

Release PCB-Investigator 7.0

Here is a short overview about the newest updates in version 7.0 of PCB-Investigator.
All Features can be tested in our free trial versions, have a look to our webpage here:
<http://www.pcb-investigator.com/en/download>

New Bare Board (DFM)

The **Bare Board Analysis (DRC)** of PCB-Investigator has been completely reworked in the new Version 7 and now offers a **broad range of design rule checks** with a dramatically increased performance!



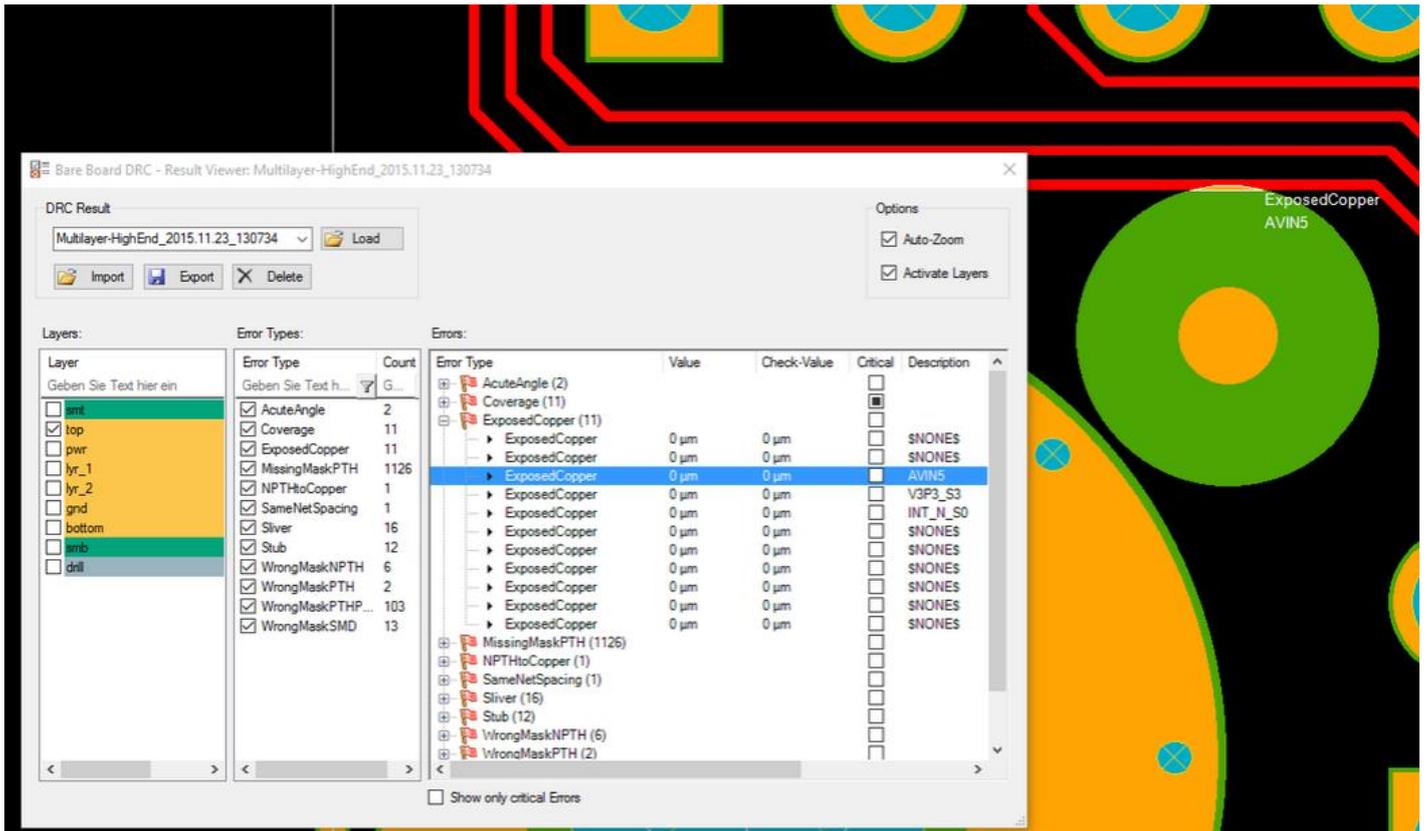
By using the new **Design Rule Check (DRC)** of PCB Investigator's Bare Board Analysis, you can easily **ensure the manufacturability** of your PCB design!

