

FORMATION PCB Investigator – niveau Avancé – Livret d'exercices en anglais



INTRODUCTION



Exclusive supplier in France of a range of software dedicated to electronics and embedded systems, EDA Expert brings its technical expertise and helps industrial companies define suitable solutions for the design and manufacturing of electronic systems. With over 20 years of experience in the field of electronics, we provide our clients with a comprehensive view from design to manufacturing, with an unbiased perspective on the software market: advanced advice and expertise, technical support, software integration and customization, and assistance with getting started.

EDA Expert is registered as a training organization under the number 11 94 08144 94 with the Prefect of the Ile de France region and referenced in Datadock under identification number 0004166. Since January 2021, EDA Expert is also certified QUALIOPi under the number FR063306-1, ensuring compliance with the standards set by the National Repository for the quality of actions contributing to skills development.



The quality certification has been issued for the categories of training actions

Furthermore, the trainers at EDA Expert are all electronics engineers with over 20 years of experience. They are certified and accredited by Altium and also hold CID certification from IPC. Furthermore, the trainers at EDA Expert are all electronics engineers with over 20 years of experience. They are certified and accredited by CID certification from IPC.



SOMMAIRE



MODULE 1: Fabrication..... - 4 -

MODULE 2: Analysis - 12 -

MODULE 3 : Net & Components - 22 -

MODULE 4 : Extras..... - 31 -

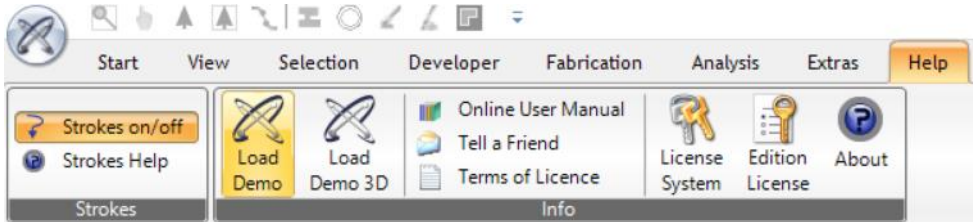
MODULE 1: Fabrication



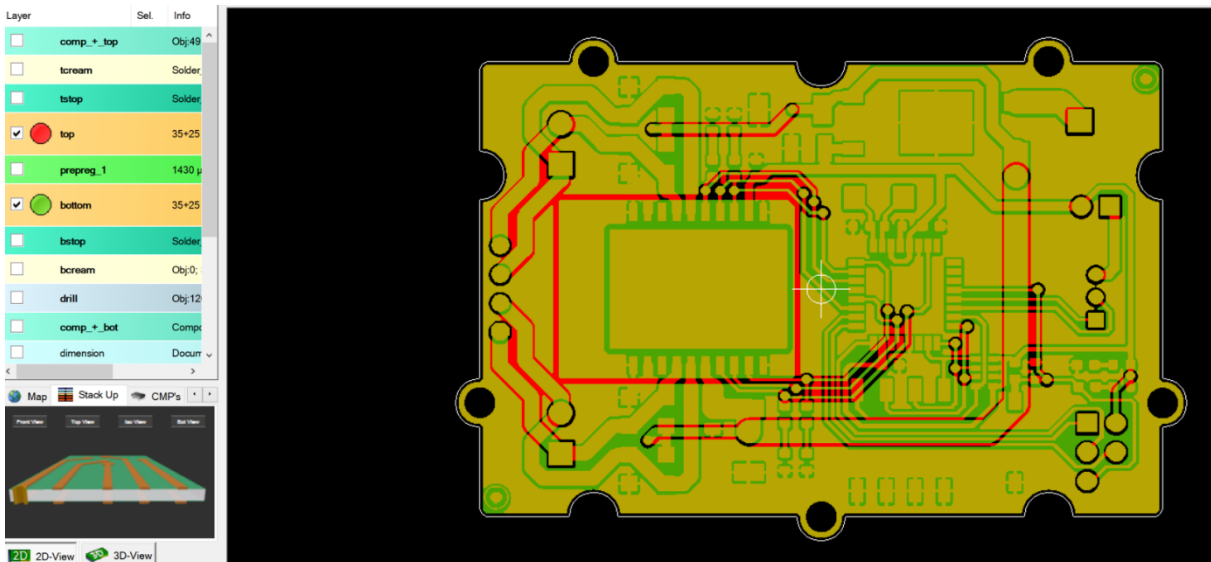


Objectives : in this module, we will show the matrix definition and the fabrication functions

From the menu **Help**, select **Load Demo** to open the demonstration project used for this training:

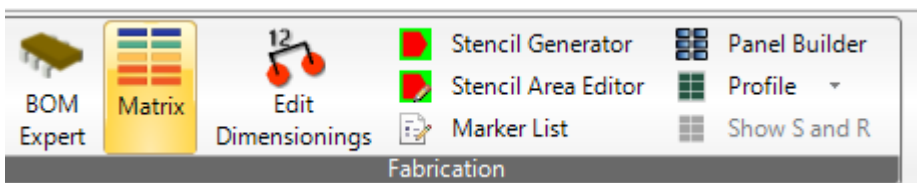


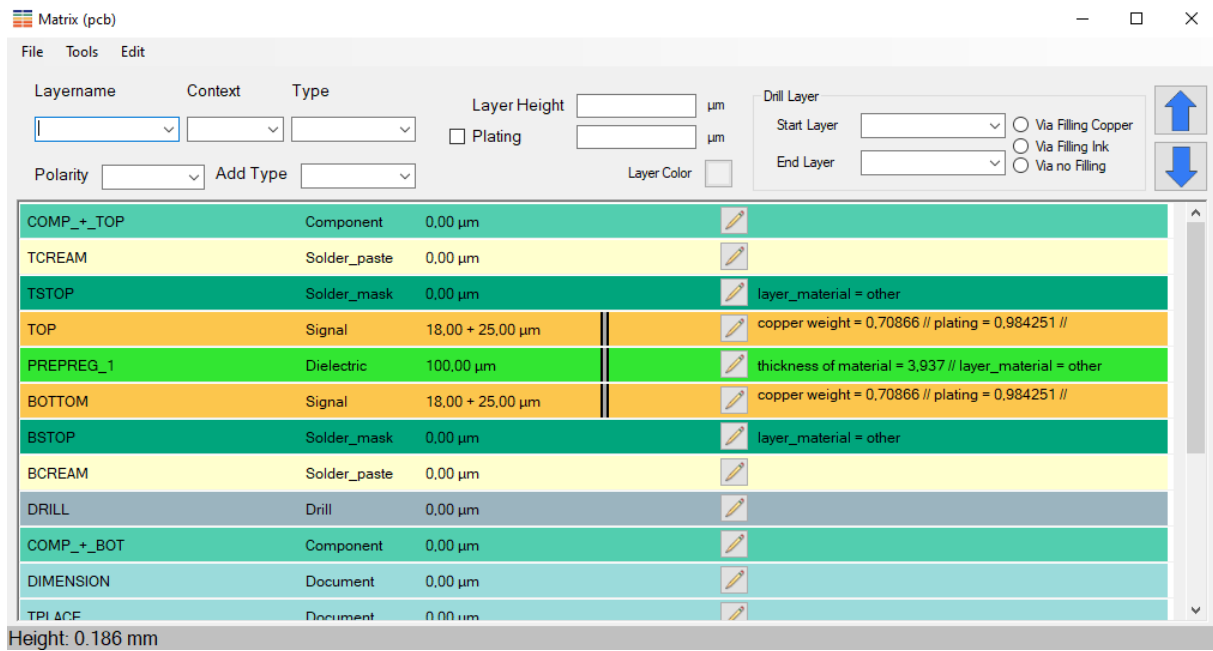
By selecting Top and Bottom layers, you will see the fabrication files:



