EASYLOGIX.DE

PCB-Investigator

Quick Start Guide

Schindler & Schill GmbH
Im Gewerbepark D33
93059 Regensburg
Germany

Tel: +49 941 568 136 20
Email: info@easyLogix.de
Web: www.easyLogix.de

Quick Start Guide PCB-Investigator - Installation

To use PCB-Investigator, the software first needs to be installed. To do this, please enter your identification data in your **client login** first: https://pcb-investigator.com/de/login.

All products purchased by you are **stored here with downloads**, including the license files (floating). You can also access **all version updates** here, making sure that you always benefit from the latest version of PCB-Investigator. To stay up to date with our updates, we recommend to **subscribe to our newsletter**.

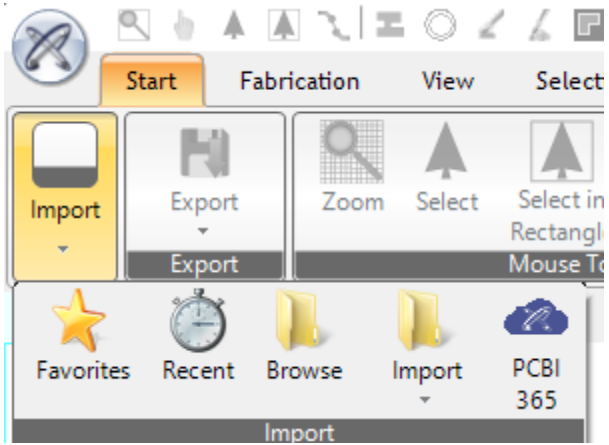
After successfully completing the download, you will have access to a zip file containing all the files needed for the installation. To perform the installation, open the **.msi** file and **follow the instructions in the installation wizard**. In the additional text file, you will find more information about the installation and possible causes for occurring problems.

Quick Start Guide PCB-Investigator – Basics

To load a new design or **job**,

click on the "Import" button. Here, you can choose between

- designs that you have marked as a favorite ("**Favorites**"),
- recently opened jobs ("**Recent**"),
- specific files in your data via the "**Browse**" function,
- an empty design via "**Import**" or,
- a design via our cloud platform **PCB-I 365** (if you are using this service).

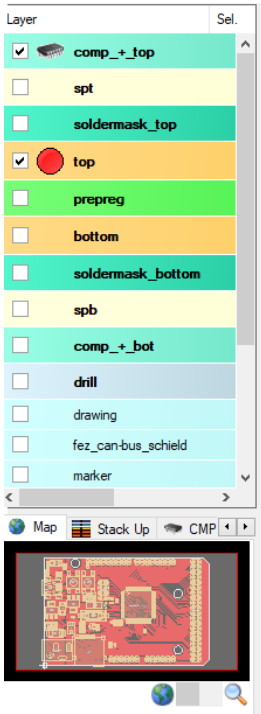


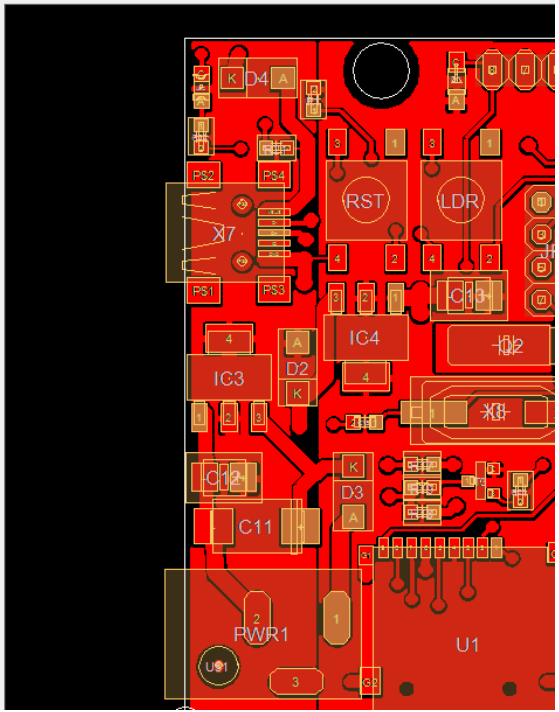
Alternatively, you can simply **drag & drop** your files into PCB-Investigator to open them.

Quick Start Guide PCB-Investigator - Basics

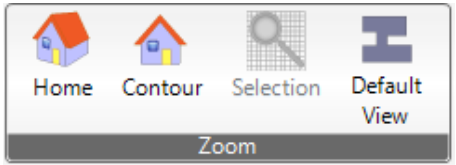
The **layer stackup** of your currently opened design is displayed on the left side of the main window.

To **show or hide layers**, check or uncheck the corresponding layer.





