



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



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>.



Basics



BOM



Working with Nets



Working with Components



PDF Synchronization



Summary

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



Installation



Basics



BOM



Working with
Nets



Working with
Components



PDF
Synchronization



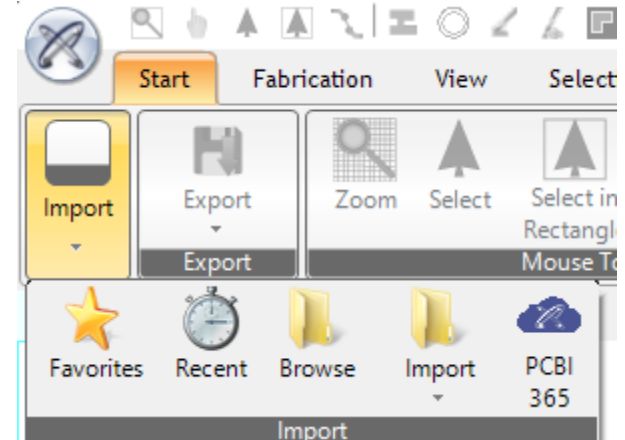
Summary

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.



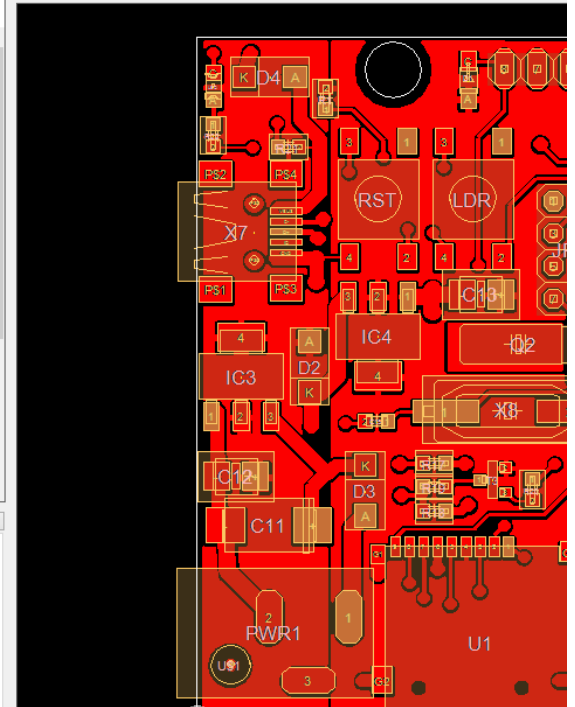
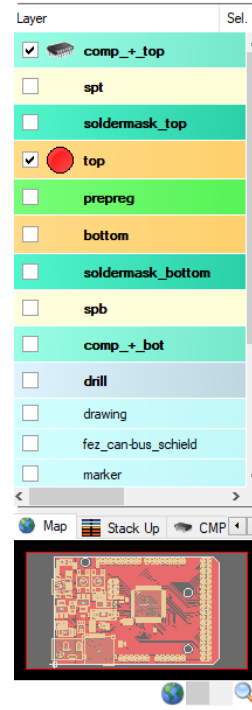
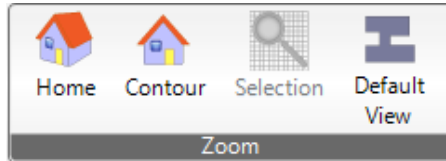
Quick Start Guide PCB-Investigator - Basics

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

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



Installation



Basics



BOM



Working with
Nets



Working with
Components



PDF
Synchronization



Summary

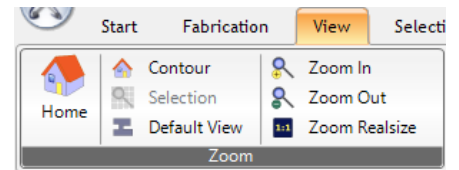
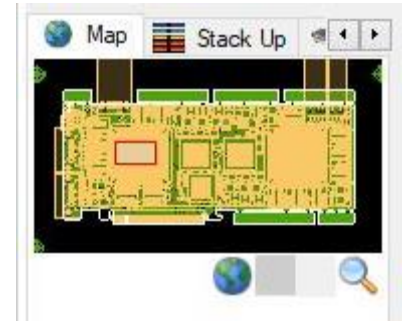
The "**Zoom**" option allows you to see your design in more detail.