A perfectly designed board not only **improves your failure rate (ppm)**, but also has a strong impact on the **manufacturing costs**! Additionally the **time to market** can be reduced by less call backs from the PCB suppliers!

The predefined rule sets can also be easily extended by adding your own rule sets, with the newly included **Rule File Manager** feature. Rule sets can be created according to your company's design rules or your supplier's requirements.

Within a few seconds, the whole design will then be checked for its producibility:

- Spacing in copper / Trace width / Slivers
- Annular Ring check for drills/laser-vias and copper pads
- Mask clearance checks for drills (plated/non-plated/laser-vias), SMD pads and test points
- Copper clearance for non-plated drills
- Drill diameters and distances
- Missing Drills / Duplicated Drills
- Coverages and exposed copper areas by wrong solder masks
- Stubs
- Open/Shorts
- ...



There is a new result viewer integrated to have a fast overview of all reported errors.

Take a look at our new [Video Tutorial](#) and [contact us](#) to get a trial version.

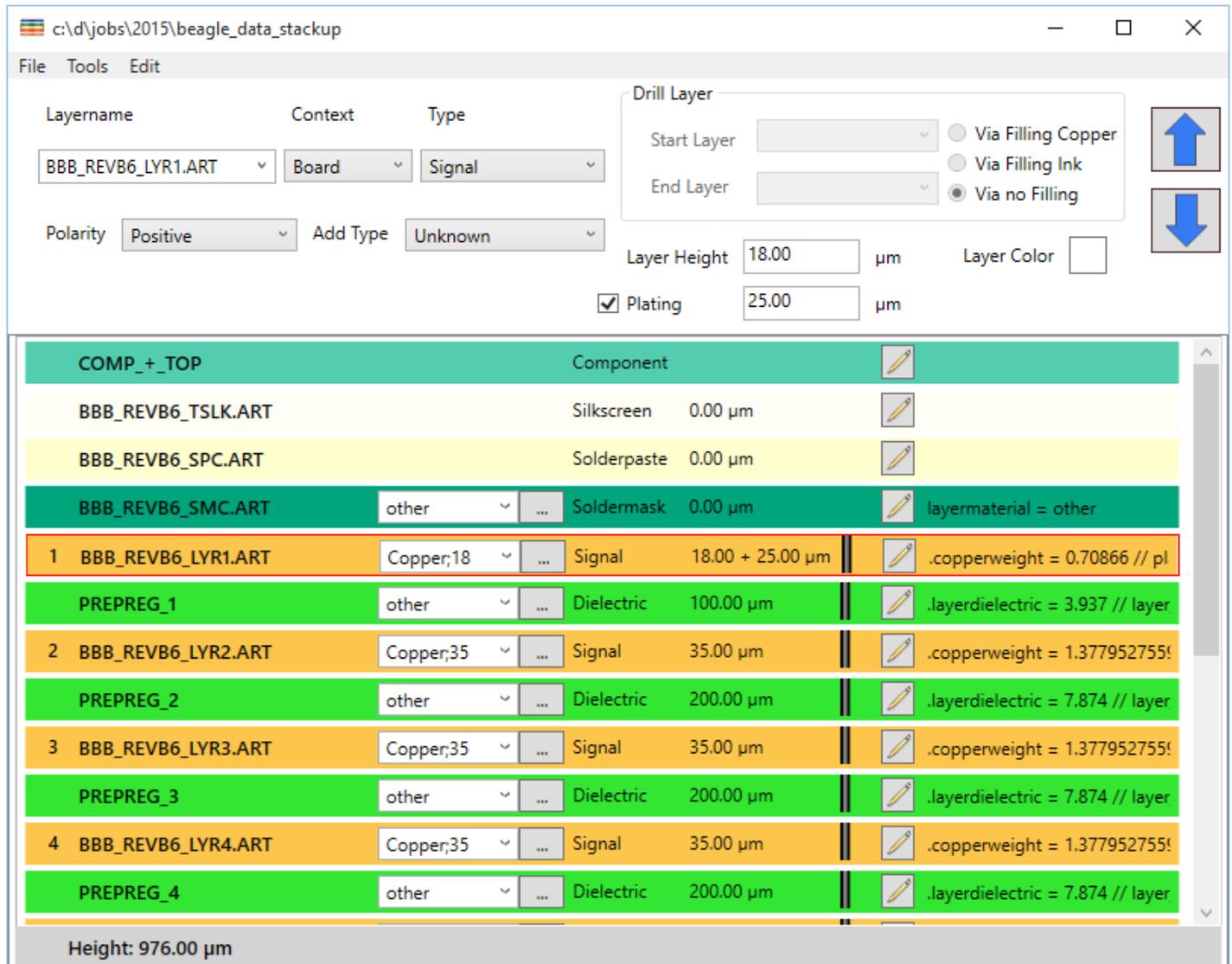
Ensure the manufacturability of your PCB design by using PCB-Investigator's DRC for your layout review!

