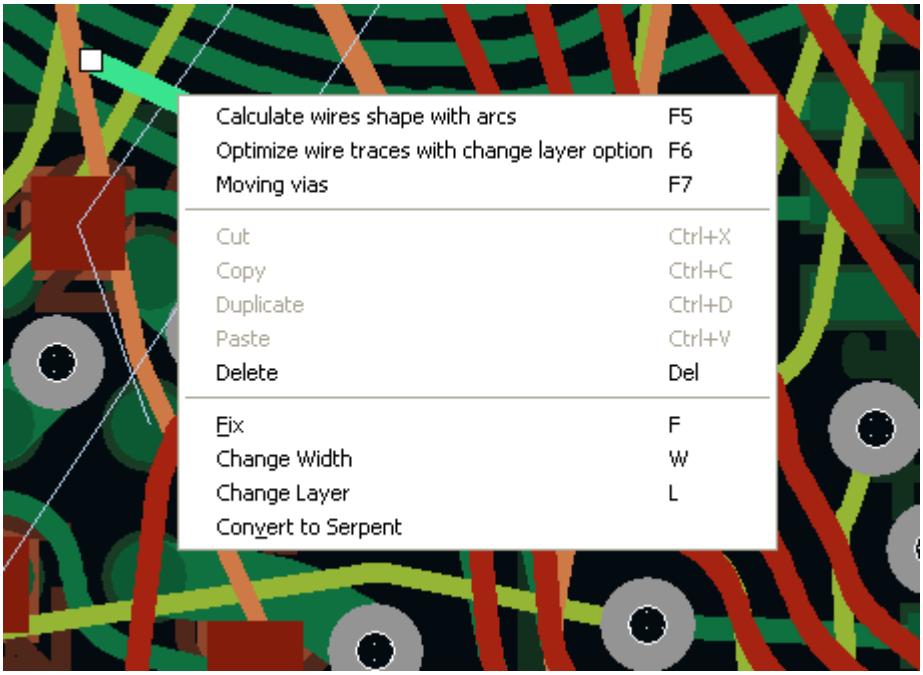
To change the layer of a wire segment, right-click it and select **Change Layer**, or press the **L** key. After that, select a name from the list that appears:



.

Vias will be added to the ends of the segment if necessary.

Another way is to use the hotkey. The layer selection menu appears at the cursor.



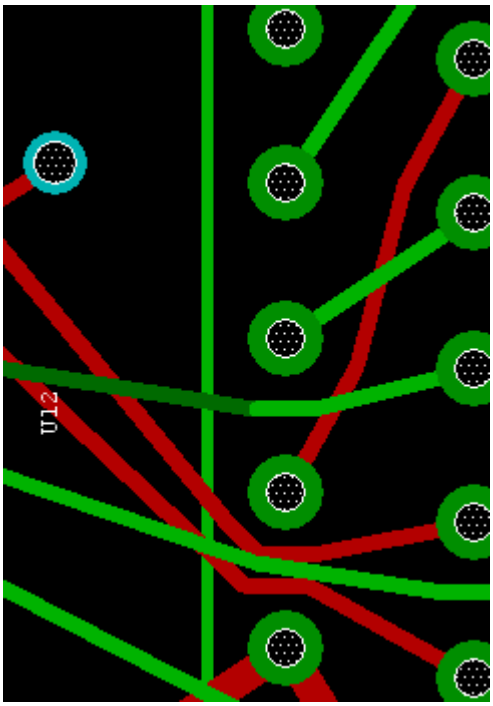
Box selection

Whether an object (component, wire and so on) is affected by box selection depends on which way the cursor is moving. If it is moving left to right (for example, from the lower left corner to the upper right), then only those objects become selected that are fully inside the selection box. If the cursor is moving right to left (for example, from the lower right corner to the upper left), this selects all objects that are at least partially inside the box.

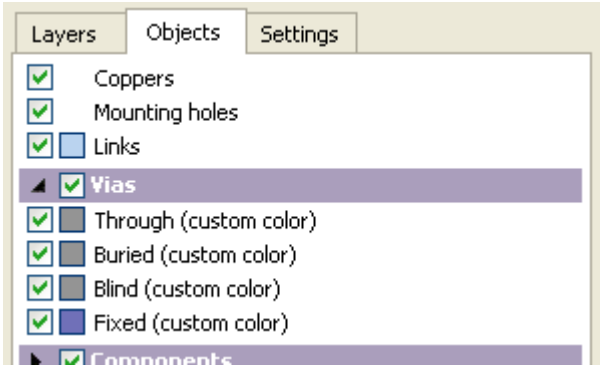
For details about other selection techniques, see Working with Components.

Working with Fixed Wires and Vias

To fix a wire segment or via in a specific location, select the object and click the **Fix** button or press the **F** key. Fixed wires are drawn in a darker shade of the layer color.



Fixed vias are also drawn in a different color, which is set in the **Objects** section in the **Display Control** panel.



To edit a fixed object (for example, reshape a wire or shift a via), first unfix it. The fixed state of wires and vias is taken into account in all modes except Placement mode.

How Special Cases are Handled

A wire is selected that is incident to a via

Fixing such a wire freezes both the wire and the via that it is incident to.

Unfixing the wire unfreezes only the selected wire itself; the state of the via does not change.

If no other wires incident to a via are in a fixed state, the via is unfixed.

A via is selected

Changing the state of the via does not change the state of any wires incident to it. This means that the following situation is possible: fixed wire—unfixed via.

A component is selected that is incident to a fixed wire

When such a component is moved, the shape of the fixed wire does not change. A flexible linking wire is created between the component and the original fixed wire.

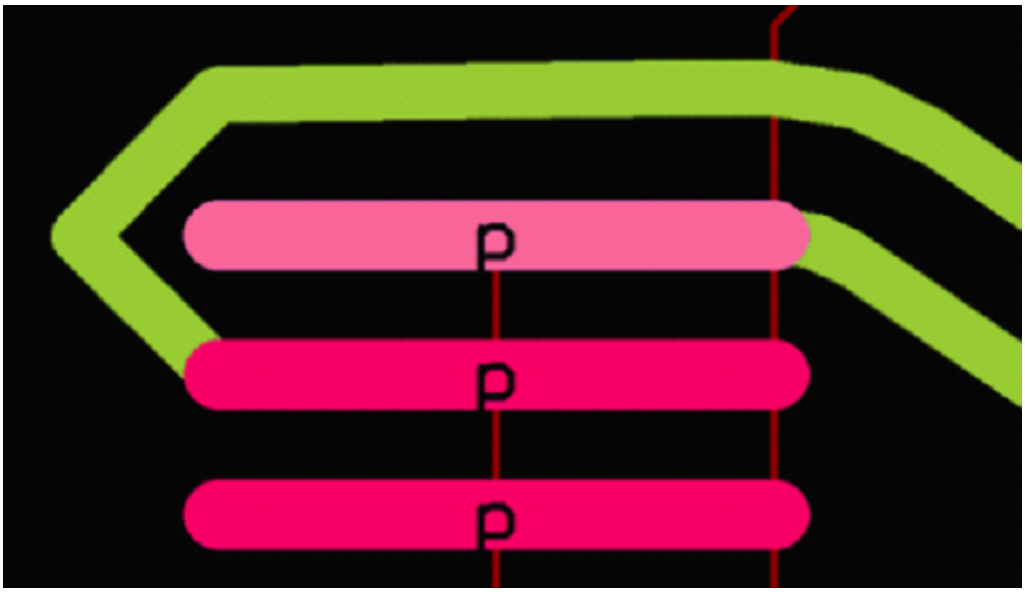
Moving components that are incident to fixed wires

If there are fixed objects in the selection, the move operation is not available.

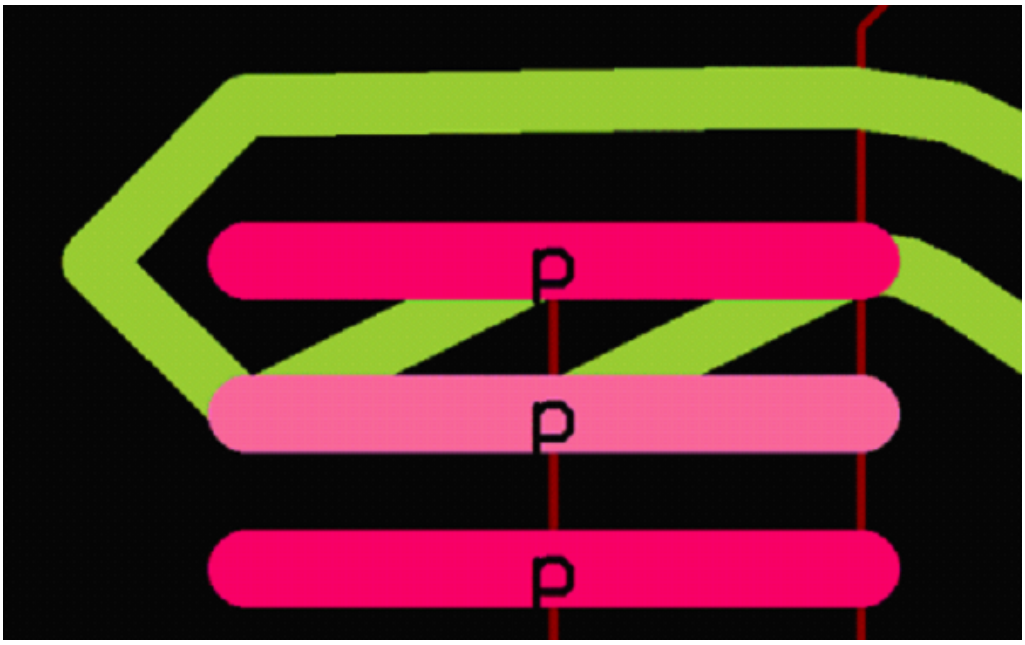
When you move a via or component that is incident to a fixed wire, an unfixed wire is created between the object you are moving and the fixed wire.

Reassigning Functionally Equivalent Pads

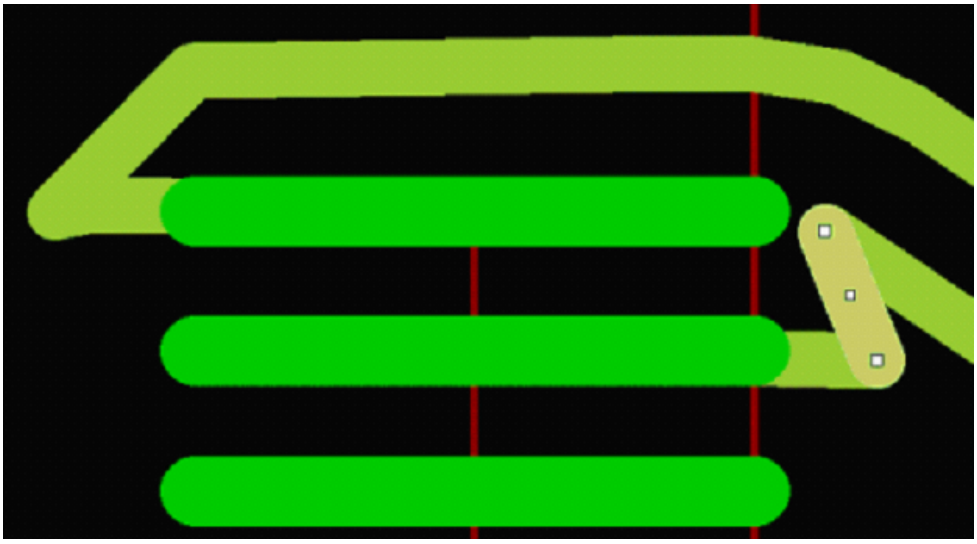
To enter reassignment mode, click **Tools ► Pin Swap** in the main menu. In this mode, groups of functionally equivalent pads are drawn in different color. Each group is marked by a letter **`[A-Z, a-z]`**. In addition, groups are easily distinguished by color (6 colors are available). If all six colors have been used, the next letter is taken and the colors are repeated. To select a pad, click it.



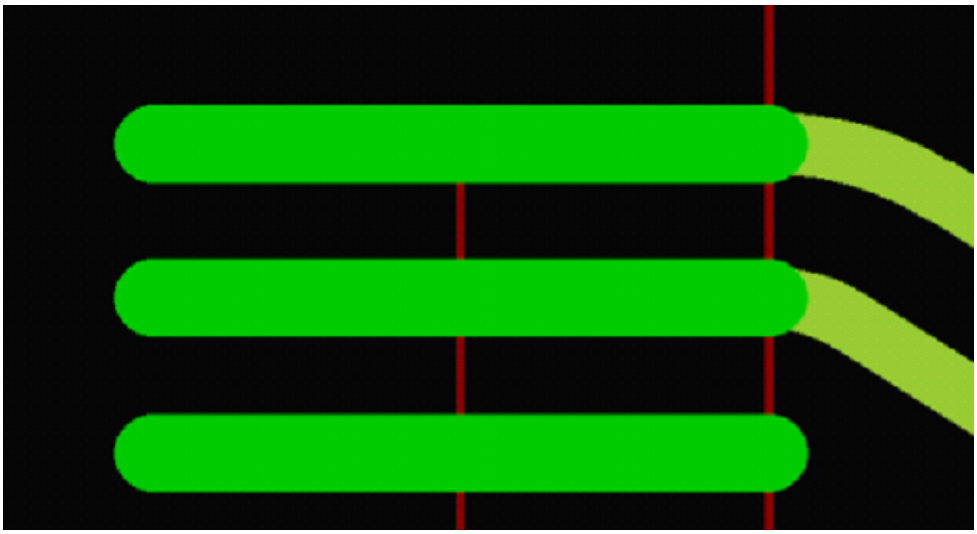
Clicking a pad in a group a second time reassigns pads. Net numbers are swapped for pads, and connected wires are switched.



Normally, you need to edit wires to eliminate intersections with pads after this.



The shapes of the wires are finalized in **FreeStyle** editing mode.



During wire switching, if the pads are planar and located on different layers, the wires remain unconnected (similar to changing the placement side of a planar component).
If the second click does not hit a pad in the selected group, the first pad is deselected, and the command is not performed.

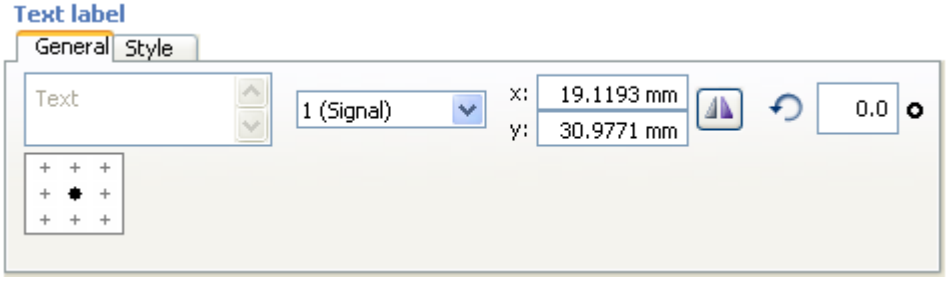
Reassignment information is written to a **.eco** file.

Sample **.eco** file:

```
;       TopoR       Version       6.0.4.0       [C:\TopoR\EXAMPLES
\Example_04\ADP_SPEC.fsx]--T10u Mar 06 10:52:21 2014
; --------
PinSwap "D6-10" "D6-12"
PinSwap "D6-12" "D6-14"
PinSwap "D6-14" "D6-10"
PinSwap "D15-1" "D15-18"
```

Working with Text Labels

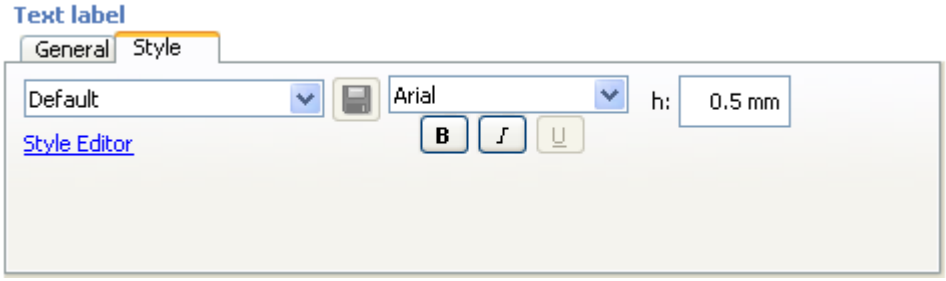
To insert a text label, use the **Create Text Label** button on the toolbar or click **Insert ► Text Label** in the main menu. Next, click where you want to place the text. A box with "Text" in it appears at the specified location, and in the **Properties panel**, the **Text label** panel is added, where you can edit text.



The **Text label** panel lets you:

Change the text color
Rotate the text by an arbitrary angle (in degrees)
Mirror the text using the **Mirror** button
Select the text style
Set the pivot position and how the text is justified

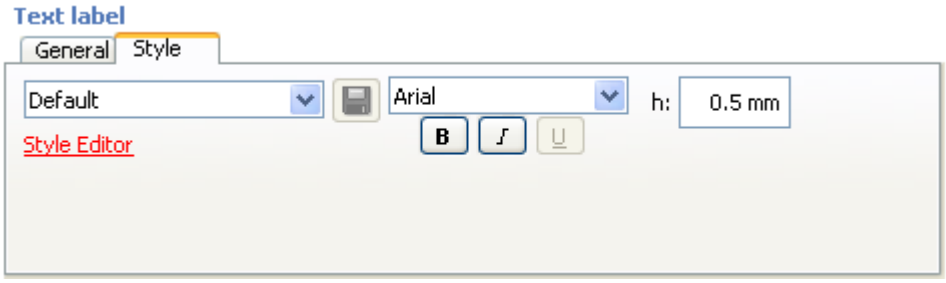
· Select the layer: the text can be on the top or bottom side (silkscreening), or on routing layers
· Launch the style editor to create a custom text style



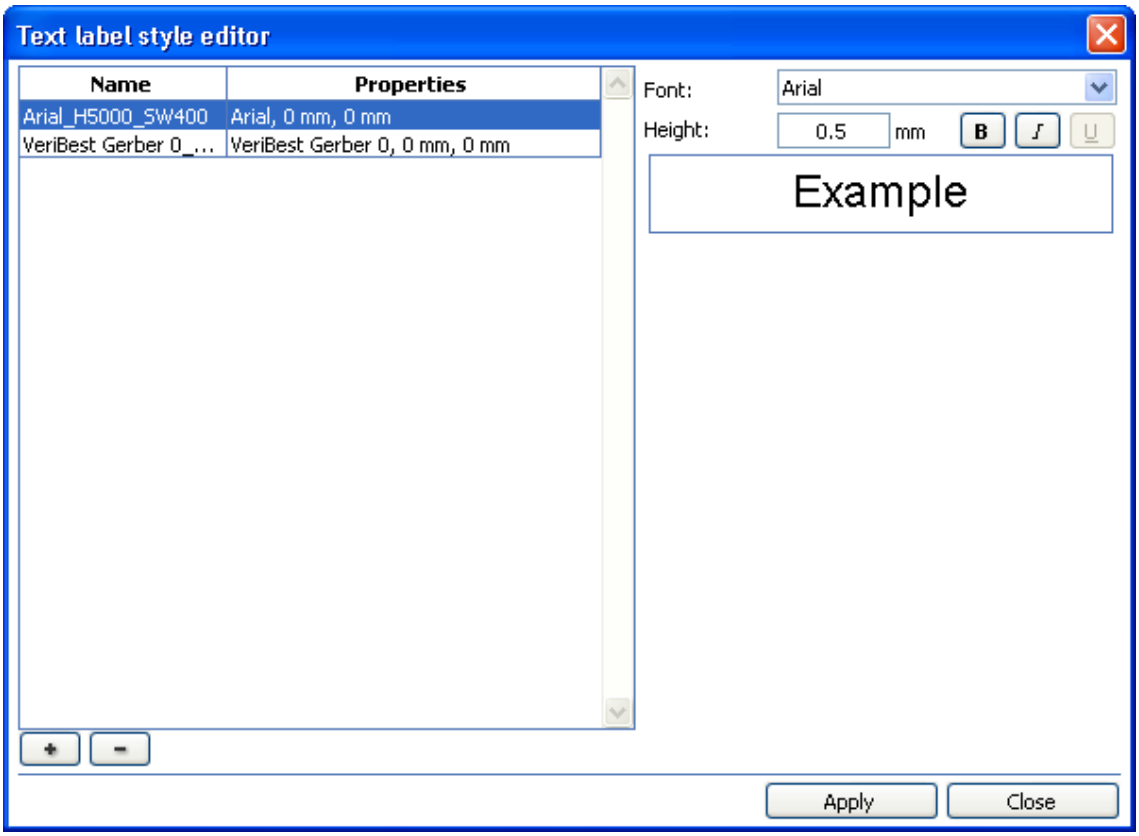
To activate your changes, press **Enter**.
When a text label object is selected, the **Properties panel** shows the parameters of the text label. When multiple text label objects are selected, changes are applied to all of them.

When multiple text labels are selected, the pivot position is not editable. Identical values of text object parameters are shown; if the values are different, the corresponding controls are blank.

Launch the **Style editor** to create a custom text style.



Text Label Style Editor



For each style, you can specify:

· Font (raster fonts are supported)

Character thickness

Character height

Select a style to automatically fill in the style options, which you can customize.

To activate your changes, click **Apply**. A style can be deleted if it is not in use by any text label.

Editing Text Labels

Whenever you change a text label parameter, the **Apply** button becomes active.

When a new style is added, the default parameters are the parameters of the style that is selected in the list.

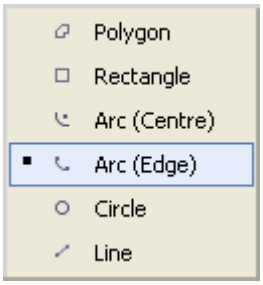
You cannot delete the default style or a style that is in use by a font.

The default style is customizable. To select a font for any style, use the corresponding dialog box.

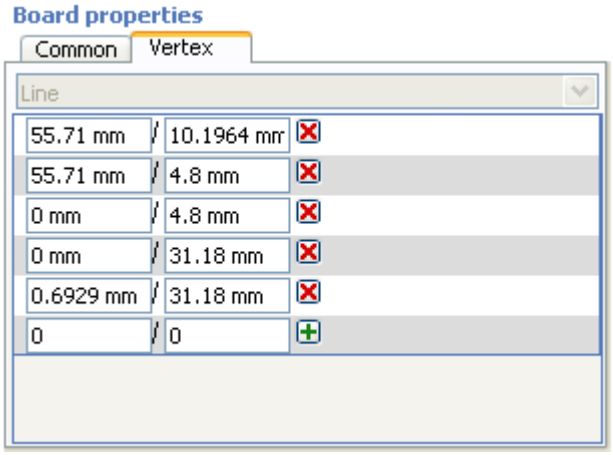
 Fonts are not duplicated. The program uses the first font with the specified name that it can find.

Creating PCB Outlines and Cutouts

To create the outline of the PCB, click the **Create keepout** button on the toolbar. This creates an outline with the currently selected shape. To change the shape, right-click the **Create keepout** button and select the shape you need in the context menu.



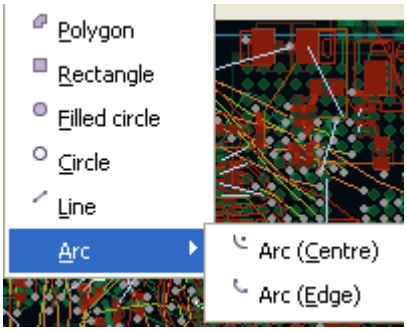
After the primitive is drawn, set the line width on the **General** tab. You can adjust the outline shape in the PCB outline editor, on the **Vertex** tab, by editing vertex coordinates.



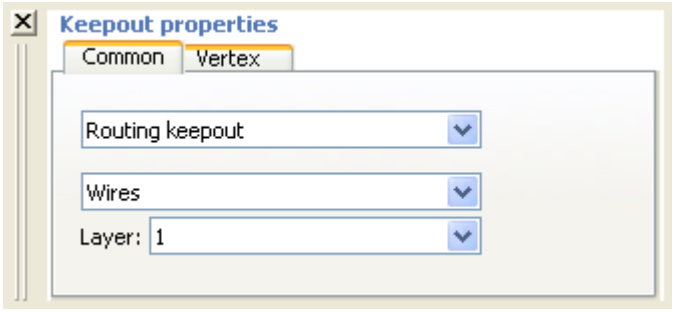
Cutouts are created likewise, but you need to set the **Role** to **Board Cutout**.

Creating Keepouts

To create a keepout, click **Insert** ► **Keepout** in the main menu and select the primitive shape you want from the drop-down menu.



After the keepout is drawn, configure its properties:



Next, set the keepout role: **Routing Keepout** or **Placement Keepout**.

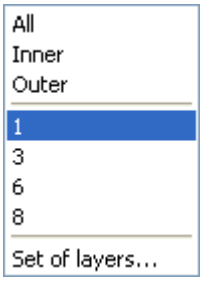
For a **placement keepout** specify the restricted side (or sides):

Top
Bottom
Both

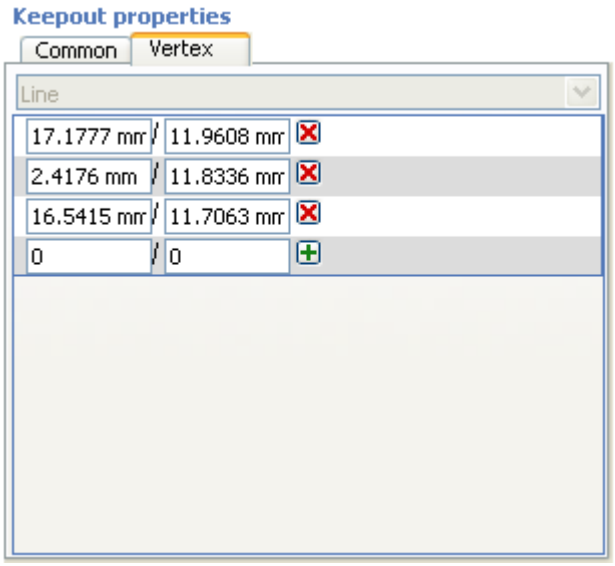
For a **routing keepout** specify what it affects:

Wires
Vias
Both wires and vias

After that, select one or more layers that the keepout applies to:

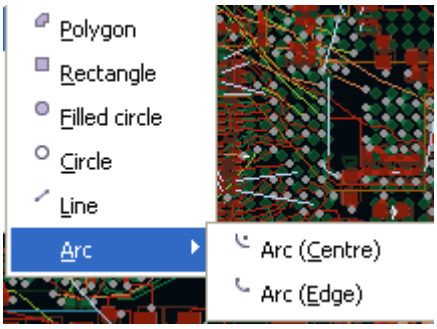


You can change shape of the keepout in the **topology editor** or in the keepout's **Properties panel** on the **Vertex** tab by changing vertex coordinates.


Keepout properties
Common
Vertex
Line
17.1777 mm
11.9608 mm
2.4176 mm
11.8336 mm
16.5415 mm
11.7063 mm
0
0

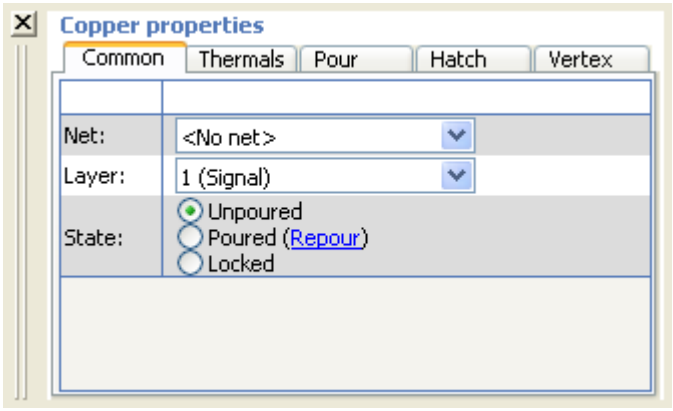
Working with Copper Areas

To create a copper area, click **Insert** ► **Copper** in the main menu and select the primitive you want in the drop-down menu:



After the copper area is drawn, configure its properties.

Common Copper Area Properties

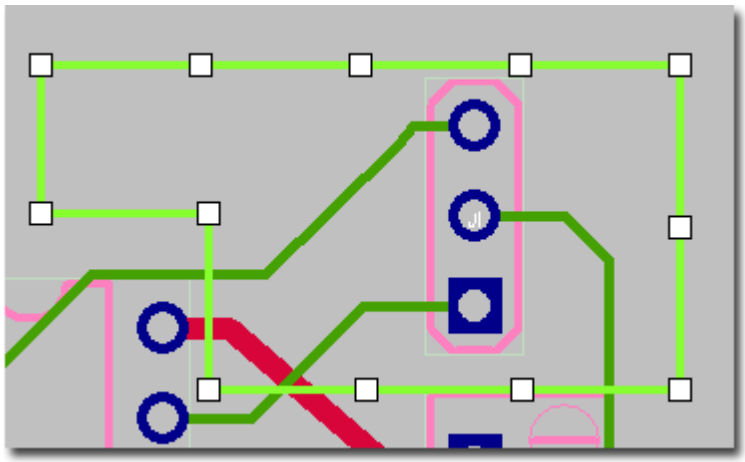


Net—Select the net that the copper area must be connected to. When the area is poured, the objects connected to the same net are not cut out of the area.

Layer—Only routable layers are allowed.

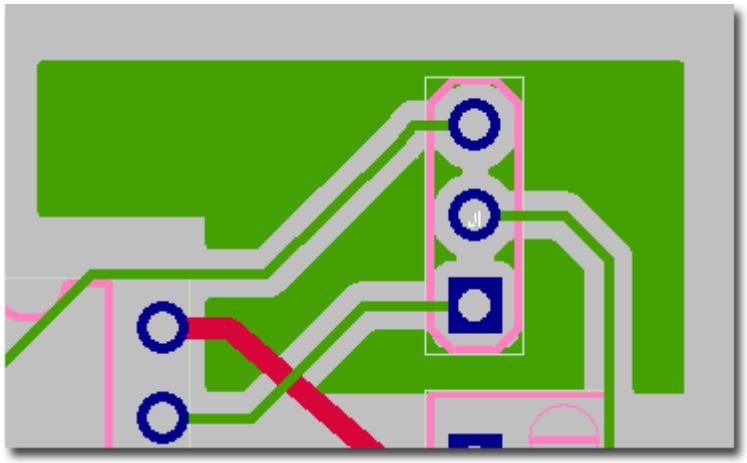
State—A copper area can be in one of three states:

- **Unpoured**—A copper area without pour is ignored by automatic tools and not checked during DRC. The outline of such an area is editable.

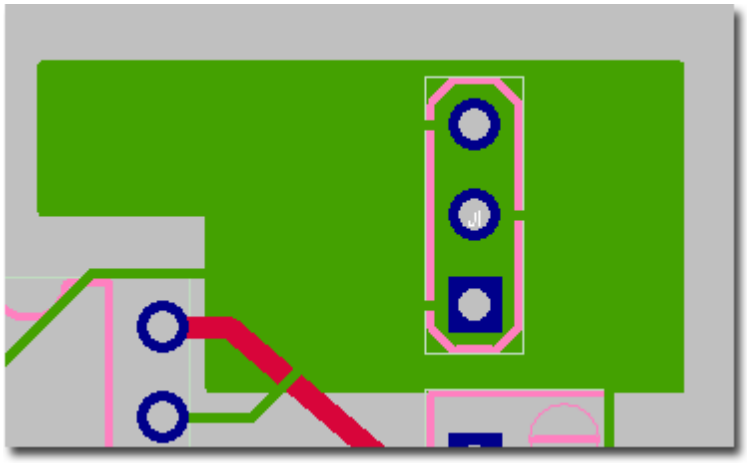


- **Poured**—If a copper area overlaps an area with components (pads, wires, etc.) in it, then the overlapping portions of the copper area are subtracted, while maintaining the required

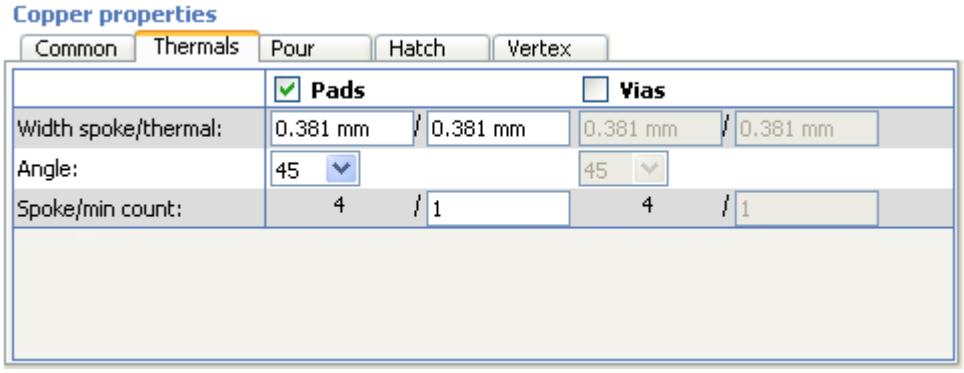
clearance. This may split the copper area into multiple unconnected parts. You can select and delete the resulting islands after pouring the area.



- **Locked**—If the previous state of a copper area was **Poured**, then locking it makes the following commands unavailable for the area: **Repour All**, **Repour Seleted**, **Unpour**. If the previous state of a copper area was **Unpoured**, then a poured area is created with the outline that you have set. Thermals are not created, and objects connected to other nets are not cut out.



Thermals

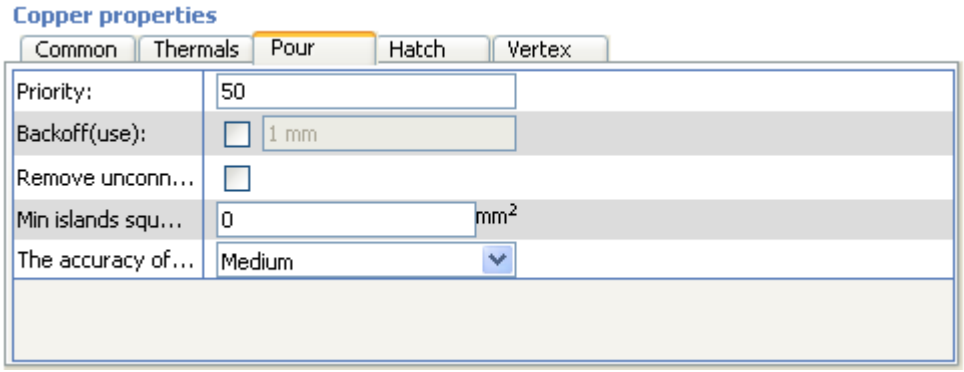


You can specify the method for connecting pads and vias to the nets they belong to: through direct connection or through a thermal.

TopoR tries to make 4 spokes in a thermal. However, if the number of spokes is found to be less than specified in the rules (for a pad or a via),a warning is displayed.

You can configure the angle of the thermal, the width of the spokes and the width of the thermal.

Pour



Priority—Controls how overlapping copper areas are subtracted at the same layer. Higher-priority areas are cut out of lower-priority areas.

Backoff (use)—Specify the clearance here, otherwise the clearance set in the Rules panel of the design properties editor is used between the copper area and objects in other nets by default. If the nets of the copper area and the subtracted object have different clearance settings, the maximum setting is used.

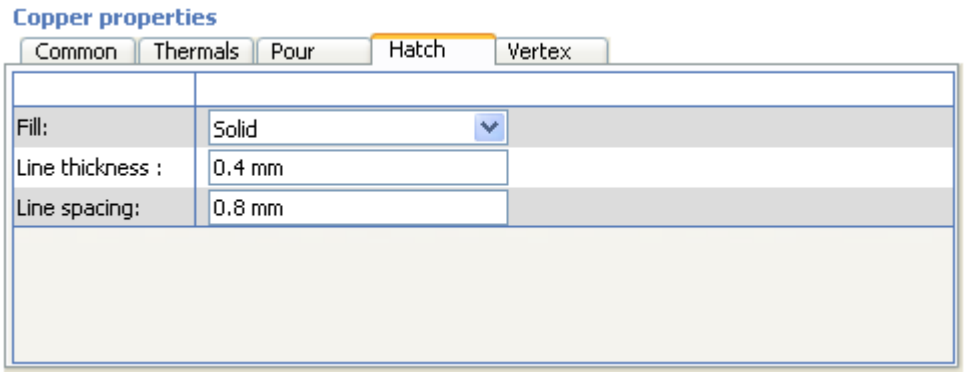
Remove unconnected islands—Enabling this option deletes any islands that do not contain pads, vias or wires from the net that the copper belongs to.

Minimum islands square—If the area of an island is less than specified, the island is removed. When you modify the properties of the copper area that affect the pour, the area is repoured.

The accuracy of approximation—Select one of three options: low, medium or high. This setting is considered when round and curved objects are cut out.

If there are changes to properties that affect the pouring of a poured copper area, that copper area is repoured.

Hatch

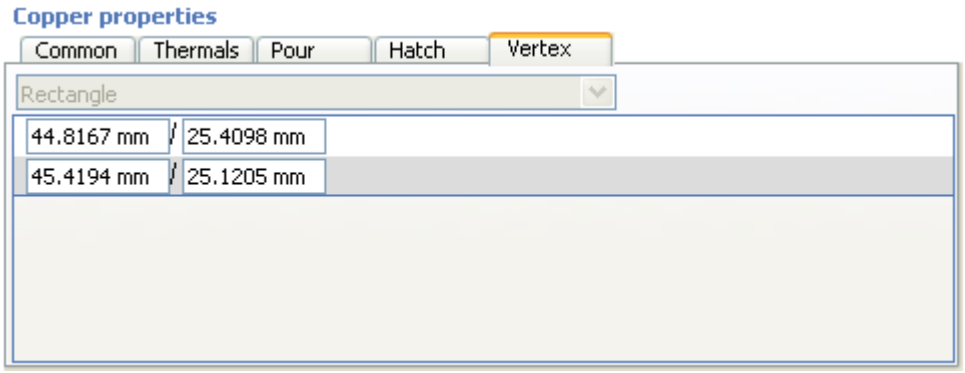


You can select one of three options: solid fill, grid hatching, and cross hatching

Line thickness—How thick the filling lines are.

Line spacing—Spacing between the edges of lines in non-solid fills.