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



Installation



Basics



BOM



Working with Nets



Working with Components



PDF Synchronization



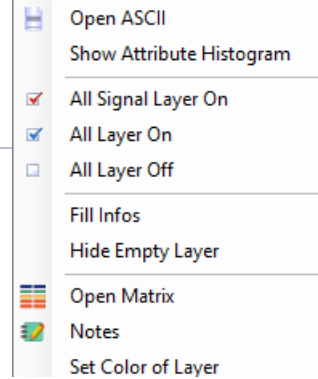
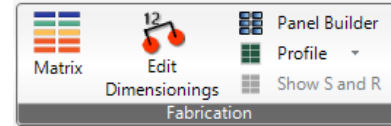
Summary

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.



File Tools Edit

Layername	Context	Type	Drill Layer
			Start Layer <input type="text"/> <input type="radio"/> Via Filling Copper
			End Layer <input type="text"/> <input type="radio"/> Via Filling Ink
			<input checked="" type="radio"/> Via no Filling
Polarity <input type="text"/>	Add Type <input type="text"/>	Layer Height <input type="text"/> µm	Layer Color <input type="text"/>
<input checked="" type="checkbox"/> Plating <input type="text"/> µm			

Layername	Context	Type	Thickness	Properties
COMP_+_TOP		Component		
SPT		Solderpaste	0.00 µm	
SOLDERMASK_TOP	other	Soldermask	0.00 µm	layermaterial = other
1 TOP	Copper:18	Signal	18.00 µm	.comment = Copper:18 // .copperwt
PREPREG	other	Dielectric	1218.00 µm	layerdielectric = 47.9528 // layer_ms
2 BOTTOM	Copper:18	Signal	18.00 µm	layermaterial = Copper:18 // .copper
SOLDERMASK_BOTTOM	other	Soldermask	0.00 µm	layermaterial = other
SPB		Solderpaste	0.00 µm	
COMP_+_BOT		Component		
DRILL		Drill		
DRAWING		Document		
FEZ_CAN-BUS_SCHIELD		Document		