By clicking on "**Default View**" (under the zoom function in the toolbar at the top), all layers are hidden at once. Only the contour of your design will be visible.



Quick Start Guide PCB-Investigator - Basics

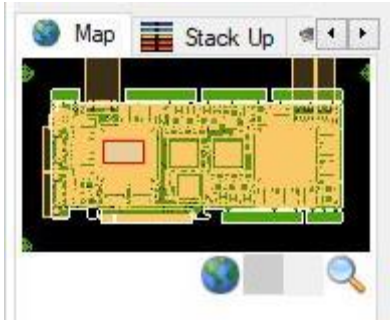
The "**Zoom**" option allows you to see you design in more detail.

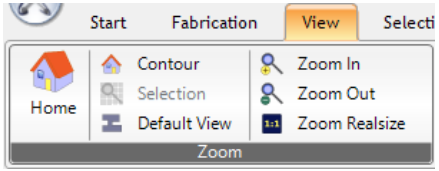
The small window below the stackup, showing you a miniature version of your design, serves as an **orientation** aid. Here, you can see by means of the red rectangle at which point in the design you are currently working.

To see specific parts of your design, use the **scroll function** of your mouse to zoom in towards the corresponding position. You can also use the small window on the left side of the start window to zoom. Simply select the point of the design you want to view more closely by dragging a rectangle to the corresponding position.

If you want to display the design in its original size again, press the "**Home**" button at the top of the toolbar.

Alternatively, you can select "**View**" in the ribbon above. Likewise, different zoom options are offered here, such as zooming in and out or displaying the "Home" screen showing the full design.





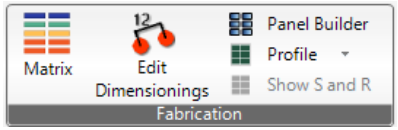
Quick Start Guide PCB-Investigator - Basics

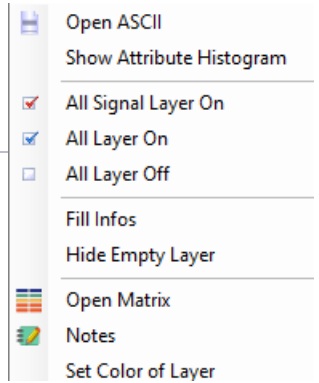
To change the matrix / layer stackup, you have to open the "Fabrication" tools in the above ribbon or right-click on the layer stack-up (left side) to open the .

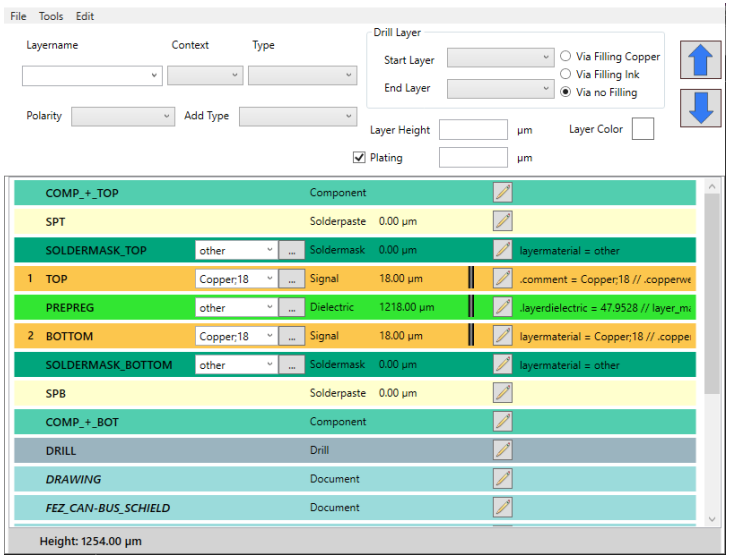
By clicking on the corresponding layer, you can **change** various **parameters** of this layer, such as layer name, context, type, layer thickness or polarity. By clicking on the blue arrows on the right side of the window, the position of the selected layer can also be changed by shifting.

To **add a layer**, click on "Edit" and select "Add Layer" to choose the type of layer. Drill layers are added specifically with "Add Drill Layer".

To **save the changes**, click on "File" and " Save Matrix". Otherwise, you will be asked to save your changes when closing the window.



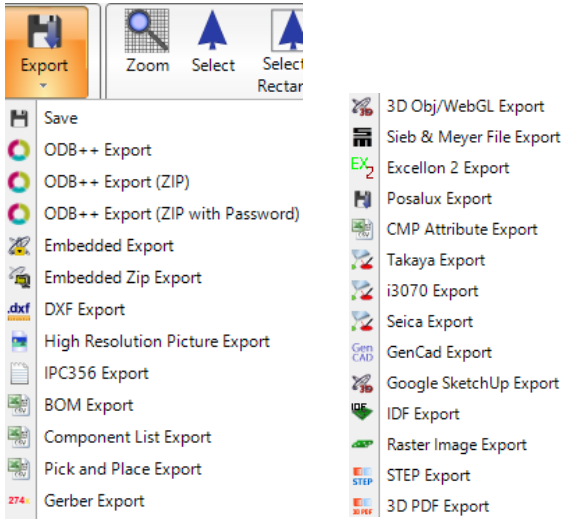




Quick Start Guide PCB-Investigator - Basics

To save changes to a design, select "**Export**" in the ribbon at the top. You can choose the type of file format you would like to use to save and export your design.