A. Matrix

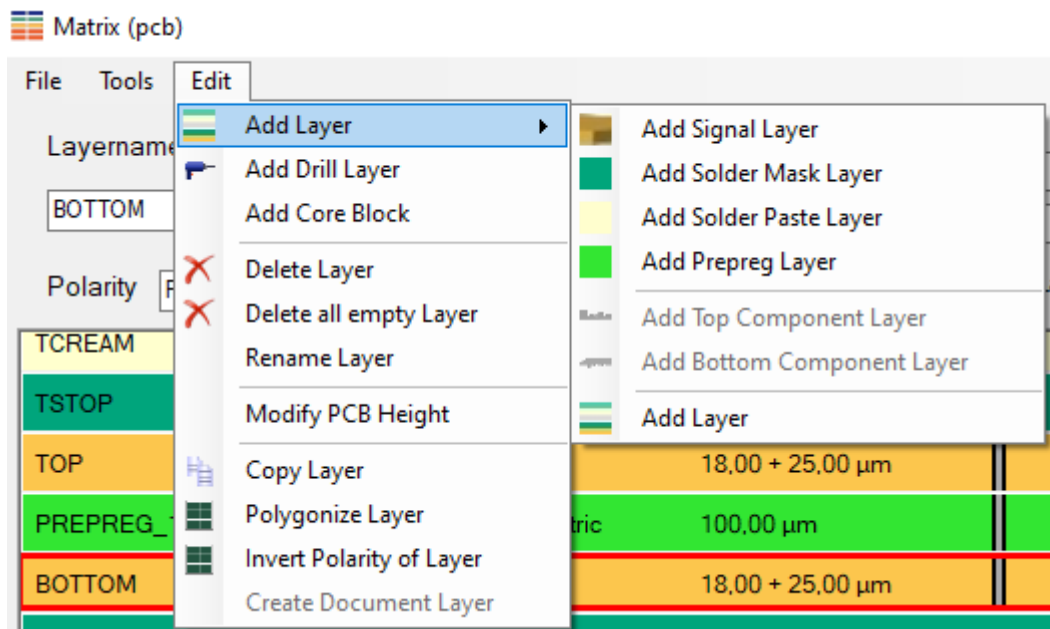
From the **Fabrication** menu, select **Matrix** for opening the matrix definition and its options:





In the window we can add new layer, change the layout and positions of the layers, create Drill and use an existing layout.

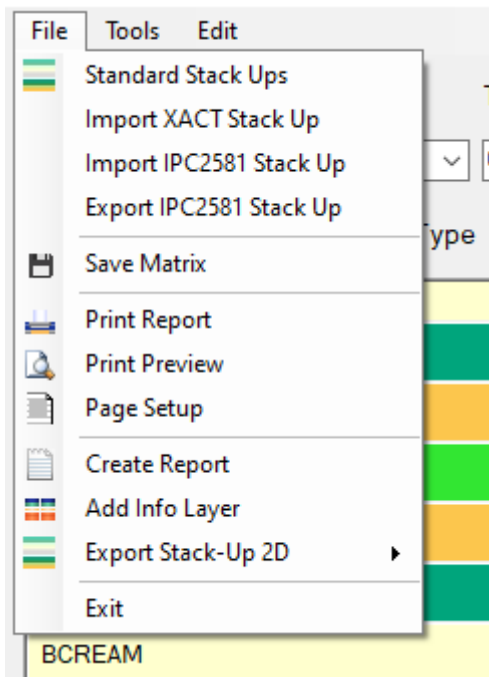
To edit layer, go to Edit menu:



We can edit the layout by adding new layers, deleting them. We can also edit the layer directly in the menu below:

Layername	Context	Type	Layer Height	18,00	µm	Drill Layer
BOTTOM	Board	Signal	<input checked="" type="checkbox"/> Plating	25,00	µm	Start Layer
Polarity	Positive	Add Type	Unknown	Layer Color		End Layer
						TOP
						BOTTOM
						<input type="radio"/> Via Filling Copper
						<input type="radio"/> Via Filling Ink
						<input type="radio"/> Via no Filling

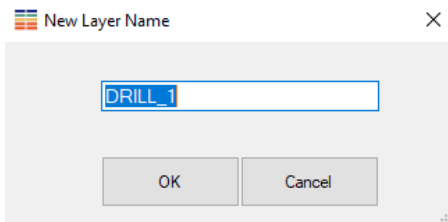
In the file menu , we can import an existing stackup, or export the one we created.



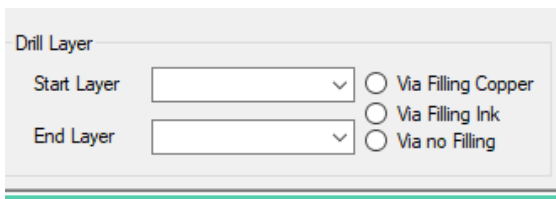
Drag a layer to change its position.