See <http://www.pcb-investigator.com/en/blog/brand-new-bare-board-analysis-drc>

New Matrix Dialog

The matrix dialog gives you easy ways to define new stackups or use existing definitions to update your data. It helps you to upgrade simple formats like gerber 274x or Sieb and Meyer files.

You have some options to modify layers in edit menu (invert layer, polygonize layer or create document layer of components).

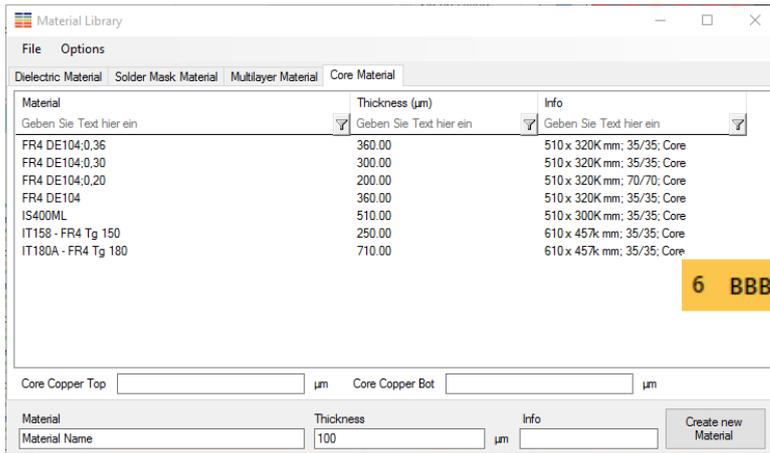


Layername	Context	Type	Height	Material	Attributes
COMP+_TOP		Component			
BBB_REVB6_TSLK.ART		Silkscreen	0.00 µm		
BBB_REVB6_SPC.ART		Solderpaste	0.00 µm		
BBB_REVB6_SMC.ART	other	Soldermask	0.00 µm	layermaterial = other	
1 BBB_REVB6_LYR1.ART	Copper;18	Signal	18.00 + 25.00 µm	.copperweight = 0.70866 // pl	
PREPREG_1	other	Dielectric	100.00 µm	.layerdielectric = 3.937 // layer	
2 BBB_REVB6_LYR2.ART	Copper;35	Signal	35.00 µm	.copperweight = 1.377952755	
PREPREG_2	other	Dielectric	200.00 µm	.layerdielectric = 7.874 // layer	
3 BBB_REVB6_LYR3.ART	Copper;35	Signal	35.00 µm	.copperweight = 1.377952755	
PREPREG_3	other	Dielectric	200.00 µm	.layerdielectric = 7.874 // layer	
4 BBB_REVB6_LYR4.ART	Copper;35	Signal	35.00 µm	.copperweight = 1.377952755	
PREPREG_4	other	Dielectric	200.00 µm	.layerdielectric = 7.874 // layer	