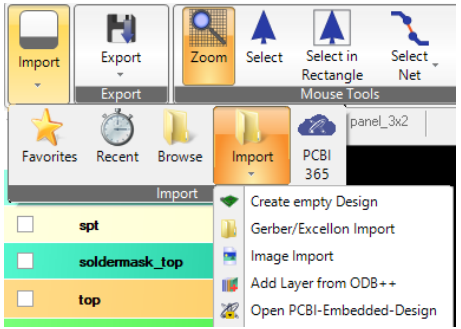
All available file formats PCB-Investigator supports are listed here for you to choose from.

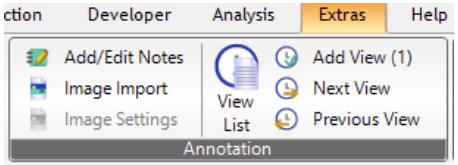


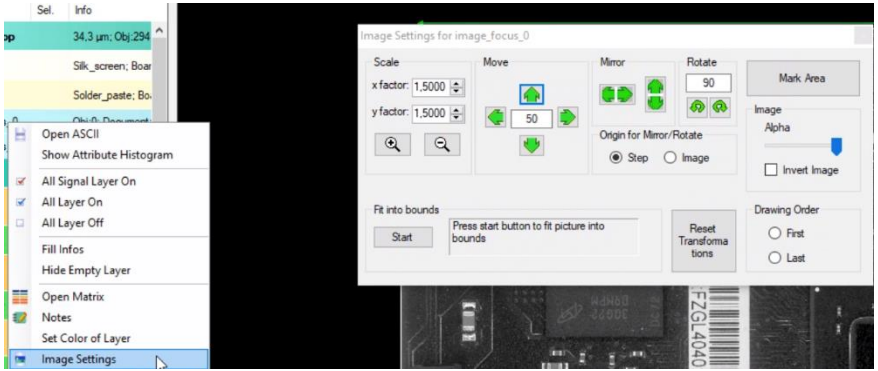
Quick Start Guide PCB-Investigator - Basics

To add data, you can basically use the Import button or drag & drop. Here, you will find numerous import options. For example, you can use CSV Components to import only components, or additional layers, additional information (attributes), netlists or entire designs.

Images can also be loaded via the "Extras" button (use "**Image Settings**" for adjusting). This serves e.g. the better processing of Gerber files by an enhancement on the basis of illustrations.







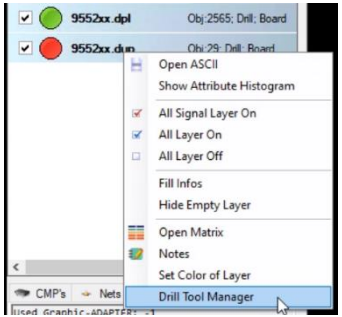
Quick Start Guide PCB-Investigator - Basics

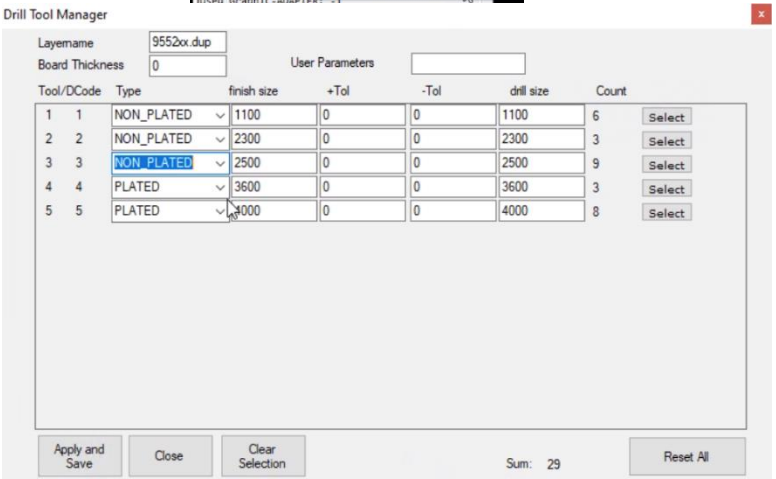
To load Gerber files, use the import function or drag & drop analogously to loading other jobs. To be able to fully use Gerber for analyses with PCB-Investigator, we offer you some **processing options**.