Vertex



You can change the shape of a copper area in the **topology editor** or in the **Properties panel** on the **Vertex** tab by changing vertex coordinates.

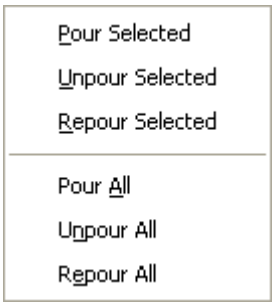
Creating Cutouts in Copper

This functionality is available only if one unpoured and unfixed copper area is selected. To create a cutout, select the **Create copper cutout** button on the toolbar. To change the current shape of the cutout, right-click the button and use the drop-down menu. Cutouts in copper can overlap or reach outside their copper area. To delete a cutout, select it and press **Delete**.

Copper Area Specifics

The line thickness for a copper area should be no less than the width of the wires connected to the it. After you have poured a copper area, it is recommended that you switch to **FreeStyle editing mode** and run the wire path optimization procedure.

The following commands are available in the **Tools** ► **Coppers** areas menu:



The **Pour All** command pours all unpoured copper areas.

The **Unpour All** command removes the pour only from unlocked copper areas.

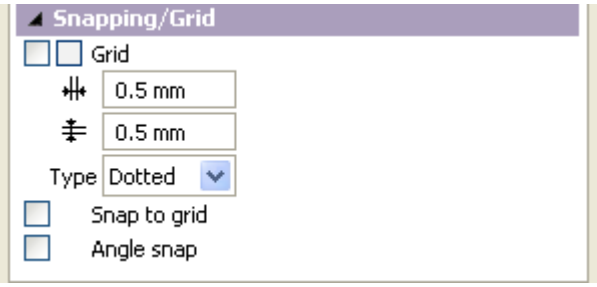
The **Repour All** command repours all unlocked copper areas.

You can modify the properties of a locked copper area. If you modify properties that affect its geometry, then the copper area is repoured.

If any copper areas are selected, the following operations are also available: **Pour Selected**, **Unpour Selected**, **Repour Selected**.

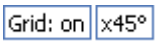
Editing Primitives on Mechanical Layers

When creating and editing lines and polygons, you can use the **Snap to angle** option, which is enabled on the **Settings** tab of the display control panel.



When snapping is enabled, the segment incident to the vertex you are moving rotates in 45-degree steps. To enable and disable snapping, press the **O** key. When you move a vertex shared by two segments, only one of the segments uses snapping. To select which of the segments snaps, press the **P** key.

When you create or edit shapes, the **Align to grid** option is also available. To turn this option on or off, press the **G** key (by default). Information about the options you are using is displayed in the right corner of the status bar. For example, if both of the snapping options are enabled, the right side of the status bar looks like this:



To select a primitive, click its outline or, if the primitive is poured, anywhere inside the outline. You can use box selection with primitives. Only those primitives are selected that are completely inside the selection bounding box. To modify the outline, reposition its markers. To move a primitive, click its outline (not a marker) and drag it.

Line

Each time you click, a vertex is added to the newly created line. To complete the line, right-click or press **Esc**.

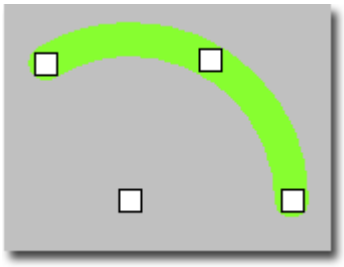
Arc

You can create an arc shape in two ways:

From center and two arc points—The first click sets the arc center, the second click sets the starting point and the radius, and the third click sets the end point (on the circle with the radius you have set)

From three arc points—The first click sets the arc's starting point, the second click sets the end point, and the third click sets the center (located between the starting point and end point defined after the second click)

The arc is drawn counterclockwise from the starting point to the end point.
Selecting an arc shows four markers:



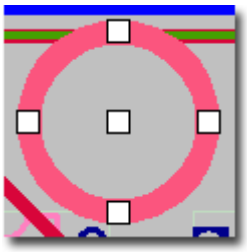
When the center marker of an arc is moved, the starting point and end point remain locked, the location of the center (and thus the arc radius) changes, the arc curvature does not change. When the middle marker of an arc is moved, the starting point and end point also remain locked, the location of the center changes, and you can change the arc curvature.
You can move the end markers of an arc in two ways:

If **Ctrl** is held down—The locations of the center and the opposite end marker are locked. The selected marker can move along the arc with the locked radius. If the **Snap to angle** option is enabled, the movement occurs in 45-degree increments.

If **Ctrl** is not held down—The location of the center is locked, and the marker movement changes the arc radius and the location of the selected marker on the arc. The opposite marker moves in a straight line towards or away from the arc center.

Circle

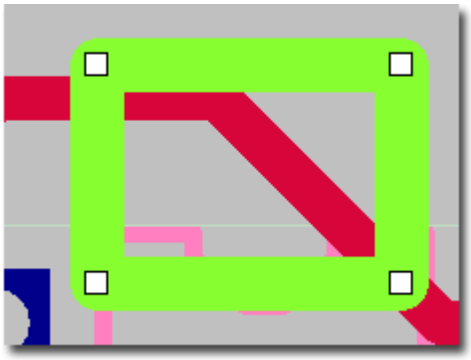
When you create a circle primitive, the first click sets the center, and the second click sets a point on the circumference. Five markers are available on a selected circle:



To move a circle, drag the center marker or the primitive outline. To resize a circle, drag any of the markers on the circumference. All of the above also applies to filled circles.

Rectangle

When you create a rectangle primitive, the first click sets a vertex, and the second click sets the diagonally opposite vertex. Four markers are available on a selected rectangle:



To resize a rectangle, drag any of the corner markers.

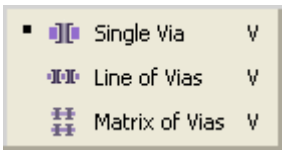
Polygon

Each time you click, a vertex is added to the polygon you are creating.
To complete the polygon, right-click or press **Esc**. If the polygon has fewer than three vertices, it is removed. If the outline of a polygon intersects the outlines of other polygons that are at the same layer and have the same role, the polygons are not merged. If the outline of a polygon self-intersects, a warning message is shown in the output bar.

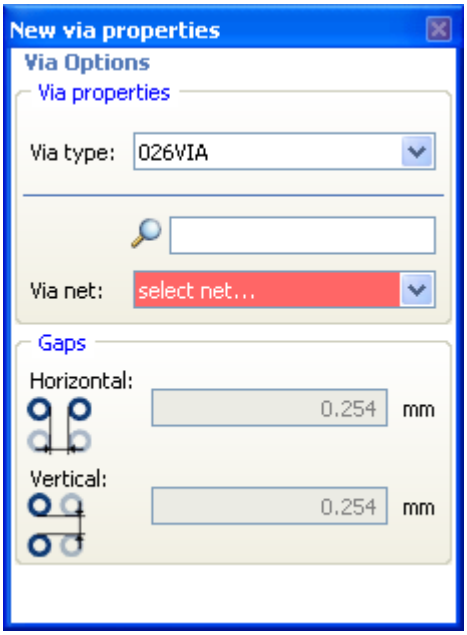
Vias

A via is added automatically when you change the layer for a wire you are routing. In other situations, add vias manually by clicking the **Create Via** button in the toolbar or **Insert ► Via** in the main menu. The latter method opens a menu with the following options:

Single Via (default hotkey: **V**)
Line of Vias
Matrix of Vias



When you select an option, the **Via Options** dialog box is shown.



You can specify the following:

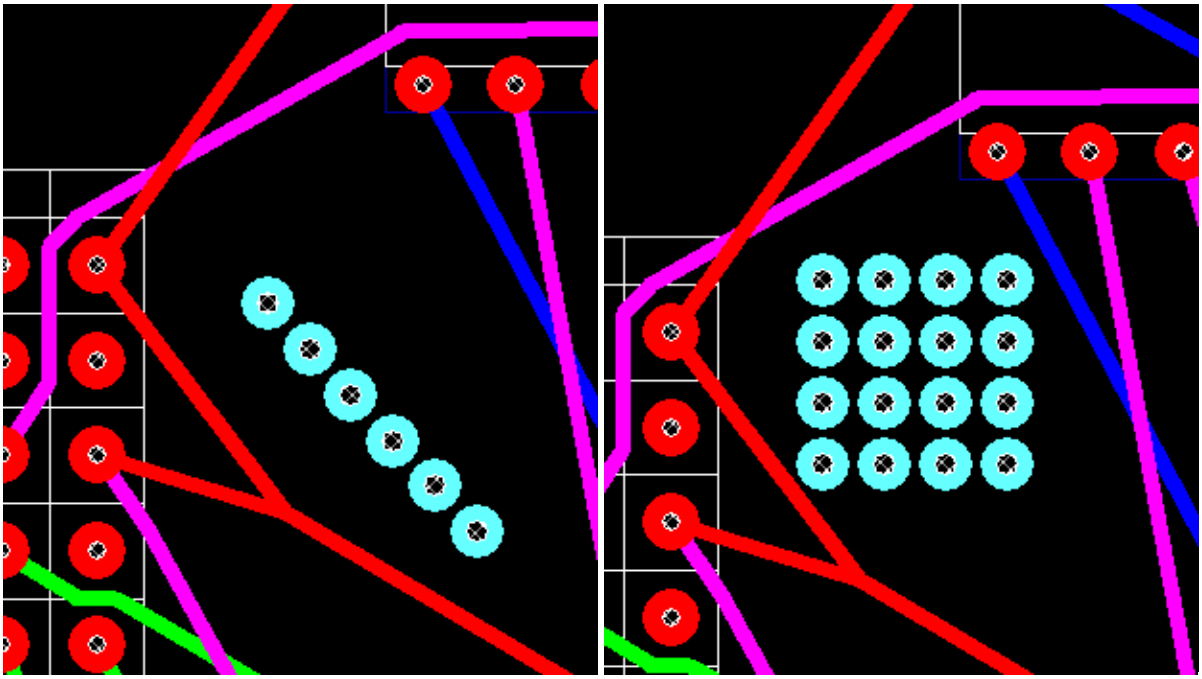
Via type—select from a list of via types)
Via net—select from a list of nets on the board)
Gaps—spacing between vias in the group

To quickly find the necessary net, use the filter box to the right of the magnifying glass icon. As you type, the list is narrowed down to names that contain the specified substring. Use the **V** shortcut key to add a single via (or line of vias, or matrix of vias, depending on the active option in the toolbar).

Both gap options are available when you are adding a matrix of vias. When you are adding a line of vias, only the horizontal gap value is available. For single vias, neither option is available.

To add a group of vias, click a location in the work area and move the pointer in the direction you need. The point where you clicked becomes the center of the first via (in a line of vias) or a corner of the bounding box (in a matrix of vias). Release the mouse button and move the pointer to add vias.

To confirm the addition, click again. A right-click in the middle of the operation cancels it.



Changing the Via Type

To change the type of a via, use the **Via properties** panel. The available types come from the list configured in the **Vias** section of the **design properties editor**.

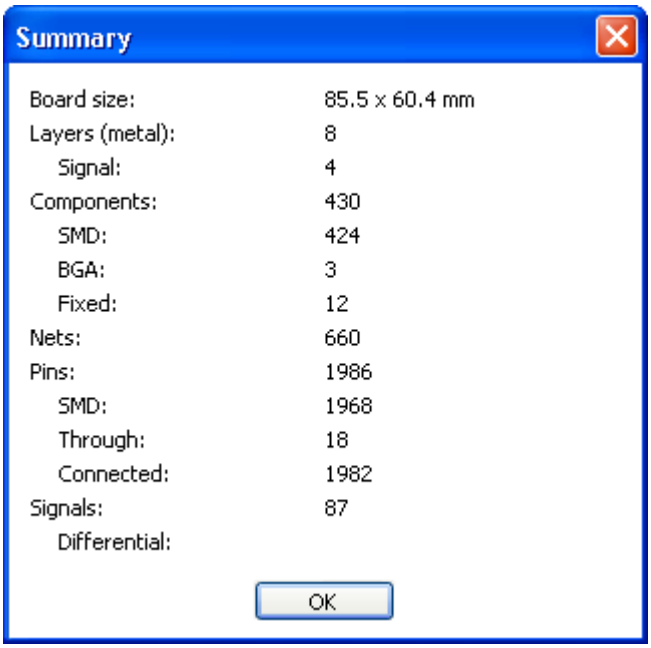
Via properties

Net:	XSIG130916
Type:	VIA0.2Rnd0.5
Layer span:	Through
Fixed:	

Via editor

Viewing Summary Information

To view the primary characteristics of the current design, click **Design ► Summary** in the main menu.



To go back to the editor, click **OK**.

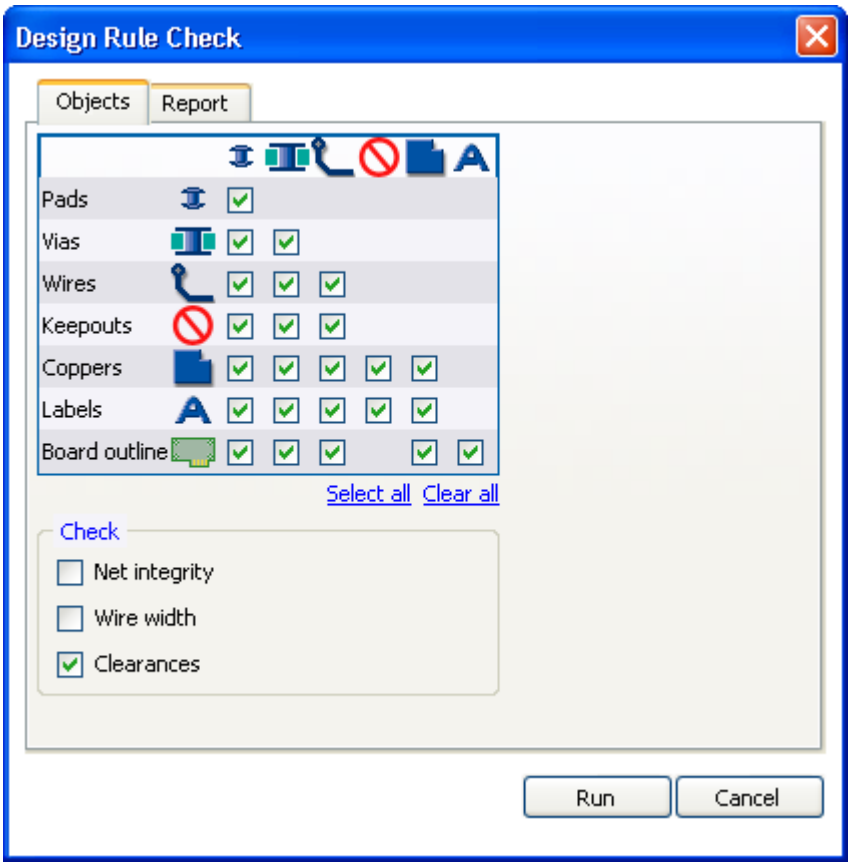
Design Rule Checking

To launch a design rule check (DRC), click **DRC** 

in the toolbar. The DRC procedure checks the board topology and creates a text file with diagnostic information.

DRC Settings

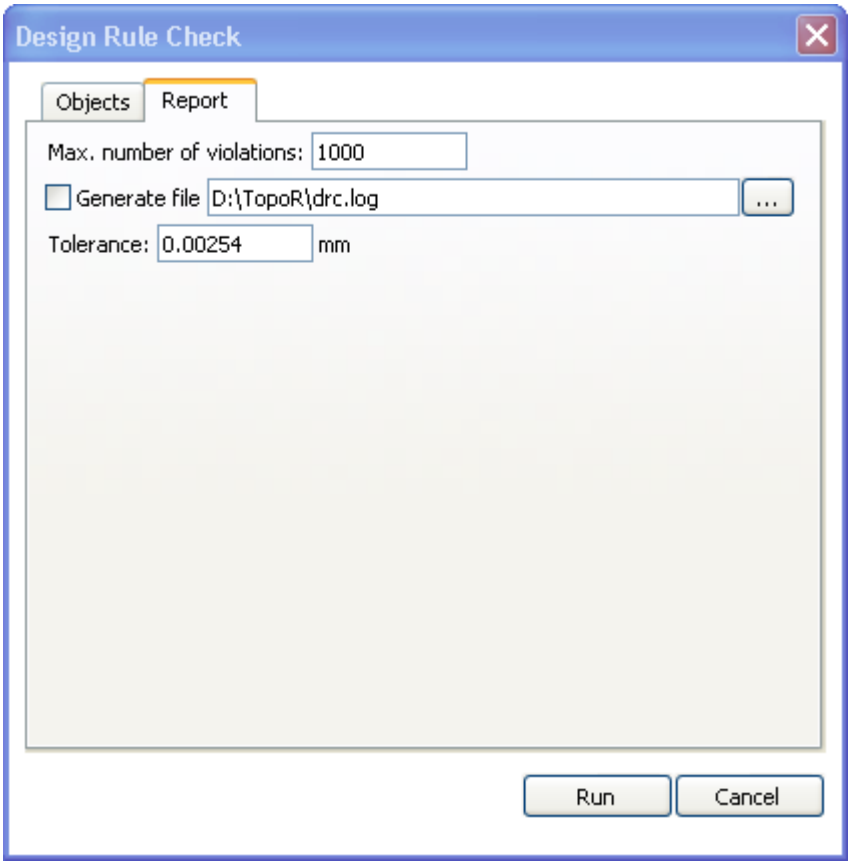
Clicking the triangle next to the **DRC** button opens the DRC menu, where you have two options: **Remove DRC Messages** and **DRC Settings**. Another way to open DRC settings is to click **Tools** ► **DRC...** in the main menu.



DRC lets you check:

Net integrity
Wire width
Clearance between topology elements (pads, wires, vias, routing keepouts, coppers, text labels)

On the **Report** tab of the settings dialog box, configure the DRC reporting parameters.



By default, the name of the report file is **`drc.log`**, and it is created in the project folder. To use a different name, select the corresponding check box and specify the new file name (and the path to the directory where it will be created).

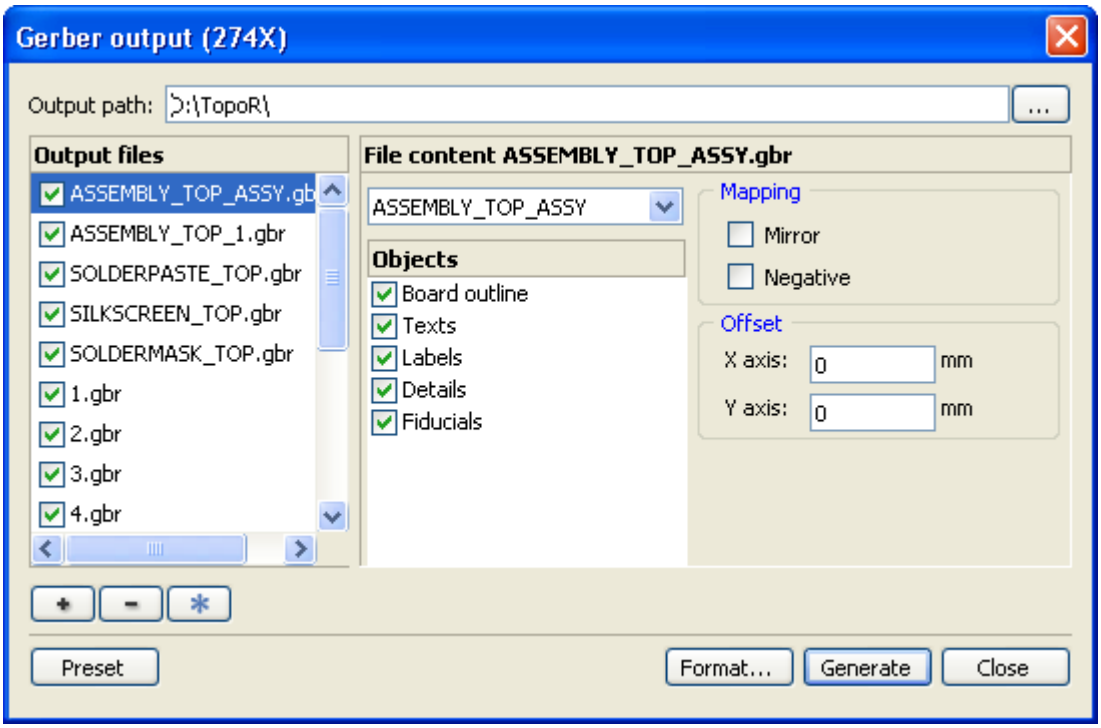
To limit the report file size, you can set the maximum number of errors it can contain (the default number is 1000). When the specified number of errors is found, the check stops.

Writing Results

The final step in working on a project in **TopoR** is to write output files.

Writing Gerber Files

Click **File** ► **Production Files** ► **Gerber** to open the **Gerber output (274X)** dialog box:



This dialog box displays two lists. The left list contains the names of files to be exported. The right list contains the layers to be included in the files.

You can edit the file name by selecting the file and clicking it again.

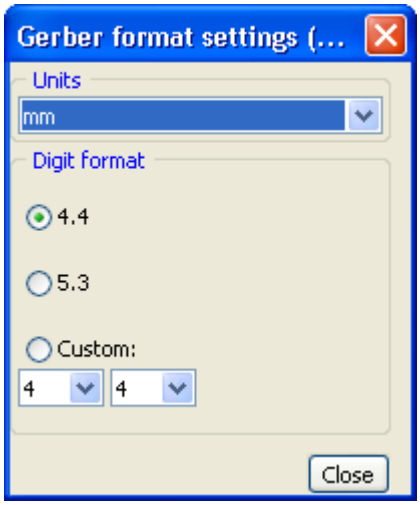
You can also mirror the output image or specify coordinate offsets.

To create a new file or delete an existing one, use the **Create File** [+] and **Delete File** [-] buttons.

The **Mark** [*] button marks all files or clears the marks from them.

Use the **Preset** button to save settings to a file or restore the default settings.

Click the **Format** button to open the **Gerber format settings** dialog box:

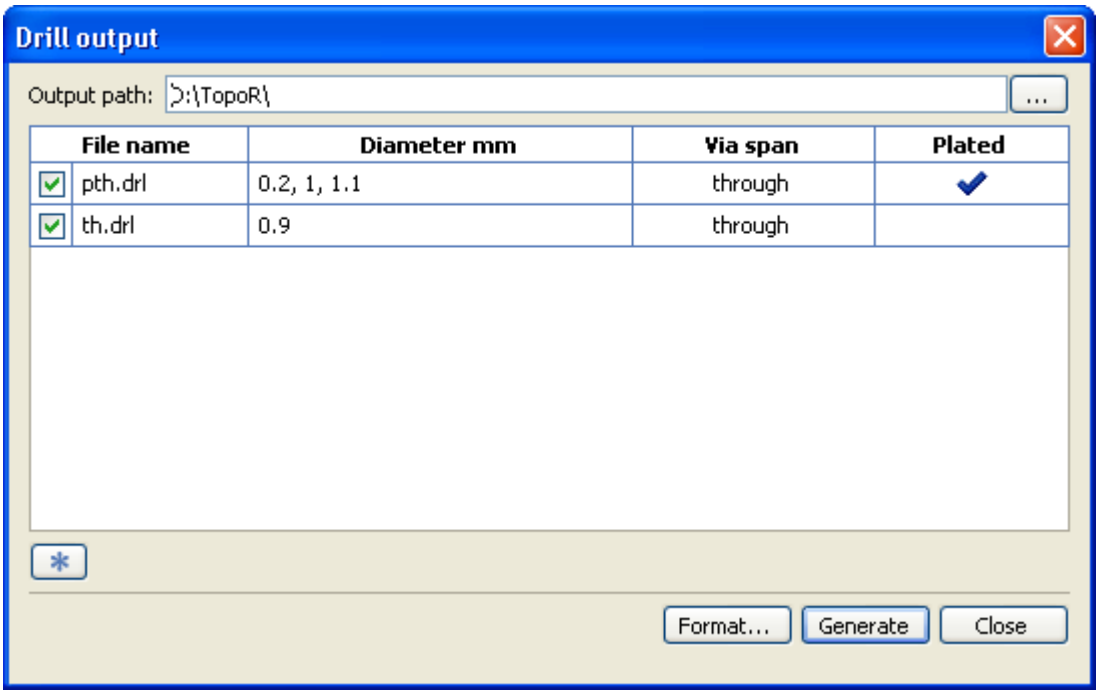


This dialog box lets you configure some parameters of Gerber output. Set the units to use (mm or mil), and specify the format for numbers (such as “4.4”, meaning four integer digits and four decimal places, or “5.3”, or a custom format).

Click **Generate** to export the files.

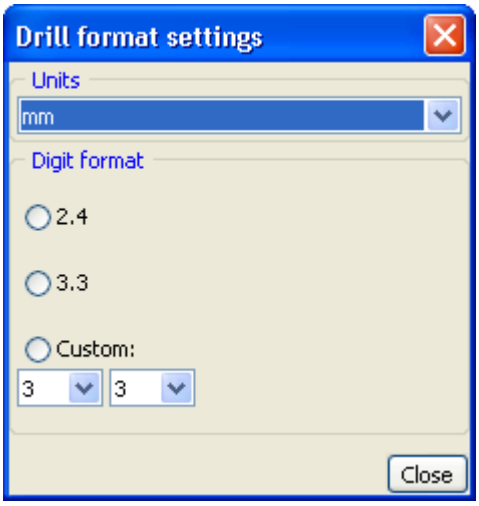
Writing Drill Files

Click **File** ► **Production Files** ► **Drill** to open the **Drill output** dialog box:



This dialog box displays two lists. The left list contains the (editable) names of files to be exported. The right list contains the layers to be included in the files.

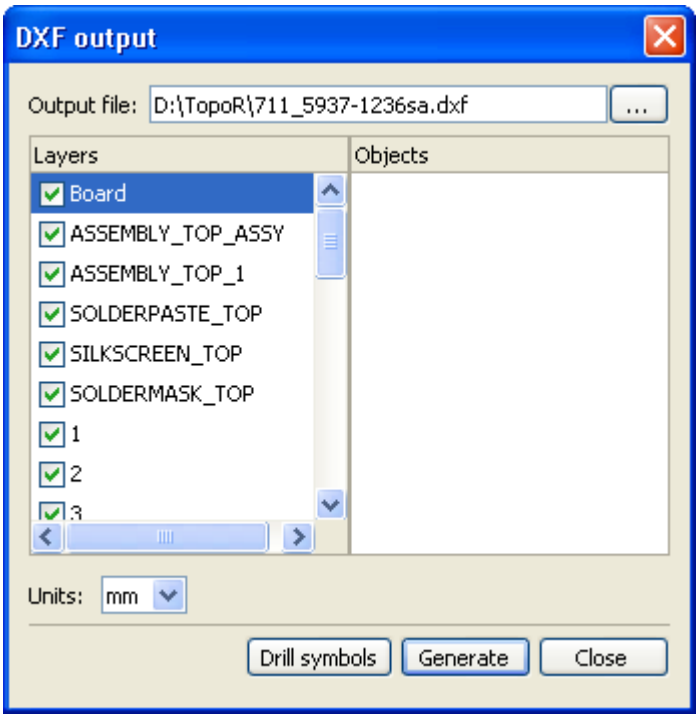
Similarly to Gerber file output settings, clicking the **Format** button lets you set up the coordinate format for the Drill file.



Specify the names and location of the files, and click **Generate**.

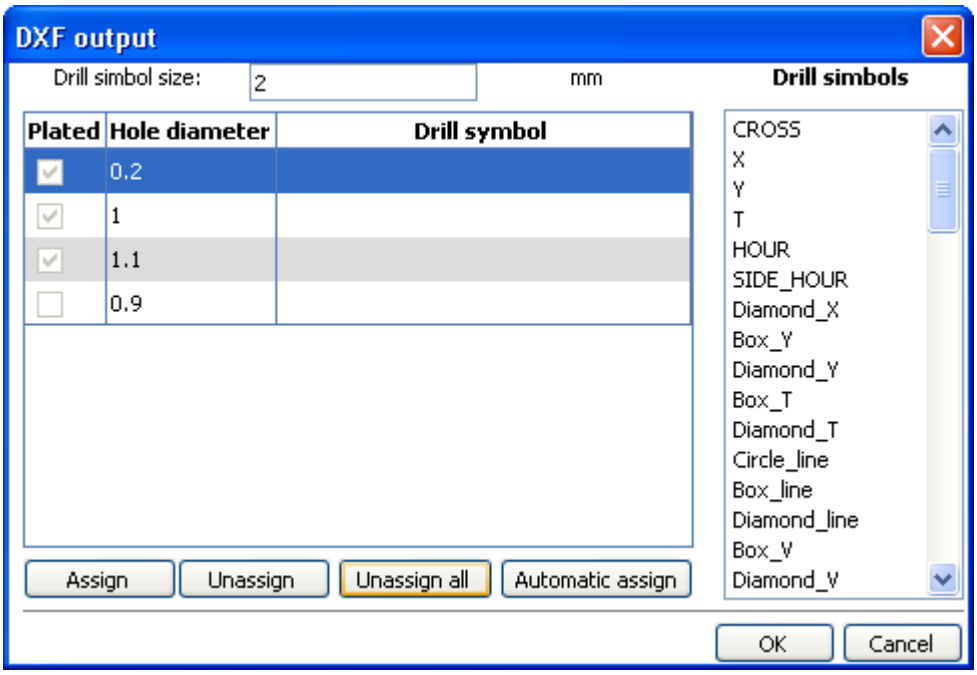
Writing DXF Files

Click **File ► Production Files ► DXF** to open the **DXF output** dialog box:



Specify the name and location of the file, and set the size of drill symbols.

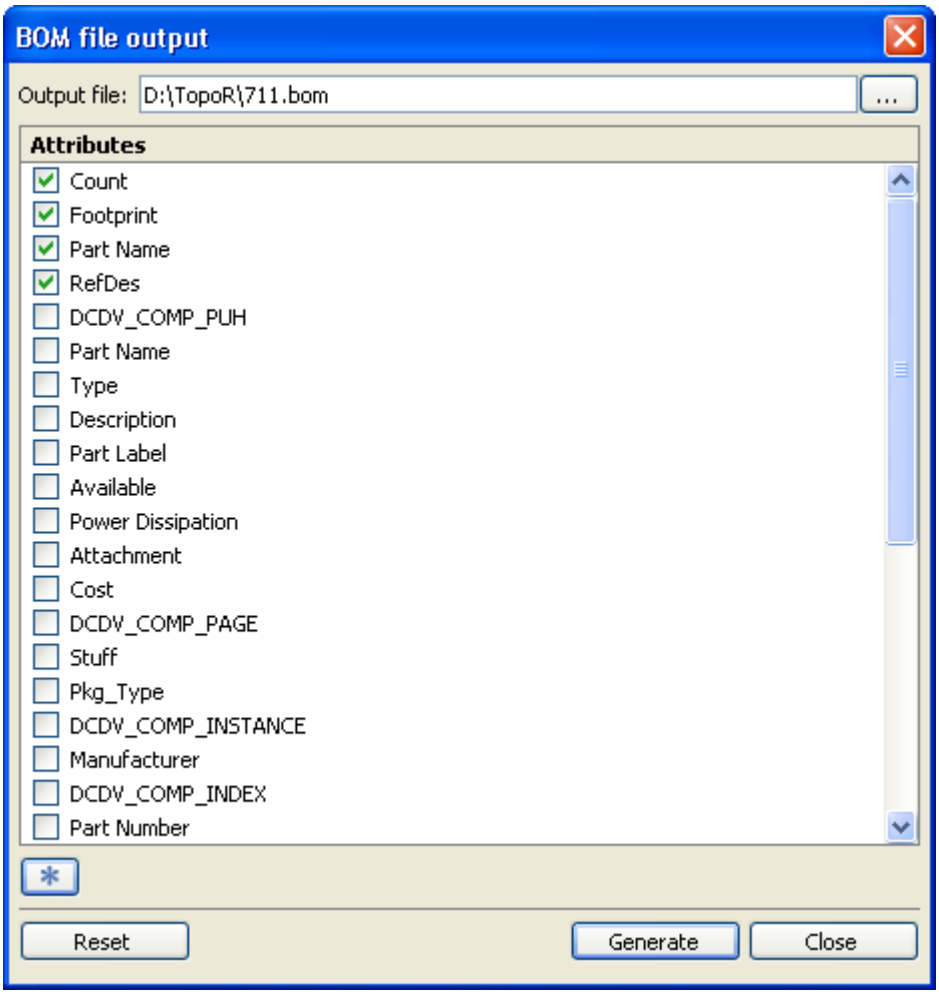
Click **Drill symbols** to open the following dialog box:

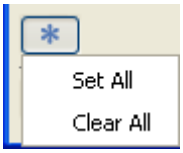


You can specify different symbols for different diameters. The symbol mapping table is exported automatically. To export the file, click the **Generate** button.

Writing BOM Files

Click **File ► Report Files ► BOM** to open the **BOM file output** dialog box:





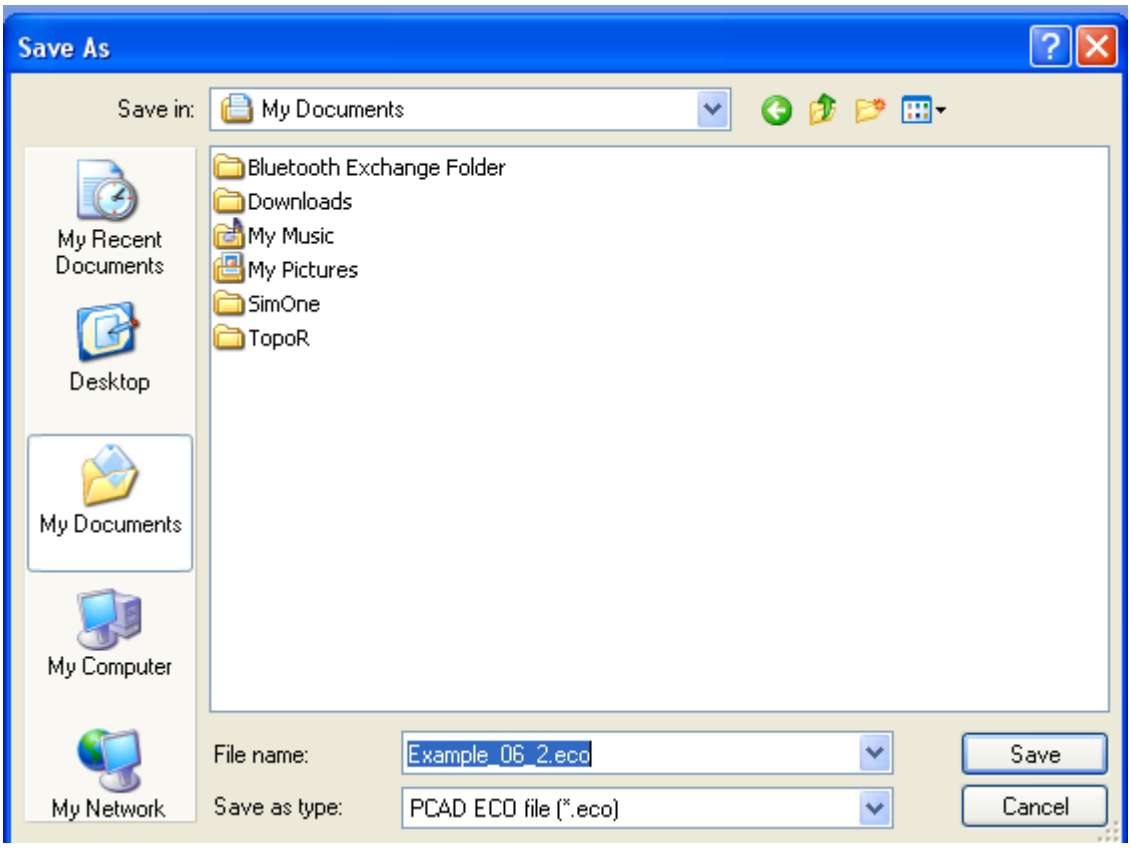
In this dialog box, specify the name and location of the file, and the following attributes:

Count
Footprint
Part Name
RefDes

The resulting file can be used as the basis of a textual design documentation set (prepared in the Kompas-3D CAD system, for example).

Writing ECO Files

Click **File ► Report Files ►ECO** to export your (automatic or manual) reassignments of functionally equivalent pads to a **.eco** file.

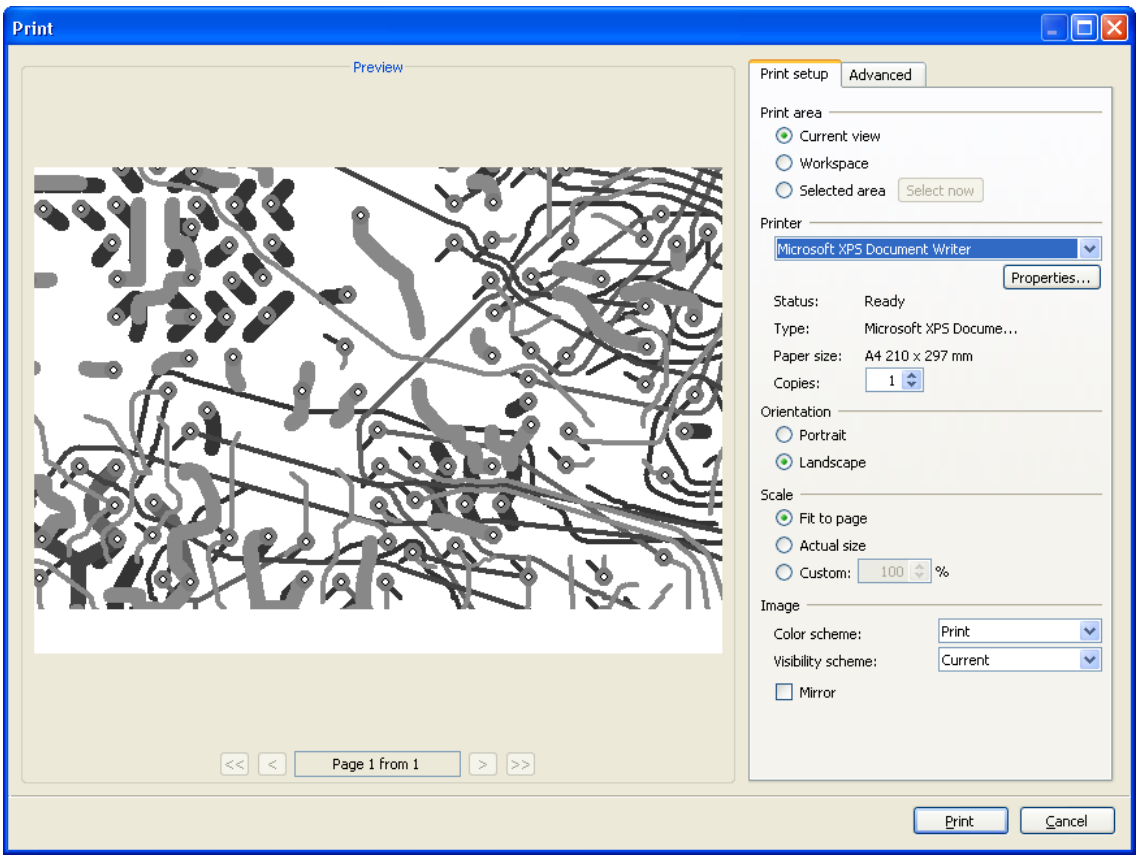


Sample **.eco** file:

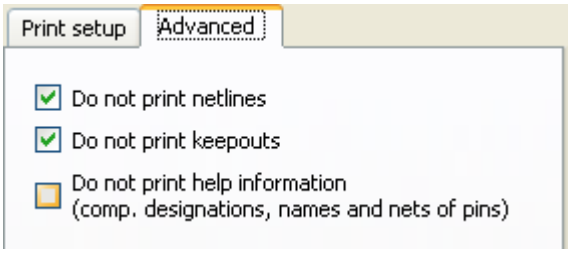
```
; TopoR Version 6.0.4.0 [D:\Topor\711.fsx]--Tue Nov 12 16:11:19 2013
; --------
PinSwap "D1-73" "D1-74"
```

Printing Routing Results

The **Print** dialog box (**File ► Print** or the hotkey **Ctrl+P**) is used for printing the design. It contains a preview panel and another panel with printing settings arranged in two tabs: **Print setup** (for general settings) and **Advanced** (for excluding specific optional elements from printing).