Height: 976.00 µm

Our new matrix dialog contains some more options:

- Material library and easy change of material
- Simple attribute changing
- Fast modify of stackup with drag and drop
- Create new standard stackups
- Exports with reports

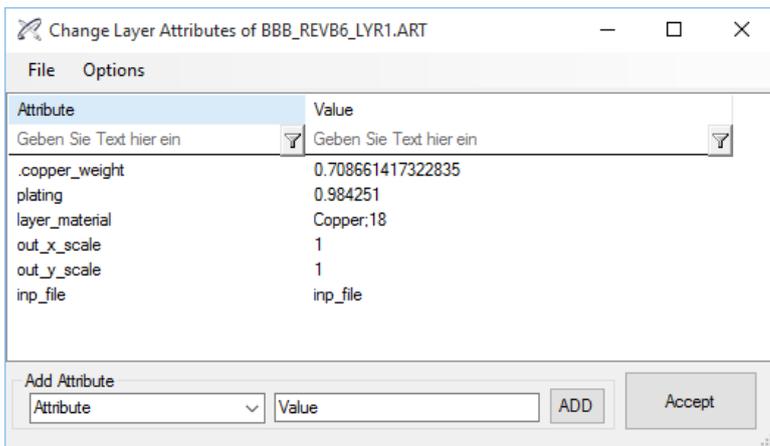


With material library you can manage your default thickness and information for all layer types.

You can add own materials and define thicknesses of dielectric-, solder mask-, multilayer- and core-material.



You can change the material with the combobox or use the library direct to select the relevant material.



In the layer attribute dialog you can change the attributes of one layer and add new attributes.

It is also possible to add user-attributes in sting format, just type your attribute name in the attribute combobox and the string value in the Value text-box.

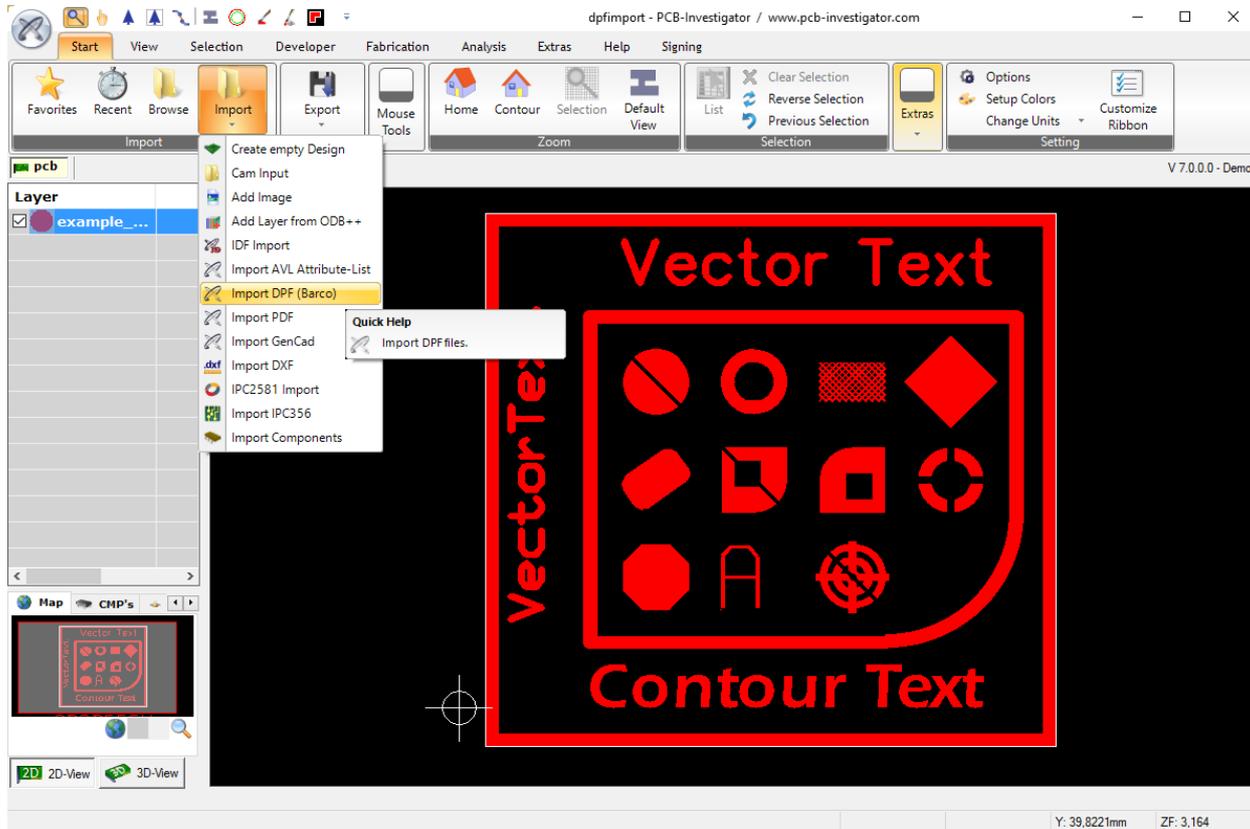
Update your data for easy analyse of production and design errors!