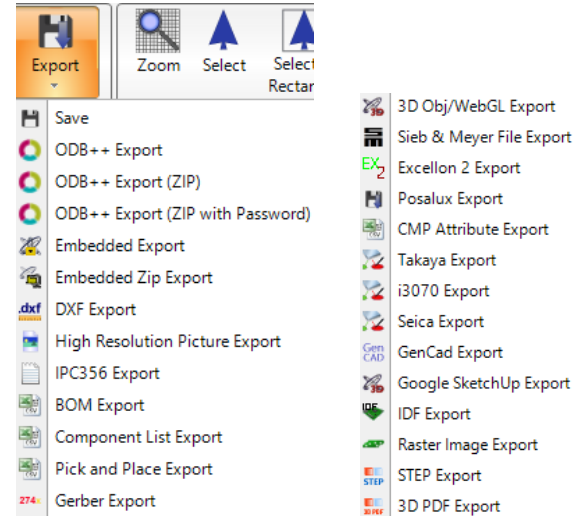
Height: 1254.00 µm

Quick Start Guide PCB-Investigator - Basics

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

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



Installation



Basics



BOM



Working with Nets



Working with Components



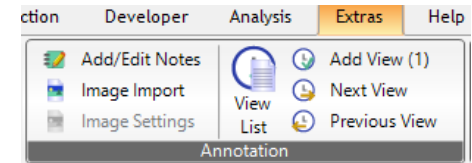
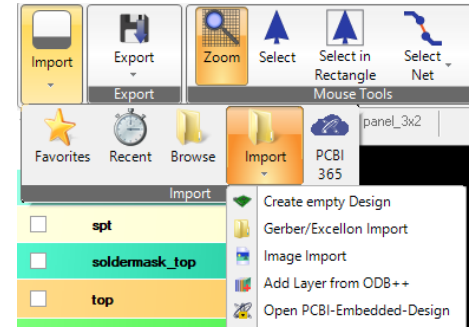
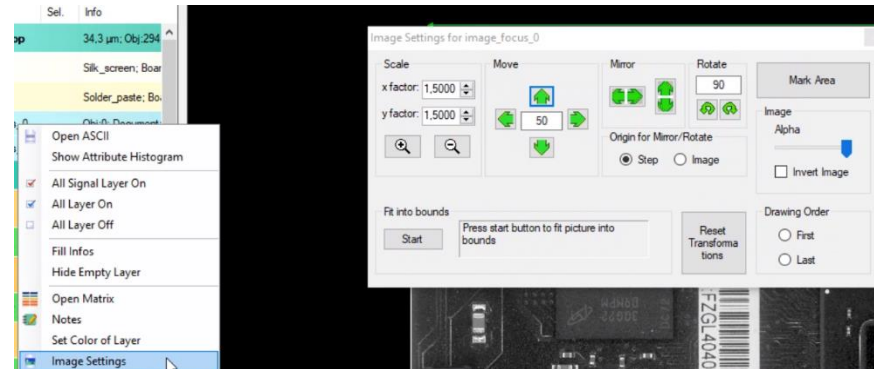
PDF Synchronization



Summary

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

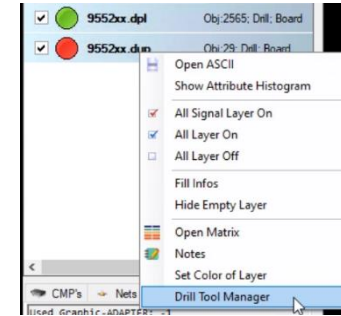
- Installation
- Basics
- BOM
- Working with Nets
- Working with Components
- PDF Synchronization
- Summary

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).



Drill Tool Manager

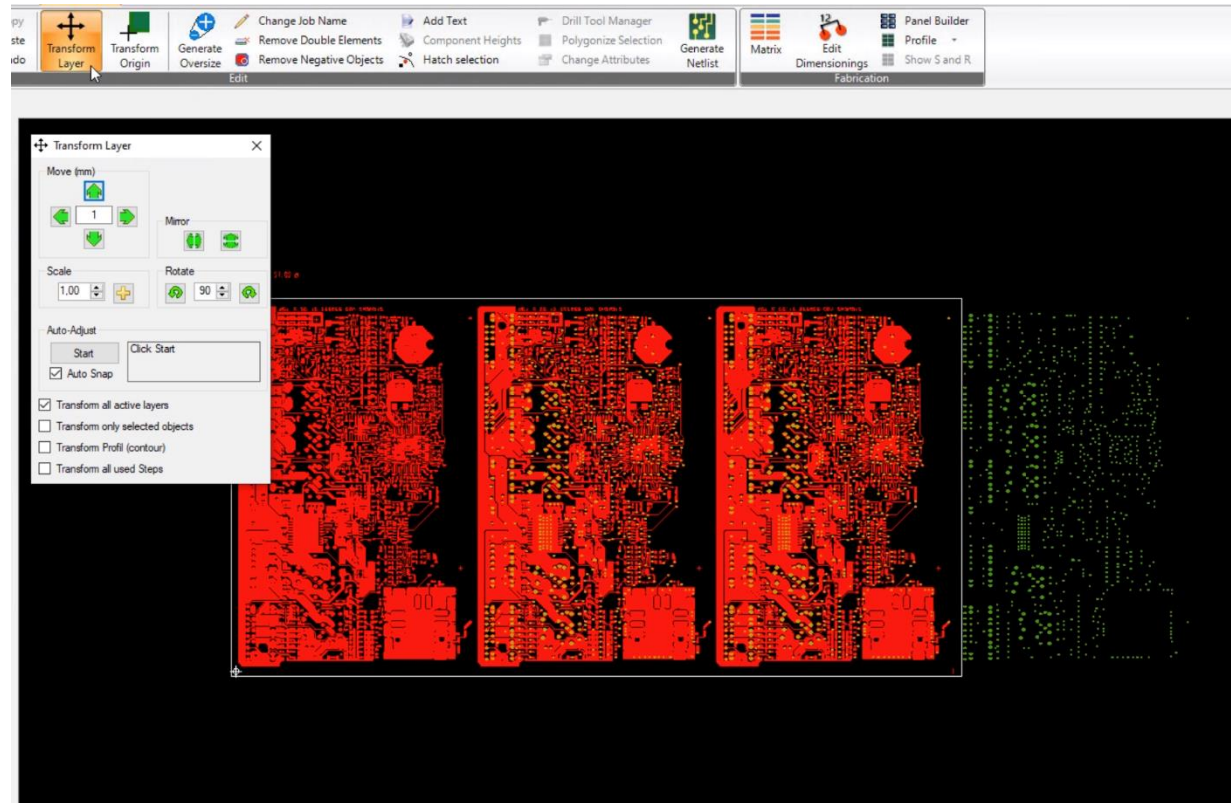
Layename: 9552xx.dup
Board Thickness: 0
User Parameters:

Tool/DCode	Type	finish size	+Tol	-Tol	drill size	Count	
1 1	NON_PLATED	1100	0	0	1100	6	Select
2 2	NON_PLATED	2300	0	0	2300	3	Select
3 3	NON_PLATED	2500	0	0	2500	9	Select
4 4	PLATED	3600	0	0	3600	3	Select
5 5	PLATED	4000	0	0	4000	8	Select

Apply and Save Close Clear Selection Sum: 29 Reset All

Quick Start Guide PCB-Investigator - Basics

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



Quick Start Guide PCB-Investigator - BOM

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

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

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

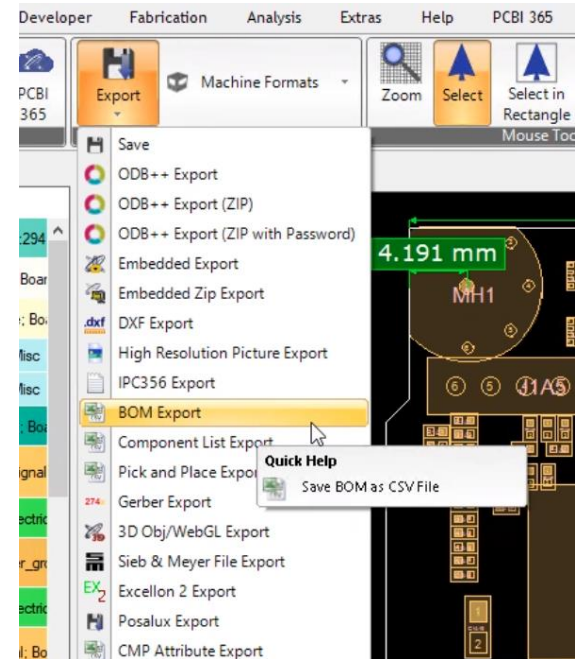
The screenshot shows the PCB-Investigator software interface. The 'Import Components' dialog box is open, displaying the 'Load Component CSV' section. A 'Quick Help' tooltip is visible over the 'Components (CSV) Import' menu item, stating: 'Import component data from csv (e.g. exported BOM.csv)'. The dialog also shows fields for Reference, Partname, Package, Pos X, Pos Y, Rotation, Minor X, Location, and Value. The 'Used Package Library' is set to 'pcb'. Below the dialog, a table displays the BOM data:

A	B	C	D	E	F	G	H
Geben Sie T...	Geben Sie T...	Geben Sie T...	Geben Sie T...	Geben Sie T...	Geben Sie T...	Geben Sie T...	Geben Sie T...
C1	C1206	...	26.47	2647	9.87	987	0
C2	C1206	...	25.22	2522	18.62	1862	90
C3	C0603	...	29.22	2922	7.22	722	180
C4	C0603	...	33.02	3302	12.72	1272	0
C5	C0603	...	2.07	207	10.07	1007	270
C6	C0603	...	34.02	3402	9.18	918	0
C7	C0603	...	25.52	2552	2.46	246	180
C10	C0603	...	31.82	3182	59.27	5927	0
C13	C0603	...	15.17	1517	40.42	4042	0
C14	C0603	...	20.77	2077	40.42	4042	0
C15	C0603	...	32.82	3282	48.22	4822	0
C18	C0603	...	8.72	872	40.72	4072	180
C19	C0603	...	31.62	3162	51.87	5187	0
C24	C0603	...	31.62	3162	55.56	5556	180

Quick Start Guide PCB-Investigator - BOM

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

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

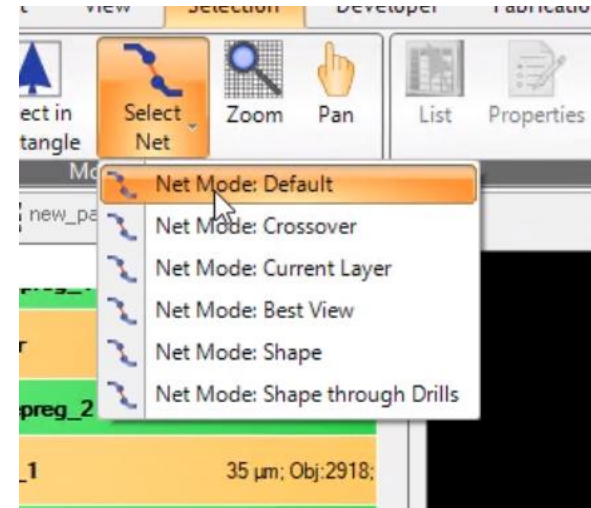


Quick Start Guide PCB-Investigator – Working with Nets

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

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.



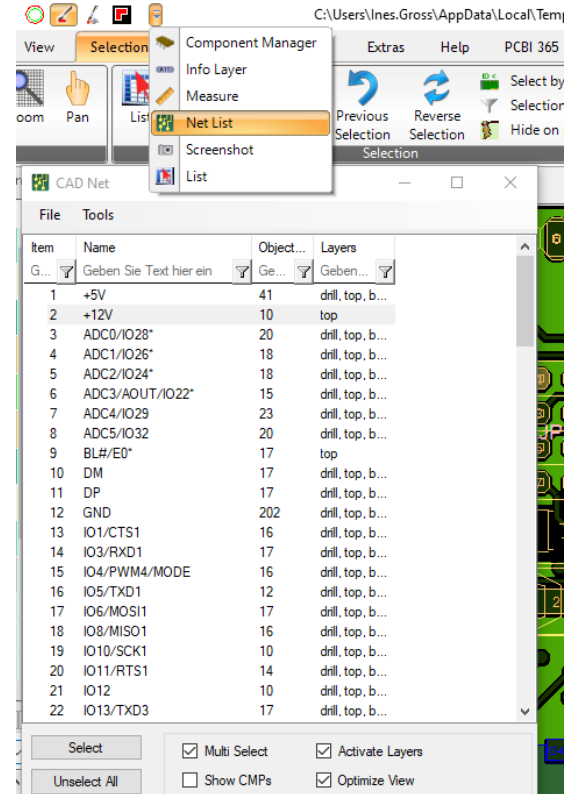
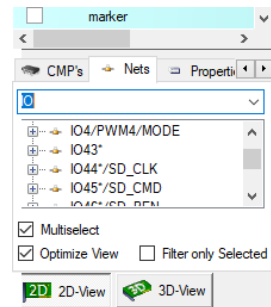
Quick Start Guide PCB-Investigator – Working with Nets

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

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".