Everything that is shown on the screen goes to print. To enable or disable the display of objects, use the display control panel (link). Therefore, you can achieve the same effect without using advanced print settings. The advanced settings are provided so that you do not have to enable or disable specific objects (such as unrouted connections) every time you print.



Print settings let you:

Specify the displayed area: current view, workspace (rectangular area containing all visible displayed objects in the project) or selected area (box selection). Clicking the **Select now** button takes you back to the workspace (previous view) so that you can select the necessary portion (use box selection, zoom in and out, etc.).

Configure the printer (specify the type, paper size, and number of copies).

Set the page orientation (portrait or landscape).

Set the scale:

- Fit the image to the page (enlarge or shrink it to match the paper size)
- Actual size (a scale of 1:1)
- Custom (a user-defined percentage; if the image does not fit in the page, it is printed over multiple pages)

Specify the image properties:

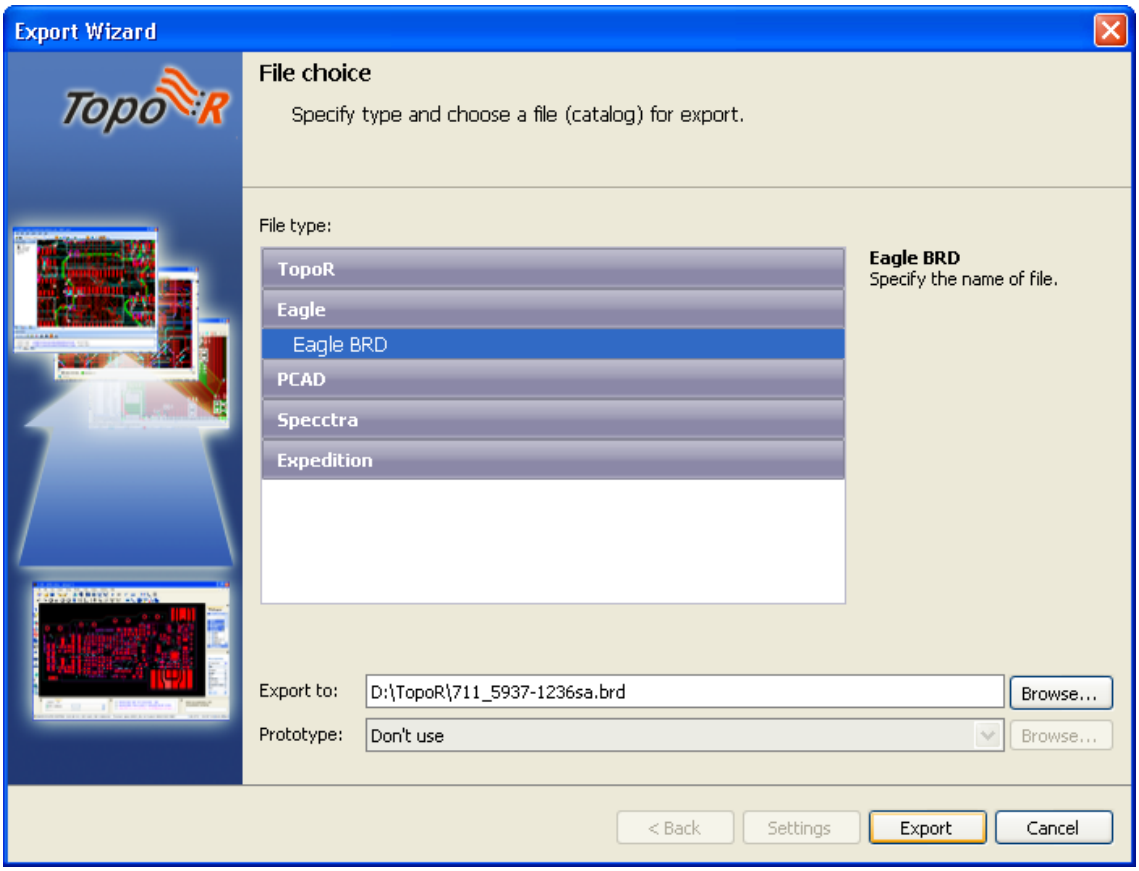
- The color scheme (print or grayscale)
- The display scheme
- Whether to mirror the image.

Exporting Routing Results

Export to the following formats is available:

TopoR PCB (`.fst`)
PCAD ASCII PCB (`.pcb`)
Specctra/Electra session file (`.ses`)
Expedition (`.hkp`)
Eagle (`.brd`)

To export results, use **File ► Export** in the main menu or the **Export document** button in the toolbar. Either action opens the **Export Wizard**:



In the **Export Wizard**, specify the following:

File format
Name of the file to export
Name of the prototype file (where applicable)

Normally, **TopoR** exports only the data it uses. This, in particular, allows you to import data in one format (such as `.dsn`), and export to another (such as `.pcb` or `.hkp`). However, since the formats differ significantly, data loss is inevitable.

If the same format is imported and exported, then using a prototype helps avoid data loss and also include information that is not used directly by **TopoR** but is contained in the original file.

When exporting a **.ses** (Specctra/Electra) file, make sure you specify the CAD system which is expected to open the file, because different systems use different modifications of the format, and even slight deviations can make the file unreadable:

PCAD, Dip Trace and OrCad use single quotations marks, while Proteus, KiCAD and Pulsonix use double quotations marks. Altium does not use quotation marks.

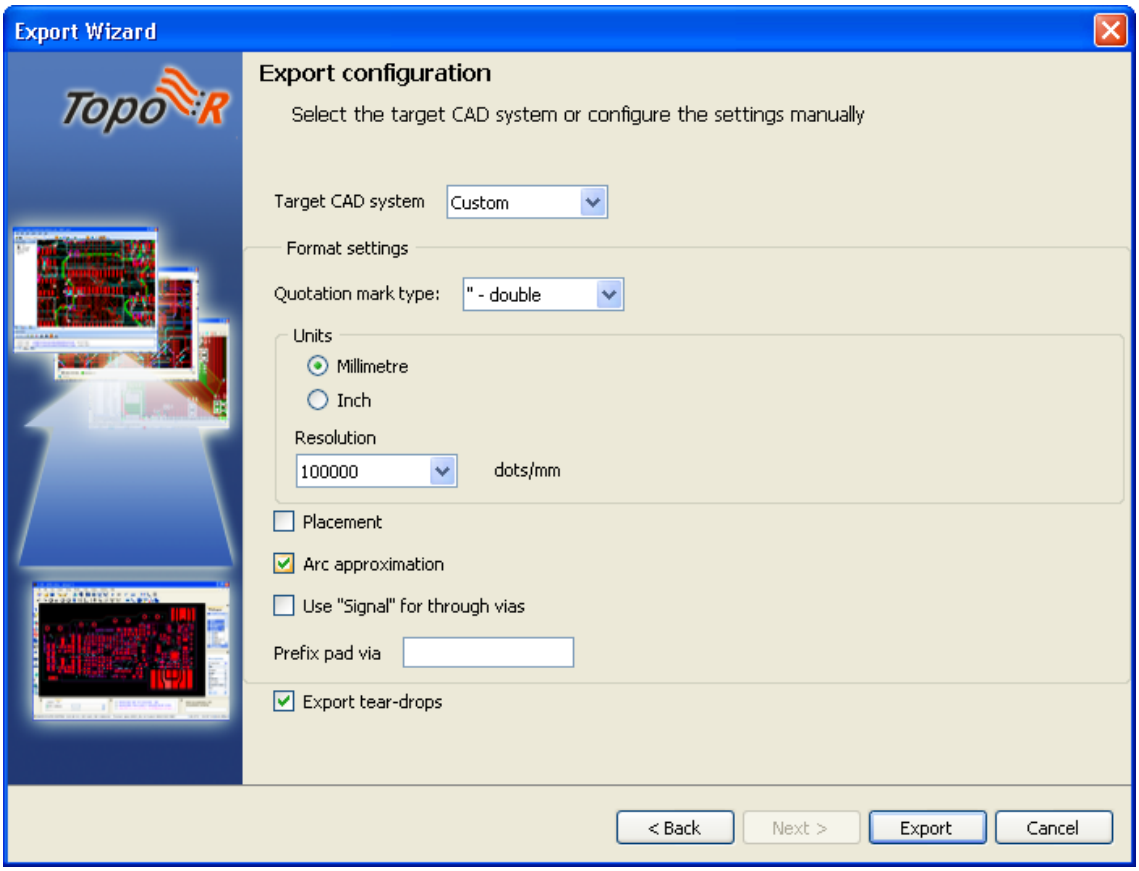
Altium, Proteus and Pulsonix do not read the section with component coordinates, so components should not be moved in **TopoR**, because these coordinates will be substituted by the old coordinates anyway.

DipTrace and OrCad create **.dsn** files and accept **.ses** files with both metric and imperial units, Altium, PCAD, Proteus and KiCAD support only metric, Pulsonix only imperial units.

Out of all the CAD systems listed in the in the menu shown below, only DipTrace can handle files where wires have arcs. While PCAD works fine with arbitrary arcs, it does not recognize them in a **.ses** file.

Altium, PCAD, DipTrace and Proteus let you specify the **signal** property for vias, meaning a common pad shape on all signal layers. KiCAD, Pulsonix and OrCad require that pads be specified on each signal layer.

PCAD requires the CCTVIA prefix for vias in a **.ses** file.



C. Advanced Features

This section is for experienced users of the software. Read on for information about **TopoR** functionality that is not covered in part B of this manual.

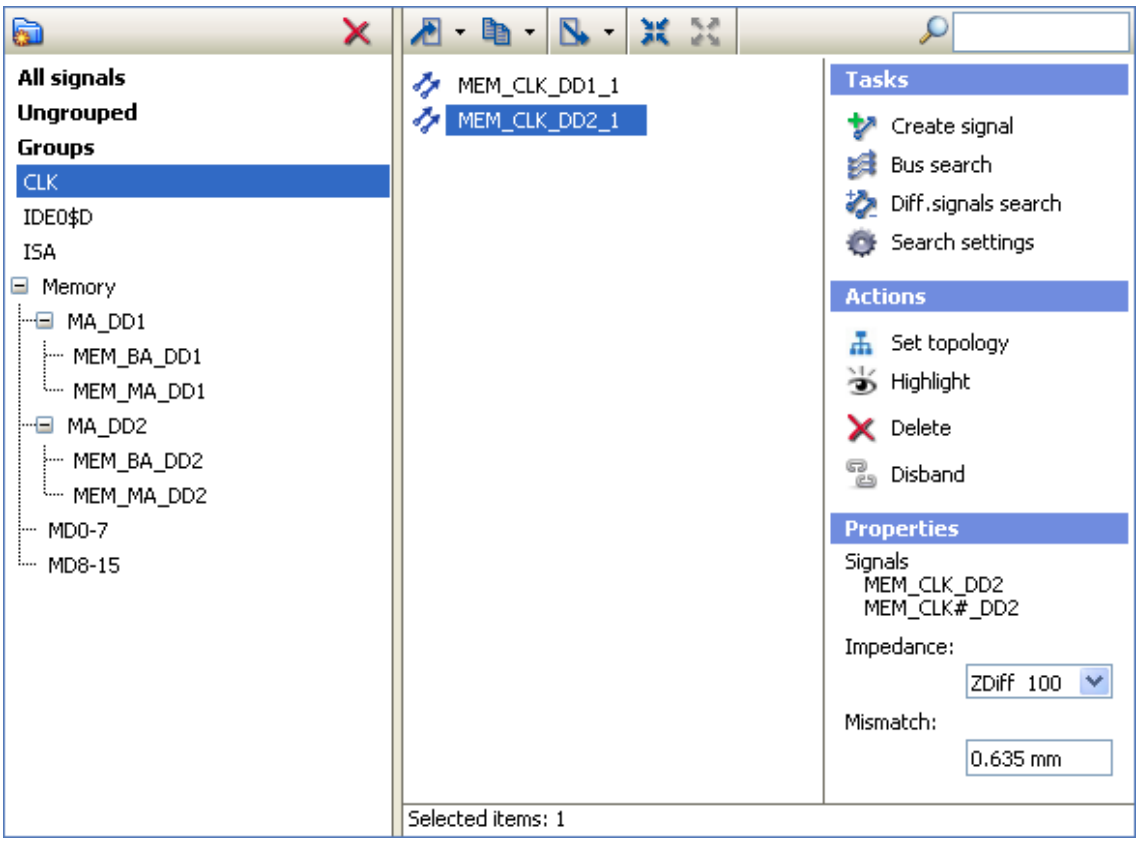
Designing High-Speed Appliances

This section describes working with the *signal* object type. This object type has been introduced to represent transfer lines for electric signals, and, compared to earlier **TopoR** versions, it provides more accurate baseline data for delay synchronization in transfer lines.

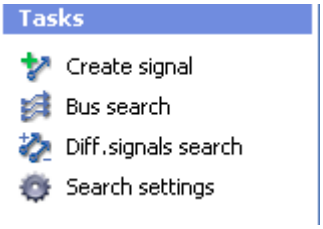
Each signal has one source and one receiver. Multiple signals can share a source. However, signals cannot share a receiver.

Introduction to Signals

Signals, differential signals (DSs) and signal groups are created and edited in the design parameter editor (**High-speed rules ► Signals**). The user interface for working with signals is the same as for object grouping. The right pane of the **Signals** panel contains three additional panels:



1. Tasks



Select one of four tasks: **Create signal**, **Bus search**, **Diff. signals search**, **Search settings**.

2. Actions

Select one of three actions: **Set topology, Highlight, Delete**.

3. Properties

View the current properties of the signal: source, receiver and impedance parameters.


Properties
Signals
MEM_CLK_DD2
MEM_CLK#_DD2
Impedance:
ZDiff 100
Mismatch:
0.635 mm

Creating a Signal

Creating a signal or multiple signals with a common source is a three-step process:

Select a source.

Select a receiver or receivers.

Configure the topology of the signal nets if necessary.

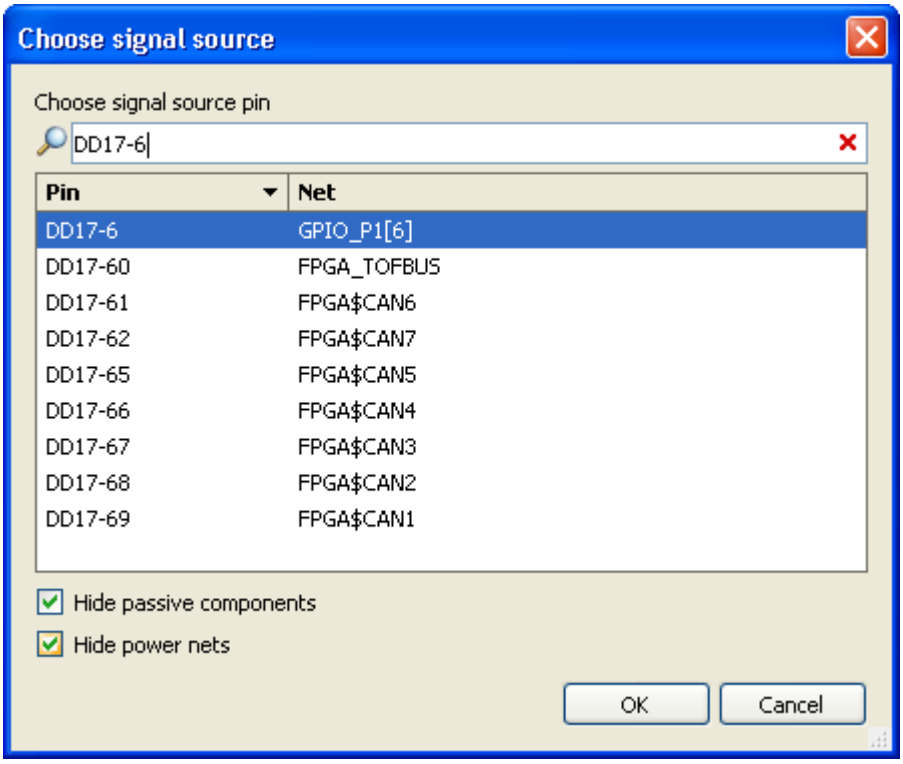
Selecting the Signal Source

To start, go to the task panel, click the **Create signal** task, specify the pad that will be the signal source in the **Choose signal source** dialog box that pops up, and click **OK**. The pad list is generated based on which options are selected:

Hide passive components—A passive component is a resistor or a two-pin gate (for example, one involved in a resistor array) with equivalent pins.

Hide power nets—The list of power nets can be modified in the signal search settings.

If a pin's net is already associated with an existing signal, then the pin is shown in a dim gray color in the list, and a note in parentheses indicates the signal that the net is associated with.

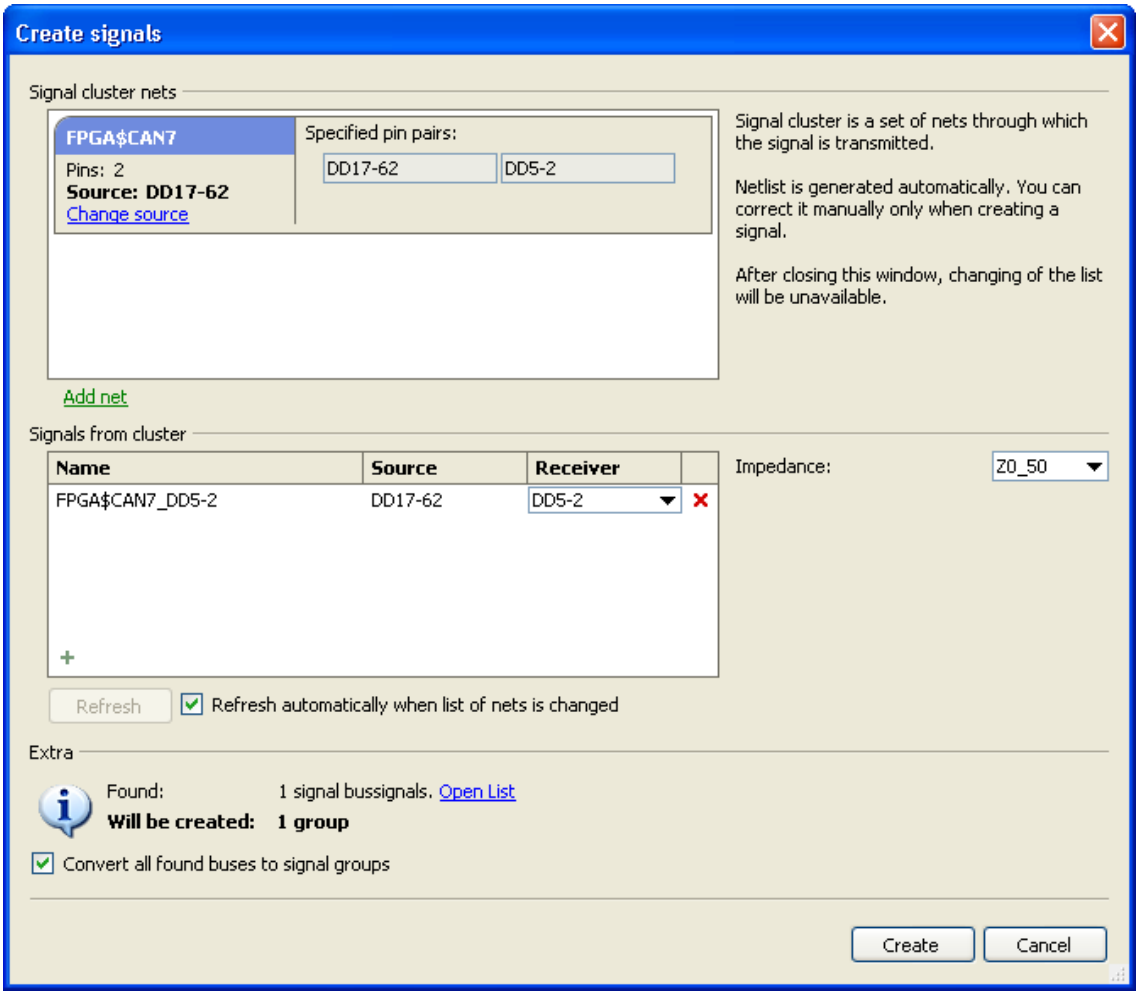


The search string can contain multiple substrings separated by spaces. The search works on the names of components, pins and nets.

Signal Cluster

Signal clusters have been introduced for combining signal nets; a signal cluster is a subschematic that contains all possible routes from the selected source to the receivers. A signal cluster contains one signal source and one or more receivers. Signal cluster nets are nets used for signal transmission.

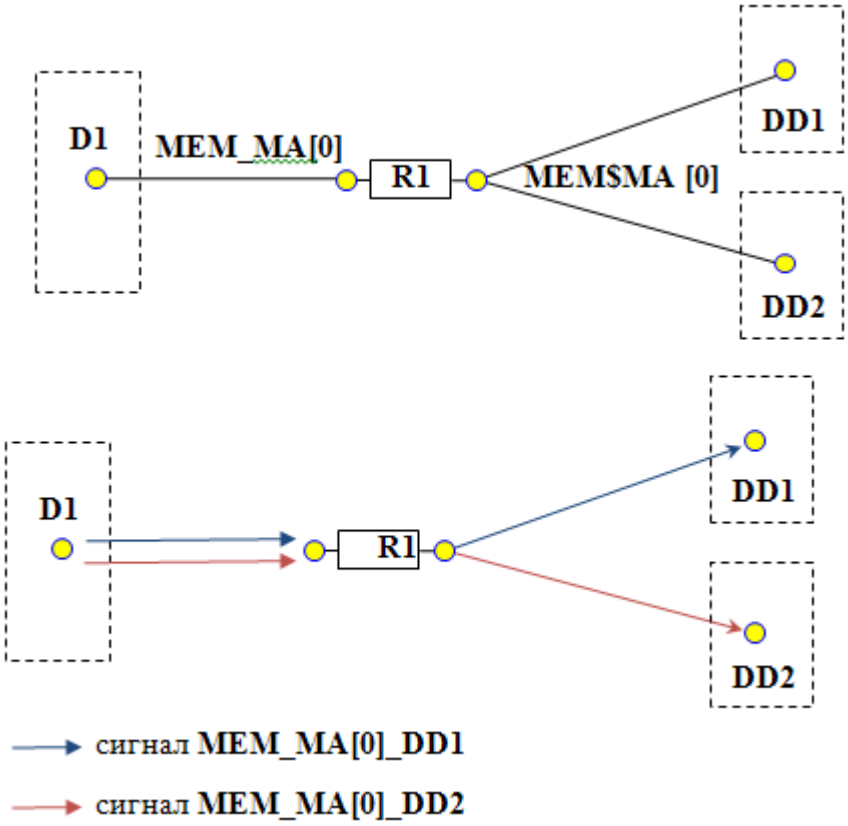
After you have selected the signal source, the **Create signals** dialog box opens. To go back to signal source selection, click **Change source**. If the source is changed, the dialog box is updated (the net and signal sets are rebuilt).



In the schematic below, selecting the pin of component D1 causes the creation of the following:

A signal cluster that contains nets 'MEM_MA[0]' and 'MEM$MA[0]'

Two signals with a common source and receivers in the pins of components 'DD1' and 'DD2'.



Signal Cluster Nets

After you have selected the signal source, the set of signal cluster nets is prepared automatically. A signal cluster includes all nets associated with the source's net through passive components, excluding the following kinds:

Power nets

Nets whose names follow the naming convention for paired signals in DSs (see Signal Search Settings)

Nets included in a different signal cluster

You can change the set of nets in a signal cluster using the following operations:

Add net—You can add any net connected to the cluster's nets through any components. You cannot add a power net or a net that is part of a different signal cluster.

Remove—You cannot remove the source net. You cannot remove a net if it is the only link between the source net and any other net in the cluster.

After you have changed the set of nets for a signal cluster, the signal list is automatically refreshed as long as the **Refresh automatically when list of nets is changed** option is selected. If this option is not selected, you can use the **Refresh** button at any time.

Signal Receiver

After you have selected the source, the signal receivers are detected automatically for the signal cluster. Receivers are the pins of active (rather than passive; see Selecting the Signal Source) components, excluding the pins of the signal source component.

A signal is automatically created for each receiver. An exception is a situation where several receivers are associated with a single component. In this case, the signal is created only for the receiver that is closest to the source.

The set of signals can be changed manually. Use the **Add Signal** button to add a signal to the signal cluster. To remove a signal, use the button that is located in that signal's table row. For each signal, you can change the receiver pin to any pin in the signal cluster, including pins that are not used by any signals.

If all pins have been turned into receivers, the button is not available.

To rename a signal, you can use the **Create signals** dialog box or the **Signals** panel. Duplicate names for signals (both regular and differential) are disallowed.

Setting the Net Topology

If a signal cluster net has more than two pins, you can create pin pairs for it. For that, click the **Add** button and select the two pins that you want to connect. To add more pin pairs, use the button.

If the pin pairs you have configured form a contour, the following message is shown on the right:
Contour is detected in net 'net_name', change pin pairs specification
Pin pairs are configured automatically if the **Create pin pairs automatically** option is selected in the **Signal search settings**. Automatic pin pairs connect the signal source to every receiver.

Impedance

Impedance (Zo) is the same for all signals in a signal cluster. You can change Zo in the **Impedance** section of the design properties editor (see also Constraints).

			Name:	Z0_50	ZDiff_100
			Z:	50	100
Layer	**Type**	**Thickness, um**		**Width**	**Width/Spacing**
1	signal	50		0.127	0.13 / 0.15
Core/Pre-preg	dielectric	100			
2	plane	18			
Core/Pre-preg	dielectric	125			
3	signal	18		0.1	0.1 / 0.15
Core/Pre-preg	dielectric	150			
4	plane	18			
Core/Pre-preg	dielectric	100			
5	plane	18			
Core/Pre-preg	dielectric	150			
6	signal	18		0.1	0.1 / 0.15
Core/Pre-preg	dielectric	125			
7	plane	18			
Core/Pre-preg	dielectric	100			
8	signal	50		0.127	0.13 / 0.15

Add Zdiff | Add Z0

To add a column, click the **Add Zo** button. In the new column, specify the required width values for each signal layer. After that, you can change the impedance for the selected signal cluster in the **Properties** panel by choosing any signal from that cluster and selecting the impedance name you want from the drop-down list. Impedance will be changed for all signals of the signal cluster at once.

Using Signals

The primary use of signals is for signal delay synchronization rules. Signals can also be controlled by rules for inter-wire clearance and rules for assignment of via types inter-wire clearance rules and rules for via type assignment.

Differential Signals

Creating a Differential Signal Manually

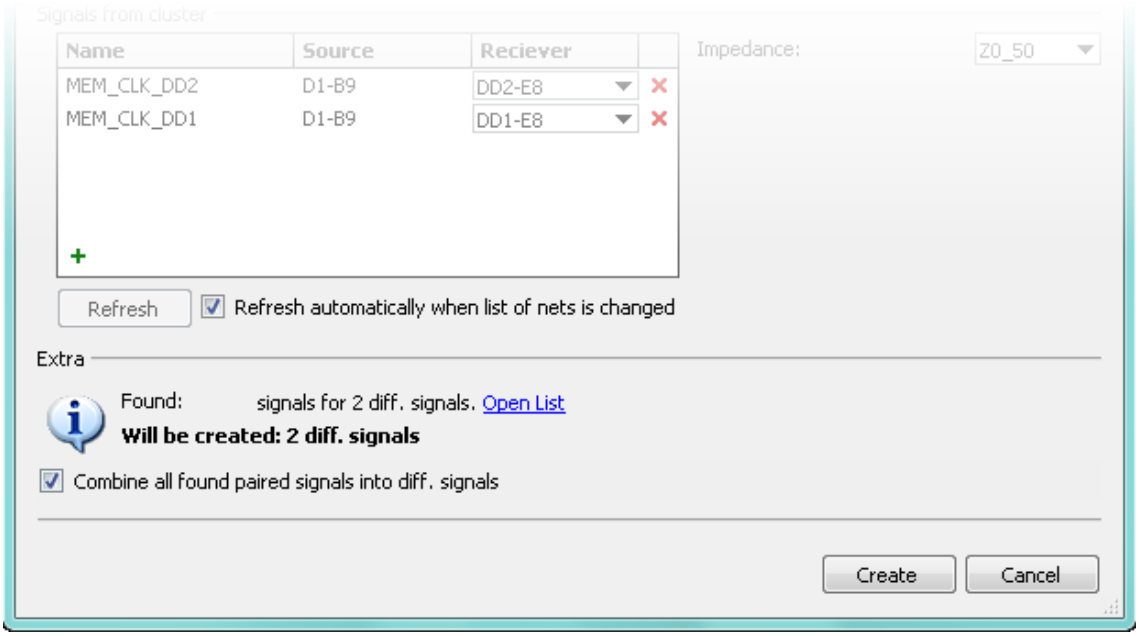
To create a differential signal manually, combine two signals. For that, select both signals in the viewing pane and click the **Generate diff. signal** action item (in the Actions panel or in the shortcut menu). Two signals can be combined into a differential signal if the following requirements are met:

1. The signals must belong to different clusters.
2. The signals must have the same number of nets.
3. The pin pairs (if any) in the signal nets must be similar.

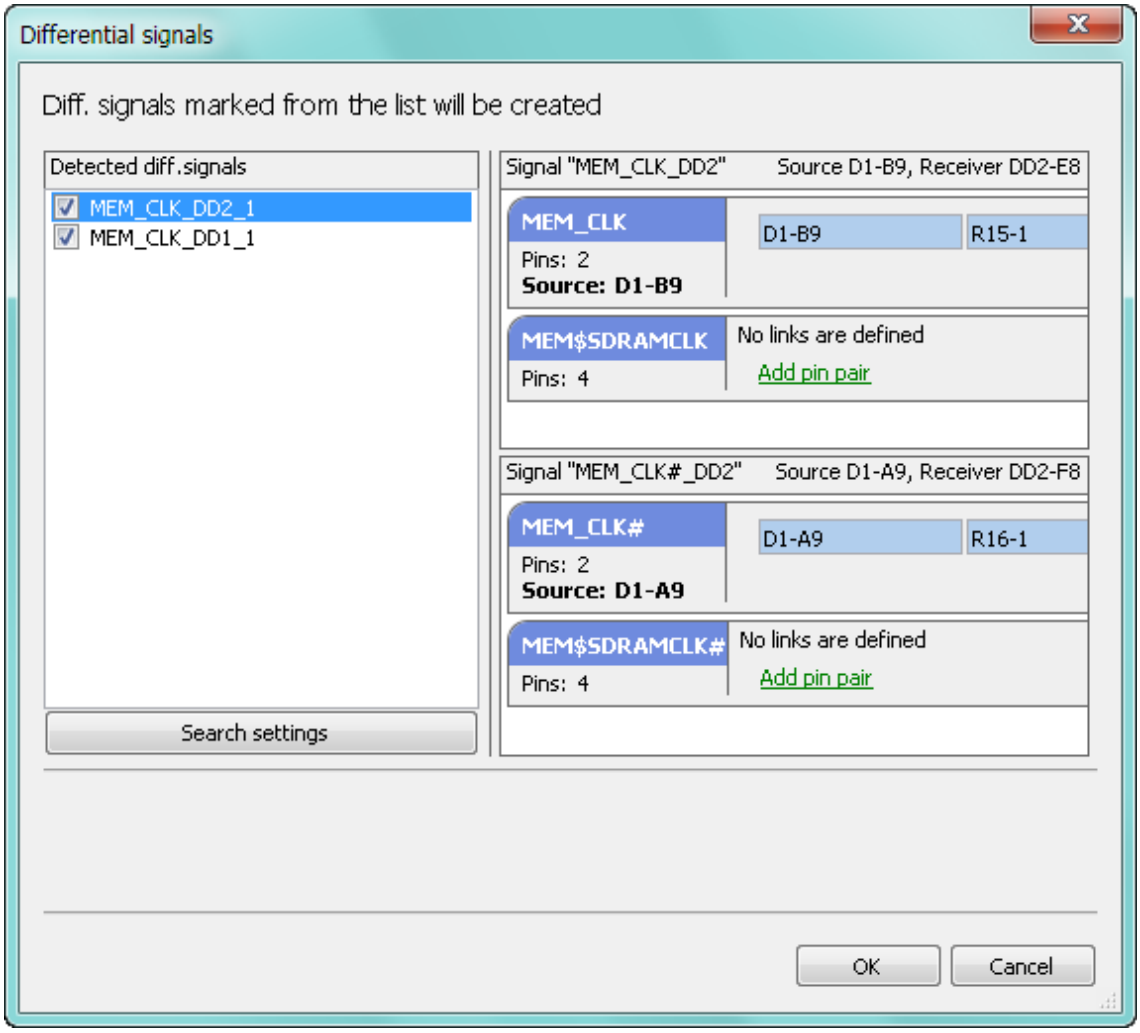
If any of the requirements is not met, an information message box is displayed, stating the reason the differential signal cannot be created.

Automatic Search for Paired Signals

During signal creation, a search for a pair for each signal in a cluster is automatically performed; this is needed for creating differential signals.
Signals are paired if their sources and receivers belong to the same components. In addition, paired signals must have the same number of nets that transmit the signal. The names of nets must follow the naming conventions for DS nets (see Signal Search Settings).
If paired signals are detected, then a message about this is displayed at the bottom of the **Create signals** dialog box.



To combine the paired signals that have been found, use the **Open list** action item; in the dialog box that opens, select the differential signals you want to create. To create all available differential signals, select the **Combine all found paired signals into diff. signals** option, which is located at the bottom of the **Create signals** dialog box.



When you have selected the differential signals you want to create, information about the number of DSs to be created is shown at the bottom of the dialog box. When you click the **Create** button, the differential signals will be created.

Automatic search for paired signals can be performed on newly-created signals. To run a search, use the **Diff. signals search** action item.

Differential Signal Parameters

The parameters of a differential signal, such as the maximum allowed difference between paired signal lengths or impedance, are configured in the properties panel (when the DS you need is selected). Similarly to regular signals, impedance is the same for all DSs with a shared source.

Working with Differential Signals

Differential signals, like regular signals, can belong to signal groups.

For DSs, you can set rules for inter-wire clearance, rules for assignment of via types and rules for signal delay synchronization.

A differential signal can be split into the two regular signals that it is made up of. For that, use the **Disband** action item. As a result, the DS is deleted.

Automatic Search for Signal Buses

When signals are created, searches for the signal bus are performed for each signal in the cluster. Bus signals are signals whose sources and receivers belong to the same components. The names of signal sources can differ in the index they use. An index is an arbitrary chain of digits, sometimes enclosed in parentheses or square brackets.

If signal buses are detected, a message about this is shown at the bottom of the **Create signals** dialog box.

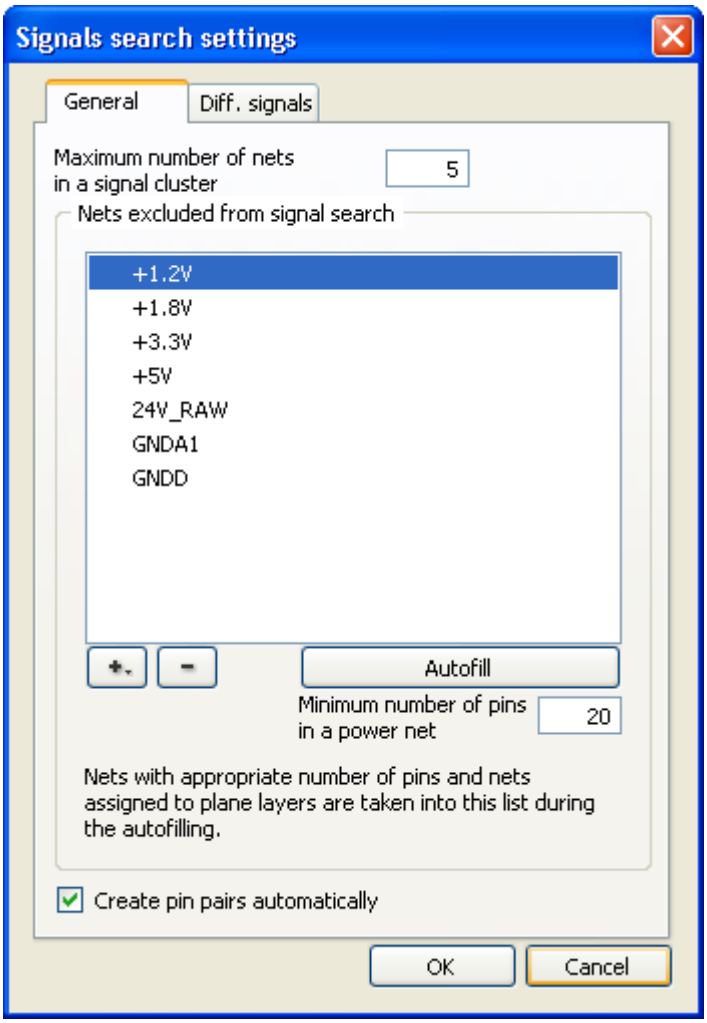
To create signal groups out of the signals in the buses that have been found, use the **Open list** action item. In the dialog box that pops up, select all buses that you want to convert to groups. To create all available signal groups, select the **Convert all found buses to signal groups** option at the bottom of the **Create signals** dialog box.