See <http://www.pcb-investigator.com/en/blog/matrix-dialog>

DPF Import

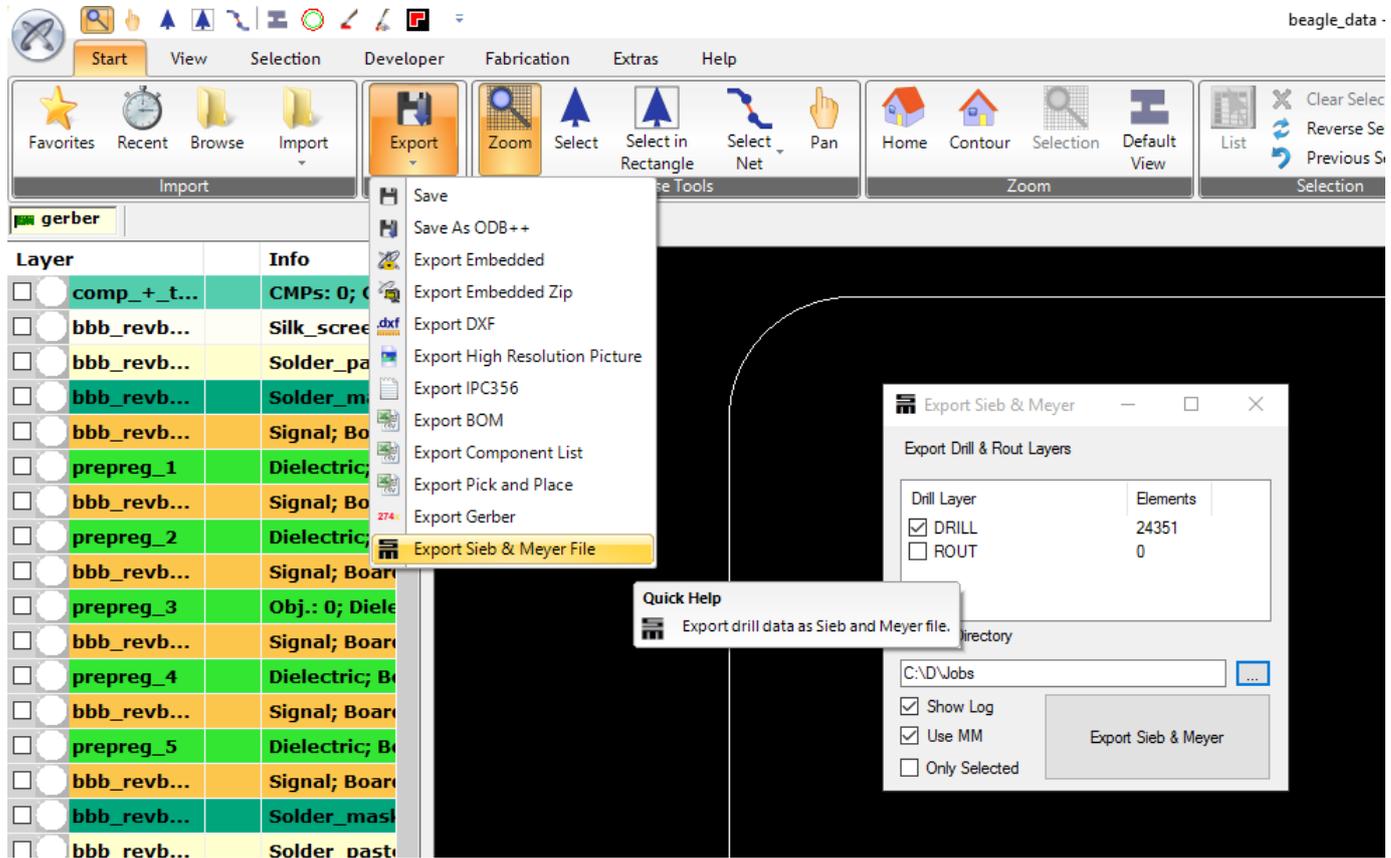
Import DPF Barco files as new format in Version 7.0 of PCB-Investigator.