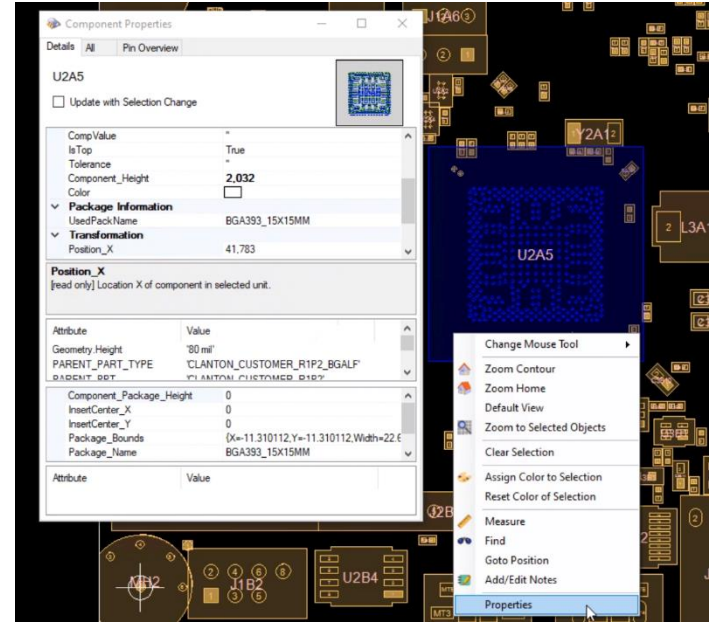
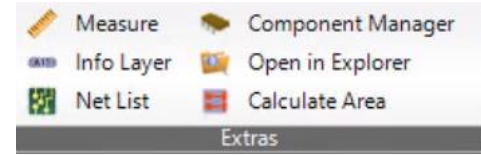
Quick Start Guide PCB-Investigator – Working with Components

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








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








Component Manager

Both mm

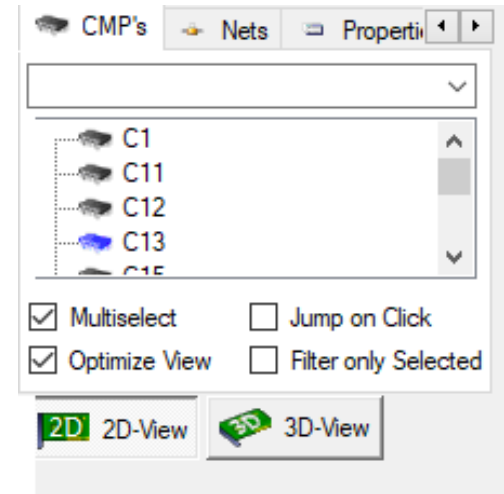
Count Top: 294; Bot: 246

ID	REFERENCE	PlacementX	PlacementY	Rotation	PartName	PackageName
0	LB1	87,757	59,906	0,000	IPD281-121	MK_EU_WEEE_9X5MM
1	L3M1	43,358	4,039	90,000	E53905-001	SML03025_4P
2	L3M2	40,589	2,870	0,000	E53905-001	SML03025_4P
3	TP2	47,219	42,723	0,000	IPD231-098	TEST_PAD_S20
4	TP1	47,828	39,675	0,000	IPD231-098	TEST_PAD_S20
5	LB3	53,061	60,935	0,000	A19177-001	628492-001
6	U2L1	53,848	33,655	0,000	G94441-001	BGA8_1_05MM
7	U2M1	53,899	30,683	90,000	G94441-001	BGA8_1_05MM
8	Y3L1	38,989	48,666	0,000	G91801-001	SMY2414_2P
9	LB6V1	77,546	36,297	90,000	A88430-001	LB_1500X500_PKG
10	J2L1	54,077	60,935	180,000	C59768-003	SKT_MPCIE_FULL_52P_LATCH
11	FB3M1	31,623	30,632	180,000	G22224-001	SMF0402
12	FB3L1	39,014	35,636	90,000	G22224-001	SMF0402
13	U1L4	86,690	50,724	0,000	D30400-001	BGA4_1_05MM
14	U1L3	90,856	45,441	0,000	D30400-001	BGA4_1_05MM
15	U1L2	87,401	44,323	180,000	D30400-001	BGA4_1_05MM
16	U1L1	89,891	50,698	180,000	D30400-001	BGA4_1_05MM
17	C2M7	51,181	7,925	0,000	644066-030	SMC1210_110T
18	C3L28	41,872	33,071	135,000	C83410-012	SMC0201
19	C3L29	40,843	33,071	45,000	C83410-012	SMC0201
20	C3L24	42,621	34,112	90,000	C83410-012	SMC0201
21	C3L26	40,081	33,884	45,000	C83410-012	SMC0201
22	C3L20	41,351	35,077	90,000	C83410-012	SMC0201