When you have selected the signal groups to create, the resulting number of groups is shown at the bottom of the **Create signals** dialog box. To proceed, click the **Create** button.

Automatic search for signal buses can also be performed on existing signals. To run a search, use the **Bus search**  task in the signal panel action pane.

Signal Search Settings

You can open the **Signals search settings** dialog box from the action pane, from the **Differential signals** dialog box, or from the **Found signals buses** dialog box.



The **Signals search settings** dialog box has two tabs. The **General** tab contains the following:

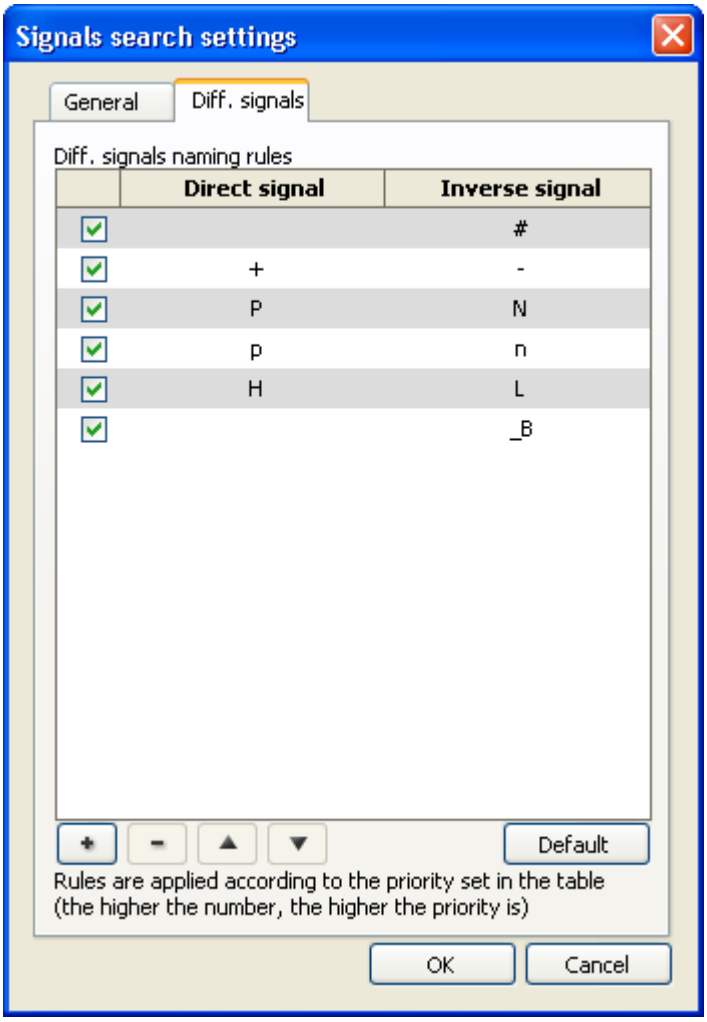
Maximum number of nets in a signal cluster—This parameter can be useful for filtering out multi-pin nets that are not explicitly included in the list of the **power nets list** below. If the specified number of nets in a cluster is exceeded, then the net is removed from the cluster, and so is the preceding net in the sequence, but the signal source net remains.

Power nets list—You can generate the list automatically or change it manually. Power nets are excluded from search when nets are gathered for a signal cluster.

Minimum number of pins in a power net—This parameter is used during automatic generation of the list of power nets.

Create pin pairs automatically—If this option is enabled, then the signal source is automatically paired with each receiver during the creation of a signal cluster.

The **Diff. signals** tab lets you edit naming rules for DS nets. These rules are used during automatic search for paired signals to combine into differential signals.



A DS net naming rule contains two substrings, one of which can be empty. Two nets match a rule if substrings represent the only differences in their names and both substrings are similarly positioned.

For example, suppose a rule contains the substrings “p” and “n”. In this case, the “NetCLKp” and “NetCLKn” nets match the rule, and so do the “pNetCLK” and “nNetCLK” nets, but the “pNetCLK” and “NetCLKn” nets do not.

The rules are listed in order of descending priority. Check boxes in the leftmost column enable and disable rules.

Action Panel

When you select objects (signals, DSs, signal groups), the following actions become available for them in the **Signals** panel of the design properties editor:

Highlight—Highlight the selected signals, DSs or signal groups in the topology editor.

Set topology—Open an editor for specifying connections in the signal's nets.

Delete—Delete the objects; when a DS is deleted, the signal pair that it includes is also deleted.

Combine into diff. signal—Create a differential signal from two regular signals (see Creating a Signal).

Split—Split a differential signal into two regular ones (and delete the original DS).

Restoring Differential Pairs

The **Tools ► Restore Diff. Pairs** menu item starts a search for wires that can be combined into a differential pair. For this procedure to work, the design parameters must define the necessary differential signals. The final step of the procedure is zipping all the differential pairs that have been found.

The primary use is to restore the list of differential pairs that were previously created in the **TopoR** CAD system and later lost due to export of the topology to a different CAD system. During re-import, such pairs will need to be detected and restored. This is done by the **Restore Diff. Pairs** procedure, and the corresponding wires are automatically zipped.

Restoration is guaranteed to work for differential pairs that were created by **TopoR** as long as arcs were disabled and clearances for the corresponding differential signals on the corresponding layer were observed. Zipping the pairs does not change wire shapes. Some pairs may already be zipped.

When the procedure is complete, one of the following messages will be shown:

Differential wire pairs found and zipped for <number> differential signals.

This means that differential pairs were restored successfully.

No differential pairs found for restoring. For details, see the "Restoring Differential Pairs" online help topic.

The procedure could not find any differential pairs to zip.

No differential pairs restored, because differential signals are not set up. For details, see the "Restoring Differential Pairs" online help topic.

The procedure cannot run, because differential signals have not been configured in the design properties.

High-Speed Rules

The **High-speed rules** section in the **Design properties editor** lets you set rules for impedances, signal delays and signal synchronization.

Impedance

You cannot delete the predefined Z0_50 and Zdiff_100 rules, but you can edit them. The impedance value is not used in **TopoR 6.1** and is only present for reference. There is no check whether the width matches the impedance.

Delay Match

The **Delay match** pane contains a table of delay synchronization rules in signal groups.

#		Affected	Tolerance
1	☑	Group of signals 'MA_DD1'	10 ps
2	☑	Group of signals 'MA_DD2'	10 ps
3	☑	Group of signals 'MD0-7'	10 ps
4	☑	Group of signals 'MD8-15'	10 ps
5	☑	Group of signals 'IDE0$D'	10 ps
6	☑	Group of signals 'ISA'	10 ps

To create a rule, click the **Add rule** button. Next, select the group of signals and set a tolerance.

To delete a rule, select it and click the **Delete rule** button. The check box next to the rule lets you exclude it from processing without removing it from the table.

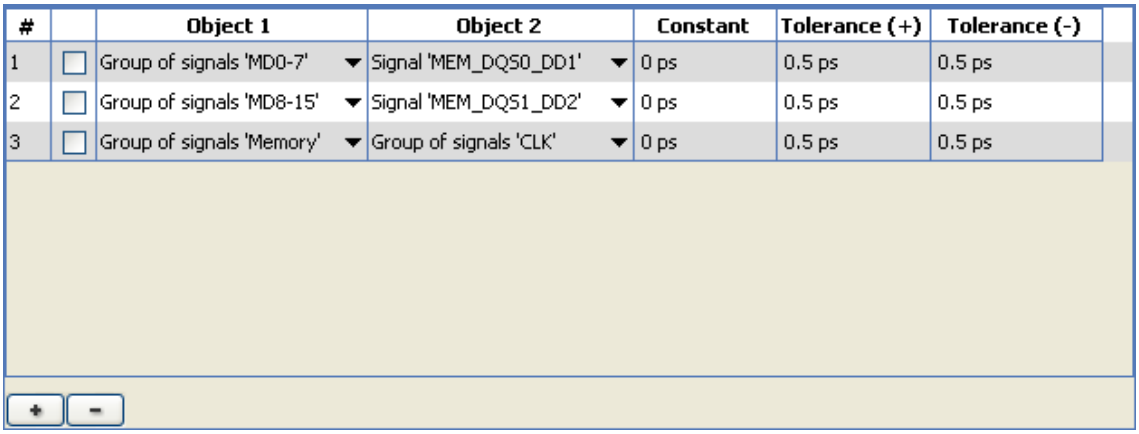
A tolerance is specified for each rule. The tolerance value should be specified in time units: picoseconds (ps) and nanoseconds (ns). Length units are also accepted, but not recommended due to the likelihood of board delay on different layers. If length units are used and the type of unit is not specified, the current units are assumed.

Delay Relation

The **Delay relation** pane contains a table with rules of two types. The first type of rule is for absolute signal delay. The absolute value is specified in the **Constant** column. The second type or rule lets you define the following relationship between two objects:

$$-tolerance^{(-)} \le Delay(1) - Delay(2) - c \le tolerance^{(+)},$$

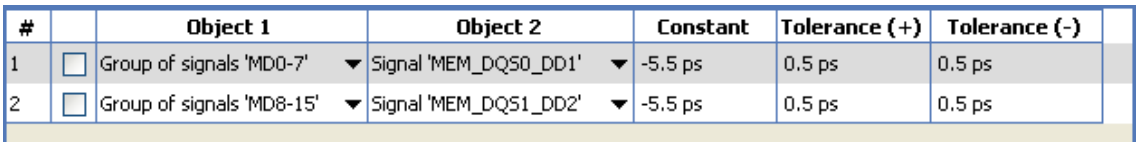
where *Delay(1), Delay(2)* are delay values for objects 1 and 2, *c* is the constant requisite delay difference, $tolerance^{(+)}$ is the upper bound for the tolerance, and $tolerance^{(-)}$ is the lower bound.



#		Object 1	Object 2	Constant	Tolerance (+)	Tolerance (-)
1	☐	Group of signals 'MD0-7'	Signal 'MEM_DQS0_DD1'	0 ps	0.5 ps	0.5 ps
2	☐	Group of signals 'MD8-15'	Signal 'MEM_DQS1_DD2'	0 ps	0.5 ps	0.5 ps
3	☐	Group of signals 'Memory'	Group of signals 'CLK'	0 ps	0.5 ps	0.5 ps

To add a new rule, click the **Add Rule** button. To set a rule of the first type (absolute delay), specify one object and a value in the **Constant** column. To set a rule of the second type, specify two objects. Objects can be signals or groups of signals.

For example, suppose you need any of the signals in three different groups to be ahead of the synchronization signal by at least 5 picoseconds and at most by 6 picoseconds. In this case, the rules can be set as follows:



#		Object 1	Object 2	Constant	Tolerance (+)	Tolerance (-)
1	☐	Group of signals 'MD0-7'	Signal 'MEM_DQS0_DD1'	-5.5 ps	0.5 ps	0.5 ps
2	☐	Group of signals 'MD8-15'	Signal 'MEM_DQS1_DD2'	-5.5 ps	0.5 ps	0.5 ps

For details about delay equalization, see Manual Delay Equalization and Automatic Delay Equalization.

Editing Techniques

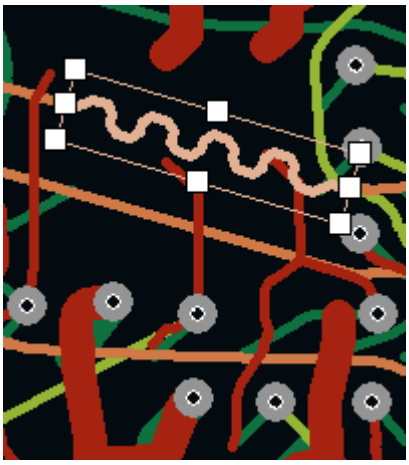
This section describes:

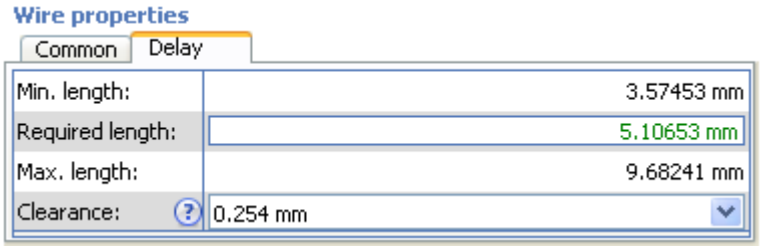
- Manual delay equalization
- Automatic delay equalization
- Routing differential pairs
- Polygonal layout
- Troubleshooting
- FreeStyle mode
- Refine tool
- Via shifting
- Inverting selected areas

Manual Delay Equalization

Before you continue reading, make sure you are familiar with the Delay Match and Delay Relation sections of this manual.

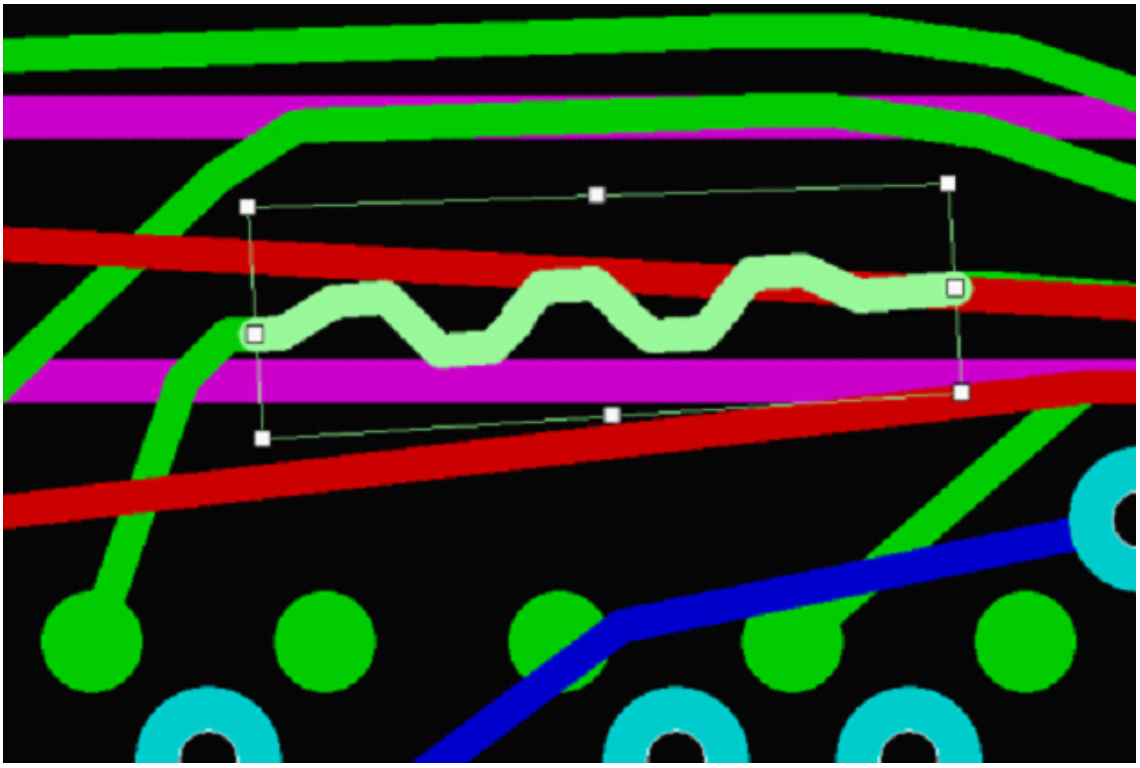
Signal delay equalization is controlled using *serpents*, which are shapes that fit inside a bounding trapezium. A trapezium (starting out as a rectangle) is created around a selected wire segment when you click the **Convert to serpent** button.



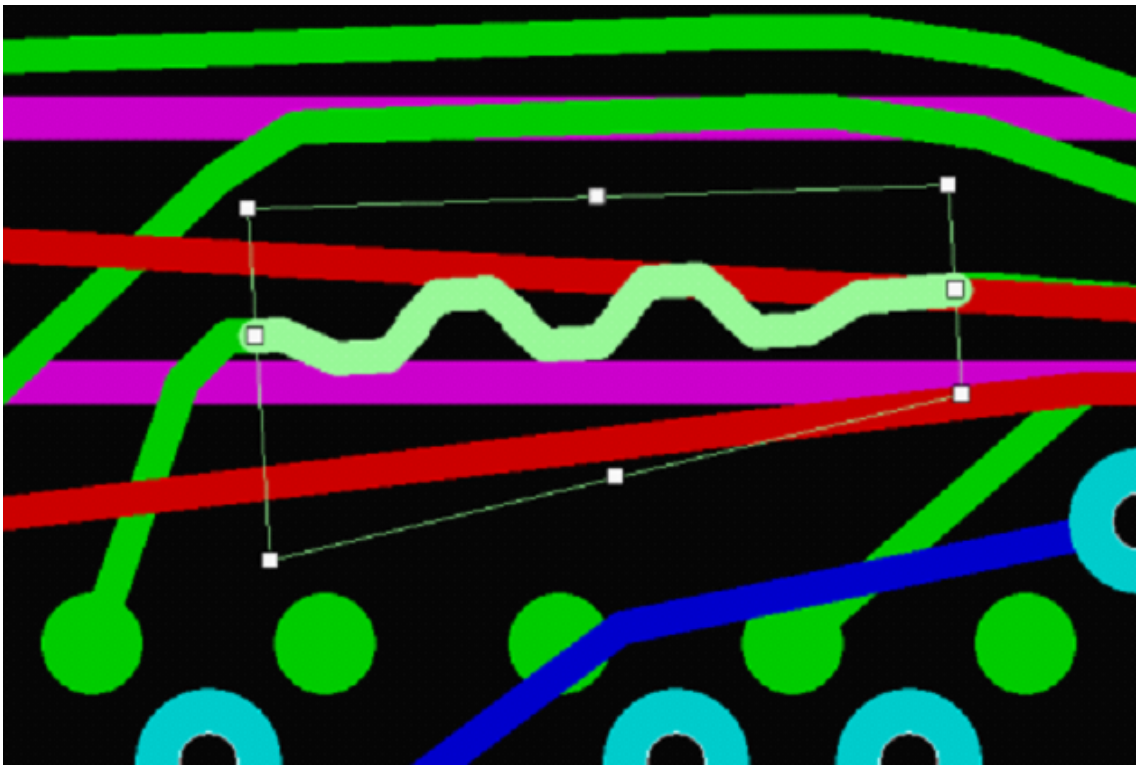


To configure the properties of the serpent, use the **Delay** tab in the wire properties panel. On this tab, you can specify the spacing between bends (twice the wire width by default) and the desired delay (this affects wire length). The minimum and maximum possible delay time (or wire length) are displayed and updated automatically for the trapezium.

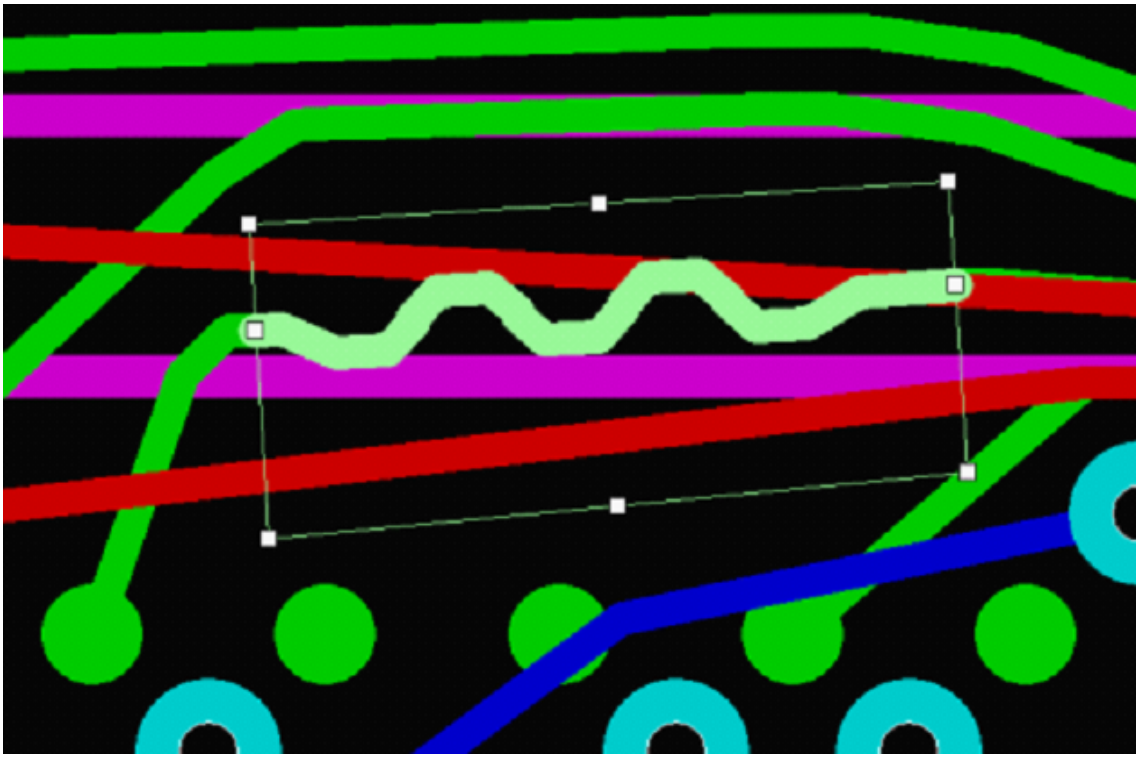
Use markers on the trapezium to set its height, base lengths and the angles of its lateral sides.



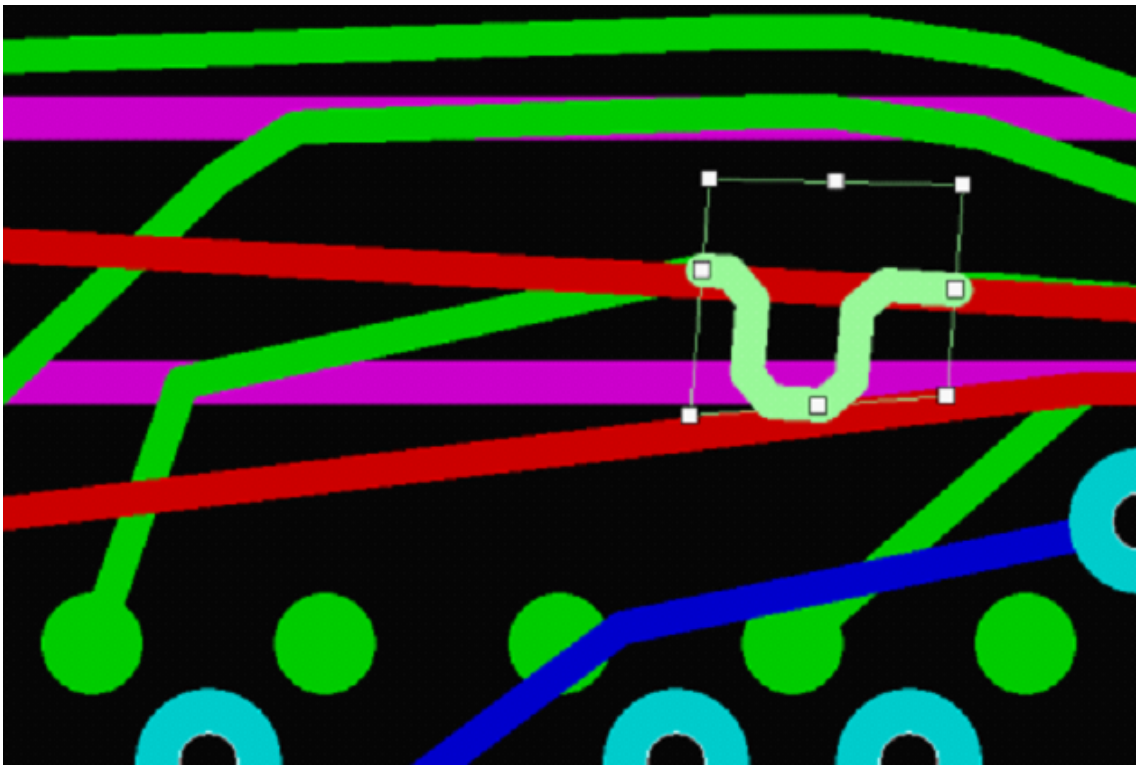
By moving the markers at the base vertices, you set the base length and the angle of the adjacent lateral side.



By moving a marker in the middle of a lateral side, you move the side along the bases.



By moving a marker in the middle of a base perpendicularly towards or away from the opposite base, you make the lateral sides shorter or longer.



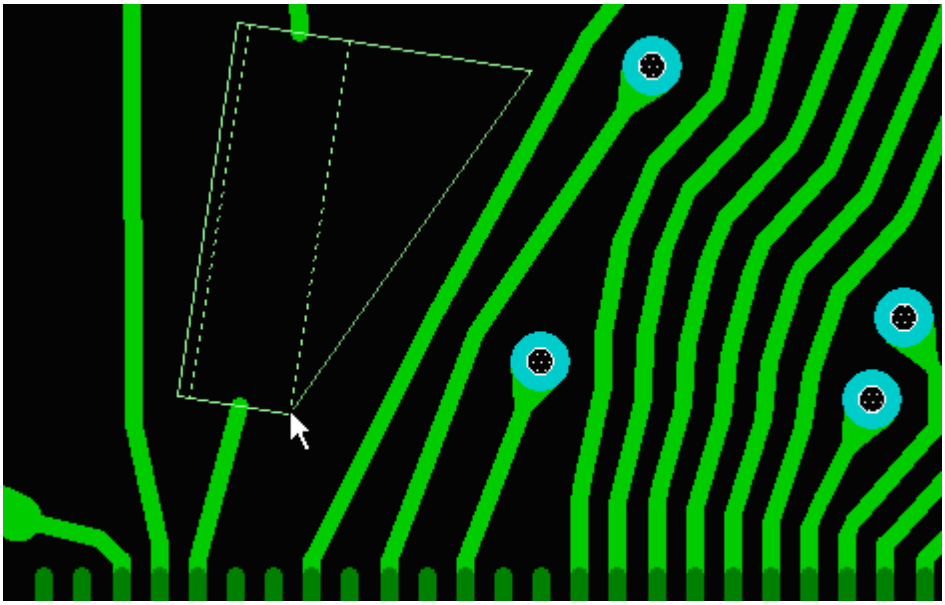
If the direction is not perpendicular, the trapezium is also rotated about the middle marker of the opposite base.

When you equalize wire lengths manually, the length and spacing of the inscribed serpent are taken from the **Delay** tab.

If the area of the trapeziums is sufficient in each of the nets you are equalizing, the specified desired length is ensured *automatically* with a tolerance of 50nm at the most.

If the area of the trapeziums is not sufficient for precise equalization, then the output bar displays a rule violation message. The message lists the nets with the grossest violations (the shortest net is listed first) and shows by how many picoseconds the value is off.

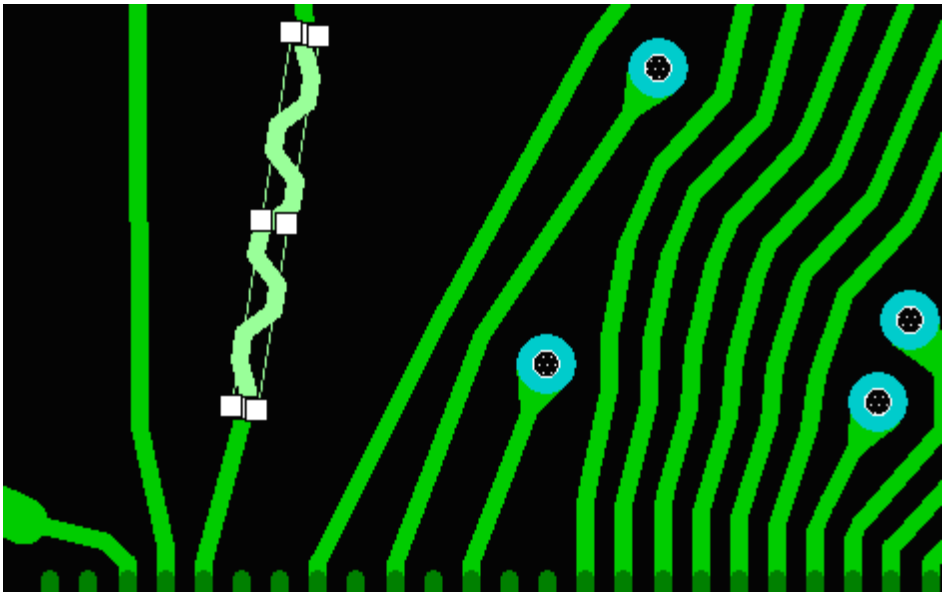
Clicking the **Convert to serpent**  button a second time removes the trapezium and leaves the serpent in place. The serpent is not a separate object now, but a collection of wire segments. Areas whose widths are less than the serpent cycle are considered narrow. Only constant-amplitude and arc-free serpents are inscribed in narrow areas, so editing narrow areas is different from editing regular areas. A change to the area due to a moved marker is shown as a solid line. A dashed line shows the actual area where the serpent will be inscribed; this is the area that is set when you release the mouse button.



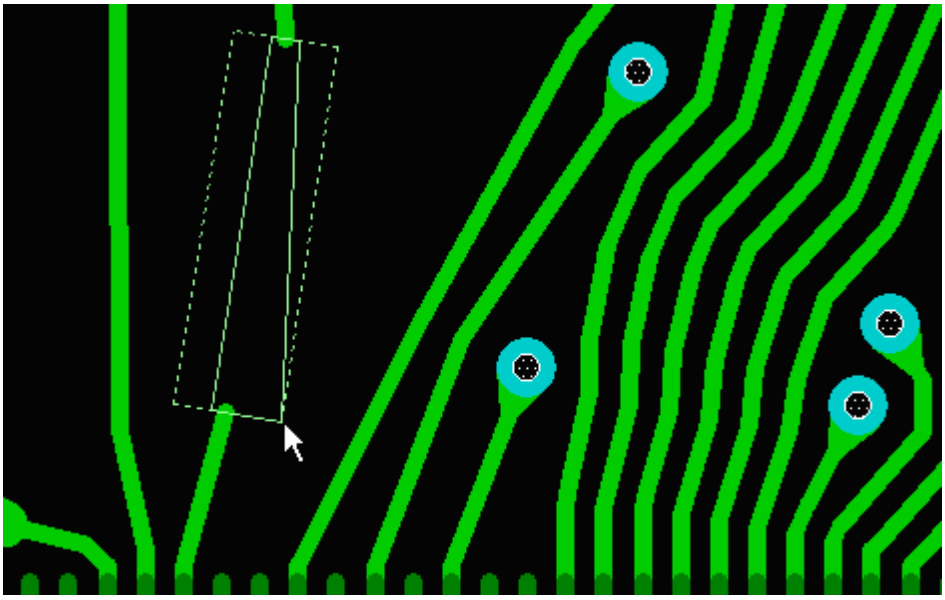
The minimum area width is 1.5 times the width of the wire.

Example

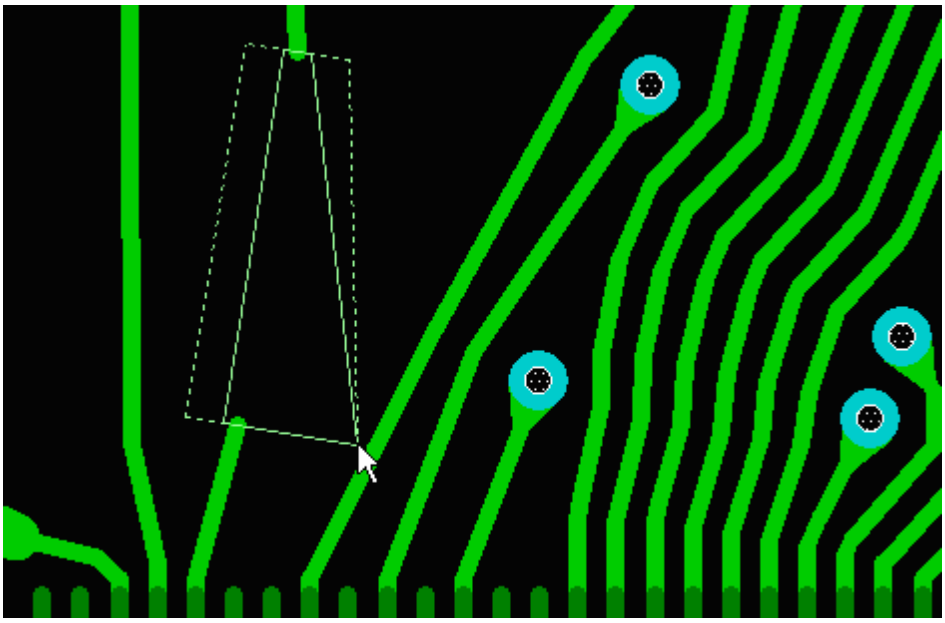
A serpent is inscribed in a minimum-width area.



During a marker move, two areas are shown. The area indicated by a dotted line changes symmetrically.



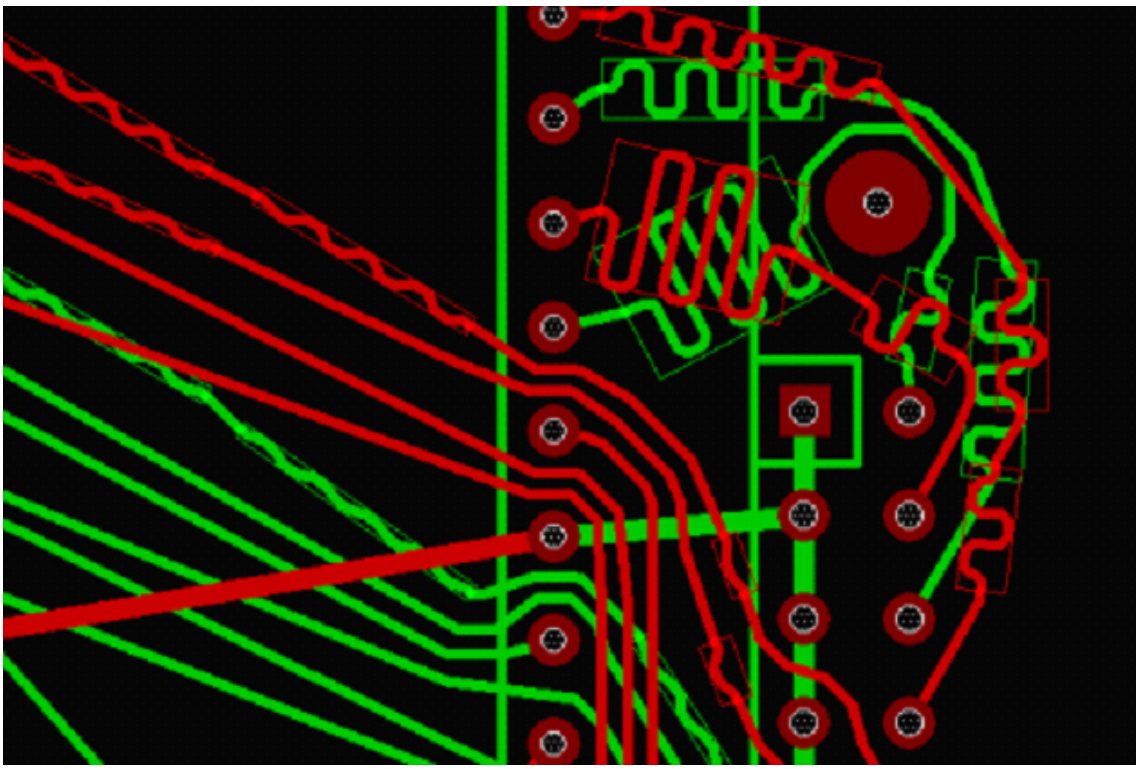
After an area stops being narrow, subsequent marker moves give it a trapezium shape.



If you need the area to stay rectangular, move the middle markers rather than the corner markers. To preserve symmetry, hold down the **Shift** key.

Automatic Delay Equalization

The **Tools ► Create serpents** menu item automatically adds serpents to wires that need equalization. When you equalize wire lengths automatically, the serpent has the specified length if there is enough room, or the maximum possible length if there is not enough room.



Routing Differential Pairs

To route a differential pair, select **Route ► Place diff pair** in the main menu. Pads that belong to different differential pairs are highlighted in different colors.

A differential pair is a set of *zipped* and regular wires. Zipped wires are shown as a zipper with two sliders. A slider always stays perpendicular to the nearest segment of a zipped wire.

Clicking one of the highlighted pads creates a pair of wires at pads that are the specified distance apart from each other. Clicking again places a slider under the cursor. From now on, the zipped wire pair is routed as a single wire.

When you approach another pair of pads, clicking one of them places a slider at the end of the last segment of the zipped wire and adds wires to connect the slider to the corresponding pads.

If you need to unzip the selected parts of a zipped wire pair, click the **Unzip wires** button. To zip the wires of a differential pair, it is necessary to allocate parts of the segments that belong to the differential pair, for example, select a segment of one wire and hold down **Ctrl** then select another segment, then press **X**, after that click the **Zip wires** button in the toolbar.

To zip all differential pairs (for example, after autorouting) press **Ctrl+A** (select all), and then click the **Zip wires** button. This may result in some violations that need to be removed manually.

Requirements that wires need to meet to be zipped

The selected segments should belong to the wires that are part of the differential signal specified in the rules.