To "complete" Gerber files with drill holes, upload the corresponding **Excellon** file. In addition, the layer stackup needs to be defined (using the **matrix**) and possibly, for completion, the components are to be uploaded by means of a **CSV** file.

Deviations in the overlay of layers can be adjusted using "**Transform Layers**" (see slide 10).

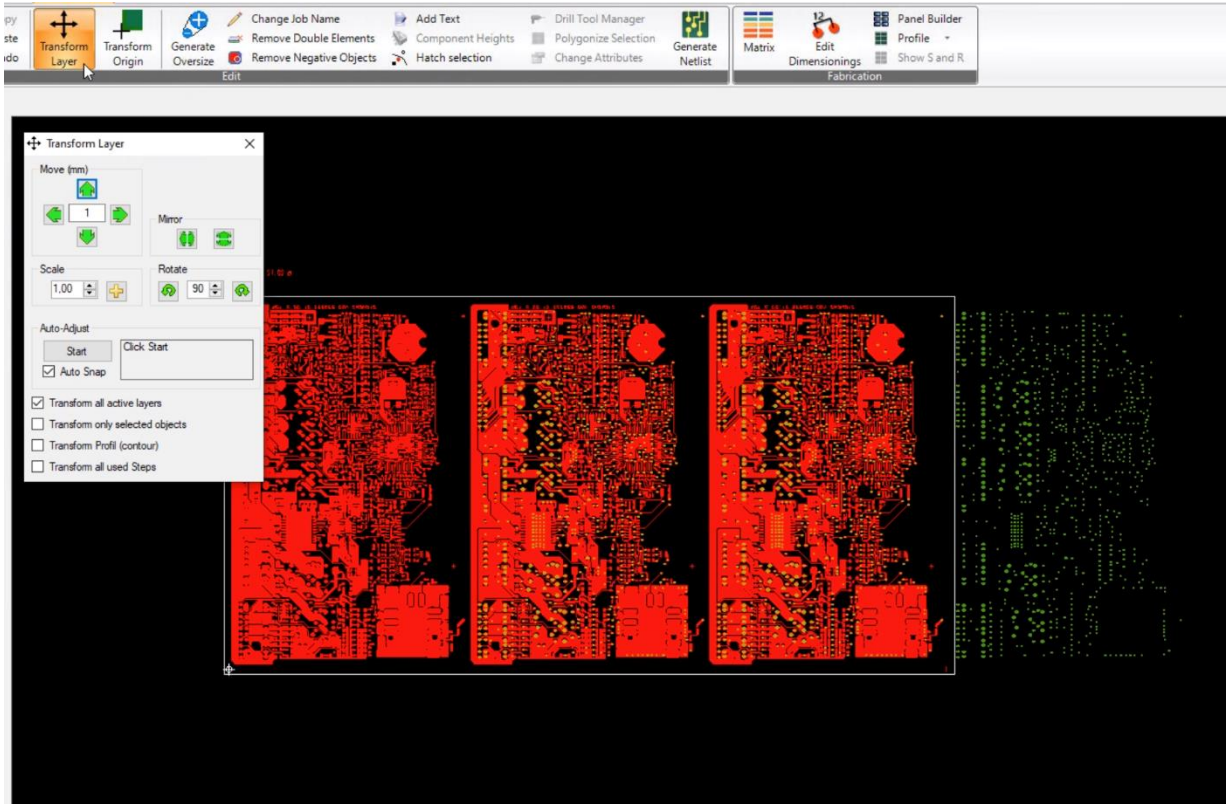
Using the **Drill Tool Manager** (Right-click on the layer stackup after a drill layer has been selected), you can define drill holes as plated and unplated (e.g. to perform a DRC analysis).