You need no options, just select the file and it will be imported directly to your current project or as new project.



Sieb & Meyer Export

Newest export format for drills and rout layers is "Sieb & Meyer", this format is added in the export menu with only easiest options like unit and export only selected objects.

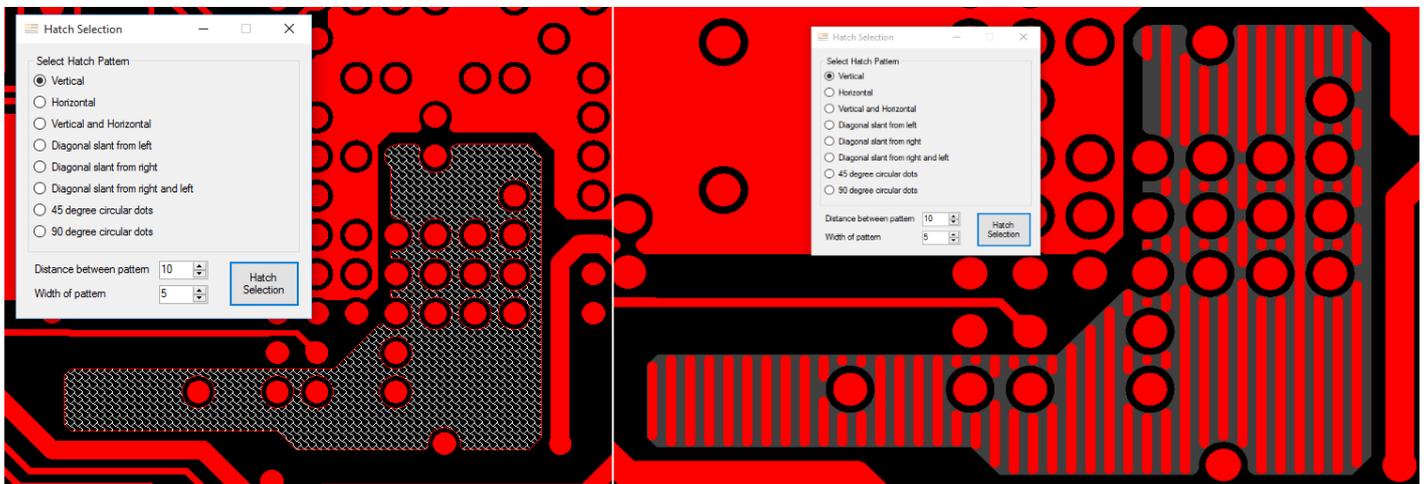
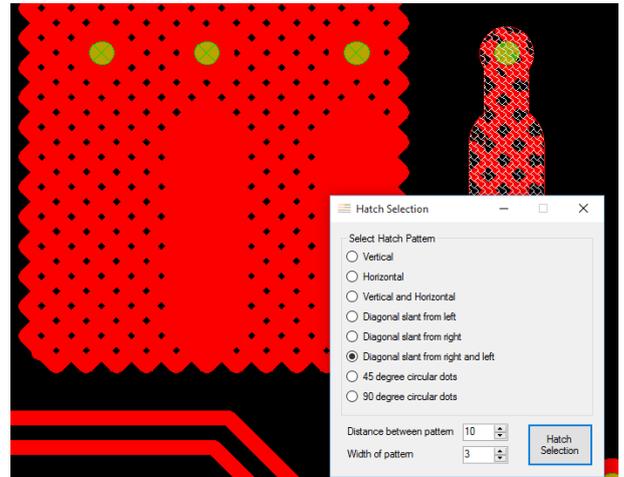