Quick Start Guide PCB-Investigator – Working with Components

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

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.

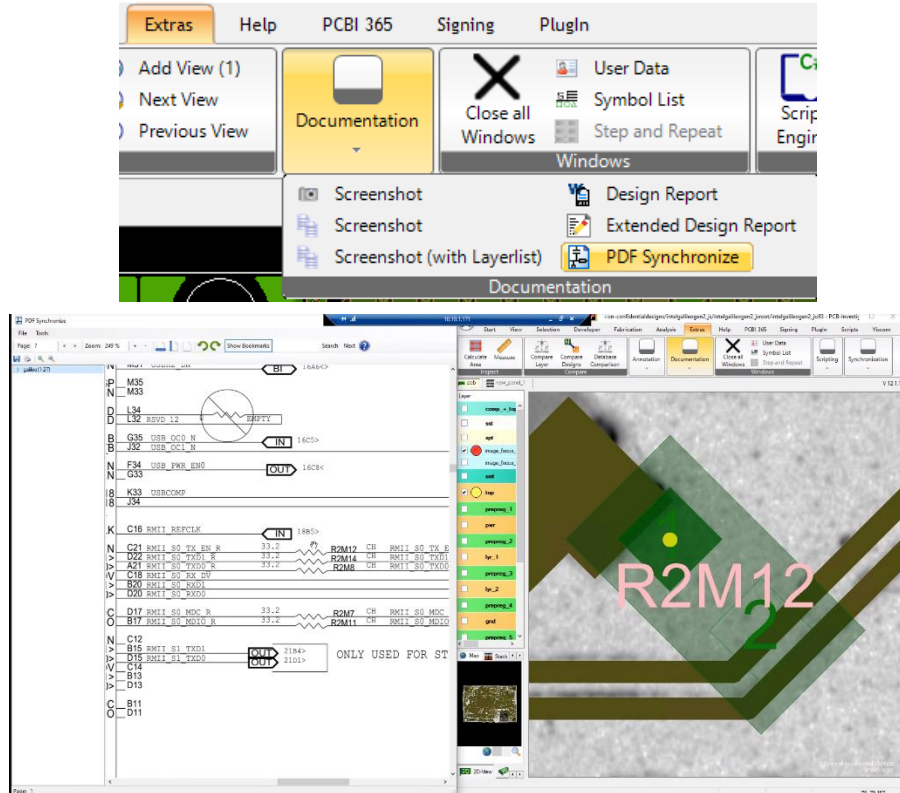


Quick Start Guide PCB-Investigator – PDF Synchronization








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

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

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

The image shows a three-step process to configure high DPI settings for the application:

- 1**: A file explorer window shows the file list for 'PCB-Investigator.exe'. A red arrow points to the 'Eigenschaften' (Properties) option in the context menu.
- 2**: The 'Eigenschaften von PCB-Investigator.exe' dialog box is open, with the 'Kompatibilität' (Compatibility) tab selected. A red arrow points to the 'Kompatibilität' tab. Another red arrow points to the 'Hohe DPI-Einstellungen ändern' (Change high DPI settings) button.
- 3**: The 'Eigenschaften von PCB-Investigator.exe' dialog box is open, with the 'Hohe DPI-Einstellungen ändern' dialog box displayed. Two red arrows point to the 'Verhalten bei hoher DPI-Skalierung überschreiben' (Override high DPI scaling behavior) checkbox, which is checked, and the 'Anwendung' (Application) dropdown menu.

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