Quick Start Guide PCB-Investigator - Basics

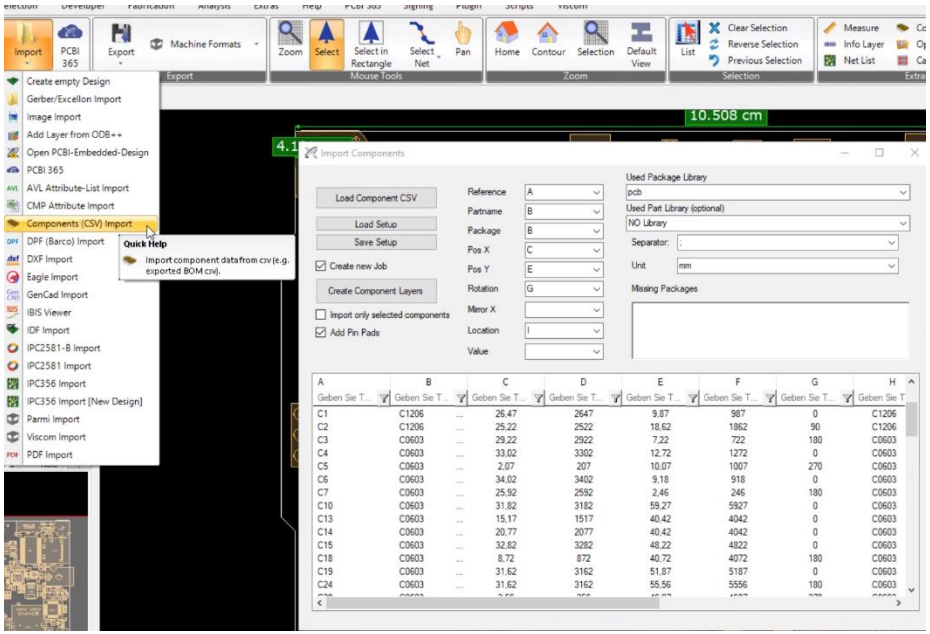
- Installation
- **Basics**
- BOM
- Working with Nets
- Working with Components
- PDF Synchronization
- Summary



Quick Start Guide PCB-Investigator - BOM

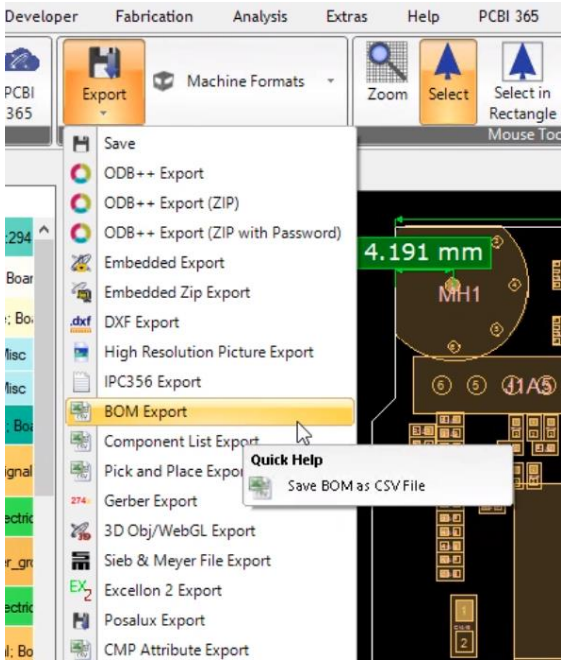
To import the BOM, you can use the **CSV Component Import**.

To view the BOM, use the **Component Manager** (like described on slide 12).



Quick Start Guide PCB-Investigator - BOM

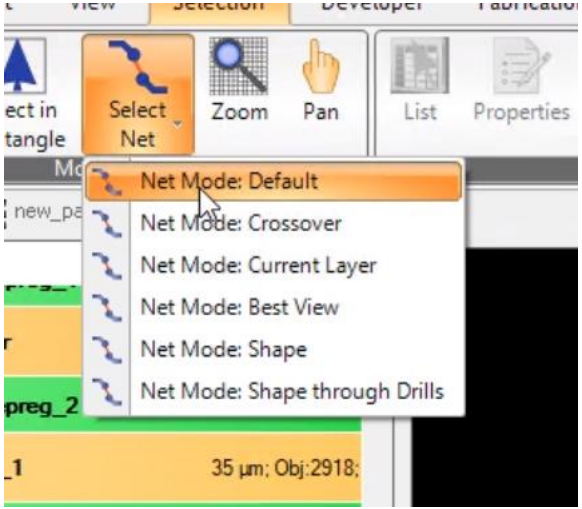
To export and **print** the **BOM**, use the "Export" tab.



Quick Start Guide PCB-Investigator – Working with Nets

To select nets, click on "Selection" in the bar above. Here, six different selection types for nets will be available. In case of missing net lists (e.g. Gerber files), especially the last two types ("Shape", "Shape through drills") are to be used.

Simply use the **left-click to select** a net in the graphical interface.