COMP+_TOP	Component	0,00 µm	
TCREAM	Solder_paste	0,00 µm	
TSTOP	Solder_mask	0,00 µm	layer_material = other
TOP	Signal	18,00 + 25,00 µm	copper weight = 0,700
TEST_LAYER	Solder_mask	0,00 µm	
PREPREG_1	Dielectric	100,00 µm	thickness of material =
BOTTOM	COMP+_TOP	Component	0,00 µm
BSTOP	TCREAM	Solder_paste	0,00 µm
BCREAM	TSTOP	Solder_mask	0,00 µm layer_material = other
	TOP	Signal	18,00 + 25,00 µm copper weight = 0,70866 // plating
	PREPREG_1	Dielectric	100,00 µm thickness of material = 3,937 // lay
	BOTTOM	Signal	18,00 + 25,00 µm copper weight = 0,70866 // plating
	BSTOP	Solder_mask	0,00 µm layer_material = other
	BCREAM	Solder_paste	0,00 µm
	TEST_LAYER	Solder_mask	0,00 µm
	DRILL	Drill	0,00 µm

We can create a new Drill layer with the layers that are going to be your drill. Go to **Edit** and select **Add Drill**, and specify the name:



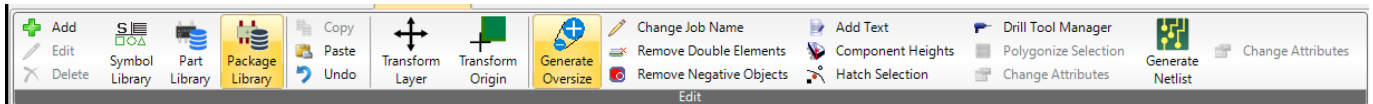
And select this new Drill layer and specify the Start and End Layers:



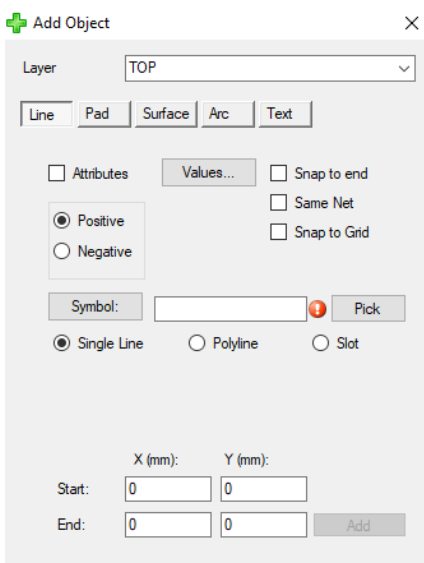
Close the matrix menu without saving the change.

B. Add object

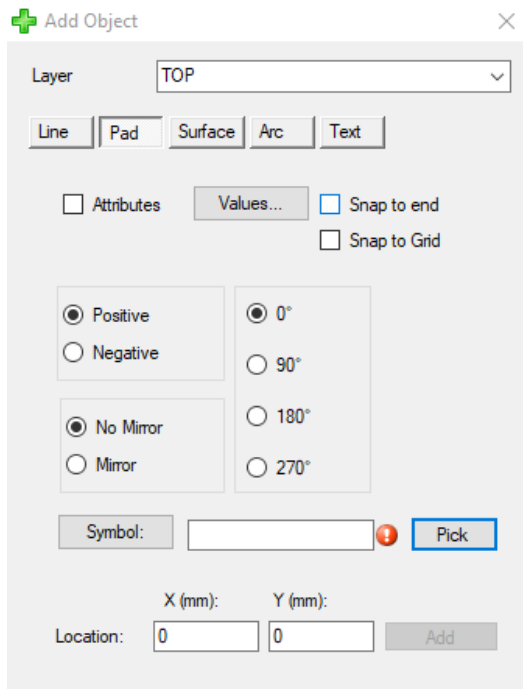
To add a new object, go to **Fabrication -> Edit**:



And select **Add**:

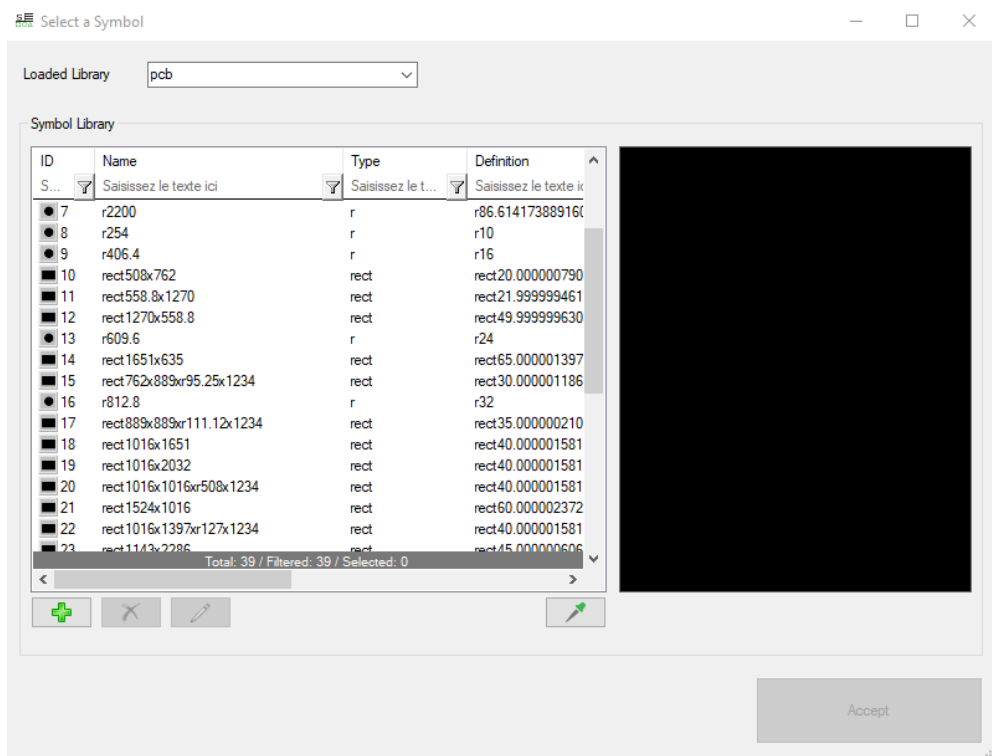


We can select the **Pad** option and choose **Symbol**:



Click on it to have more options or write directly on the blank space if you know the reference.