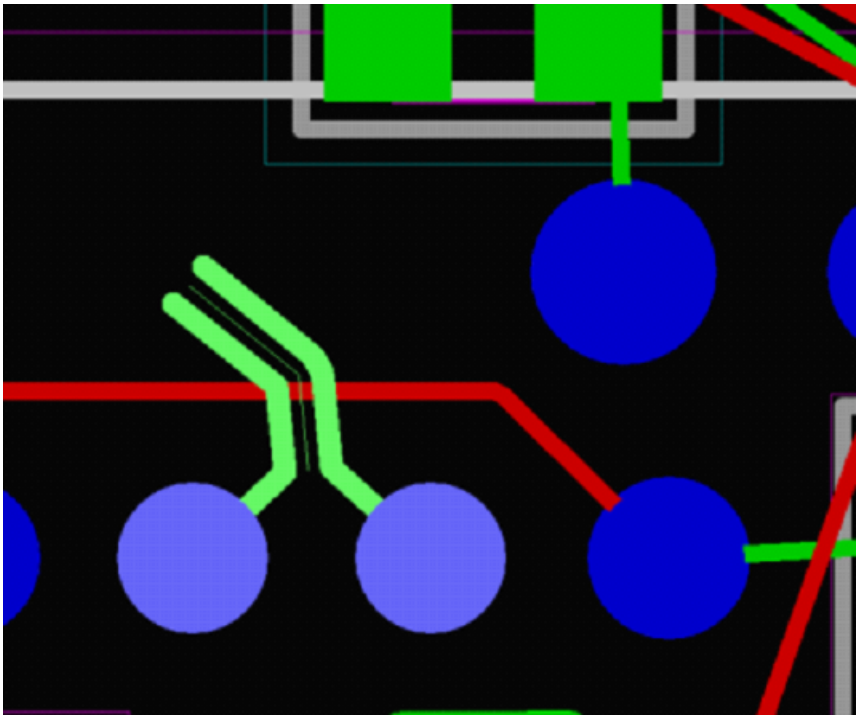
The wires should be on the same layer.

The widths of the wires should observe the rule specified for the impedance.

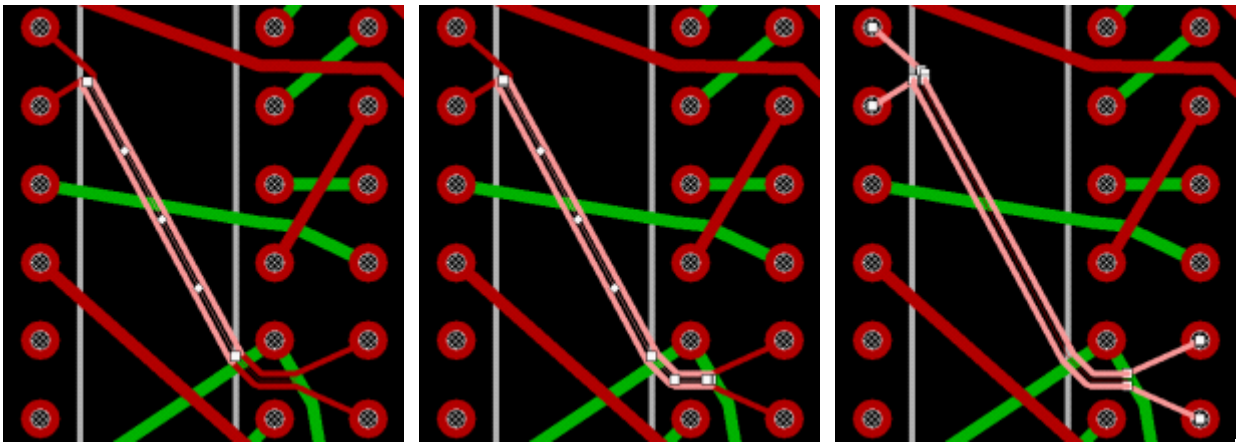
The straight length of each of the wires must be no less than the width of the zipped wire pair (according to the rule for differential pairs that is twice the wire width plus clearance) multiplied by three.

In the wire where the straight length of the selected segments is less than in the other wire, those selected segments are deleted, and a zipped pair replaces the longer selected segments of the other wire.

When you select a differential pair, the selected segment is highlighted. If the selected segment belongs to a zipped wire, then pressing the **X** key selects the zipped portion up to the slider. Pressing **X** again selects both wires in the differential pair, as shown in the following screenshots:



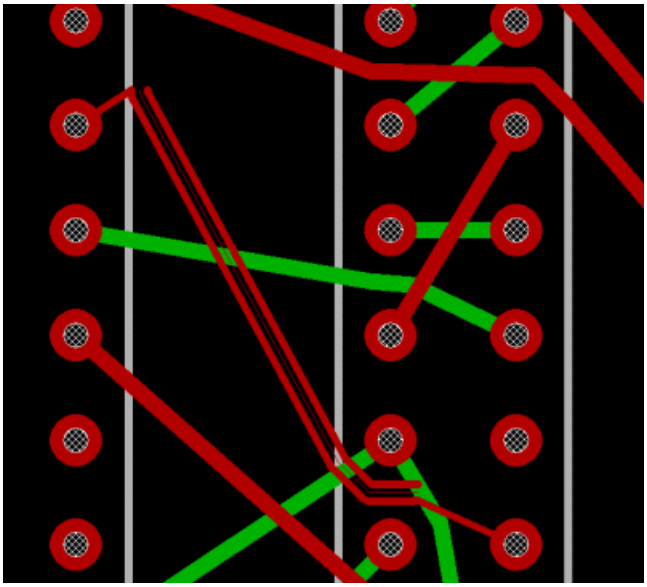
If the selected segment belongs to an unzipped wire, then pressing the **X** key selects the segment up to the slider. Pressing **X** again selects the entire wire, as shown in the following screenshots:



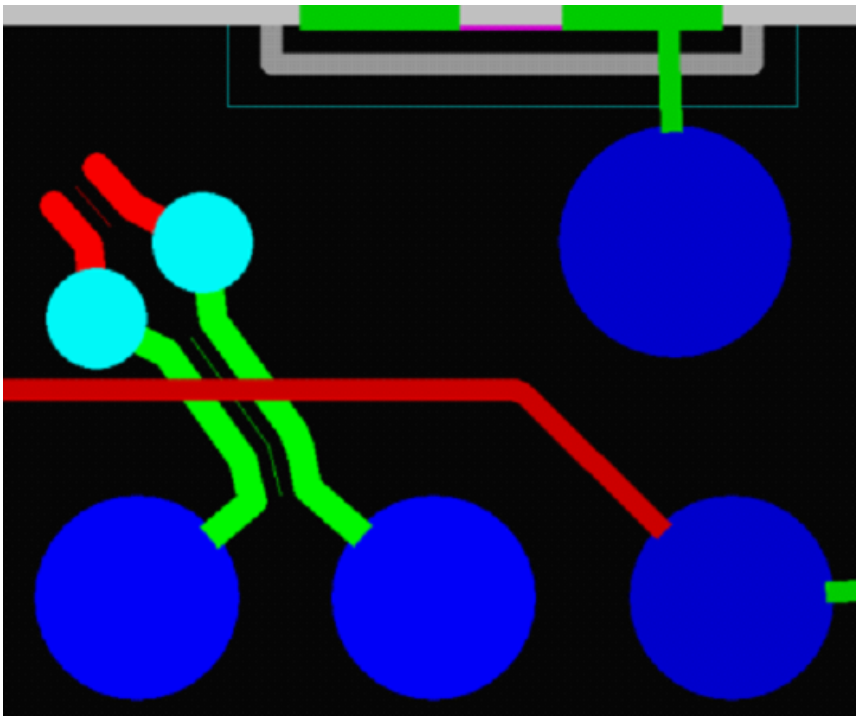
Editing

Editing of selected segments of a zipped wire (moving segments, straightening selected parts) is similar to editing regular wires. The widths of zipped wires can be modified only in **Design ► Parameters ► High-Speed Rules ► Impedance**.

When you delete a selected net (by pressing **Delete**), only unzipped portions are removed.



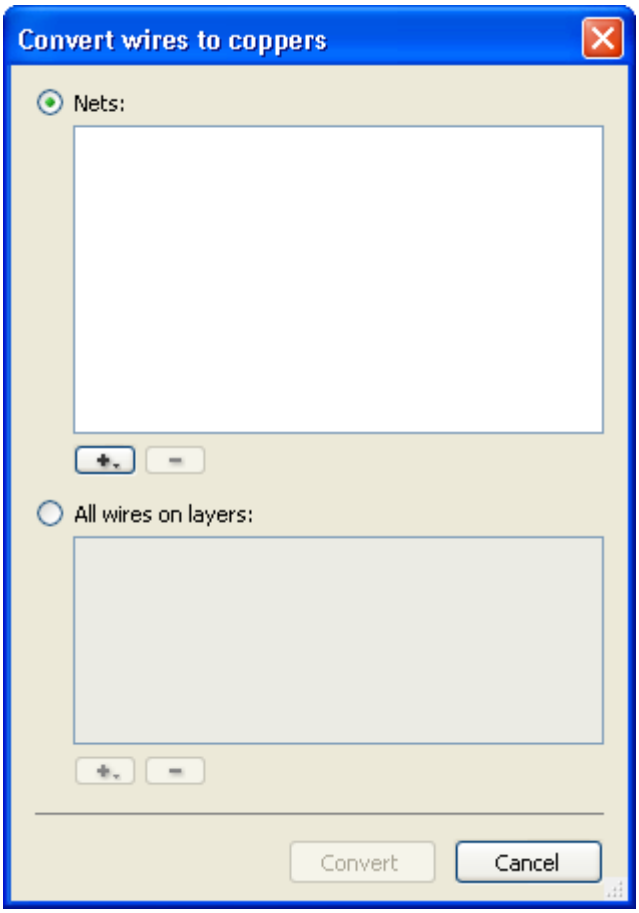
To switch layers, press the **L** key and select the next layer. As a result, a slider is automatically added on the current layer, and a pair of vias is added on the new layer, as well as wire segments connecting the vias and the corresponding slider pads.



To remove overlap between the wires of a differential pair, use the net swap feature. For that, in the properties of the zipped wires, click the **Swap Nets** button, and then click **Apply**.

Polygon Routing

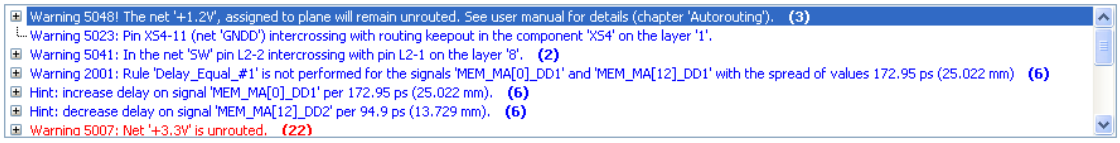
This feature is useful for power boards. To make polygons out of wires on a routed board, run the **Tools ► Convert wires to copper Areas** procedure.



In the **Convert wires to copper Areas** dialog box that opens, select the nets where to convert wires. If you select the **All wires on layer** option, you can select one or more layers and convert all of the contained wires at once. The wire-to-polygon conversion starts when you click the **Convert** button. The completed conversion can be rolled back using the standard undo mechanism (**Ctrl+Z**).

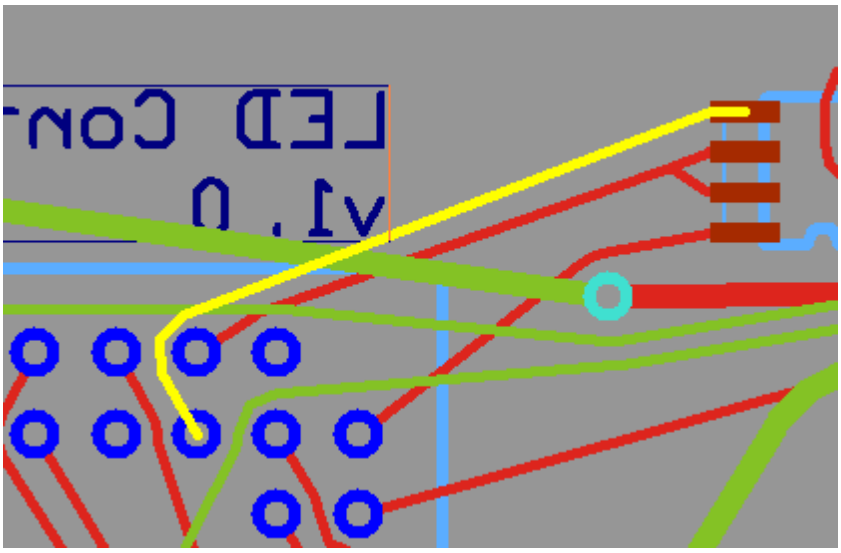
Error Messages

Warnings, errors and design rule violation reports are shown in the **message bar** in a tree, grouped by type. If there are multiple messages of the same type, their count is displayed in parentheses.



To read a message in full, you do not have to expand the message bar; you can simply hover the cursor over it.

Double-clicking a message centers the view on the problem spot and highlights the relevant objects.



The message bar contains warnings about non-fixed elements that are over the edge of the board. At the same time, clearance violations are highlighted. The listed elements should be either fixed or moved to eliminate violations. This makes the warnings and highlighting disappear.

Clicking the line that marks a DRC violation highlights the conflicting topology elements, and a diagnostic message appears in the properties panel.

FreeStyle Editing Mode

The **FreeStyle** editor helps improve a previously routed PCB layout performed not only in **TopoR** but also in any other router. In this mode, you can move components, vias and T-points. Optimal wire geometry is immediately calculated automatically.

In the action pane, select one of two shape calculation methods: with and without arcs.

Not all CAD systems accept arcs. For example, P-CAD (versions prior to 2004) accepts only arcs where the angular values are multiples of 0.1°; OrCad and Specctra multiples of 90°. Signal integrity programs such as HyperLynx cannot handle arcs either. Therefore, it is preferable to export to other CAD systems with arc-like wires disabled.

TopoR lets you move elements around on the routed board, while preserving layout integrity and maintaining the specified clearance. Moving components, vias, and T-points can help decrease wire length, observe clearance (eliminating bottlenecks), and reduce the board area used up by components.

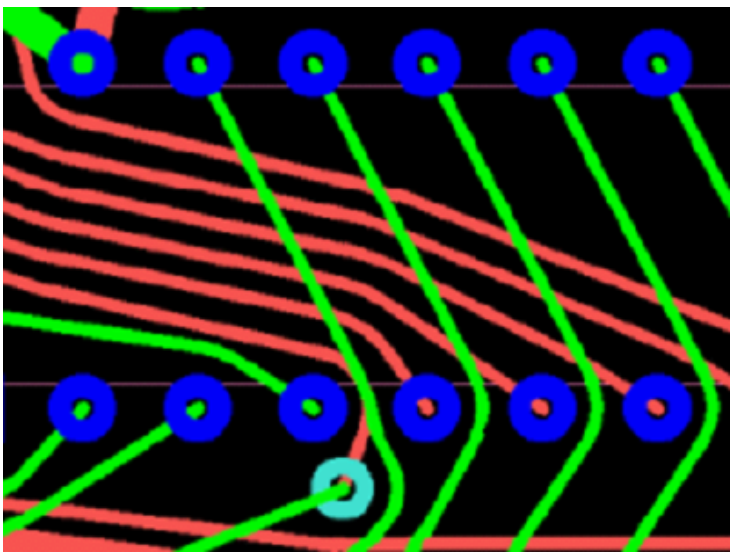
FreeStyle mode has the following submodes:

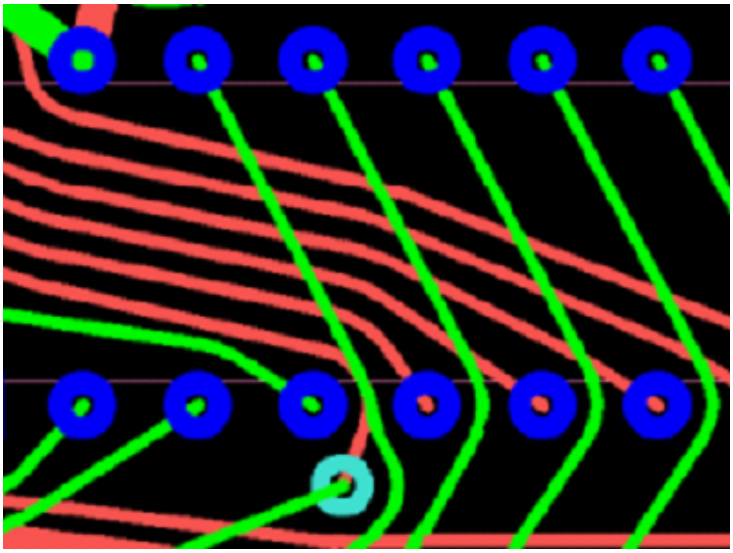
Push none—move objects without complying with the specified clearance. In this case no other objects are moved.

Push vias—move vias and wire T-points, spacing them appropriately.

Push components and vias—move components, vias, and T-points, spacing them appropriately.

When you move components, vias, and T-points manually in **Push none** mode, constraint-violating moves are not prevented. For example, you can pull a via between microchip pins.

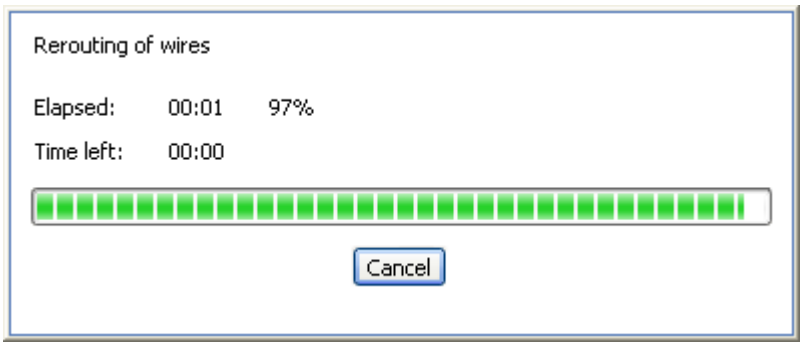




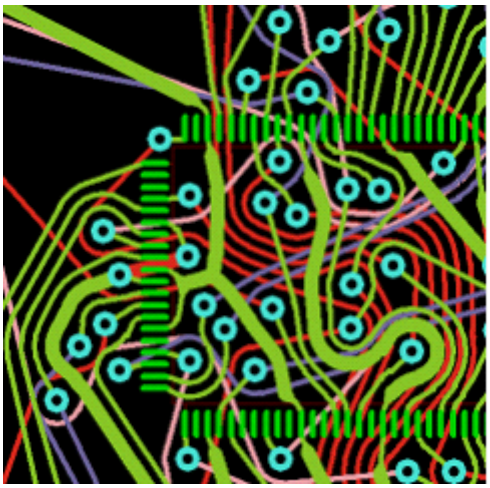
Element moving is done automatically, vias and T-points are moved to their calculated optimal positions. If a component is not supposed to be moved automatically, you should fix it first.

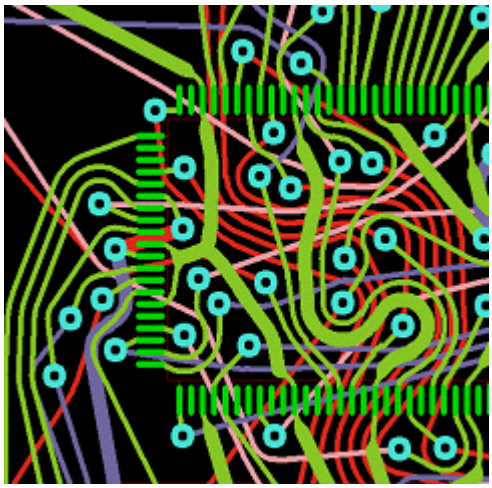
Refine Tool (F6)

During automatic movement of components and vias, wire geometry is modified automatically without modifying the topology. This causes some wires to be placed in a suboptimal fashion. To correct such situations, use the ***Refine*** tool, which optimizes automatically moved wire paths. To run this procedure, use the button in the topology editor's action pane.

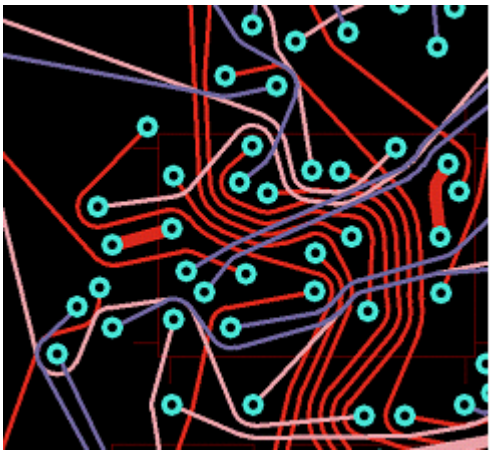


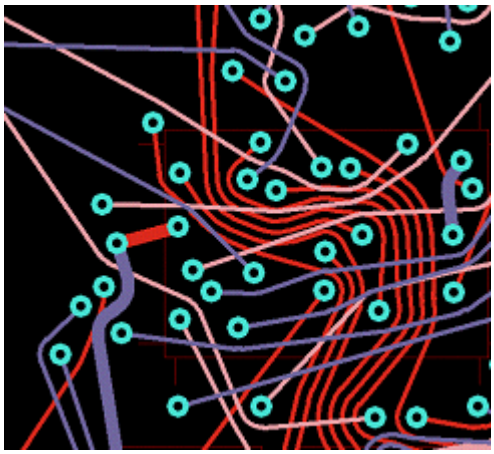
The screenshot on the left shows a portion of a board before ***Refine*** is applied. The screenshot on the right shows the result of the procedure.





Here are the same fragments without the wires on the top layer:





Notice that the optimization has shortened some wires.

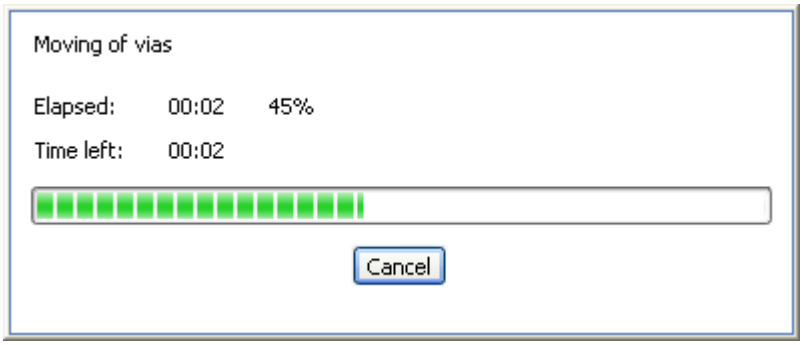
The ***Refine*** tool has two submodes: with layer change for wires enabled and disabled. To select one of these submodes, right-click and use the context menu. In the first submode, the procedure

can change not only the wire geometry but also its layer (if the wire connects through pads or vias). In some cases, this decreases the number of vias. The ***Refine*** tool is most effective when designing multi-layered circuit boards. This use of the ***Refine*** tool is especially helpful in designing multi-layer boards

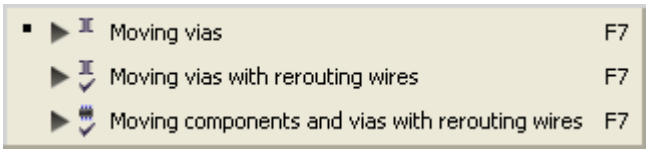
Redundant vias left behind by the ***Refine*** tool are deleted automatically.

Via Moving (F7)

Another tool in the action pane is a procedure that enhances the placement of vias. To run it, click the ▶ button. The short name for this tool is **Shift Vias**:



The shifting mode has three submodes:

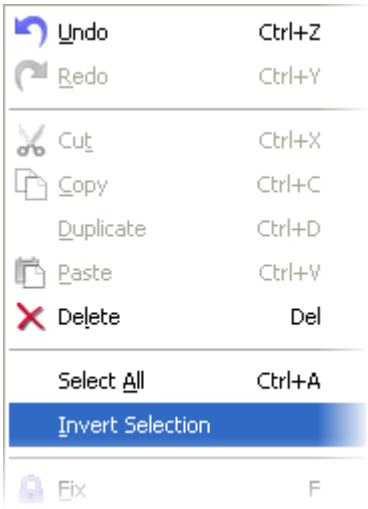


Pressing the **F7** key launches the current (most recently used) submode; to select which submode is active, right-click and use the context menu.

In the first submode, vias are moved without wire rerouting, and all components stay where they are. In the second submode, wires can be rerouted in addition to via moving. Finally, in the third submode, moving can involve not only vias but also components.

Inverting the Selection

If you need to keep only a few wires, you can select the elements you want to leave and use the **Edit ► Invert Selection** item in the main menu. After that, remove all the topology elements that have become selected as a result.



Data Recovery

TopoR automatically backs up data so that it can be subsequently recovered. Data is autosaved for every project you open. When the project is closed normally, the backup data is deleted.

Autosaving of **`.fsx`** files is automatic (for any project that is open). Autosaving of routing data is enabled when you run autorouting and disabled when autorouting is completed.

Opening a Project after Abnormal Termination

If you experience a program failure while working on a project, the next time you open the project, **TopoR** will check that the autosave file is more recent than the main project file. If this check confirms abnormal termination, then notification is displayed.

Likewise, **TopoR** can notify you that you can recover autorouting data.

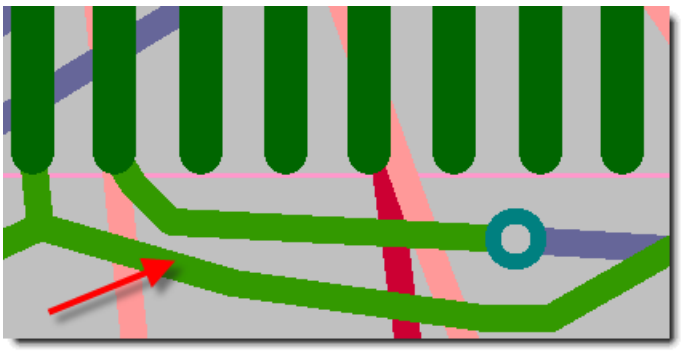
TopoR Workflow Specifics

For efficient use of **TopoR**, consider the following distinctive characteristics of the system:

Imprecise detection of the optimal wire shapes
Choice of “desired” spacing
Peculiarities of pad matching
Peculiarities of using DRC during and after routing
Ways to fix wires
Manual control of optimization progress
Specifics of working with copper areas

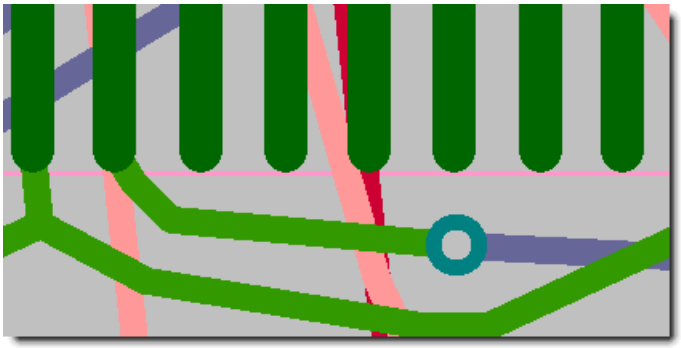
Imprecise detection of the optimal wire shapes

TopoR implements automatic calculation of the optimal wire shape. However, due to the complexity of the calculations, there may be some imprecision on densely-packed boards.



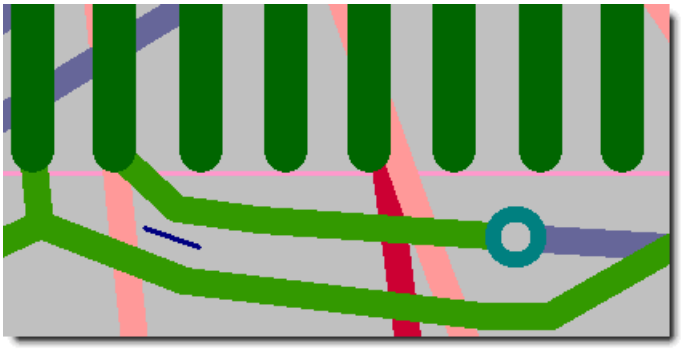
The “miscalculated” wire is marked with an arrow

Such imperfections are normally corrected by shifting any of the objects nearest the wire.



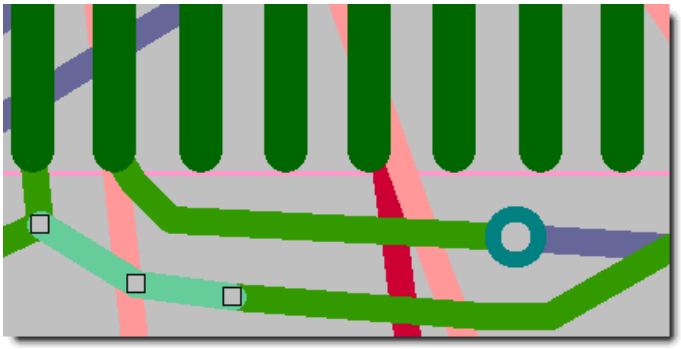
Corrected: the via has been moved

Another possible solution is to add an object, such as a routing keepout, in close proximity to the miscalculated wire.



Corrected: a routing keepout has been added

Finally, another method is to correct the wire manually and then fix it (or fix just the corrected segment of it).



Corrected: manually edited and fixed

Choice of "desired" spacing

The desired spacing for all rules should be set higher than the minimal value.

Peculiarities of pad matching

At the placement stage, if there are identical pads on the opposite sides of the board, it is important that their positions match exactly. This will make the autorouter's job much easier.

Peculiarities of using DRC during routing

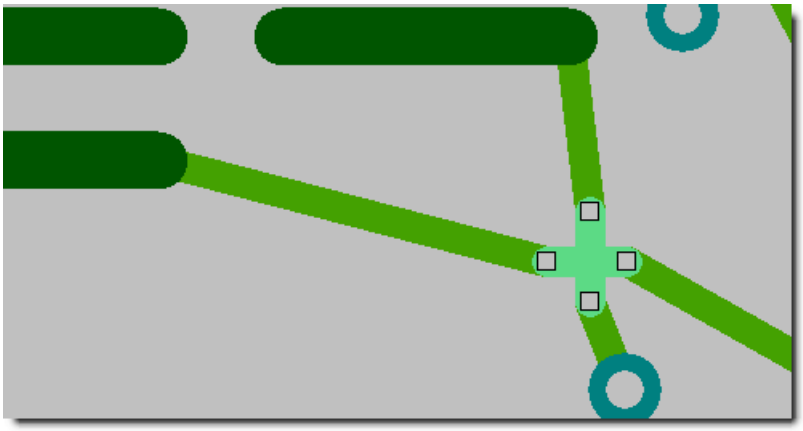
If the source `.pcb` file and the project that is opened have different design constraints, then connection routing uses the constraints set for the project.

Peculiarities of using DRC after routing

The autorouter in **TopoR** routes all connections, even where this causes constraint violations. Most violations are eliminated by procedures for automatic via shifting (or component shifting) and wire path optimization.

Ways to fix crossed wires

In the presence of some errors (such as when wires from different nets cross topologically or when wires overlap pads or vias) automatic procedures cannot be launched. If you really need to connect different nets, you can do it in several ways. At the connection spot, cross two fixed wires or two fixed copper areas or a fixed copper area and a fixed wire.



Nets are connected by fixing crossed elements

Manual control of optimization progress

You should not stop optimization until the value in the Round column of the table has reached 5–10.

While **TopoR** is frequently coming up with new variants, stopping it is not recommended. The rightmost column in the variant table shows the ratio of the total wire length to the number of vias. Practice shows that good results are achieved at the ratio of 25mm/via (1000mil/via). When there have not been any new variants in a long time, the program recommends stopping the topology optimization. The **Autorouting may be stopped** message is shown in the variant view.

Specifics of working with copper areas

The copper line should be set to be at least as wide as the wires that connect to the copper.

After you have poured a copper area, it is recommended that you run the wire path optimization procedure.

Polygons should be poured at the final stage, after routing has been tweaked manually and errors have been corrected.

In a poured copper area, the autorouter does not place vias from other nets.

Automatic component and via moving does not work inside poured polygons.

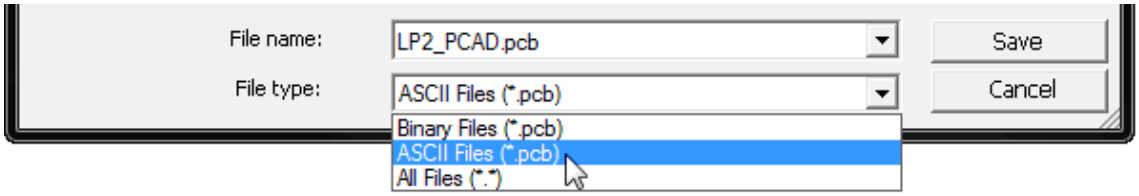
Working with Other CAD Software

This section describes the specifics of data interchange between **TopoR** and particular CAD systems through file export and import. The term import is used to mean getting the output of other software into **TopoR**, and export means the reverse.

P-CAD 2000 – 2006

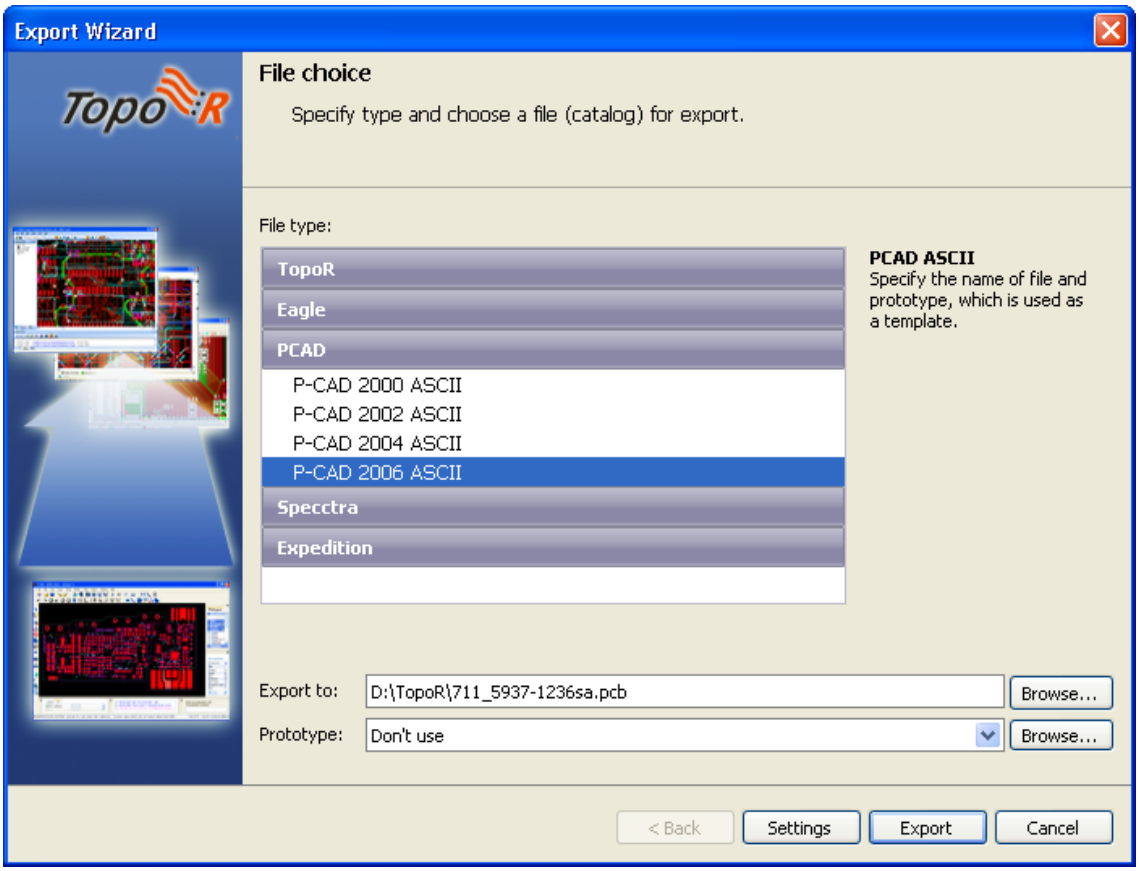
Import

P-CAD can save two types of files: binary **.pcb** and ASCII **.pcb**. **TopoR** works only with ASCII **.pcb**. Therefore, a project done in P-CAD should be saved in ASCII **.pcb** format before you import it into TopoR. In the P-CAD main menu, click **File ► Save As**, and in the file type dialog box, select **ASCII Files (*.pcb)** and click **Save**.



Export

If you need to bring the routing results back into P-CAD, export the file from **TopoR** to ASCII **.pcb** and specify which version of P-CAD it is compatible with. For example, for P-CAD 2006 the file selector in the export wizard looks like this:



You can also use the **.dsn** or **.ses** format for the conversion, but ASCII **.pcb** is a richer format, which makes it preferable.

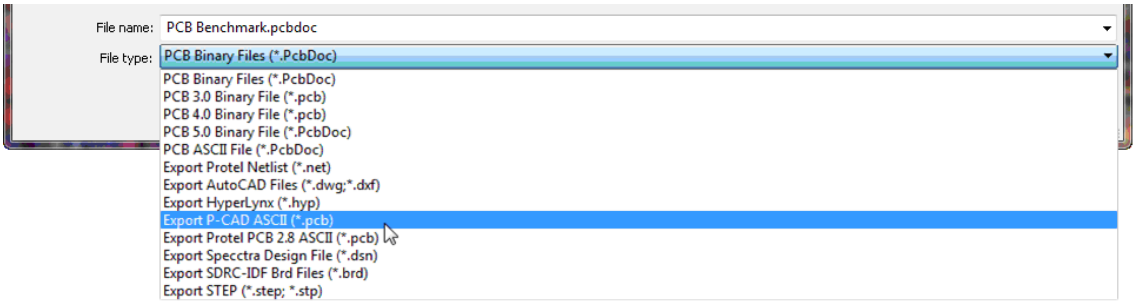
For example, if you exchange data through **.dsn**, then the pin/gate swap information is lost (P-CAD does not save it).