Hatch Pattern

The hatch pattern option is hidden in customize ribbon, first you need to add this option to your menu than you can open this new dialog to fill selection with lines.

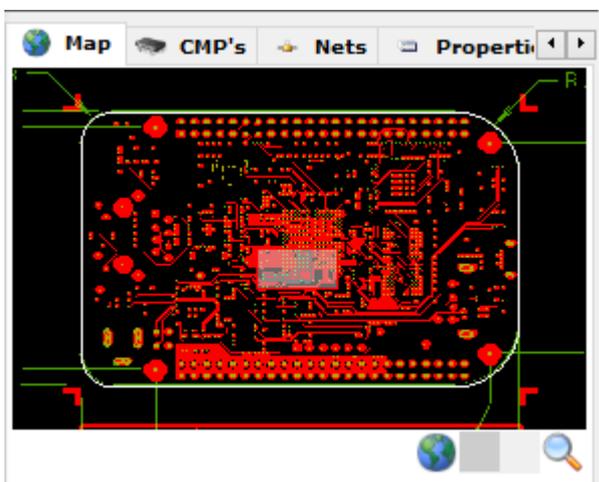
You can select each type of object, but each object will be hatched for itself.

You have many options like vertical, horizontal, both or diagonal and many more.



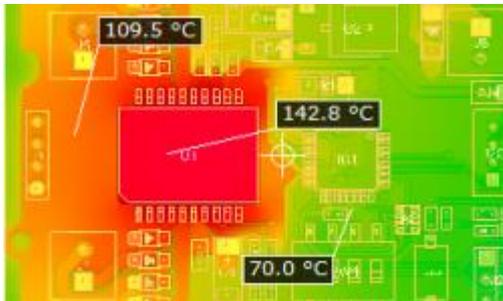
Map View with Image

An small update of the overview map in the left down corner with all visible layers drawn in the map.



Physical Simulation

PCB-Investigator Physics is the perfect tool to simulate the **physical behaviour** of your Printed Circuit Boards during development phase.