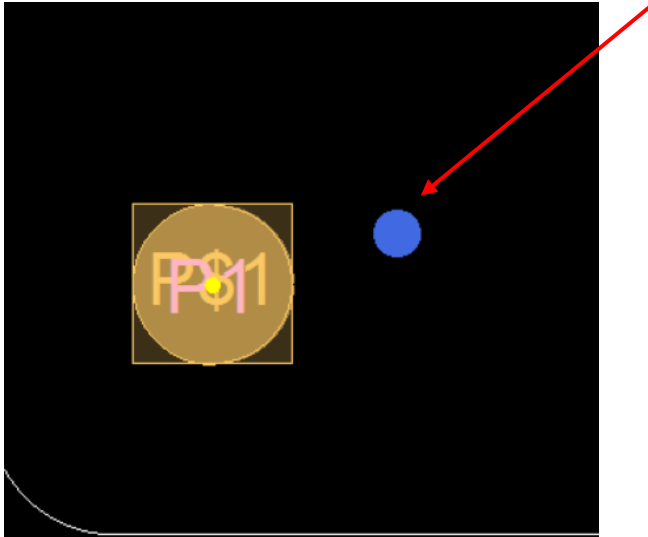
When you click on the button **Symbol**, a window shows up to select the Pad:



After selecting and accepting, click on the button **Add** to add this pad on the layer (by default, at the origin) :

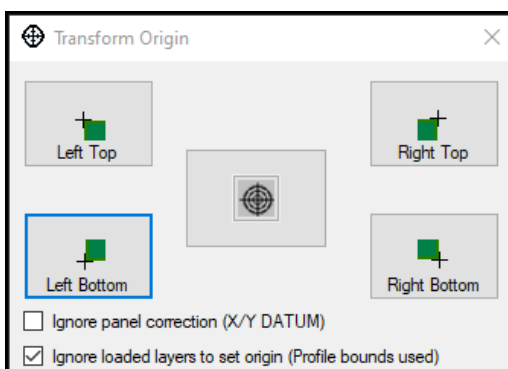
Location: X (mm): 0 Y (mm): 0 Add

Without selecting the pad from the list, you can also choose the existing pad from the fabrication files, and move your mouse to the location, double click to place this pad:

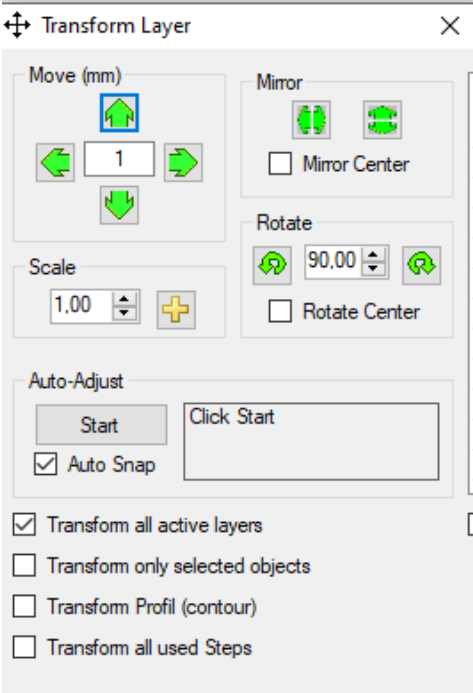


C. Transform Origin / Layer

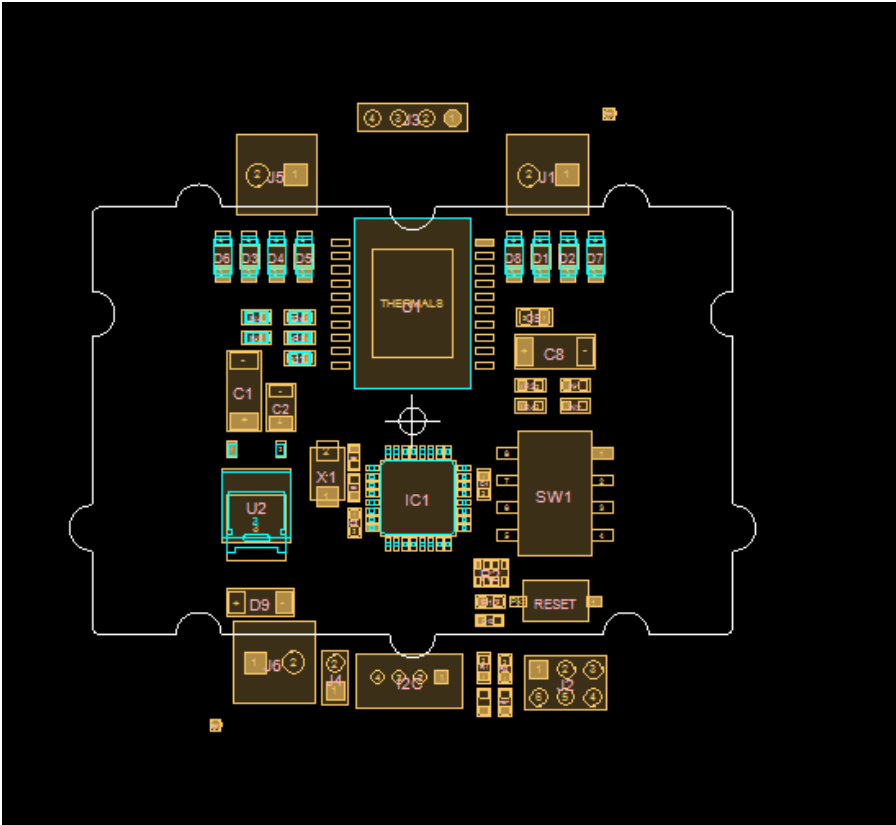
From **Fabrication** -> **Edit** -> **Transform Origin**, we can change the origin of our PCB. this will help to calculate better. Depending on the placement of the origin, the values can be more precis.



Also, we can change the layers positions and rotate them by selecting **Transform Layer**



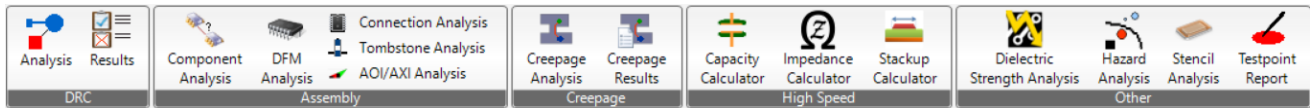
For instance, when clicking on the rotate 90° on the right, all the active layer will turn 90°:



MODULE 2: Analysis



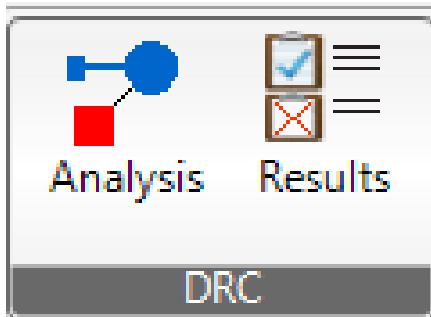
From the menu **Analysis**, there is different types of analysis we can perform:



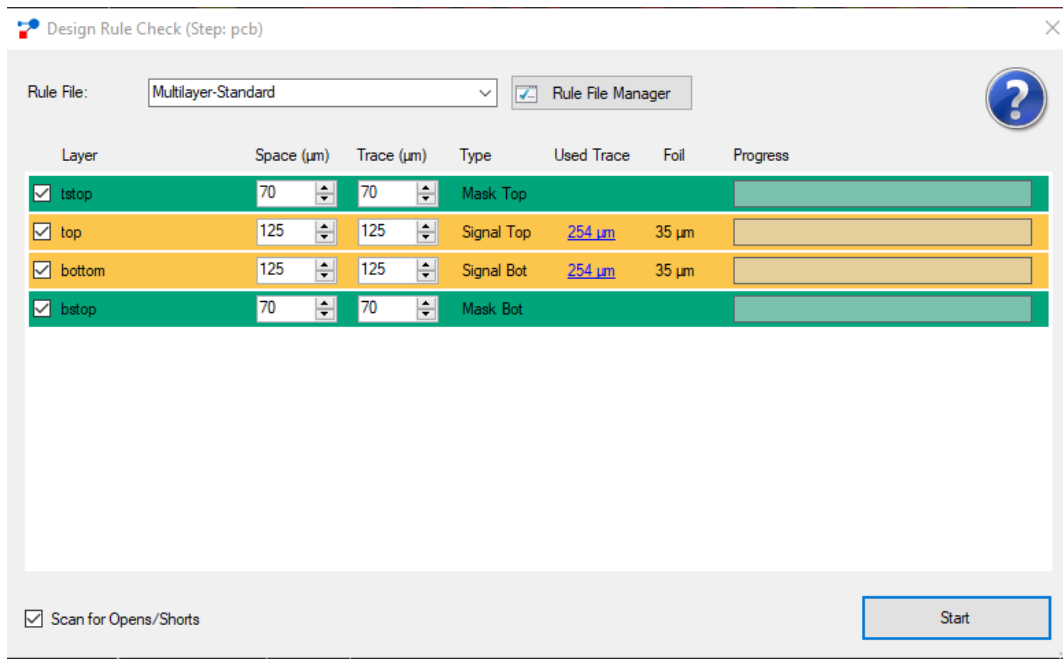
NB: Types of analysis depend PCB Investigator's Editions, please check your edition and available analysis.

A. DRC

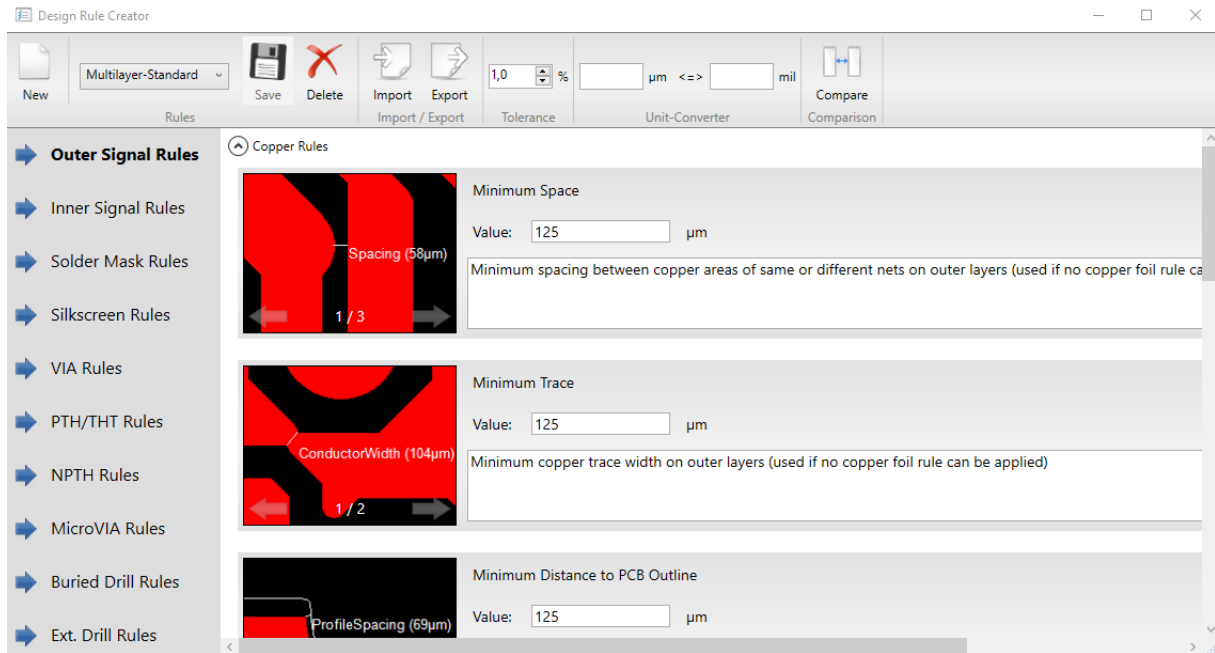
Let's starting with the DRC analysis.



When clicking on the Analysis button, a new window appears:

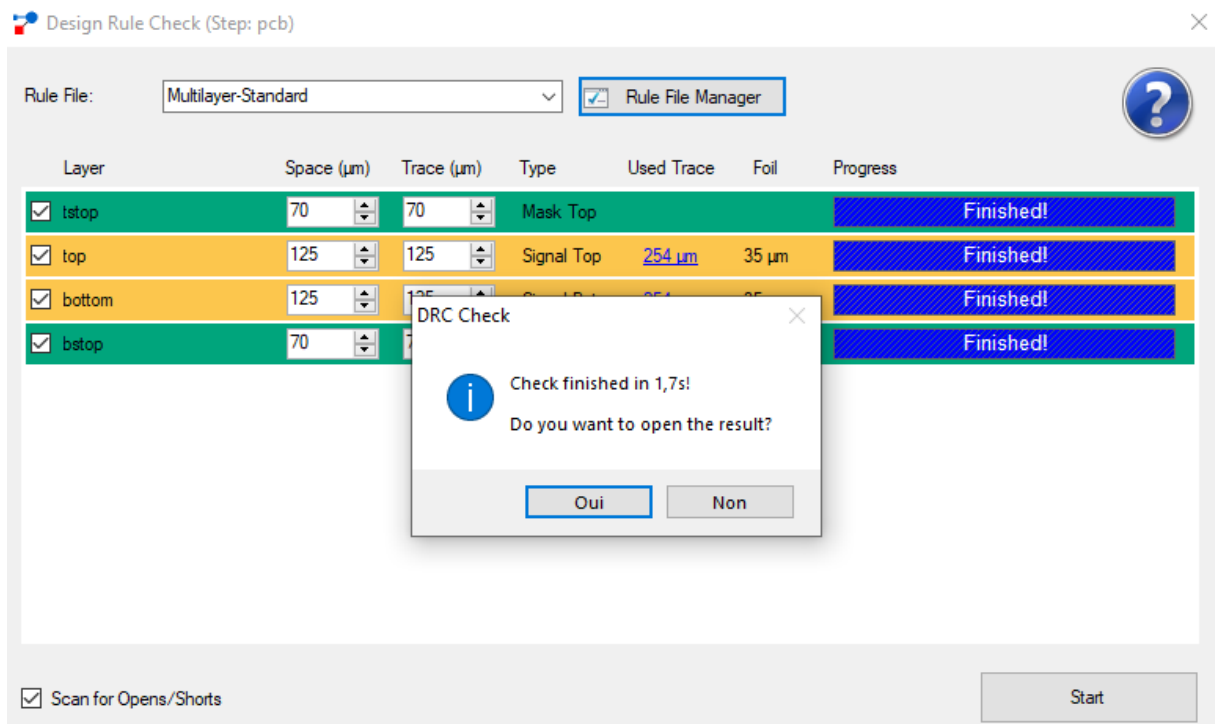


Select Rule File Manager:

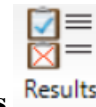


You can modify the rules with your specifications.

After finishing the modifications, click on **Save**, close the window, then click on **Start**:

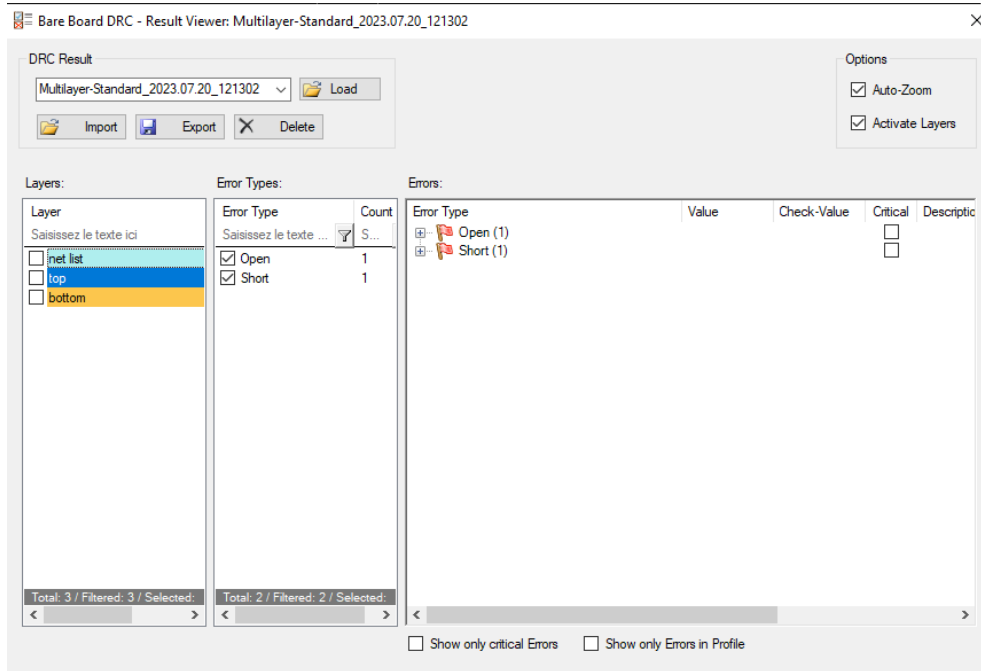


Click on **Yes** to visualize the results.

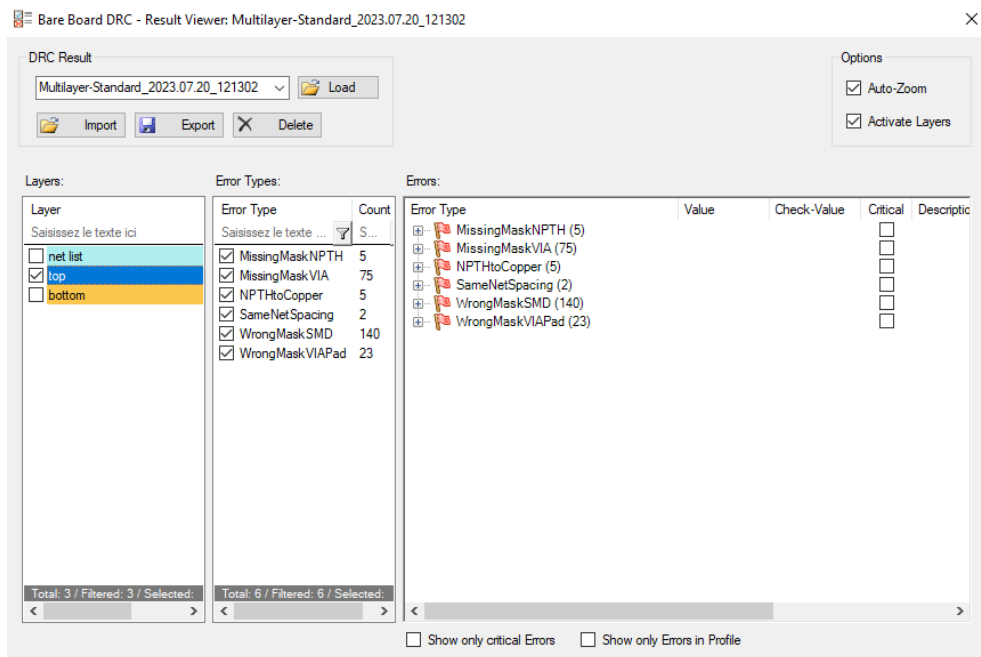


NB: you can also close the window and click on the button **Results** to see the results later.