Altium Designer

Import

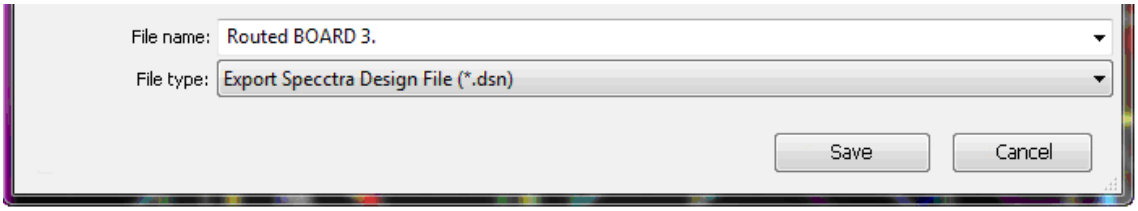
Before importing a project made in Altium Designer, save the project in ASCII PCB format:

1. Click **File ► Save As** (or **File ► Save Copy As**) in the Altium Designer main menu.
2. In the file type dialog box, select **Export P-CAD ASCII (*.pcb)**.
3. In the file name dialog box, change the extension from **.pcbdoc** to **.pcb** and click **Save**.

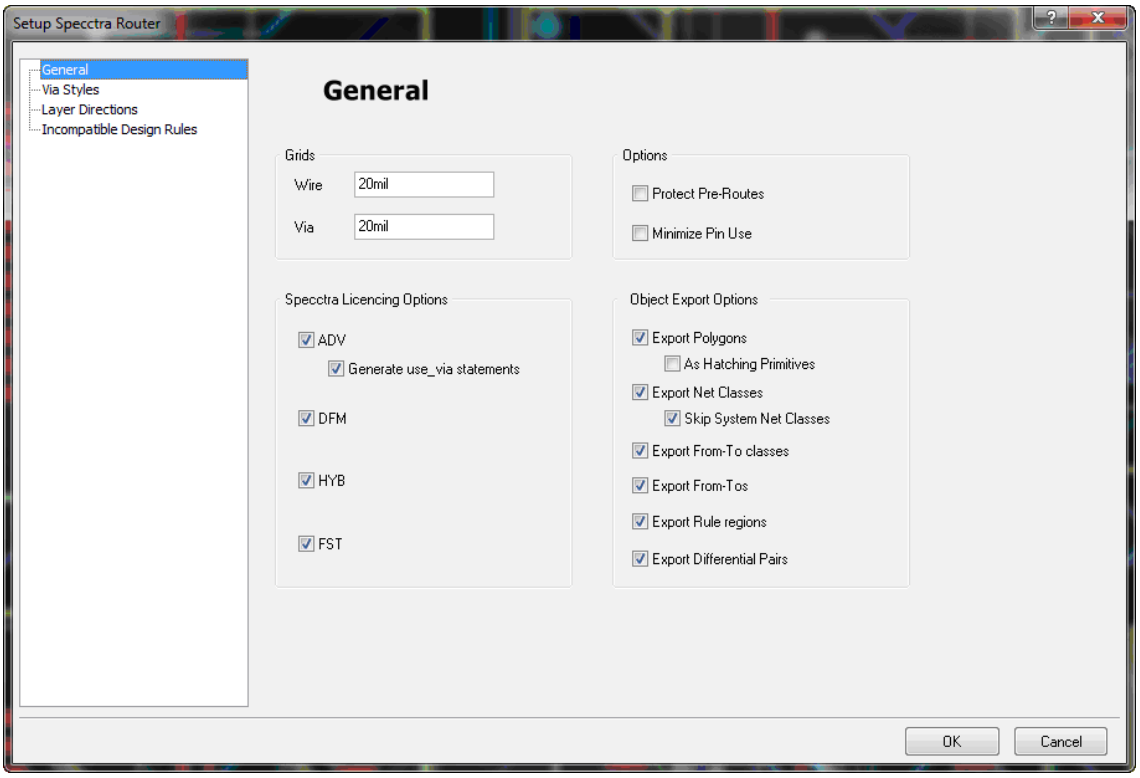


Alternatively, you can save a **.dsn** file:

1. Click **File ► Save As** (or **File ► Save Copy As**) in the Altium Designer main menu.
2. In the file type dialog box, select **Export Specctra Design File (*.dsn)**.
3. In the file name dialog box, change the extension **.pcbdoc** to **.dsn** and click **Save**.



A configuration dialog box is displayed. Go to the **General** tab and make sure the settings are configured as in the screenshot below (except the Grids section). Do not change the settings on other tabs.

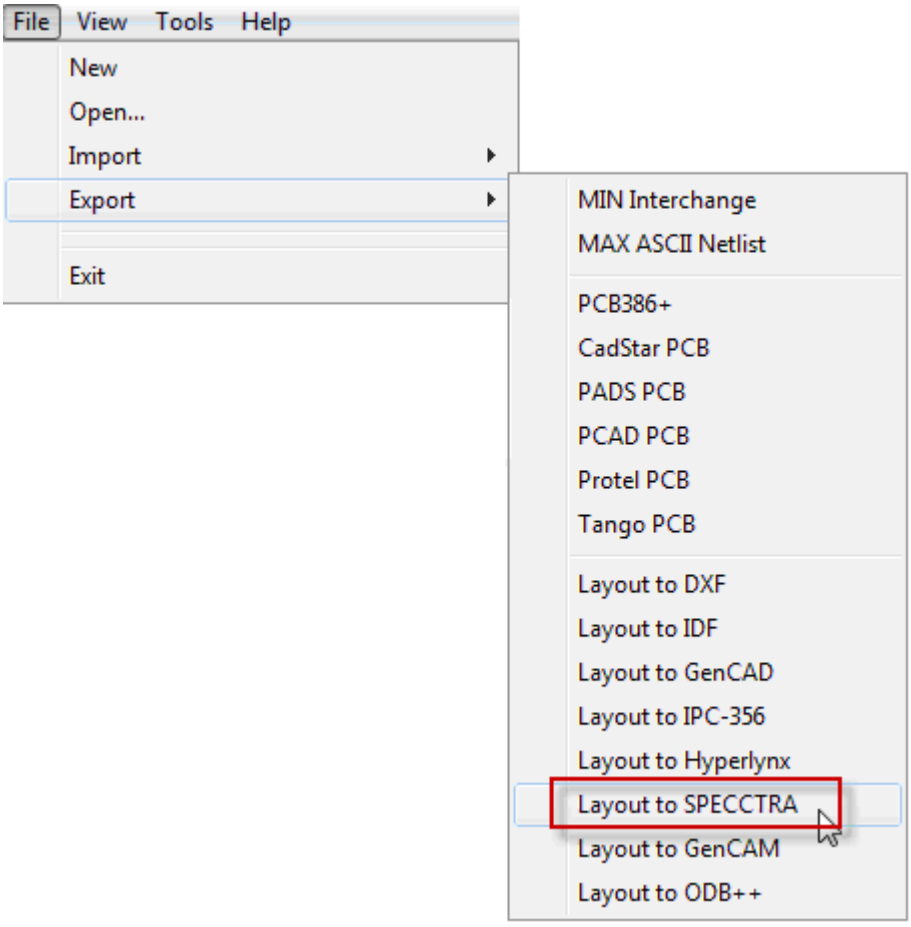


The board outline should be drawn in Altium Designer using lines (arcs) on the **Keep-out Layer**, not specified as a room.

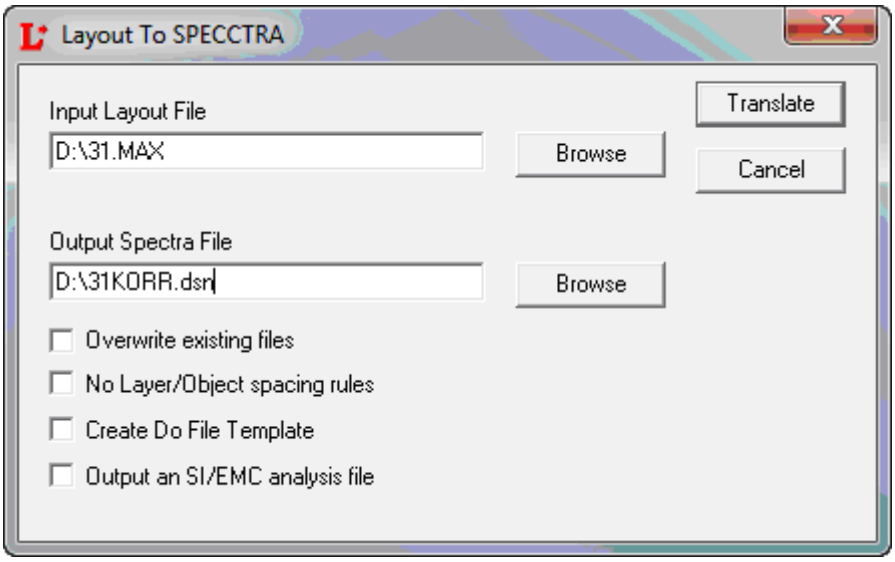
Cadence OrCad (Layout)

`Import`

To transfer a project from OrCad to **TopoR**, export it as a **`.dsn`** file. For that, first launch OrCad Layout or OrCad Layout Plus without opening the project. In the OrCad main menu, click **File ► Export ► Layout to SPECCTRA**.



In the **Layout to SPECCTRA** dialog box that opens, specify the input and output files correctly:



Use the **Browse** button next to the **Input Layout File** box to specify the **`*.max`** project file. In the **Output Spectra File** box, change the extension from **`.cct`** to **`.dsn`**. Then click **Translate**.

The resulting file is ready for editing in **TopoR**. However, the data in the file is not complete (see DSN Format Specifics for details), so you should note a few caveats.

1. Information about whether or not components are fixed is lost. You need to fix them in **TopoR** again.
2. If the placement of a component overlaps a polygon, then this polygon is stored separately from the component body. Such components should be fixed and never moved; alternatively, you can place a pad in place of a polygon when placing an overlapping component.
3. This is how polygons, keepouts and other auxiliary parts (classified as obstacles in OrCad) are translated:

Board outline: **Obstacle ► Board outline**.

Any number of board outlines can be output for use in **TopoR**, and a bounding outline is added that encompasses all of them; you can safely remove this outline.

Polygon: **Obstacle ► Copper pour.**

Retrievable by **TopoR** only on backup layers.

Unperforated polygon: **Obstacle ► Copper area.**

Retrievable by **TopoR**. Conversely, to transfer a **copper pour** from OrCad to **TopoR** as a polygon, change its property to **Copper area**, and later in **TopoR** open the copper area properties, go to the **General** tab, click **Select copper** and click the **Repour** button.

Perforation in a polygon: **Obstacle ► Anti-copper**.

Not retrievable by **TopoR**.

In OrCad, change the **Anti-copper** property to **Board outline**. Then in **TopoR**, repour the polygon to create an island within the pretend board outline. Next, click this island to select it, and in the properties of the island click **Convert into copper**. This converts the island to a separate polygon. After that, you can delete the selected island and the board outline. The result is a perforated polygon.

Keepout: **Obstacle ► Route keepout** or **Route-via keepout**.

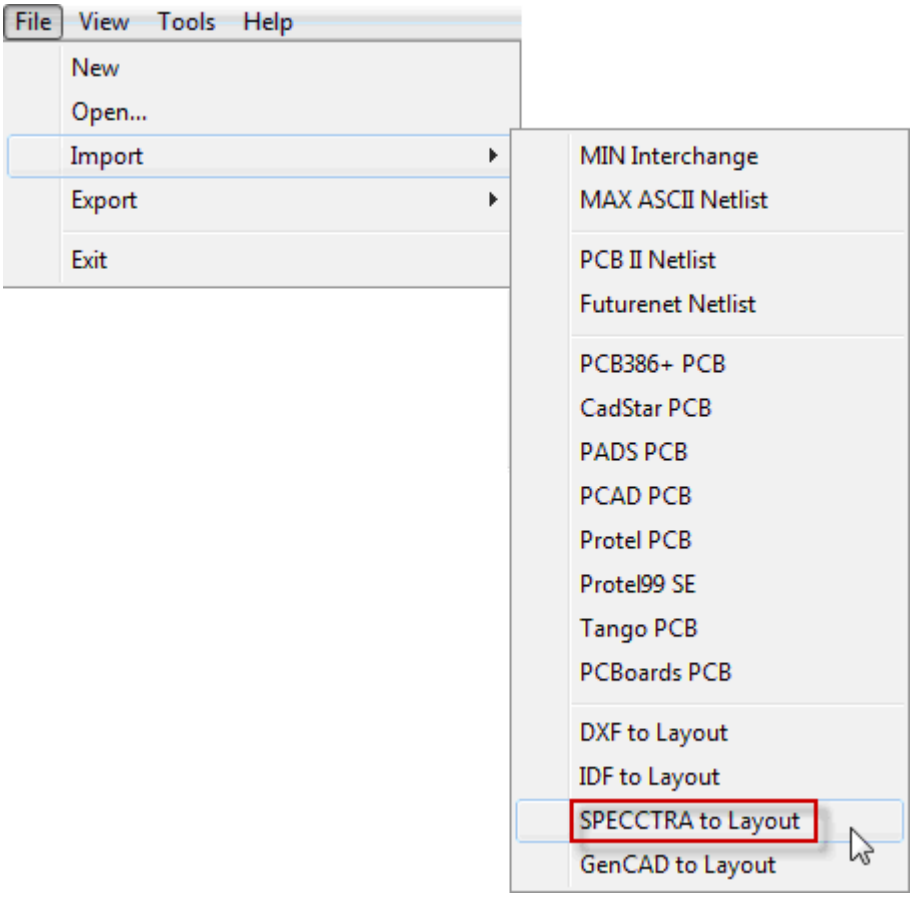
Retrievable by **TopoR**. The OrCad **Global layer** setting is treated as a keepout for all routing layers in **TopoR**; the Top layer corresponds to the Top layer, etc.

Other types of **Obstacle** (**Insertion outline, Detail, Free track**) are not retrievable by **TopoR**.

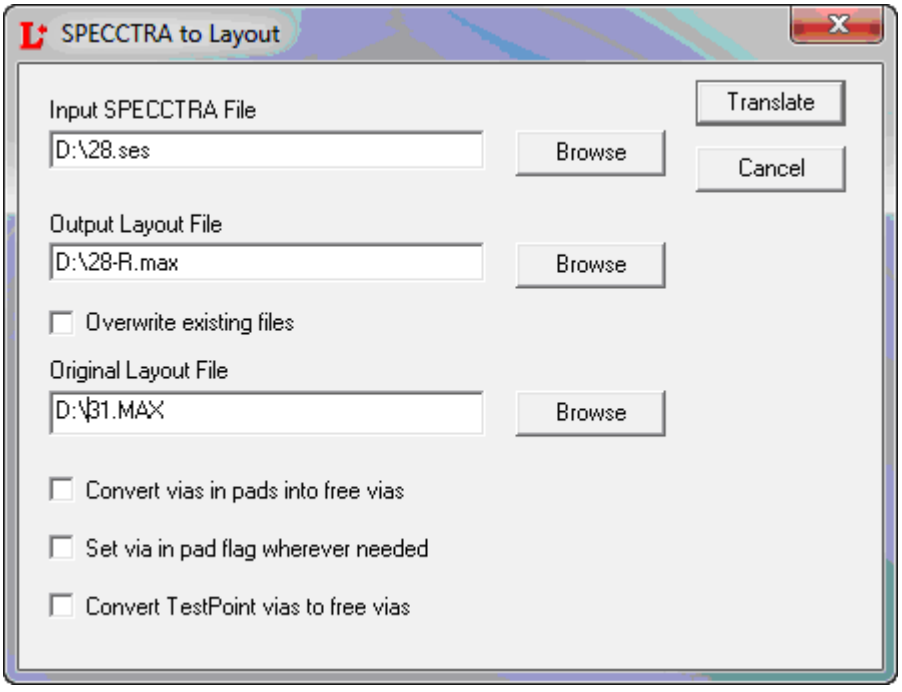
If there are polygons on the board, the recommended method is to use two OrCad files. In the first file, make all the necessary adjustments. Save the file under a different name and prepare the polygons and other primitives for export to **TopoR** by changing their properties. Use the second file to create a **`.dsn`** file. After you have finished working with the file in **TopoR**, import the resulting **`.ses`** file into OrCad using the first file with the “correct” polygons as an **Original Layout File**. This spares you the effort of readjusting the properties of the primitives.

`Export`

To bring the **`.ses`** file back into OrCad from **TopoR**, first launch OrCad Layout or OrCad Layout Plus without opening the project. Click **File ► Import ► SPECCTRA to Layout** in the main menu.



In the **SPECCTRA to Layout** dialog box that opens, specify the input and output files correctly:



Use the **Browse** button next to the **Input SPECCTRA File** box to specify the **`.ses`** file. In the **Output Layout File** box, verify the name of the resulting **`.max`** file. In the **Original Layout File** box, specify the original **`.max`** file. Then click **Translate**.

Some information is not written to the **.ses** file, so certain changes are not retrievable by OrCad. The following data cannot be retrieved:

1. Newly created layers. OrCad will lose the wires routed on such layers
2. Board layout changes.
3. Changes to pads and vias (or addition of new types).
4. Via matrices created in **TopoR**. In fact, the matrix is written to the **.ses** file, but OrCad deletes all vias but one as duplicates. You have the option to create this matrix in OrCad using the Free Via feature. **TopoR** treats such vias as single-component FreePads and makes sure they are not lost in conversion.

Interchange

When creating a library in OrCad, pay attention to pad coordinates (or verify them in the project). Even though OrCad lets you have five or more decimal places in coordinate values, you should not use more than four digits after the decimal point, because that is the precision used in **TopoR**. If there are five or more digits (a zero in the fifth decimal place is acceptable), a rounding error will occur during data transfer into OrCad after routing in **TopoR**. OrCad will consider some of the nets unrouted, even though they may be routed. You should not ignore such errors, because you might miss an actual unrouted net. In addition, if you export the project for adjustments in **TopoR**, all of the "unrouted" wires will not be written to the **.dsn** file and will be lost.

To check and correct pad coordinates in an existing OrCad project, consider using the **Tool ► Footprint ► Select from Spreadsheet** menu item. Before this, make sure precision is configured correctly: In **Options ► System Settings**, the **Display Resolution** option should be set to 0.00001. Otherwise, fewer decimal places may be displayed than there really are.

Other considerations:

1. Do not use Russian characters and spaces in file names (OrCad will fail to read them).

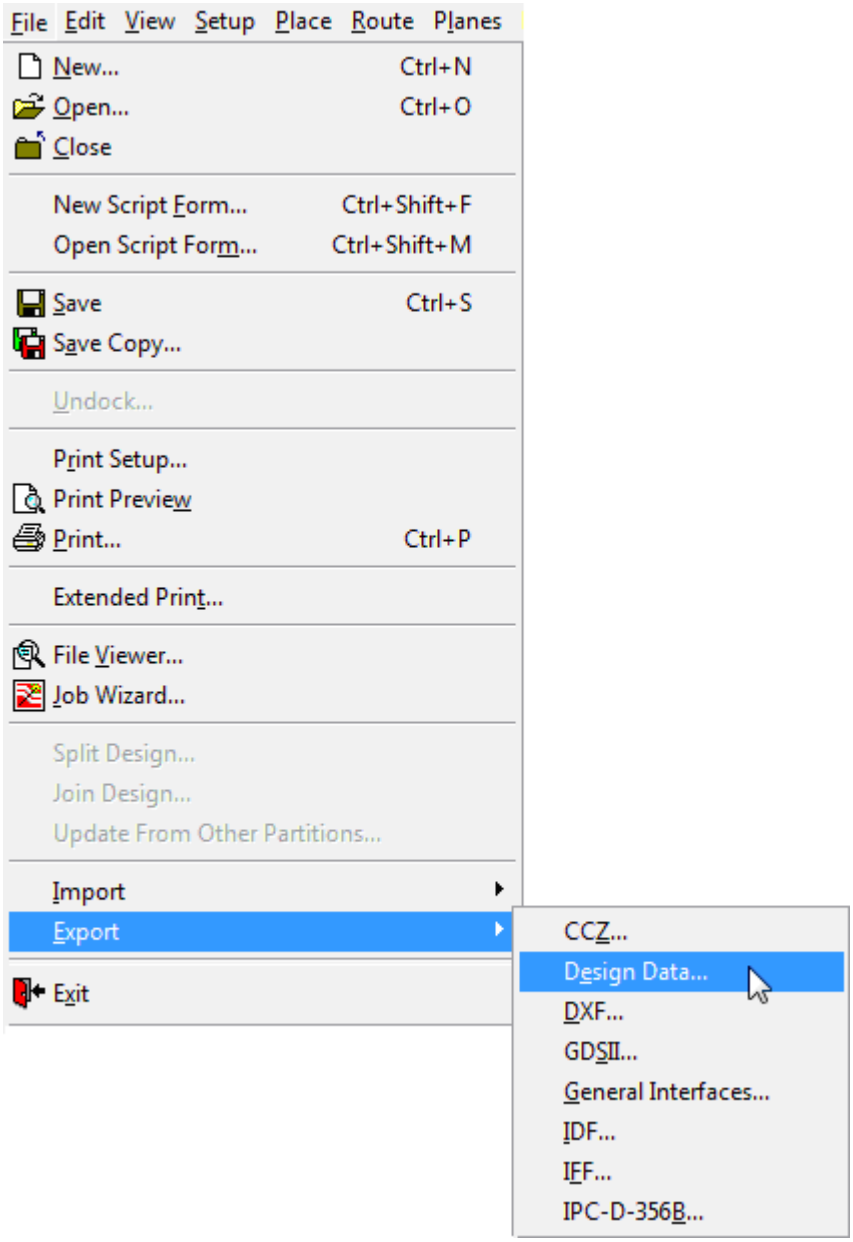
2. When files are converted back and forth, net width and clearance need to be reconfigured in **TopoR**. Therefore, you should configure them in OrCad in the first place. To specify net clearance, go to **Options ►Global Spacing**. The clearance set there will be treated as minimal by **TopoR**. For the nominal clearance, either leave the value set automatically by **TopoR**, or change it if necessary. Different clearance values are supported for different board layers (this can be configured in OrCad). To specify net width, go to **View Spreadsheet ►Nets**. The **Conn Weight** value is used. **TopoR** will build nets by group and set a width for each group. This net width value will be treated as minimal in **TopoR**.

3. If a pad is not connected to any net in an OrCad design, then a keepout for all layers is written to the **.dsn** file in its place. The diameter of the keepout is equal to the diameter of the pad. **TopoR** shows online DRC errors in such areas, but final DRC shows no errors. You can either ignore such errors or select the keepout outline (it is obstructed by the pad) and delete it.

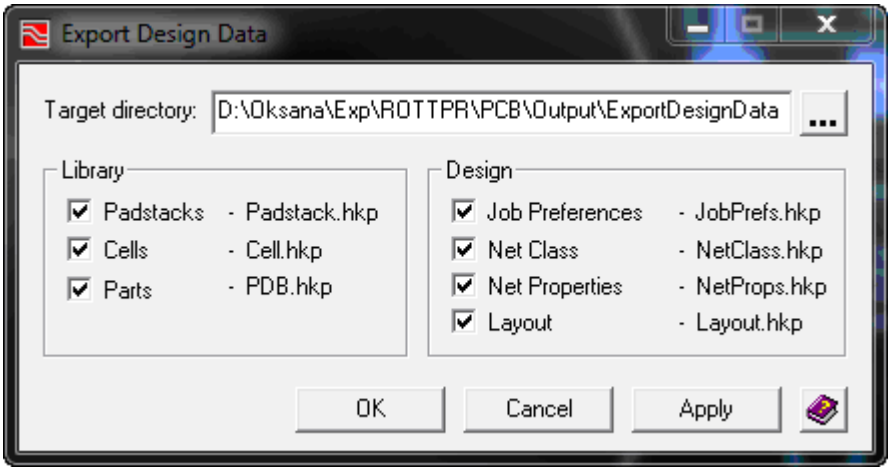
Mentor Graphics Expedition

`Import`

A project done in Expedition is exported as a group of **`.hkp`** files. For that, click **File ► Export ► Design Data** in the main menu.



A dialog box is displayed where you should select the check boxes next to all the available files, select the directory to export them to, and click **OK**.

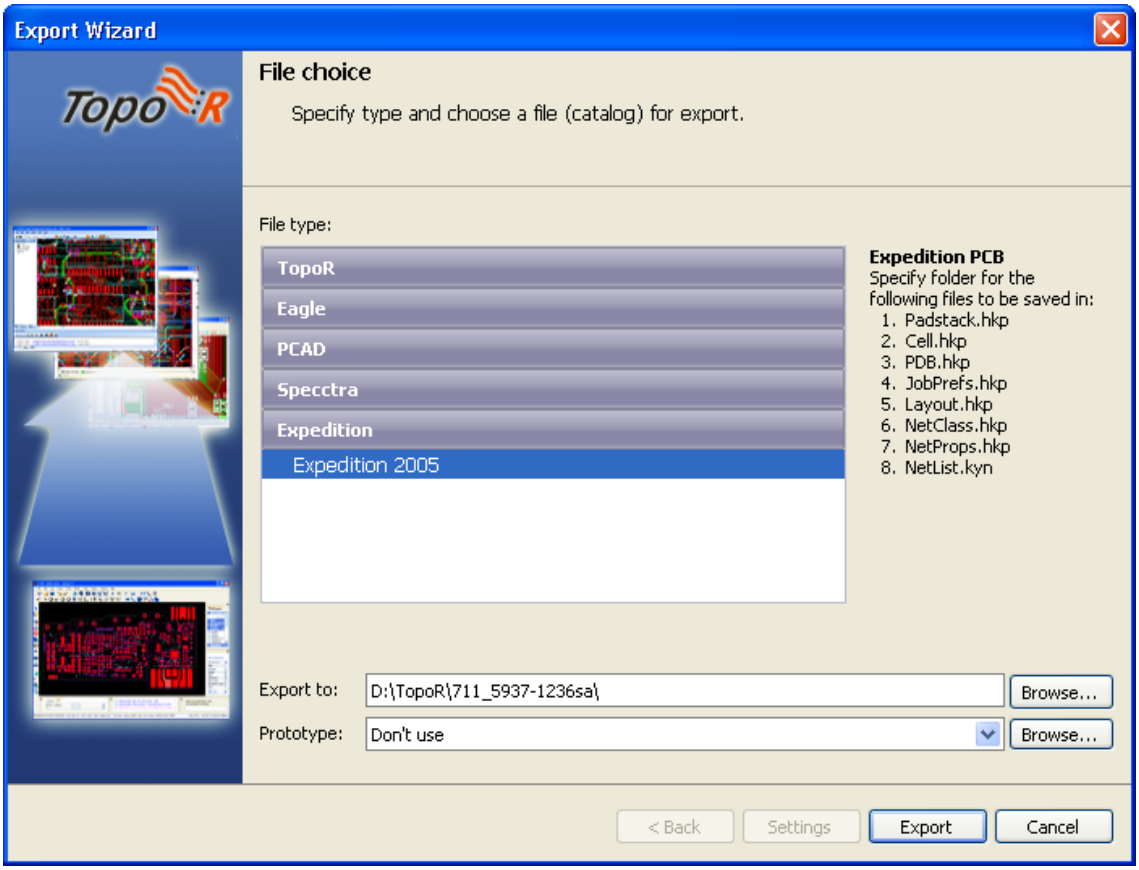


In older versions of Expedition (2007.2), **`.hkp`** files were written in plain text format and could therefore be easily imported into **TopoR**. In newer versions (2007.8) they are written in compressed binary format and need to be decoded before import. A converter utility for this purpose is shipped with Expedition. In a default installation, the path to the converter is as follows:

`C:\MentorGraphics8\2007.8EE\SDD_HOME\common\win32\_bin\DataConvert.exe`

`Export`

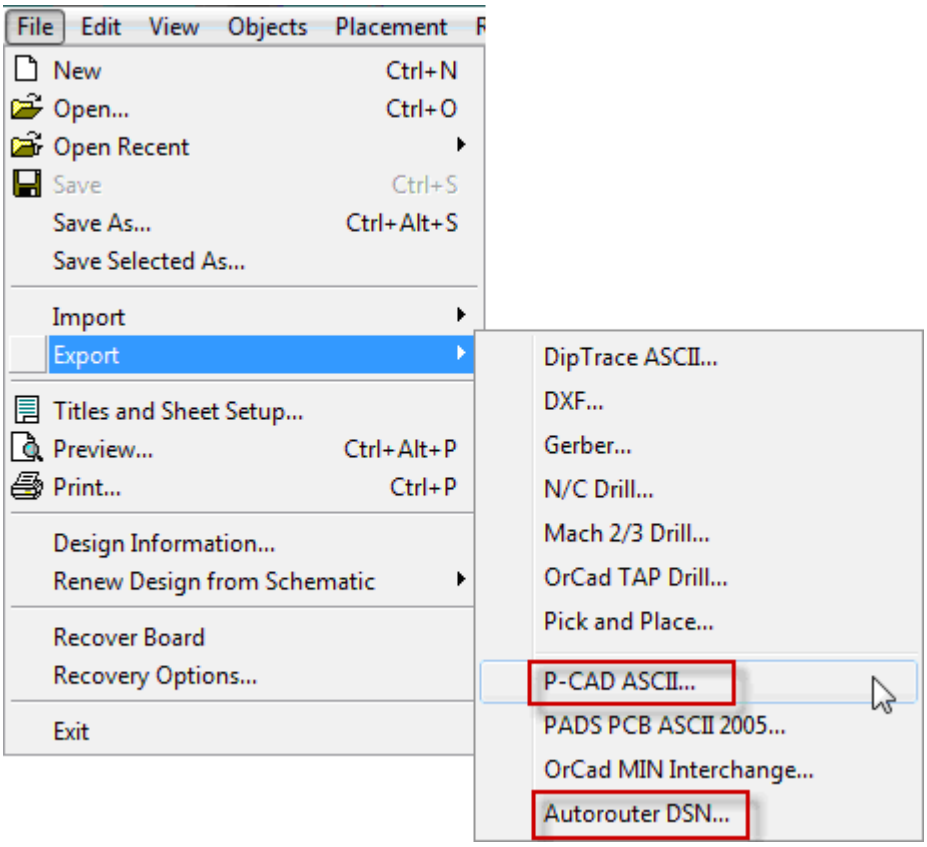
To export data to Mentor Graphics Expedition, click **File ► Export ► Expedition ► Expedition 2005** in the main menu.



DipTrace

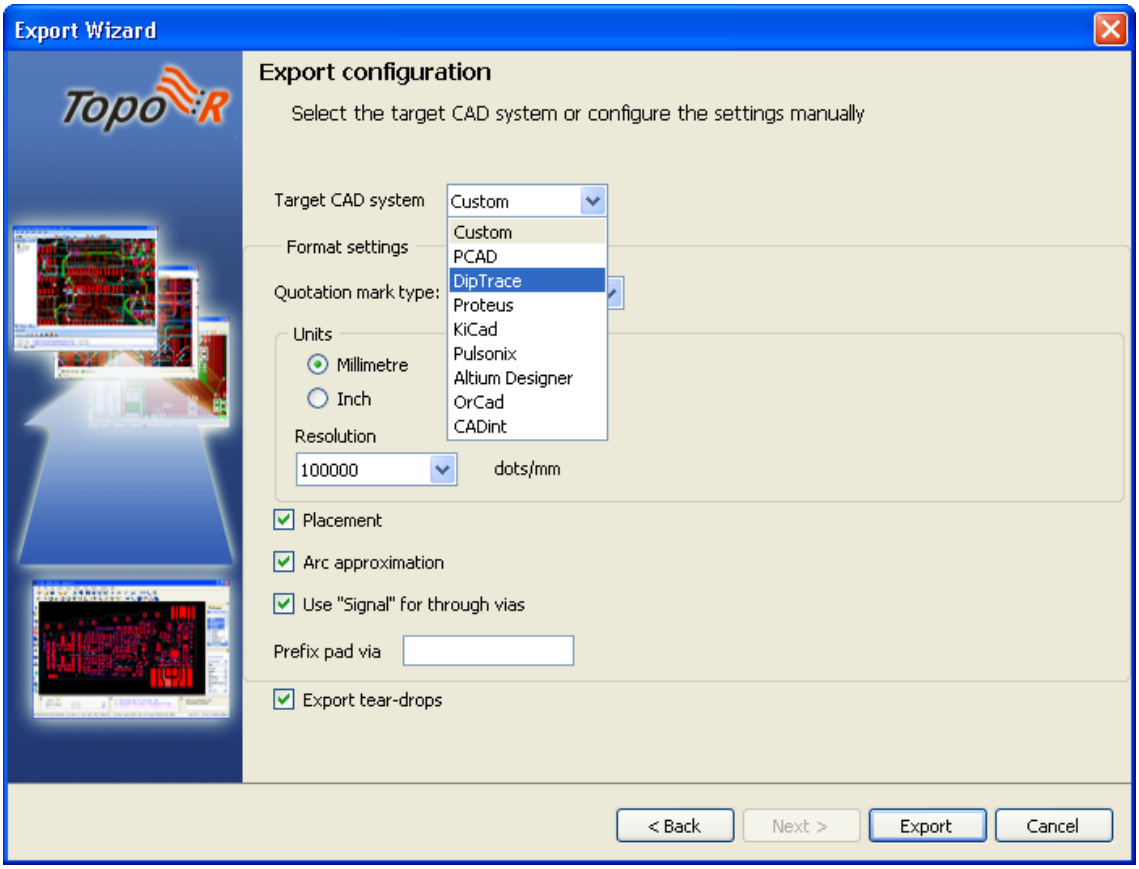
Import

Before importing a project made in DipTrace, save the project in ASCII format (extension **.pcb**) or as a **.dsn** file. In the main menu, click **File ► Export ► P-CAD ASCII** or **File ► Export ► Autorouter DSN** and click **Save**.

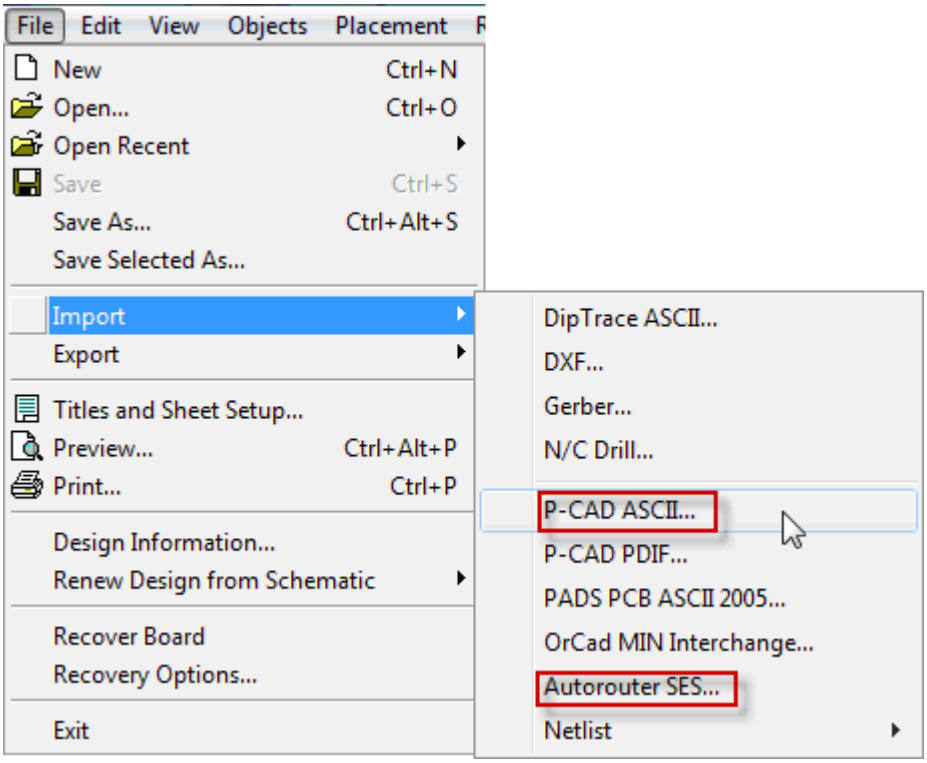


Export

After routing in TopoR, you can use the file in DipTrace again. For that, export it as a **.pcb** file from **TopoR** (select P-CAD in the export wizard). Alternatively, if you imported a **.dsn** file, export it as a **.ses** (Specctra) file, selecting DipTrace in the export settings.



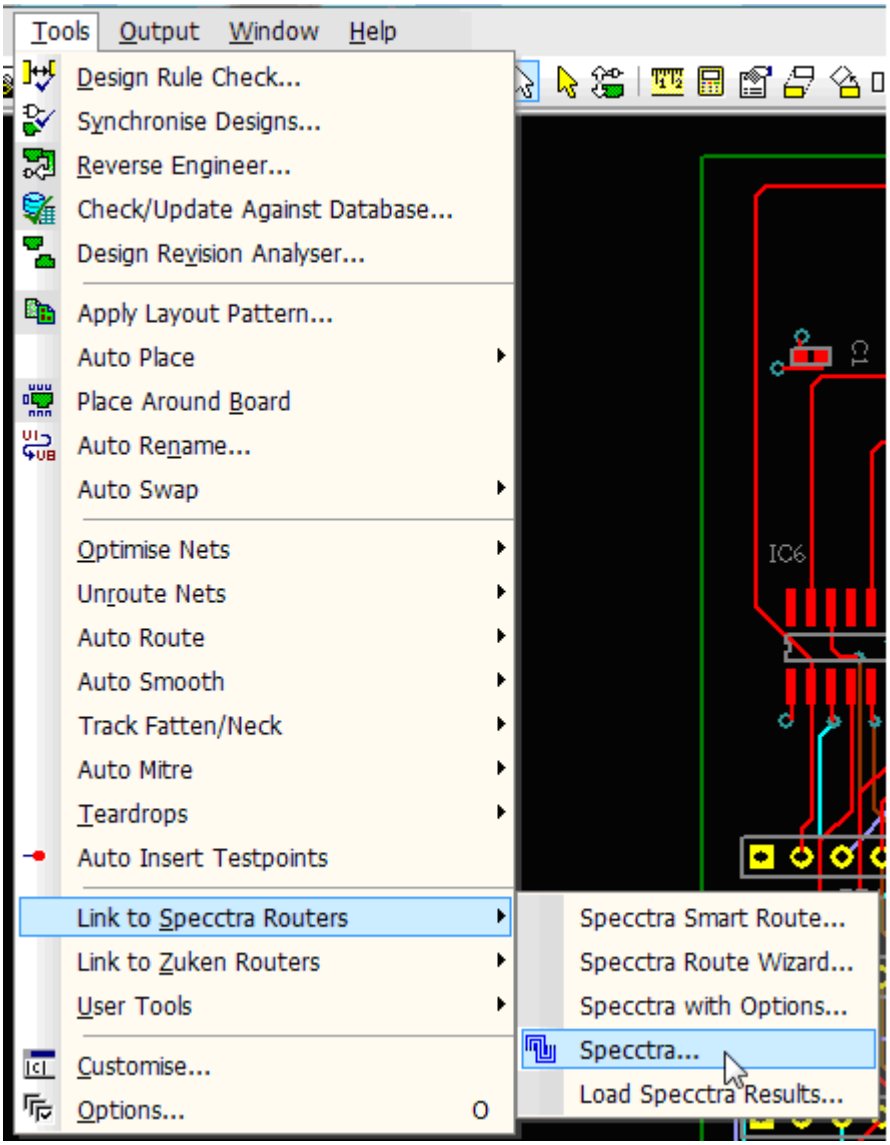
The resulting file can be imported into DipTrace by clicking **File ► Import ► P-CAD ASCII** or, in the case of **`.dsn`**, **Autorouter SES**.