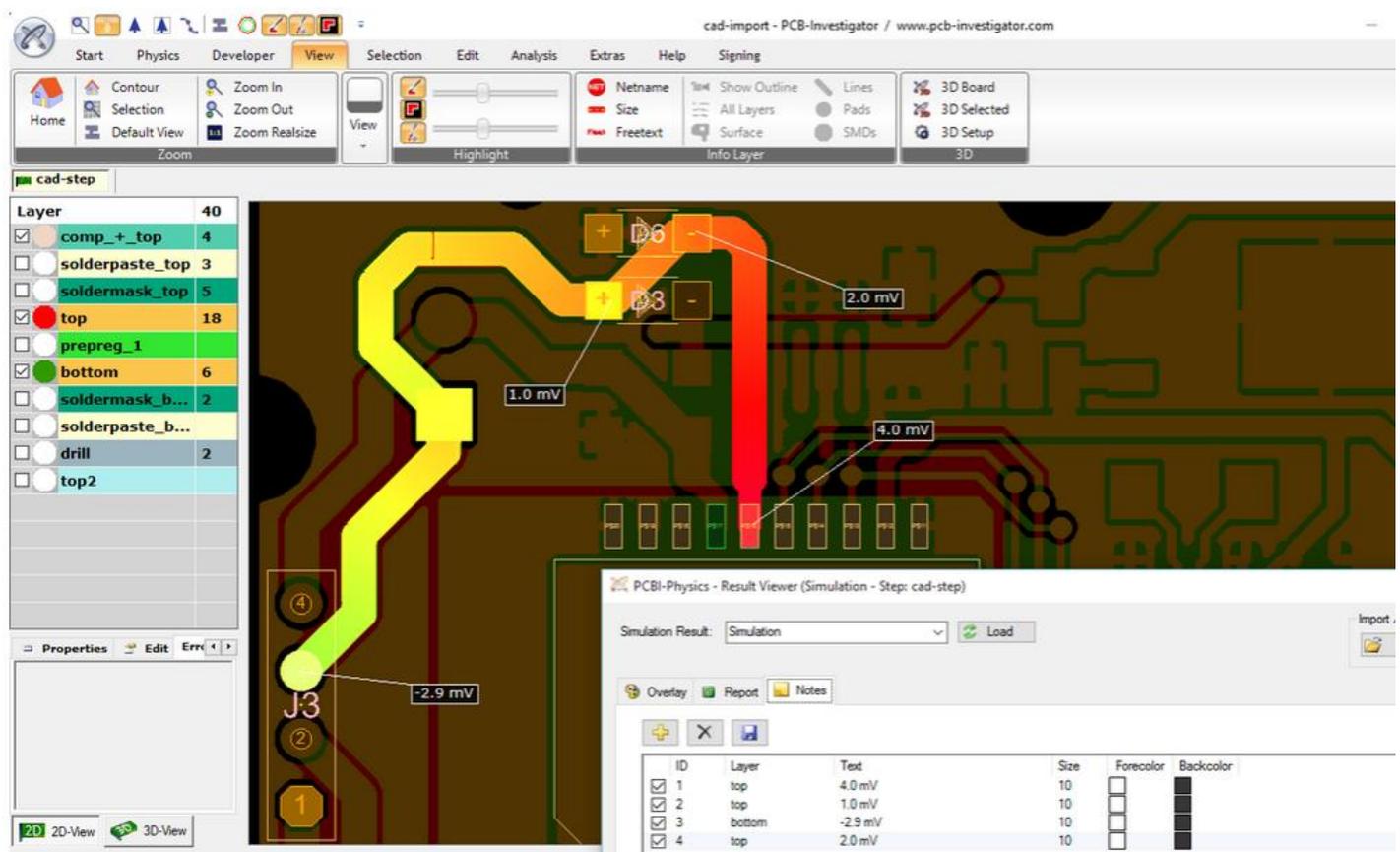
It enables you to find **thermal hotspots**, critical trace **resistances** and **voltage drops** that are too high, even before prototyping begins!

With the built-in editing functions of PCB-Investigator Physics it's even possible to **optimize the layout** and stack-up to achieve the best possible physical behaviour with only a few clicks!

Save valuable time and prototype costs with the simulations of PCBi-Physics!

To get information about the physical behaviour of your Printed Circuit Board during operation, PCB-Investigator Physics enables you to simulate the following physical properties:

- The **Temperature** raise at each location caused by power loss of components or by high currents
- The **Current Density**, e.g. at copper bottlenecks or in drills
- The **Voltage Drop** and **Copper Resistance** between any pins on any layer



The screenshot displays the PCB-Investigator Physics software interface. The main window shows a PCB layout with voltage drop simulation results overlaid on the traces. A red trace shows a 4.0 mV drop, a yellow trace shows 1.0 mV, and a green trace shows -2.9 mV. A component J3 is also labeled. The interface includes a menu bar, toolbars, a layer list on the left, and a 'PCBi-Physics - Result Viewer' dialog box at the bottom right.

ID	Layer	Text	Size	Forecolor	Backcolor
1	top	4.0 mV	10	<input type="checkbox"/>	<input type="checkbox"/>
2	top	1.0 mV	10	<input type="checkbox"/>	<input type="checkbox"/>
3	bottom	-2.9 mV	10	<input type="checkbox"/>	<input type="checkbox"/>
4	top	2.0 mV	10	<input type="checkbox"/>	<input type="checkbox"/>

Welcome to the new world of Simulations - with PCB-Investigator Physics!

(This is a own product, [contact us](#) to get a trail version)

See <http://www.pcb-investigator.com/en/blog/brand-new-physics-simulation>