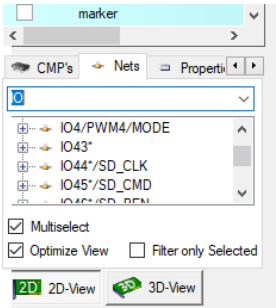


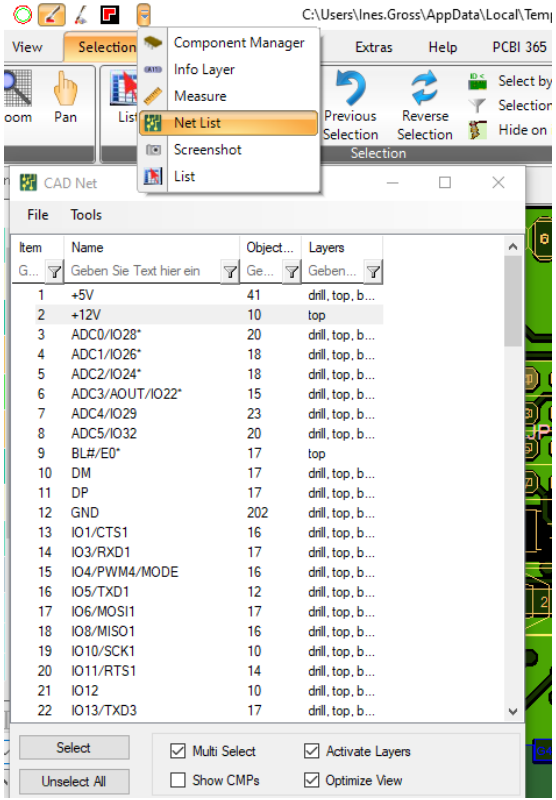
Quick Start Guide PCB-Investigator – Working with Nets

To search for nets, use the search function below the stackup menu on the left side. First, select "**Nets**". Then enter the corresponding search term to find a net.

This function can additionally be used to track the net structures of your design.

You can also use the **Netlist dialog** for this purpose. The best way to open it is via the access button on the top of the page (small blue arrow). In the Netlist dialog, you can create a new netlist under "Tools" and "Generate Netlist".



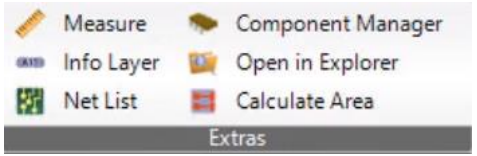


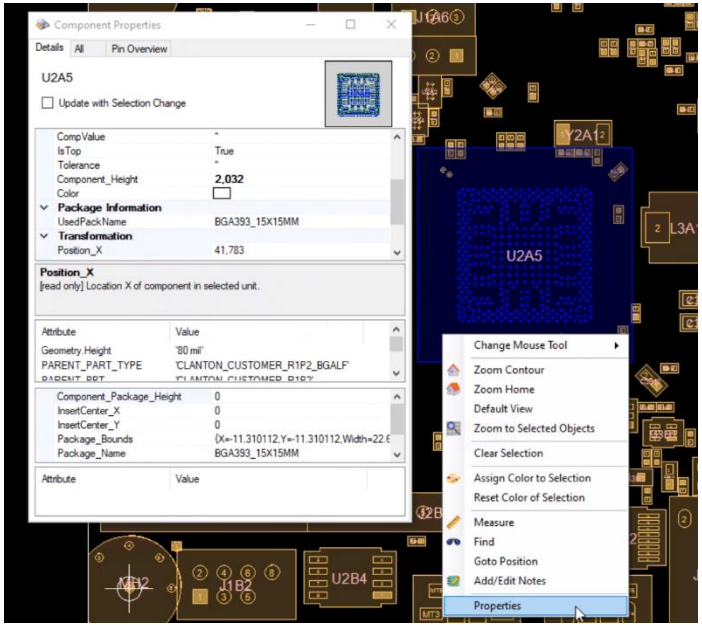
Quick Start Guide PCB-Investigator – Working with Components

The **selection of parts** is performed similarly to the selection of nets using a simple mouse click on the corresponding component in the graphical interface.

The **Component Manager** gives you access to all components in the design (see slide 16). The quickest way to open is to click on the small blue arrow in the bar at the top or you access it via the "Extras" tab.