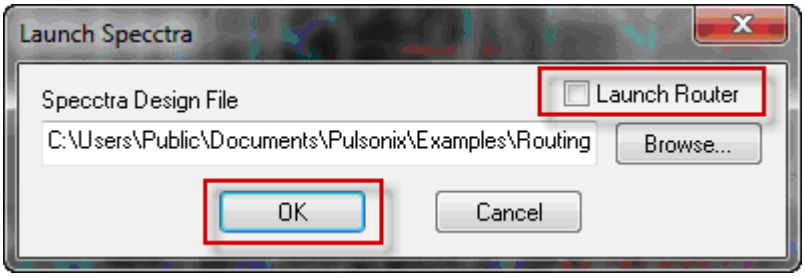
Pulsonix

Import

Transfer of files from Pulsonix to **TopoR** is best done using the **.dsn/.ses** format. For that, open the board in Pulsonix and click **Tools ► Link to Specctra Routers ► Specctra** in the main menu.

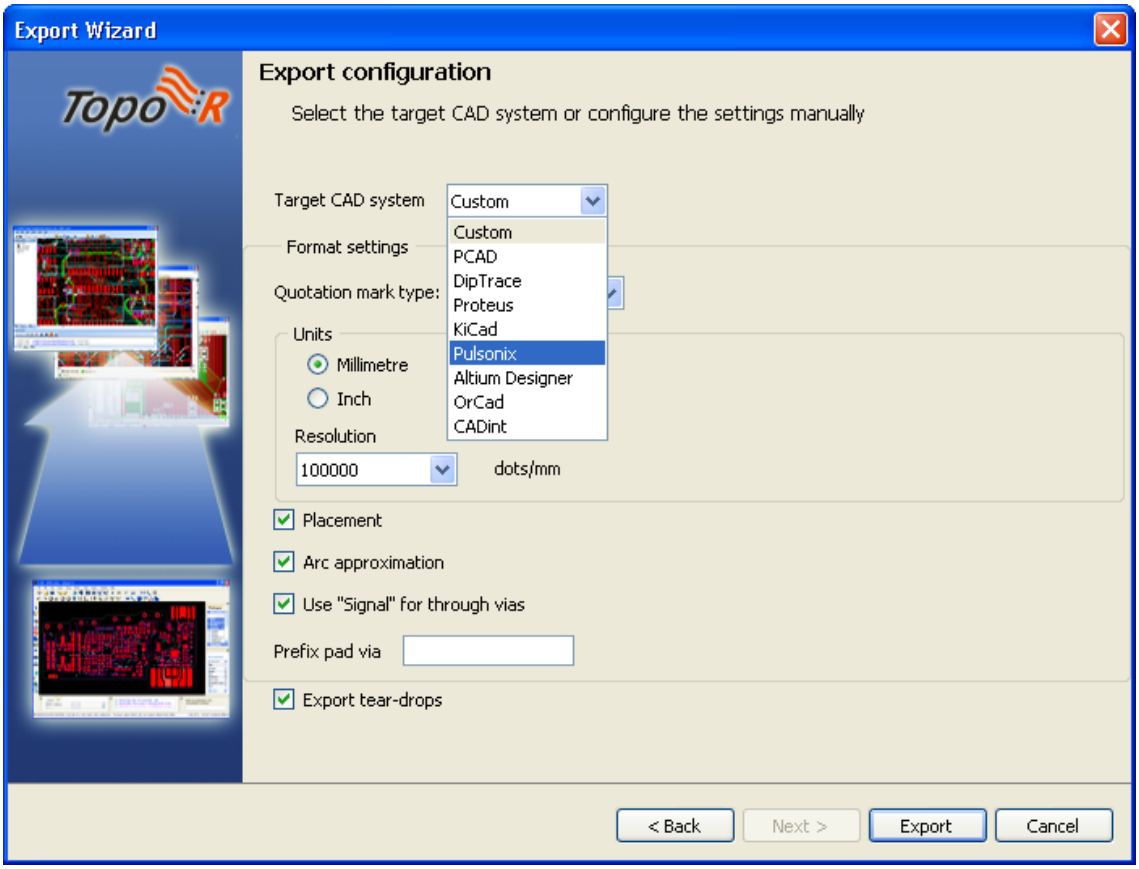


In the dialog box that opens, specify where to save the **.dsn** file. Make sure the **Launch Router** option is turned off.

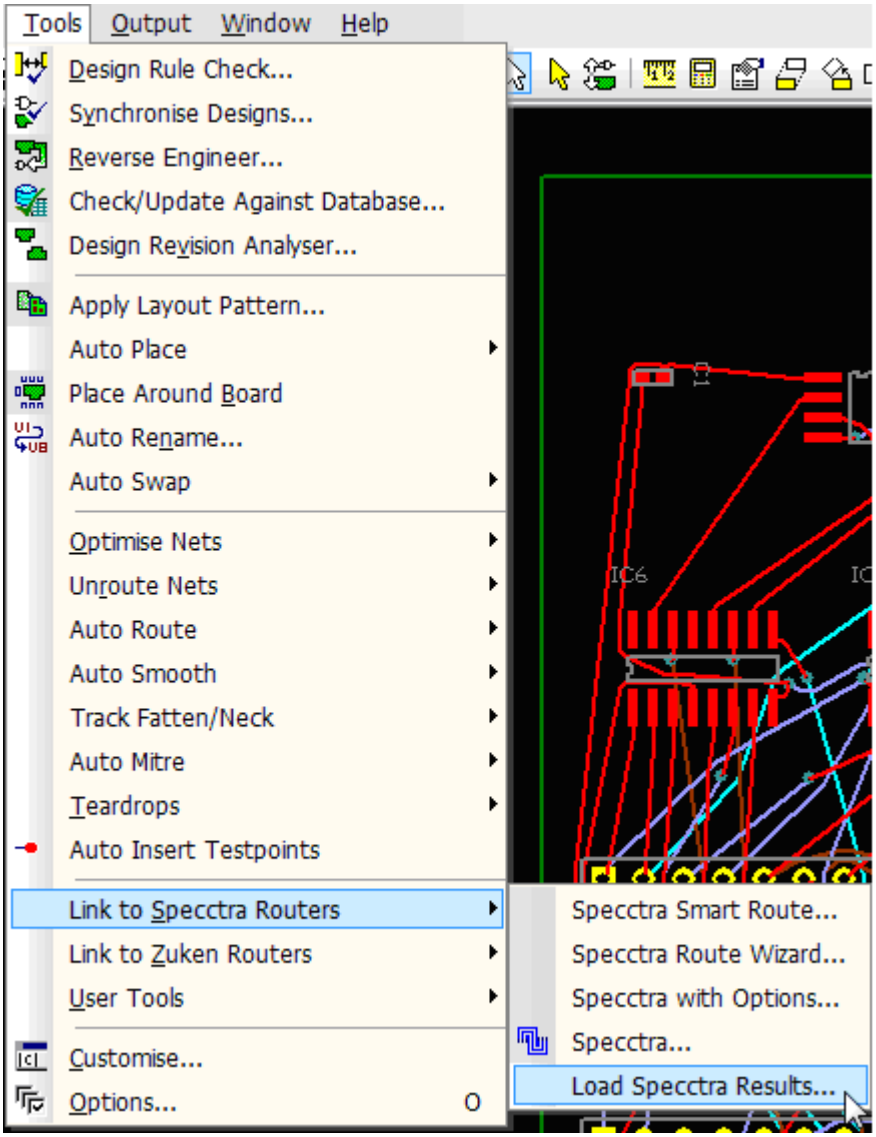


Export

To bring the file back into Pulsonix, export it from **TopoR** as a **.ses** (Specctra) file, specifying Pulsonix as the target CAD system in the export wizard settings.



To append the resulting **`.ses`** file to the original design, click **Tools ► Link to Specctra Routers ► Load Specctra Results** in the Pulsonix main menu.

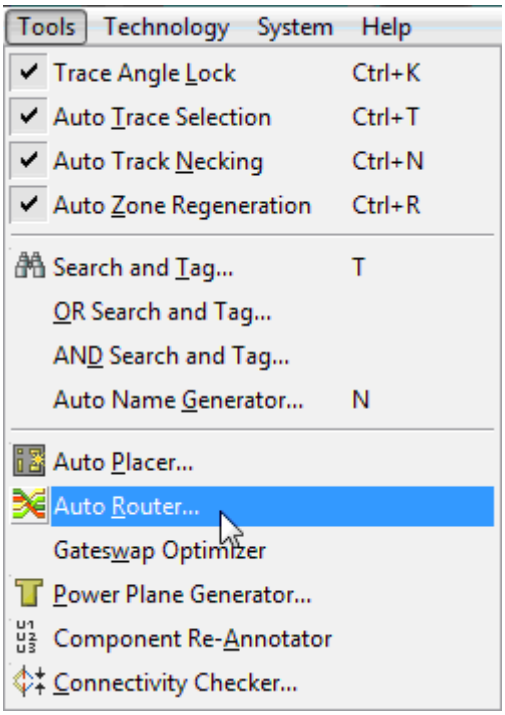


Pulsonix does not read component coordinate tweaks from the file, so do not move components around in **TopoR**.

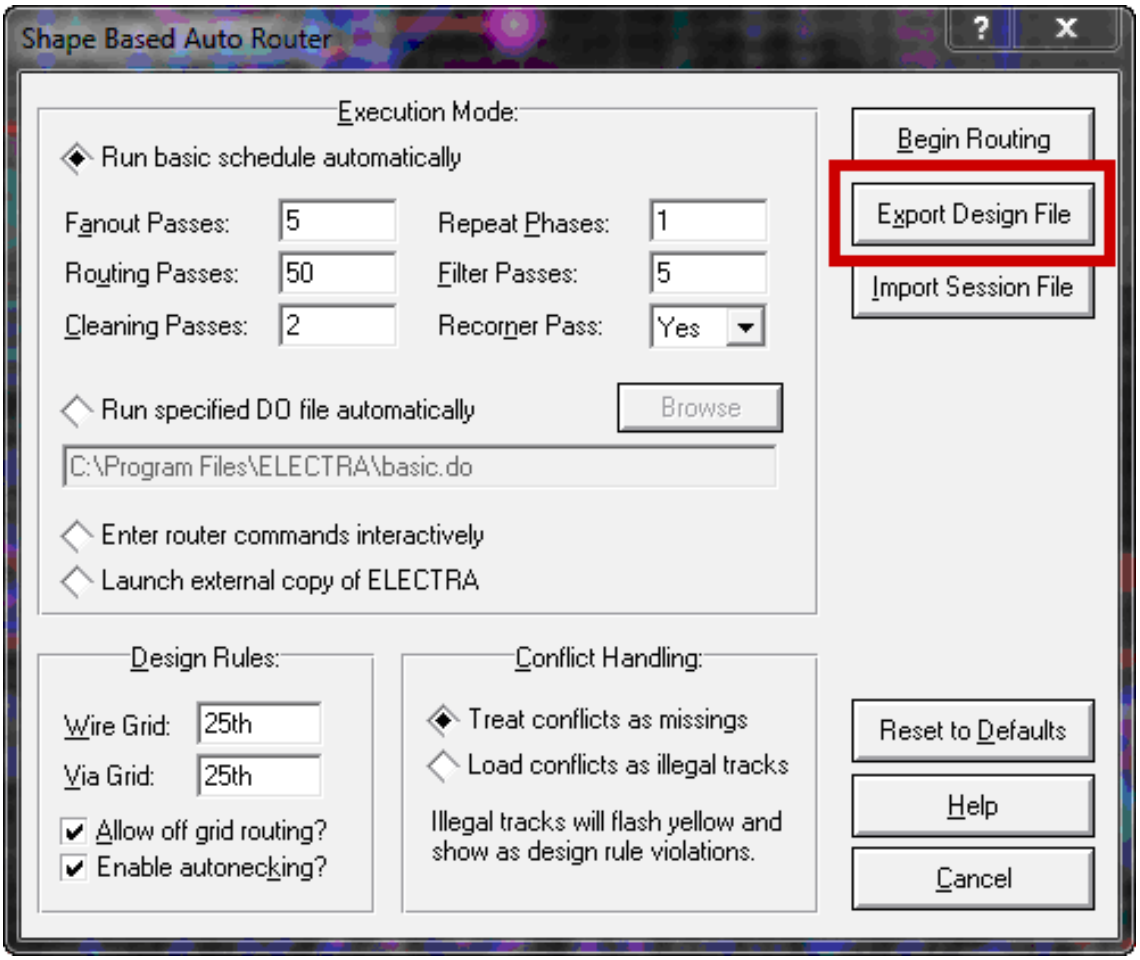
Proteus ARES

Import

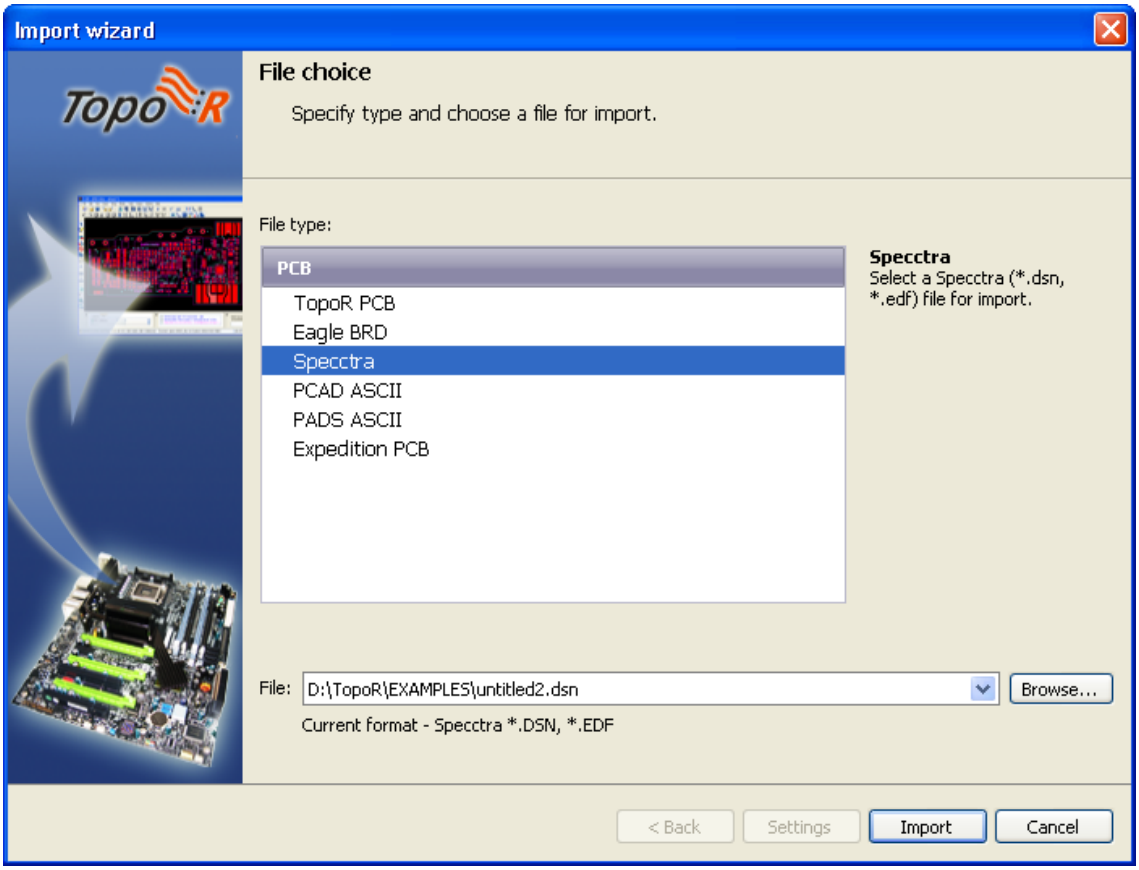
To export files from ARES, first click **Tools ► Auto Router** in the main menu.



Click **Export Design File**.

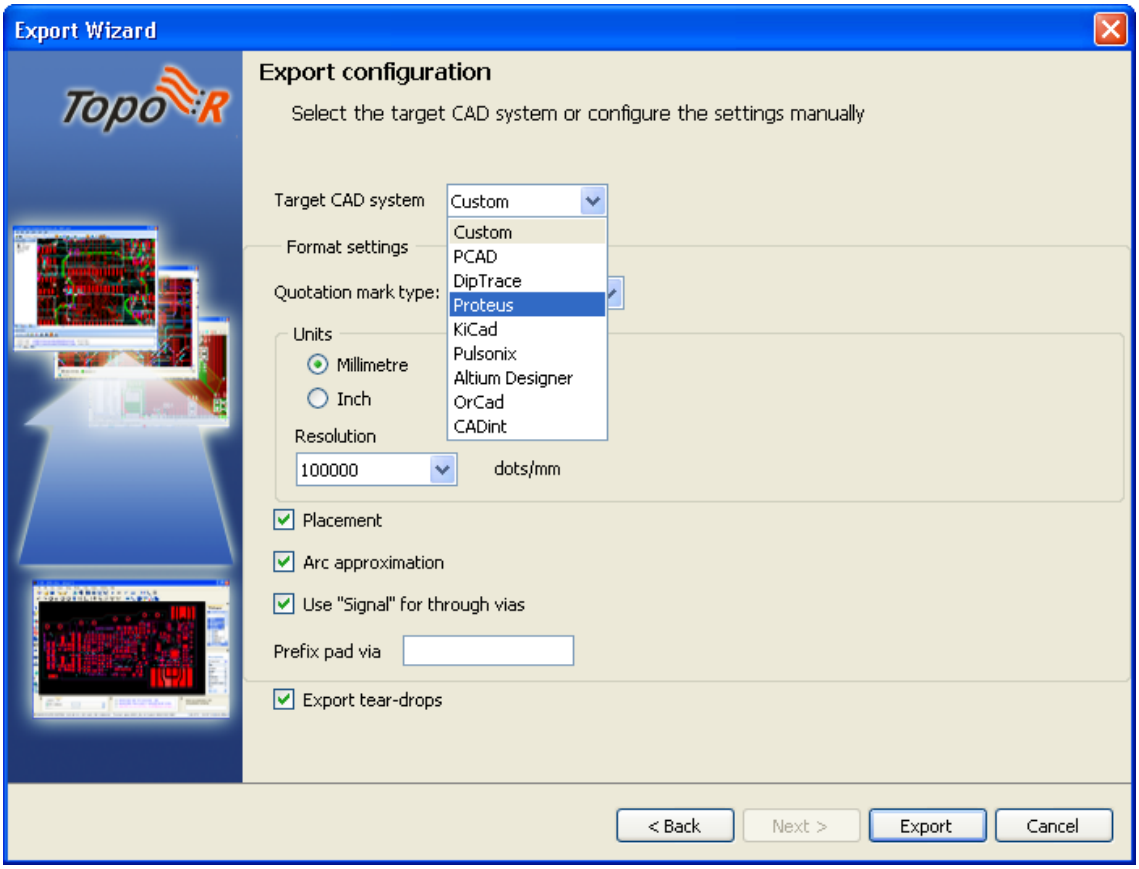


Save the file as **.edf**. To import a **.edf** file into **TopoR**, use the Specctra file type.

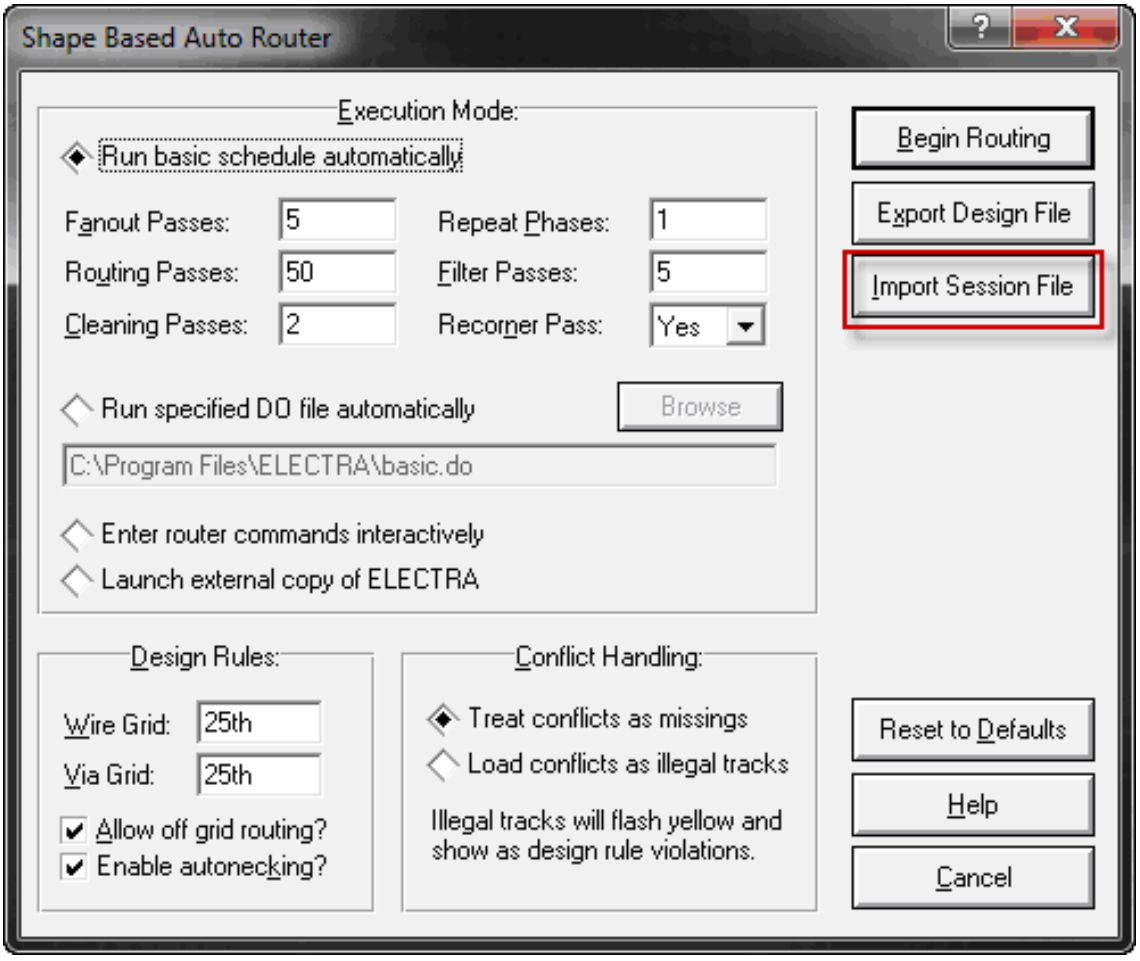


Export

To bring the routing results from **TopoR** back into Proteus, export the project from **TopoR** as a **.ses** (Specctra) file, click **Settings** and select Proteus as the target CAD system.



In ARES, click **Tools ► AutoRouter** in the main menu, and click **Import Session File**.



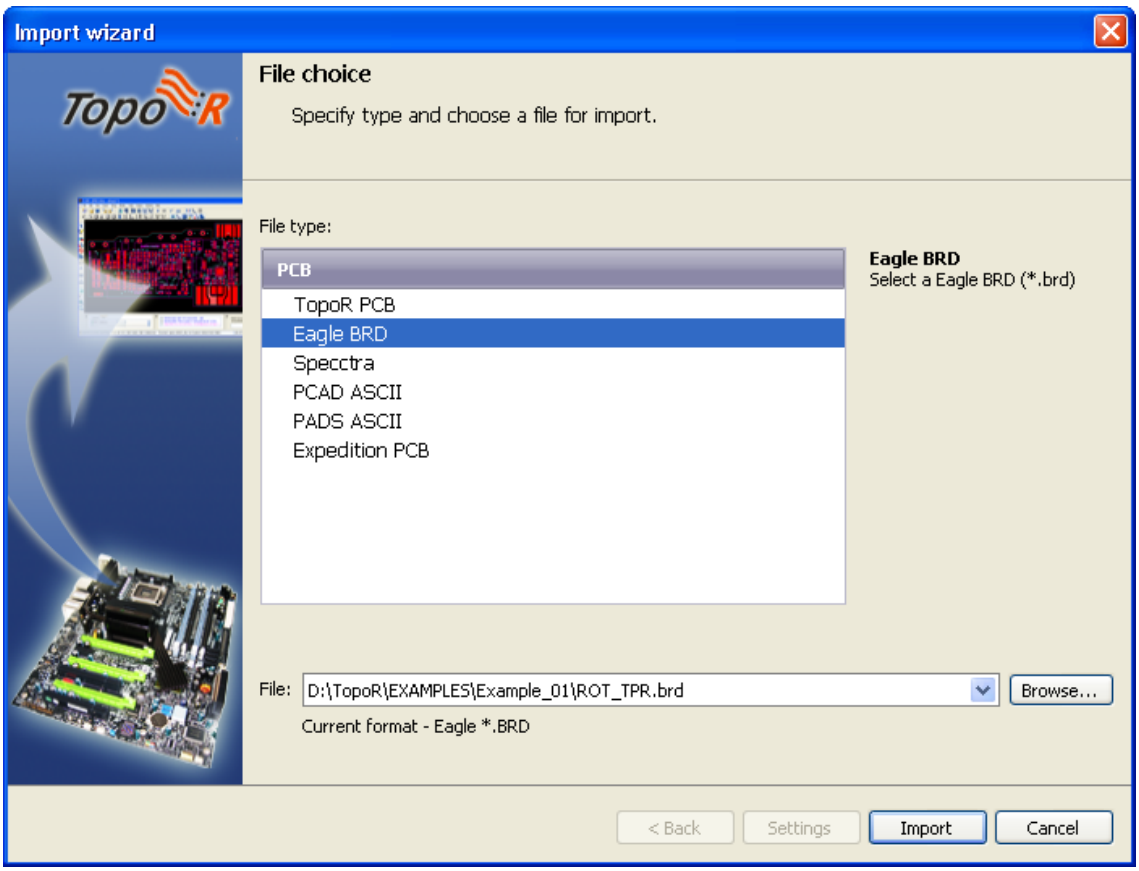
Load the **.ses** file.


Proteus does not read component coordinate tweaks from the **.ses** file, so do not move components around in **TopoR**.

Eagle

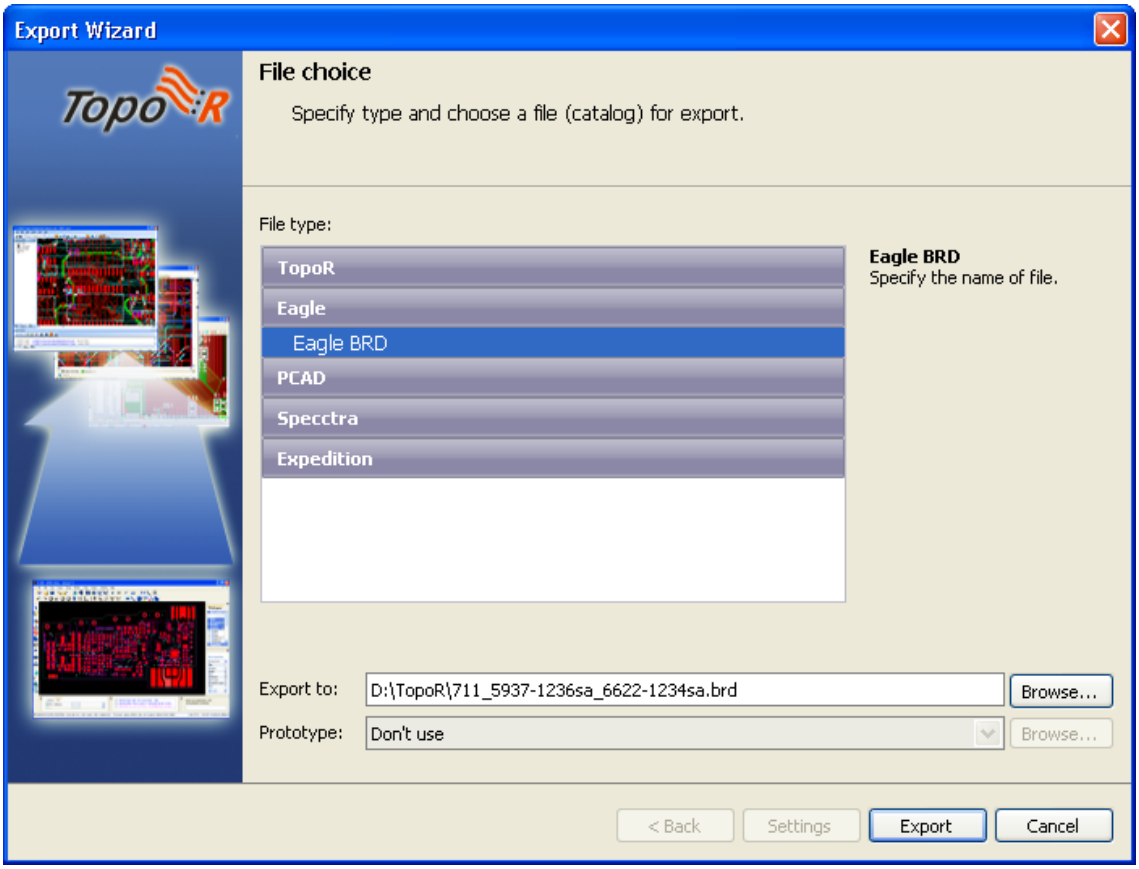
`Import`

Transfer of files from Eagle to **TopoR** is done using the **`.brd`** format. For import, click **File ► Import ► Eagle BRD** in the main menu.



`Export`

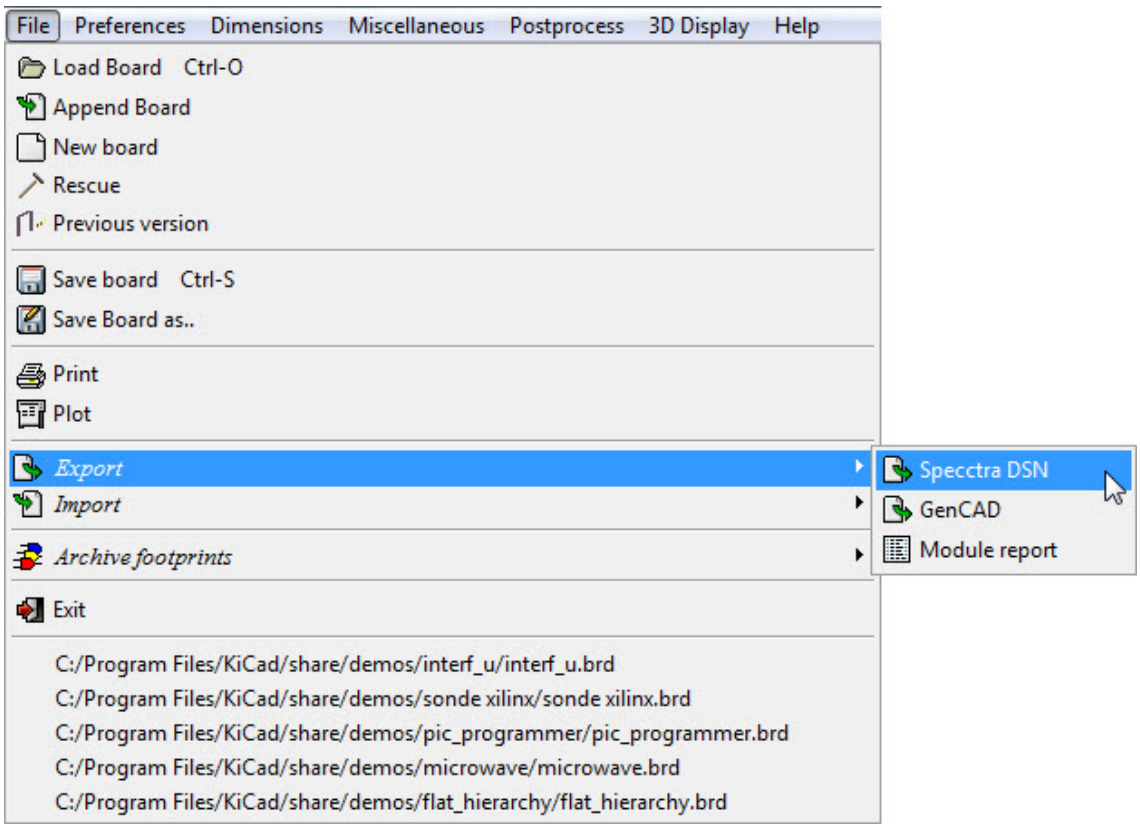
To export a file to Eagle, click **File ► Export ► Eagle ► Eagle BRD** in the main menu.


Export Wizard
TopoR
File choice
Specify type and choose a file (catalog) for export.
File type:
TopoR
Eagle
Eagle BRD
PCAD
Specctra
Expedition
Eagle BRD
Specify the name of file.
Export to:
D:\TopoR\711_5937-1236sa_6622-1234sa.brd
Browse...
Prototype:
Don't use
Browse...
< Back
Settings
Export
Cancel

KiCad

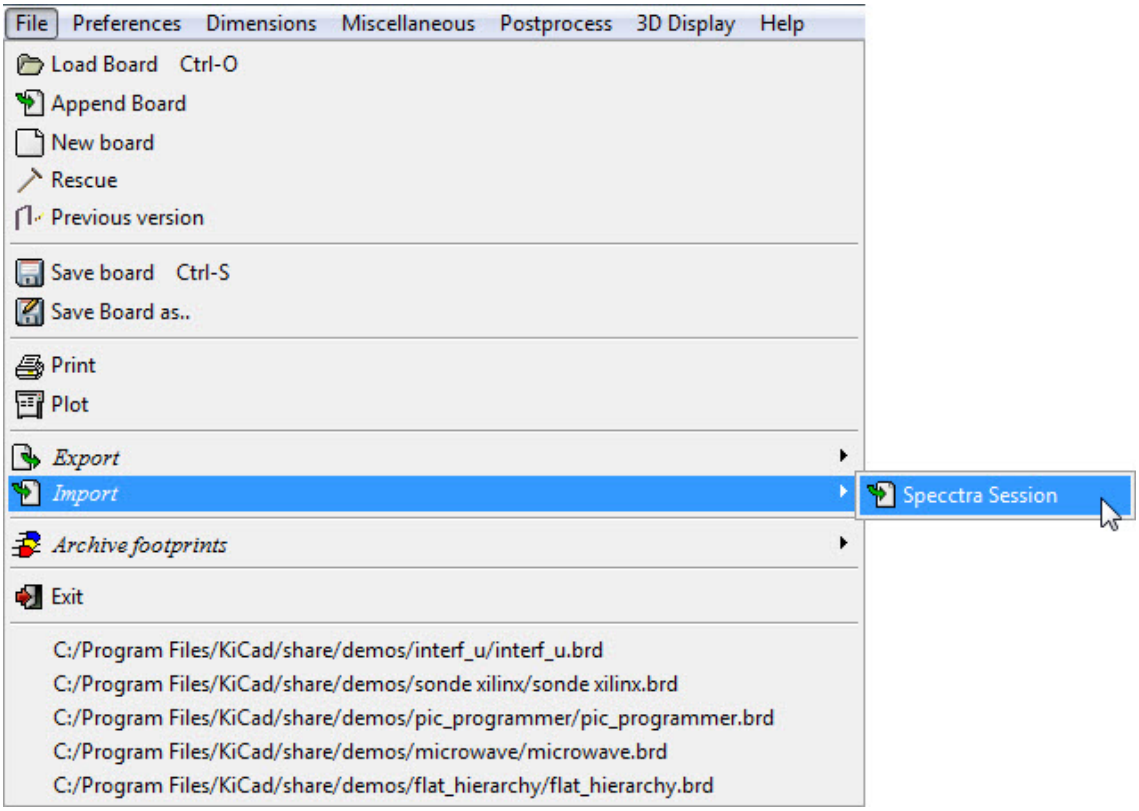
Import

To transfer data from KiCad to **TopoR**, use the **.dsn** format. To begin, open a **.brd** file in KiCad and click **File ► Export ► Specctra DSN** in the main menu.



Export

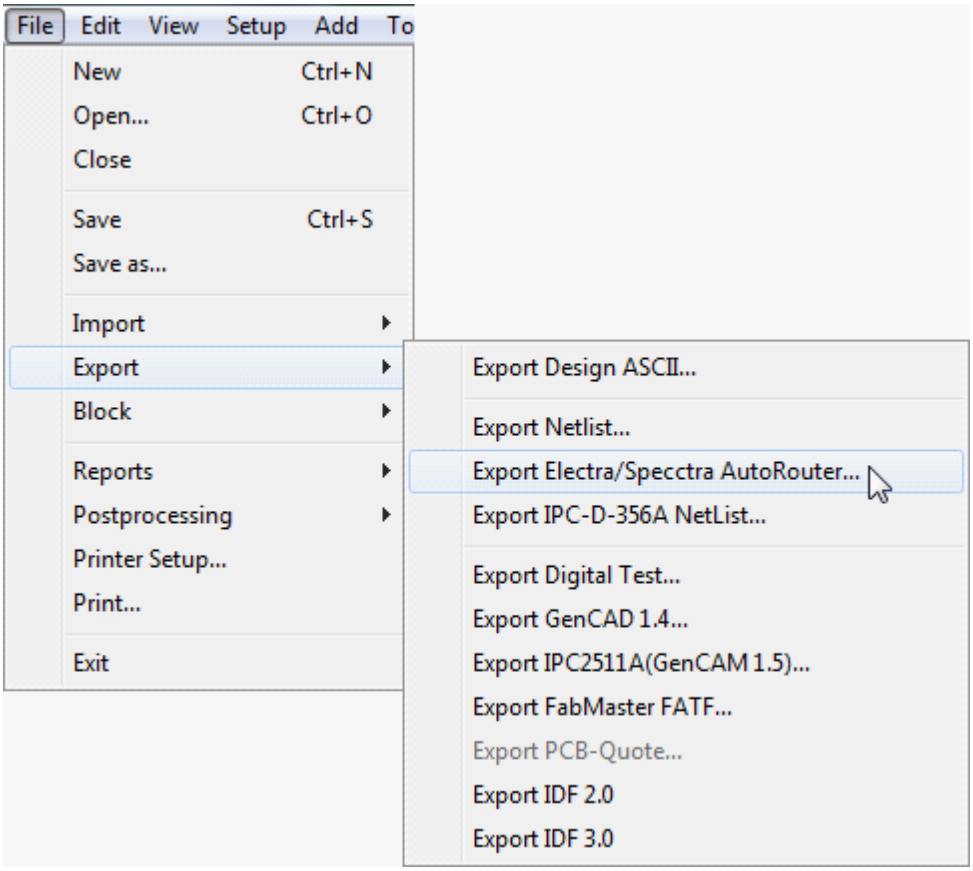
To export from **TopoR** to KiCad, use the **.ses.** format. In KiCad, click **File ► Import ► Specctra Session** in the main menu.


File
Preferences
Dimensions
Miscellaneous
Postprocess
3D Display
Help
Load Board Ctrl-O
Append Board
New board
Rescue
Previous version
Save board Ctrl-S
Save Board as..
Print
Plot
Export
Import
Specctra Session
Archive footprints
Exit
C:/Program Files/KiCad/share/demos/interf_u/interf_u.brd
C:/Program Files/KiCad/share/demos/sonde xilinx/sonde xilinx.brd
C:/Program Files/KiCad/share/demos/pic_programmer/pic_programmer.brd
C:/Program Files/KiCad/share/demos/microwave/microwave.brd
C:/Program Files/KiCad/share/demos/flat_hierarchy/flat_hierarchy.brd

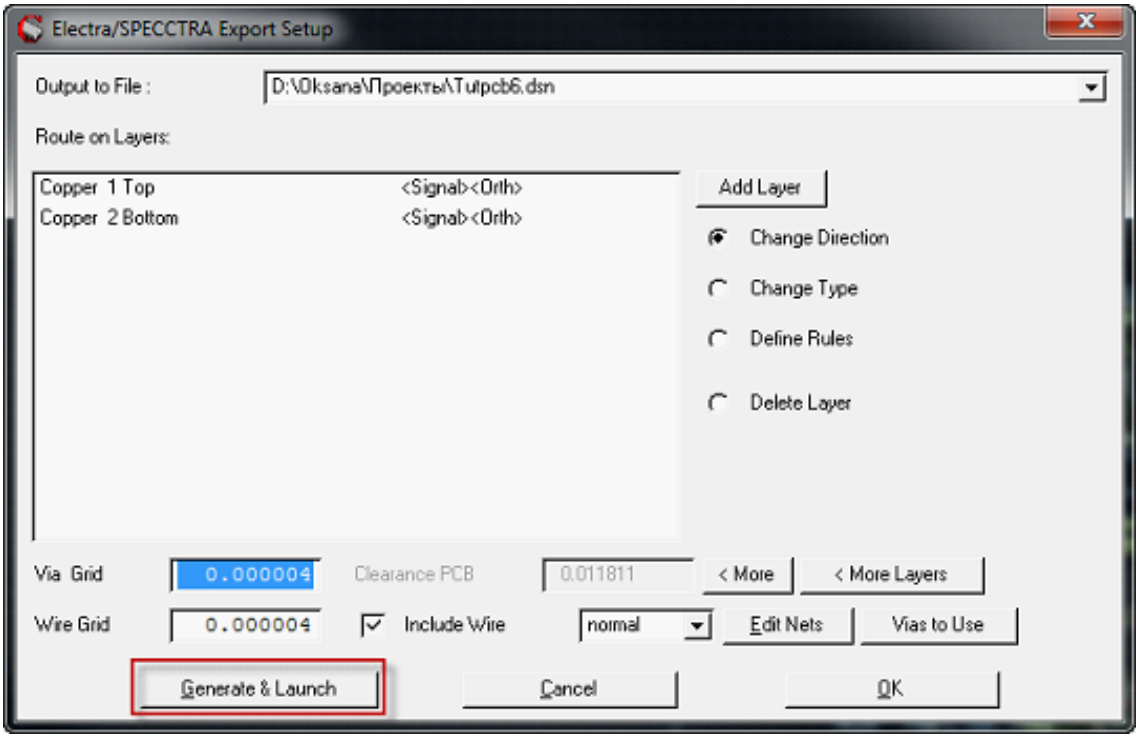
CADint

Import

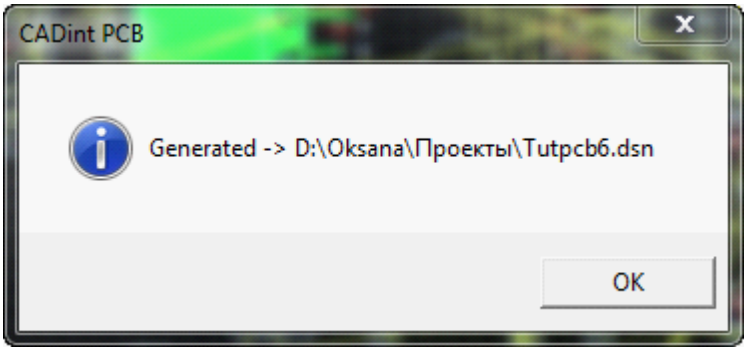
File interchange between CADint and **TopoR** is done through the **.dsn/.ses** intermediary format. To begin, in CADint, open a **.scl** file and click **File ►Export ►Export Electra/Specctra AutoRouter** in the main menu.



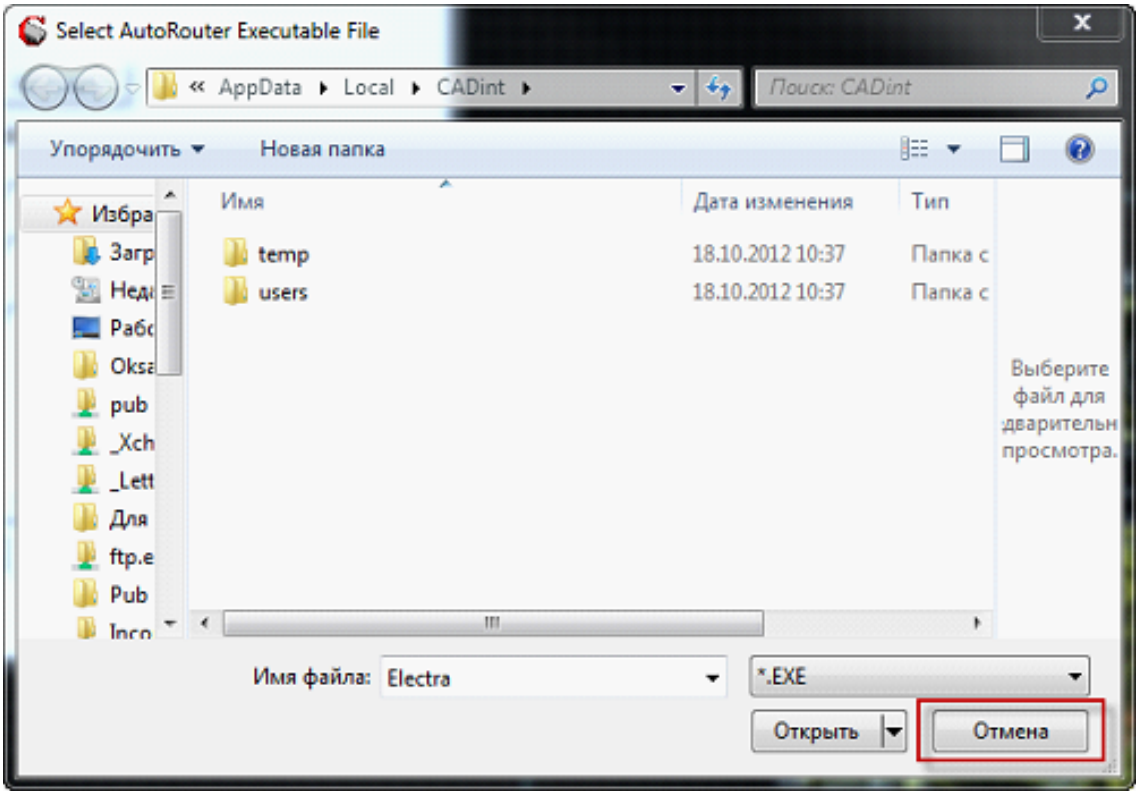
In the dialog box that opens, click the **Generate & Launch** button.



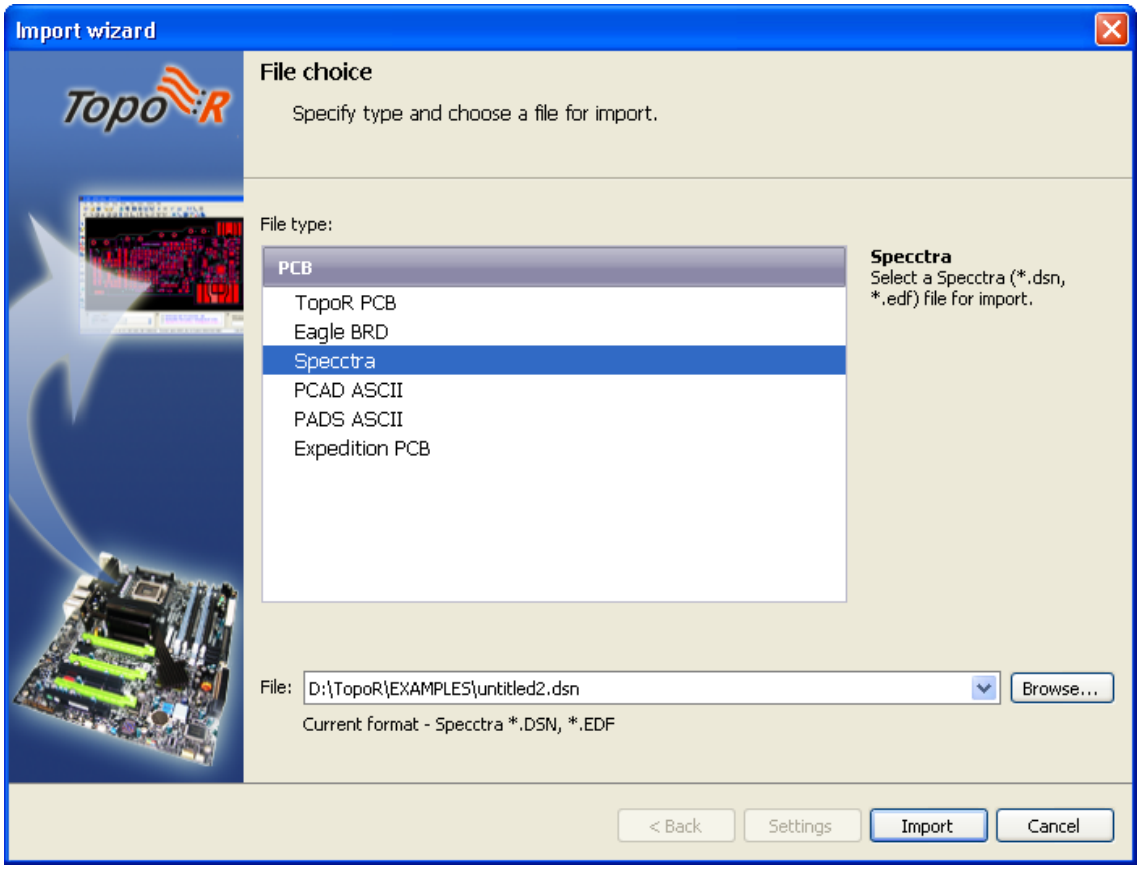
After a **.dsn** file has been generated, the following message will pop up. Click **OK**.



Next, the **Select AutoRouter Executable File** dialog box opens, where you are prompted to launch Electra. Click **Cancel**.

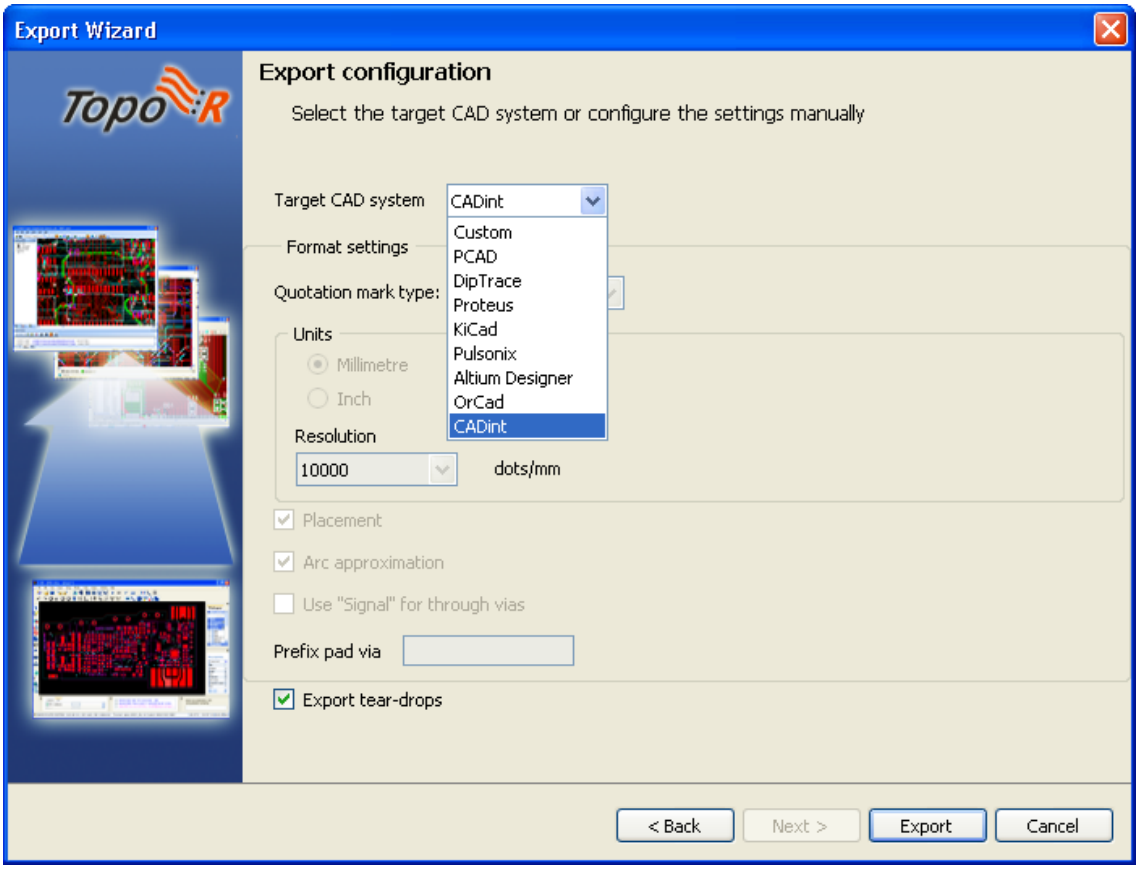


To import a **.dsn** file into **TopoR**, select the Specctra file type and click the **Import** button.



`Export`

To transfer routing results from **TopoR** to CADint, in **TopoR**, select export to a **`.ses`** file (the Specctra file type). Then in the **export wizard** select CADint as the target system. Click the **Export** button and save the result.


Export Wizard
Export configuration
Select the target CAD system or configure the settings manually
Target CAD system
CADint
Custom
PCAD
DipTrace
Proteus
KiCad
Pulsonix
Altium Designer
OrCad
CADint
Format settings
Quotation mark type:
Units
Millimetre
Inch
Resolution
10000
dots/mm
Placement
Arc approximation
Use "Signal" for through vias
Prefix pad via
Export tear-drops
< Back
Next >
Export
Cancel

DSN Format Specifics

The DSN format was designed for data exchange with the Specctra router and has a number of limitations:

1. The via diameter is not defined. (**TopoR** sets the default via diameters.)

If you are planning to output **.dxf** or Gerber files from **TopoR**, you should configure the via diameter in advance.

2. Information about mask and paste layers is not defined.

If you are planning to output Gerber files from **TopoR**, you should create such layers and primitives on them in advance.

3. Text is not stored, but the bounding rectangle around it is preserved, defined as a keepout.

This is sufficient for correct routing, but the text contents cannot be edited.

4. Component labels must not contain apostrophes (**'**) or quotation marks (**"**), because one of these characters is reserved for special uses (either one, depending on to the CAD system that writes the **.dsn**) and will not be parsed. This can cause duplicate component names and other problems.

D. Reference

The sections in part D contain useful tips, FAQs, button and default hotkey lists, an index spanning all of this manual, general operation recommendations and a detailed example of designing a relatively simple printed circuit board.

Useful Tips

Before you start, consider creating a working directory for the project and put the original file in it. This will make the program save the project file and intermediary result files in the same directory, which makes it easier to find files.

It is difficult to achieve good results with an initial component layout where no routing has been done. Normally, the process of component placement and routing involves several stages, as follows:

1. Initial component placement.
2. Connection routing.
3. Detection of areas with excessively dense wire and via patterns.
4. Changes to component placement.
5. A new iteration of connection routing.

And so on, until the required result is attained.

Frequently Asked Questions

General Information about TopoR

Is TopoR like other PCB design software? Will experience with other CAD software help me learn TopoR faster?

TopoR is fundamentally different from other CAD software. It's possible that knowledge of another CAD system may even slow you down.

How complex are the boards I can lay out in TopoR?

The current **TopoR** configurations can lay out boards with up to 32 routing layers.

Who uses TopoR today? Are there any problems in production?

The software is used successfully in businesses and higher education institutions in Russia (from St. Petersburg to Petropavlovsk-Kamchatsky), the Ukraine (Kiev, Kharkiv, Zhytomyr, Severodonetsk, Vinnytsia, Odessa), India (Heydarabad, Ahmedabad, Haryana), and also in the USA, Kazakhstan, Macedonia and other countries.

Purchasing, Support, Learning

Is there a network license for the software? How many workstations is it valid for?

If you buy five or more licenses, you get a network key without any additional payment.

The documentation is insufficient for me; how else can I learn to work in TopoR?

The easiest way is to ask any questions that arise on the forum or send them to the addresses listed in the Contacts section of the website: http://www.eremex.com. In the Tutorials section you can find short educational videos. In addition, we can offer instruction in learning centers in St. Petersburg and Moscow. You only need to fill out an application form.

What kind of technical support do I receive after purchase?

Within one year of purchasing a version of the software, you will get all updates free of charge. You can take a free course in a learning center in St. Petersburg or Moscow. Online support will prioritize your requests higher than those from demo version users.

How long is the software license valid?

The license never expires.

System Requirements

What are the minimal hardware and software requirements for running TopoR?

PC-compatible computer with an Intel® Pentium® III 1000 MHz CPU

Operating system: Microsoft® Windows® 2000 (SP3), XP (SP2 or SP3), Vista, Windows 7, Windows 8

Microsoft® Internet Explorer 5.0

Windows Installer 3.0 or higher

RAM: 512MB.

100MB hard disk space

SVGA monitor and graphics adapter, 256 colors, 1024x768.

Mouse with a scroll wheel

Can I use other OSs, such as Linux or UNIX?
Currently, you can run only the Lite version in Wine. Support for Linux in **TopoR** is planned.

Import and Export

Can I work in TopoR exclusively, without using other CAD software?
No. At this time, **TopoR** does not implement a comprehensive design cycle, although development in this area is ongoing.

Data from what CAD systems can be transferred to TopoR?

P-CAD 2000-2006—through the ASCII PCB format
Altium Designer—through the ASCII PCB format
Cadence OrCad—through the **`.dsn`** format
Mentor Graphics Expedition—through **`.hkp`** files
Mentor Graphics PADS—PADS ASCII PCB (for versions from PowerPCB V3.5 to PADS Layout V2005.2)
DipTrace—through the ASCII PCB format
Proteus ARES—through the **`.dsn`** (**`.edf`**) format
Eagle—through the **`.brd`** and **`.dsn`** formats
KiCad—through the **`.dsn`** format
CADint—through the **`.dsn`** format
Pulsonix—through the **`.dsn`** format

Can I export the project I make in TopoR to other CAD software?
Yes. The program supports import and export of the following file formats:

PCAD ASCII PCB versions 2000, 2002, 2004, 2006
PADS ASCII PCB versions 3.5, 4.0, 5.0, 2005.0
DSN/SES (SPECCTRA, Electra)

In addition, DXF output is possible for interchange with mechanical CAD software and GERBER and DRILL output for putting boards in production.

The documentation says that the signal data I add can be lost during export to other CAD software. But I have been exporting and importing without losing any signals. Why is that?
Some CAD systems can correctly save and process information about signals created in **TopoR** as long as they correspond to a single two-pad net (or differentials signals used in two two-pad nets). If the signal net consists of more pads, this data is usually lost.

If I import a project form another CAD system where wires are arc-like, can I be sure that TopoR will leave the arcs unchanged?
In a freshly imported project, yes. However, if you start editing the project in **TopoR**, the wire shapes will change.

Settings

How do I route a board on one layer?
Open the routing panel by clicking **Route ► Autorouting** in the main menu, and go to the **Settings** tab. Turn on single-layer routing for the layer you need. Check that all planar components are placed

on the layer you are going to route wires in. The program does not automatically change the placement side.

What's the purpose of the "nominal clearance" and "nominal width" parameters?
In the design properties editor, you can set up rules for wire width and spacing. There are two options: the minimal value and the nominal value. **TopoR** tries to route wires using the nominal width (and makes thin wires only if an alternative does not exist or is significantly longer) and minimizes the number of segments that are thinner than the nominal width.


Unless you have special requirements, set the minimal and nominal width to the same value.

The minimal wire-to-wire clearance value is specified for the narrowest spots on the board, for wires passing between pads. If you use multi-row sockets (or other components), pay close attention to the minimal width and minimal clearance to avoid blocking connection to pads. The nominal clearance is used wherever wires can be set apart.


When you set up rules, always set the nominal clearance to be slightly higher than the minimal.

I'd like to be able to swap pins in components. How do I configure this in TopoR?
Early on, at the stage where you create component patterns, configure pin equivalence in the asset library. **TopoR** only reads the existing equivalence data; you cannot define it for components in **TopoR**.

An example of a board that contains components with equivalent pins is shipped with **TopoR** and is located in the **`EXAMPLES`** folder: see examples 2 and 4. For details about working with functionally equivalent pins, see the Reassigning Functionally Equivalent Pads section in the manual.

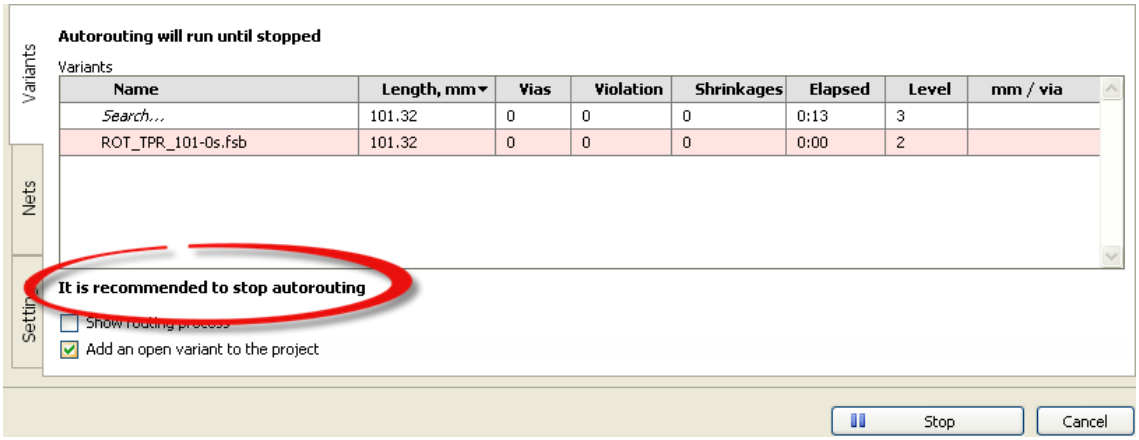
To view the pins that have been swapped, generate a **`.eco`** file (**File ► Report Files ► ECO** in the main menu).

Placement

Is automatic two-side placement possible?
In **TopoR 6.1** a component's placement side is not changed during automatic placement.

Routing

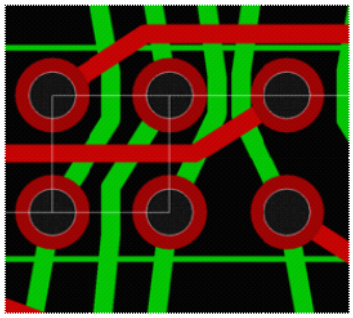
Why has TopoR been routing for hours and is still not ready, even though I just wanted to try it out on a small board?
TopoR will keep routing indefinitely until you stop the process (using the **Stop** button).
To indicate when it is time to finish, the program displays a message that suggests stopping the optimization.



After you have stopped routing, select a variant in the table by double-clicking it and open it in the editor.

Why does TopoR route multiple wires between component pins where only one wire can fit, even as TopoR itself detects a violation in that spot?
Check that the wire width and clearance rules are configured correctly. For details, see the answer to the question “What's the purpose of the “nominal clearance” and “nominal width” parameters?” in the **Settings** category.

If the configuration is such that a wire cannot fit between the pads and there is no other way, then the wire will still pass there, generating a violation. **TopoR** routes all nets without exception.



Can I perform grid-based and orthogonal routing?
In automatic mode, **TopoR** routes wires at arbitrary angles. However, if you want, you can enable grid snapping and route wires orthogonally in manual mode.

I routed the board, changed the wire width in the design properties editor and started routing again. But the wire width is the same in the resulting project—why is that?
There are two ways to perform autorouting: full rerouting of the project from scratch and refining optimization of a previously discovered variant.
1. To reroute a project from scratch, remove all existing routing before you run **autorouting**. This will apply the wire width rules set up in the **design properties editor**.
2. To optimize a project, run autorouting from the context menu of one of the saved routing variants. This will apply only those wire width rules that have already been used in the project (manually thickened wires will also be taken into account).

Editing

What is the right way to work with copper areas (polygons)?

The way to work with polygons is currently as follows:
1. If the polygon is not supposed to overlap other nets, pour it before autorouting. This will prevent any nets crossing it.
2. If the polygon is intended for pouring all remaining free space on the board and it is acceptable for wires to cut through it, do not pour it before autorouting. Pour it only after all DRC violations caused by wires have been eliminated. When you move nets or vias inside a previously poured polygon, repour it every time to recalculate clearances.
3. Polygons are best poured at the final stage of the design.

Can I change the pattern and add a net to the existing topology?
Currently **TopoR** is strictly a router with a topology editor. It does not have an asset library editor or schematic editor. TopoR is not a comprehensive design system, so you cannot change the pattern, add a component, add a net or delete a net.

TopoR has done the autorouting, but generated some DRC violations. How do I get rid of them?
On densely-packed boards where components cannot or must not be moved, you should clear the **Weak check** option in the autorouting settings. If the board has enough room and components can be moved, it is convenient to fix the sockets and mount holes, launch autorouting with the **Weak clearance check** option enabled. The result can be fine-tuned using automatic procedures.

First activate **FreeStyle** mode's **Push components and vias** submode (or, if components should not be moved, **Push vias** submode). Most violations are normally eliminated automatically. The remaining violations are corrected manually. Wires are rerouted in manual mode, and vias and components are most easily moved automatically in **FreeStyle** mode. After localized manual edits, consider using automatic procedures again, because they facilitate manual work. The website offers very useful video tutorials on manual wire routing.

Why do the shapes of the wire I route keep changing? Is there a way to disable these changes?

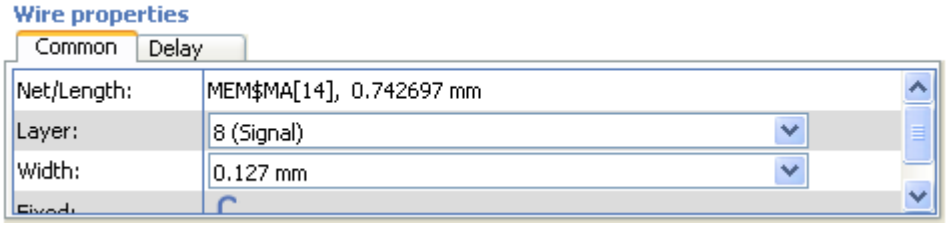
When **FreeStyle** is activated, wire shapes are recalculated, but their topological paths remain the same. For example, a wire will never jump over a pad to go around it on the opposite side. The situation is different when automatic procedures are used that optimize wire paths. In that case, wires really can turn in different places or change their layers, and so on. Therefore, automatic procedures should be used at the early editing stages rather than during fine-tuning.

Now think of the reasons you may want to leave wires unchanged:
1. If you want to reroute wires to increase spacing, this can be done by setting the nominal wire-to-wire clearance. This will make them stay further away from one another where there is enough room. The nominal clearance is specified in the **design properties editor** on the **Net clearance** tab (see the answer to the question “What's the purpose of the “nominal clearance” and “nominal width” parameters?” in the **Settings** category).
2. If your reasons are different, wires can be fixed in the topology editor by clicking the **Fix** button on the toolbar or pressing the corresponding hotkey (by default, **F**).

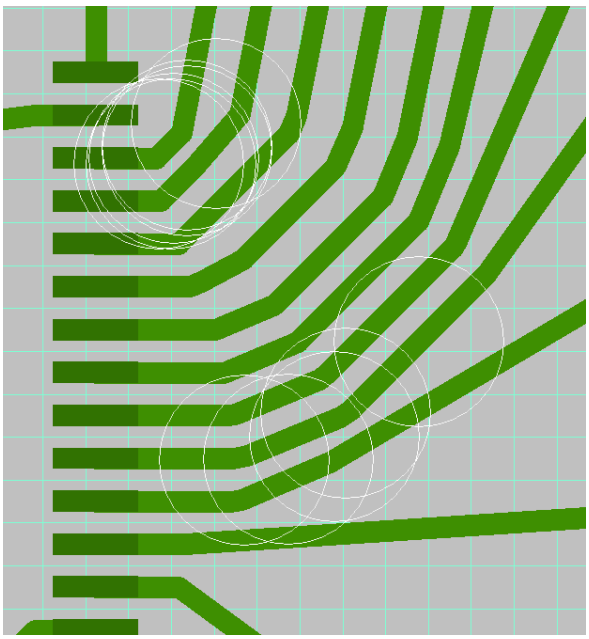
You cannot change wires manually in **FreeStyle** mode, because they are rebuilt automatically according to the specified rules. This helps maintain clearance during component shifting.

After I've done the routing, there is still some room on the board, so I want to make the wires wider without rerouting the entire project. What's the fastest way to do it?

In the **selection filter** deactivate all filters but wires. Press **Ctrl+A,** to select all wires. In the **Wire properties** panel, change the width value to the one you need. All wires will change width.



Why does TopoR generate DRC violations even though there is plenty of room?



TopoR implements automatic calculation of the optimal wire shapes. However, due to the complexity of calculations on densely-packed boards some precision issues may occur. They are usually present around microchips with small and close-together planar pads. One tip that goes a long way in avoiding these issues is to make the nominal clearance higher than the minimal. If the problems appear anyway, there are several ways to correct them:

1. Move away any of the objects near the wire (most often, vias).
2. Manually correct the wire shape and fix the necessary wire segment.
3. Add a routing keepout close by the miscalculated wire.

Errors

In my design, two pads from different components cross. They belong to the same net, and I don't mind their crossing. However, TopoR reports a pad overlap error (ID 3004) and doesn't let me run autorouting. How do I get rid of the error?

Fix both components that the overlapping pads belong to. The error will go away, and you will be able to run autorouting.

Hotkey List

Shortcut	Description
F1	Help
Alt + Q	Message bar
Alt + Enter	Properties panel
Alt + D	Display settings panel
Ctrl + F	Search panel
Ctrl + Q	Project panel
F4	Design properties editor
Ctrl + O	Open file
Ctrl + S	Save file
Ctrl + P	Print
F5	Calculate wire shapes
F6	Reroute wires
F7	Move
R	Route wires
V	Create vias
F8	Move in FreeStyle mode
Ctrl + M	Change distances
Ctrl + Z	Undo
Ctrl + Y	Redo
G	Enable or disable grid snapping
O	Enable or disable angle snapping in 45-degree increments
P	Link a different incident segment
Ctrl + X	Cut to clipboard
Ctrl + C	Copy to clipboard
Ctrl + D	Duplicate
Ctrl + V	Paste from clipboard
Del	Delete
F	Fix or unfix
L	Change layer
S	Change placement side
W	Change wire width
C	Straighten wire

Z	Pour or unpour selected copper areas
Shift + Space	Turn 90°
Space	Turn –90°
Num +	Zoom in
Num -	Zoom out
Alt + →	Next view
Alt + ←	Previous view
Ctrl + B	Show board
`	Enable or disable all metal layers
N	Enable or disable links
ESC	Deselect / Selection mode
Ctrl + A	Select all
X	Cycle selection
Backspace	Undo last operation
Q	Select new target
Alt + Space	Switch nets in differential pair
Shift + Q	Route wire automatically
[0..9]	Metal layer 1 .. Metal layer 9. Bottom metal layer (0)
Scroll wheel	Zoom workspace
Right mouse button	Pan workspace

Index

Autorouting
CAD
  Altium Designer
  CADint
  DipTrace
  Eagle
  Electra
  Expedition
  KiCad
  Kompas
  OrCad
  PADS
  P-CAD
  Proteus
  Pulsonix
  Specctra
Constrains
Editing
  FreeStyle
  pads
  design properties
  keepout properties
  text label styles
  topology (manually)
Equalize delays
  automatically
  in group
  manually
  mutually
File formats
.asc
.brd
.cct
.dru
.dsn
.eco
.edf
.fsb
.fst
.fsx
.hkp
.max
.pcb
.pcbdoc
.pcb
.ses
.ulp
High-Speed Rules
Importing designs
Labels
Layers
Net properties
Output
  BOM files
  Drill files
  DXF files
  ECO files
  Gerber files
  printing results
Padstacks
Signals
  adjusting delays
  combining into a differential signal
  creating
  deleting
  differential
  highlighting
  impedance
  routing a pair
  searching for buses
  searching for pairs
  selecting the source
  setting the topology
  splitting
  using
Toolbar

About the TopoR PCB Format

The current version of the format is 1.1.2. The *Topor PCB Format Specification* for developers is available for download from the website: eda.eremex.com/downloads.

Working with single-layer borders

TopoR offers specialized features that greatly facilitate routing of single-layer PCBs. The recommended scenario for single-layer PCB routing is as follows:

1. After you have imported a project, check the routing rules carefully. The recommended nominal clearance to set is 1.5 times the minimal allowed clearance.
2. Make sure that the board outline is set. It is better to define the outline as a single polygon than a series of lines.
3. To prevent components running into one another during automatic placement, set the component outlines. If component outlines are defined in custom layers, make sure you select the **Consider during placement** option for those layers in the **Layers** section.
4. Perform component placement. Importantly, you should fix the components whose positions are predefined; and conversely, unfix the components that you don't mind being moved.
5. **Autorouting ► Options**

a. Enable the **Single-layer routing** option and select the layer to route.
b. Enable the **Weak clearance check** option. This is needed because during single-layer routing wires normally pass between components, and it is difficult to know in advance where components should be wide enough apart to make room for wires and how many wires should fit in there. If you leave the **Weak clearance check** option cleared, TopoR, will try to closely follow the existing clearance rules. For example, if you need to route four wires between components, but only three fit in, then the fourth wire will take a detour (which can interfere with the routing of other wires) or use a jumper. By enabling the **Weak clearance check** option you permit the router to move unfixed components apart slightly, so TopoR will be able to route all four wires at the requisite location.
c. If arcs are acceptable, enable the **Use arcs** option.
d. If necessary, enable the **Reassign functionally equivalent component pads** option.
e. Do not enable the **Reuse existing layout as a starting point** option.

6. Launch autorouting and wait until the router stops turning out new variants frequently (it should take about a minute). The program shows a hint when the time is right: **Autorouting may be stopped**. Stop the procedure.
7. Select a routing variant. For single-layer routing, the best variant is usually the one that has the fewest vias (the number of vias equals twice the number of jumpers), because when you go on to edit the layout, it will be easier to shorten a long wire by adding a jumper than to remove a jumper.
8. Open the variant in the editor. Because weak clearance checking was used, there may be clearance violations. In the toolbar, right-click the triangle, select **Move vias and components and reroute wires**. This moves components further apart in dense areas and closer together in spacious areas. It is recommended that you perform the operation multiple times.
9. In the toolbar, right-click the triangle and select **Move vias and reroute wires**. The vias will be placed in a more reasonable fashion. This is a useful operation to perform (by pressing **F7**) after each minor manual edit.
10. Correct the nets that were routed poorly, move specific components to optimal locations and, if necessary, change the orientation of components. Before you transform a component, consider removing the wires incident to it; this generates net lines, which can be helpful visually.
11. Perform correct manual routing for those nets that must have a particular topology, and enable the **Fix flexibly** option for these nets. This option locks the net topology, including the via locations, but does not lock the wire shapes. The autorouter recognizes this option and keeps the topology of flexibly fixed nets unchanged; for example, it does not complete the routing of such a net if it has not been fully routed.

12. Check that the jumpers turned out straight and that no components interfere with bridge assembly.
13. Run **DRC**. Confirm that there are no errors.

Steps 6 through 13 should be repeated multiple times; normally, 5 or 6 repetitions are enough for the layout to "come together" in one of the subsequent variants. If you had 18 jumpers after the first iteration, you may have none after the fifth. This process is more like a captivating puzzle game than hard engineering work.

 A jumper is simply a piece of wire (normally, non-insulated). Specialized bridging chips are not used in TopoR, and you cannot specify a predefined set of jumper lengths.

 Some CAD systems (such as Proteus) import only wires and vias from the DSN format and ignore changes to component placement.