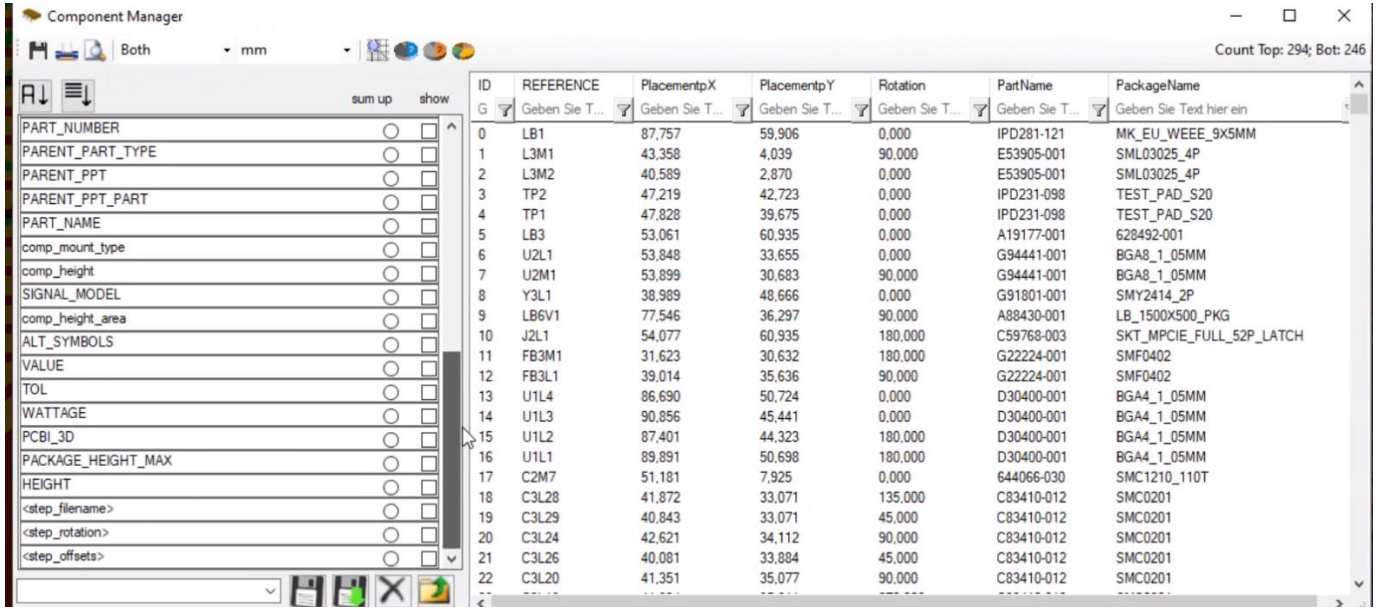
In order to access detailed **component properties** of any particular part, select a part by clicking on it. Right-click and a small window will open, click on "Properties" and you will get all the part properties (if available in your design or completed by our **Part Library**).





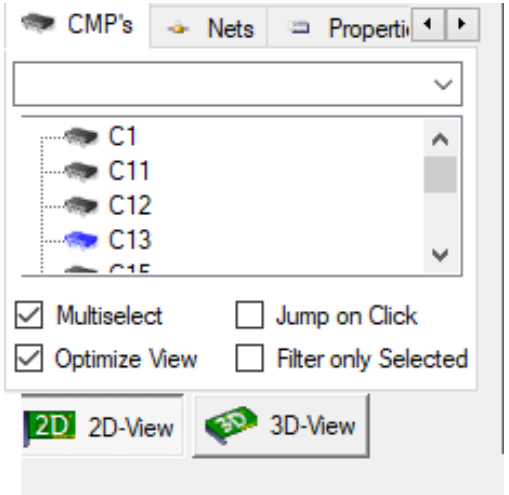
Quick Start Guide PCB-Investigator – Working with Components

- Installation
- Basics
- BOM
- Working with Nets
- **Working with Components**
- PDF Synchronization
- Summary



Quick Start Guide PCB-Investigator – Working with Components

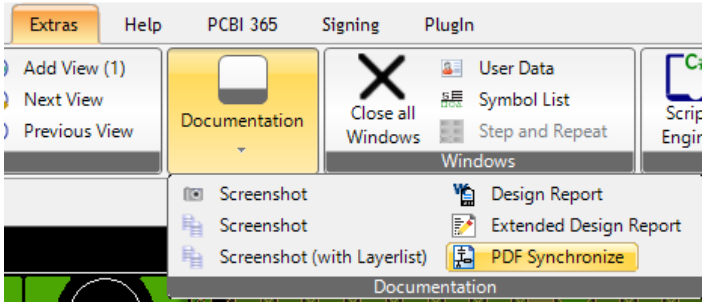
To search for components, proceed in the same way as for the Net search. Switch to the "**CMPs**" tab in order to carry out a new search for a part in the search bar using the corresponding part name.

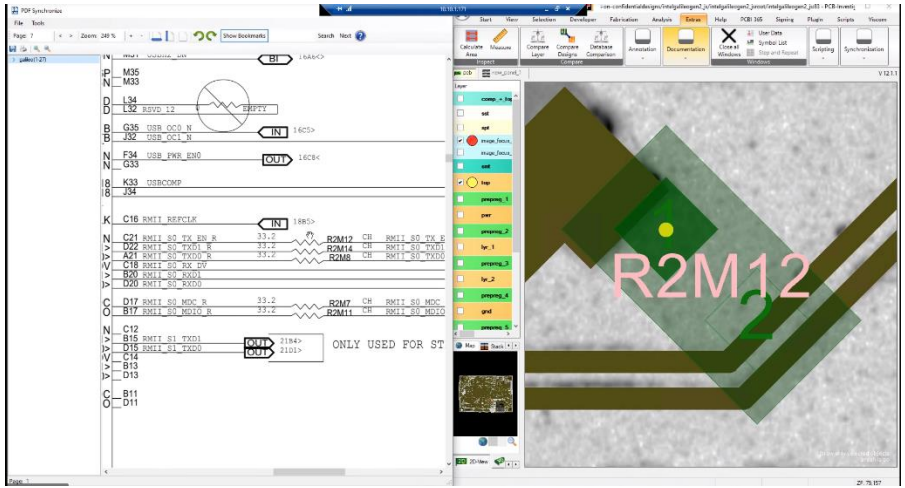


Quick Start Guide PCB-Investigator – PDF Synchronization

Use the **PDF Synchronization** tool (tab "Extras") to synchronize or match your design and the corresponding PDF document.

By right-clicking on the respective location (e.g. net or component) in the graphical interface of PCB-Investigator, the associated location within the PDF document will also be highlighted for alignment purposes.





Quick Start Guide PCB-Investigator - Summary

This **Quickstart Guide** is intended to give you a **brief overview of the main basic functionalities of PCB-Investigator** and their handling.
In case of further questions regarding the usability of PCB-Investigator, we are offering an **advanced online manual** for you. Please visit https://manual.pcb-investigator.com/.
The manual explains many **additional features and plugins** of PCB-Investigator step by step.

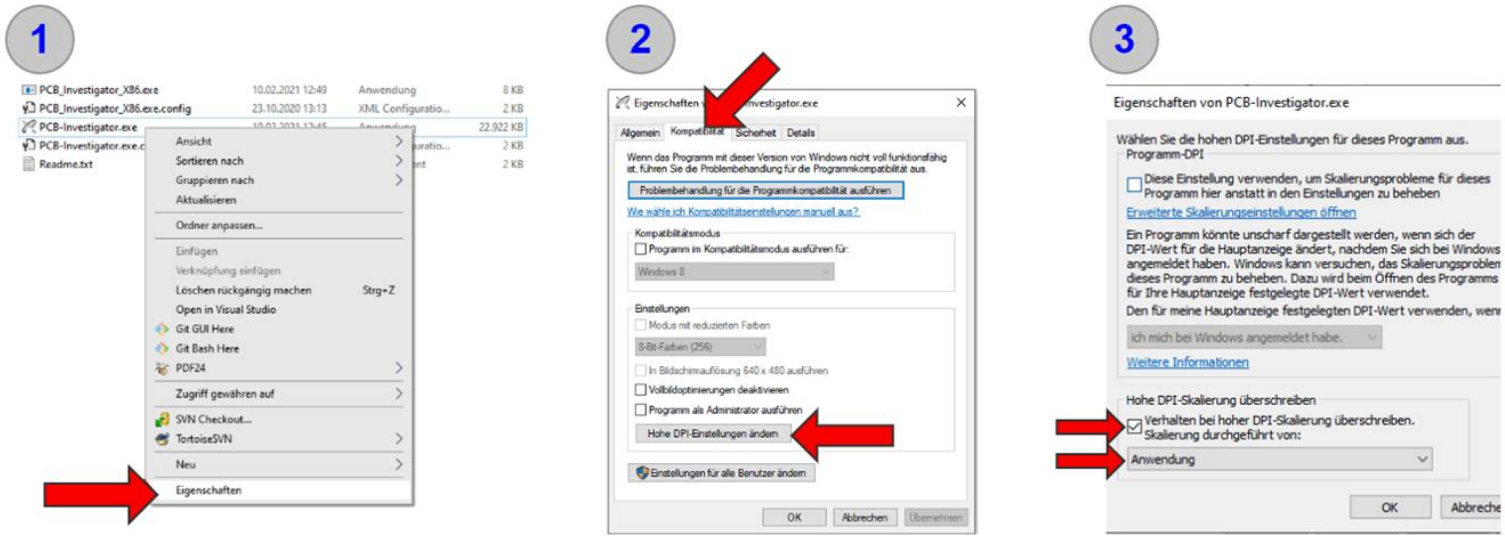
If you have any particular queries regarding the handling of PCB-Investigator, we will be happy to answer them personally, either by e-mail or by phone.

Help

Scaling

Setting the DPI for PCB-Investigator.exe.
For this purpose, the Compatibility tab must be selected in the options "Change high DPI settings" in the properties.
Then switch to "Application".

- High DPI Einstellungen



Software Portfolio

Useful Links:

PCB-Investigator
www.pcb-investigator.com

PCBi-Physics
www.PCBi-Physics.com

Native Board Import (3D Interface to CATIA, SiemensNX, SolidWorks, SolidEdge)
www.sts-development.biz

GerberLogix
www.gerberLogix.com

Online Gerber Viewer
www.Gerber-Viewer.com

Software Development, CAD Converter, data connection
www.easyLogix.de

Get in touch,
info@easylogix.de
Guenther Schindler
Tel. +49 941 568 136 26