From DRC Result, you can load a former analysis:



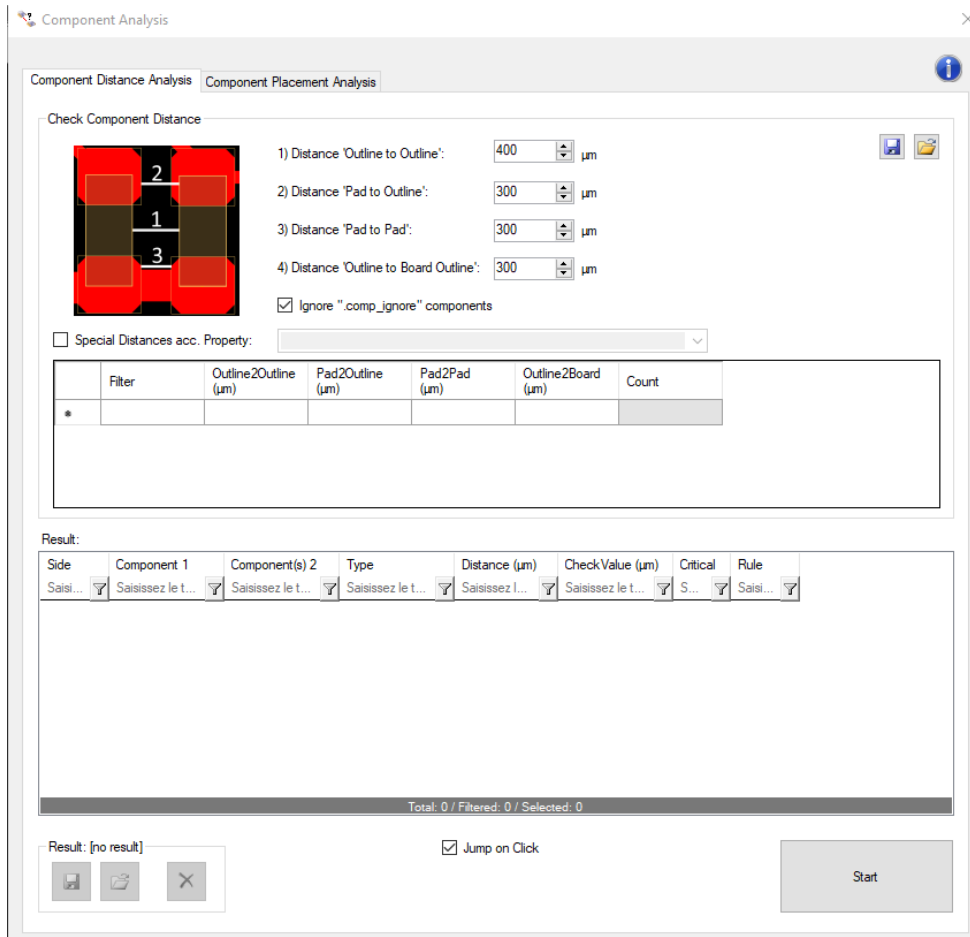
Choose top or bottom, select Error Types, and Errors to view the errors on the fabrication files.



Close without saving.

B. Component Analysis

Select Component Analysis from the menu Analysis:



We start with choosing the parameters we'd like to apply, then click on start to get the results.

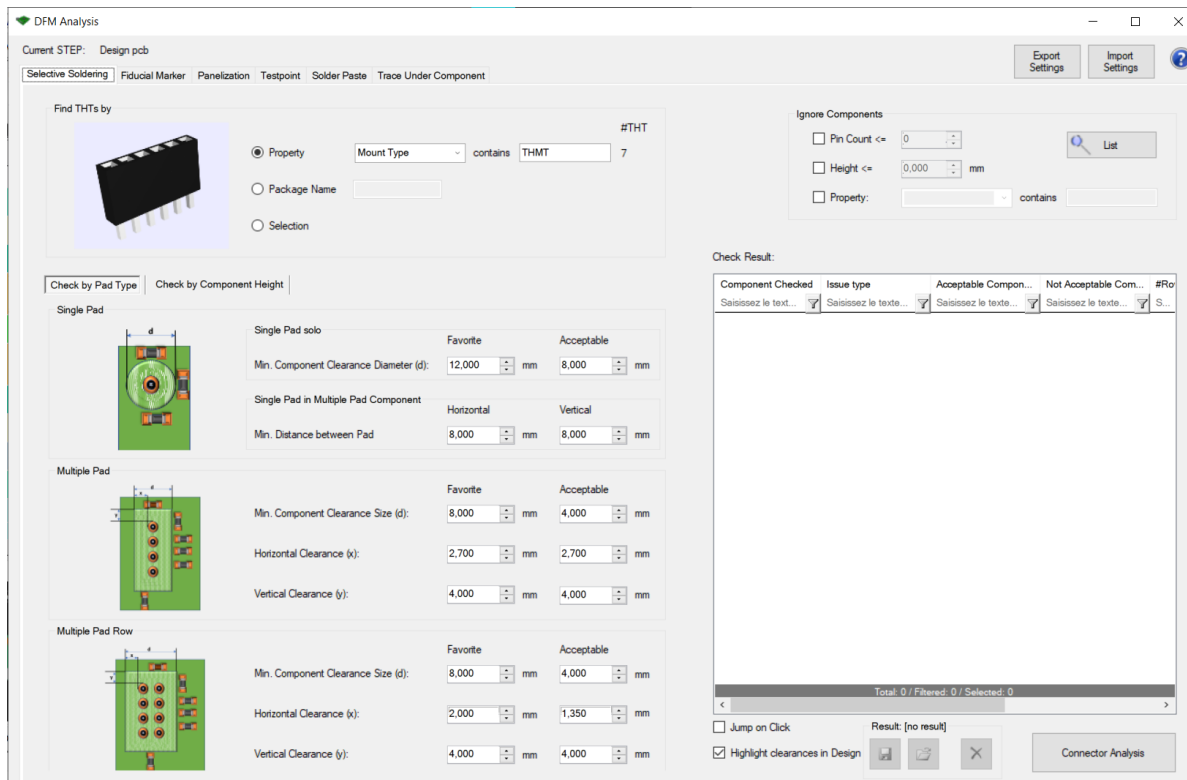
Result:

Side	Component 1	Component(s) 2	Type	Distance (µm)	Check Value (µm)	Critical	Rule
Top	C6	C9	Outline2Outline	254	400	<input type="checkbox"/>	Default
Top	R9	RST	Outline2Outline	318	400	<input type="checkbox"/>	Default
Top	X1	C6	Outline2Outline	369	400	<input type="checkbox"/>	Default
Top	X1	C9	Outline2Outline	369	400	<input type="checkbox"/>	Default
Top	C2	C1	Outline2Outline	381	400	<input type="checkbox"/>	Default

C. DFM analysis



Select **DFM Analysis** from the menu **Analysis**:



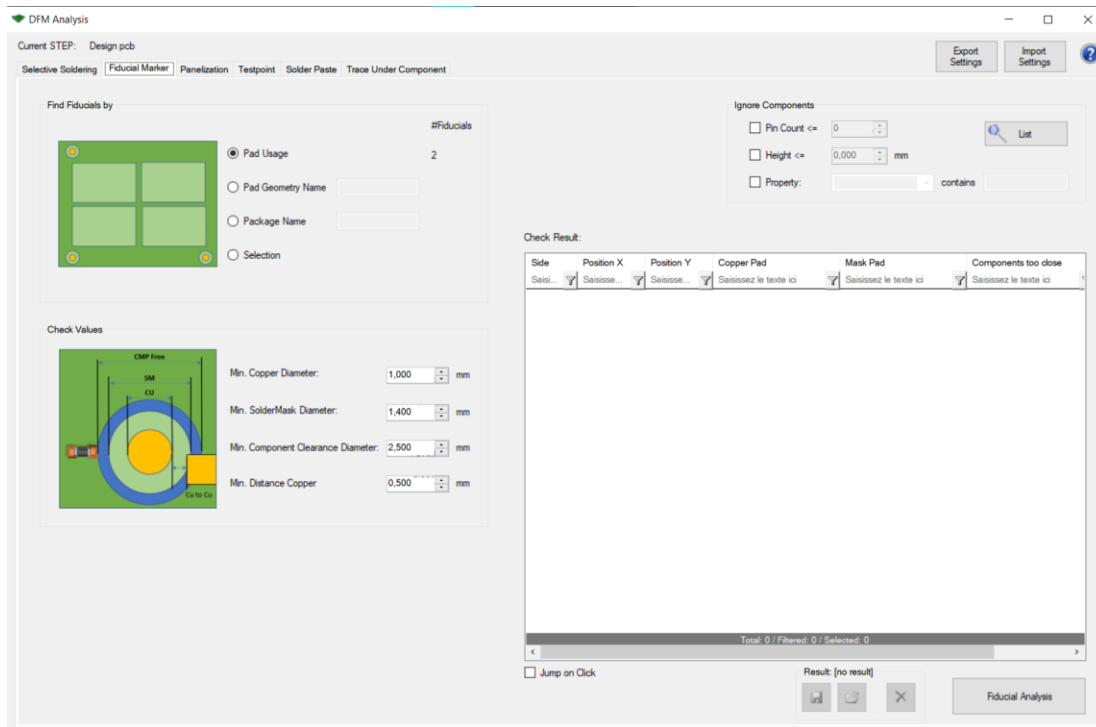
1) Selective Soldering.

This checks whether the set will fit at the given location or whether the pins of the component may be too close to the opposite side. For the settings, we start by modifying the single pad, multi pad and multi pad row. Those 3 are distinct. For each case, we can individually define the settings. That will be used for the analysis. The pictures are intended to illustrate these variables. Each variable is determined by two values, the favorite or optimum value and the “acceptable” value up to which manufacturability is still guaranteed.

The results show up in the table on the right. It is exportable into HTML format. If the component is labeled yellow that means it is defined in the range, we set. But if it's red labeled, it's far from the range defined.

2) Fiducial Solder Mark

This analysis allows you to check whether the markers set for AOI analysis can be implemented according to the selected specifications.

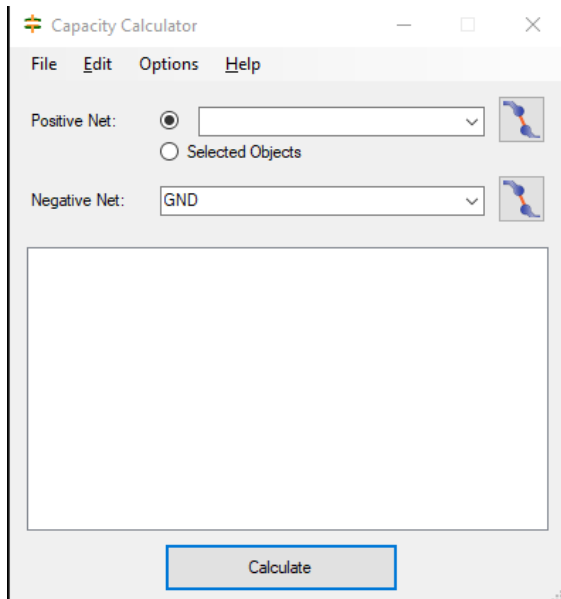


To do this, we start by looking at the fiducials in the design and knowing which are the options to use from the 4 options on the list.

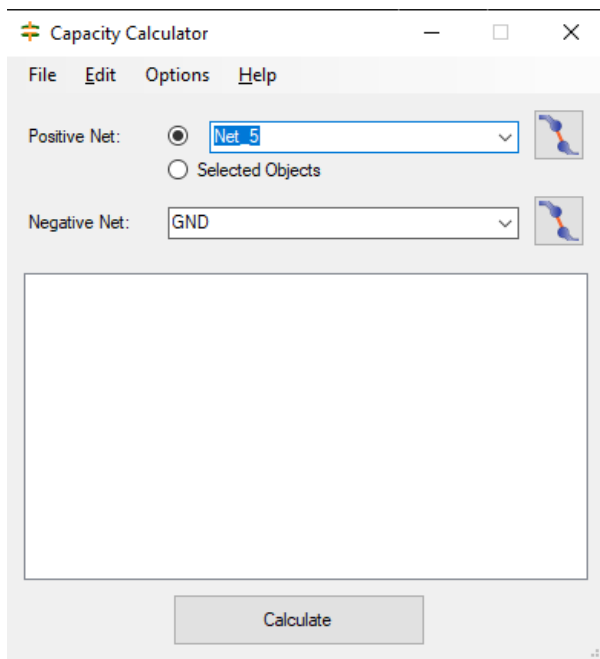
D. Capacity / Impedance Calculator

1. Capacity Calculator

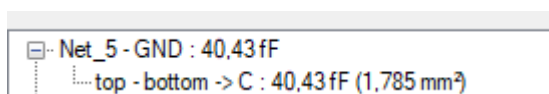
From the menu **Analysis**, select **Capacity Calculator**:



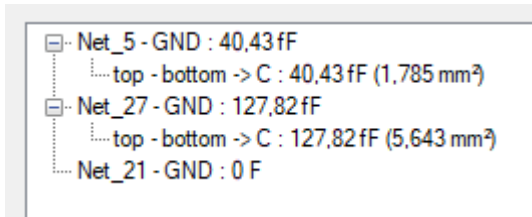
Add Positive Net from the list, or click on the button  and select the net from the fabrication file. Then click on the button **Calculate**:



We got our result:

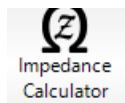


We can have more results of other nets if we keep selecting nets and calculate again.

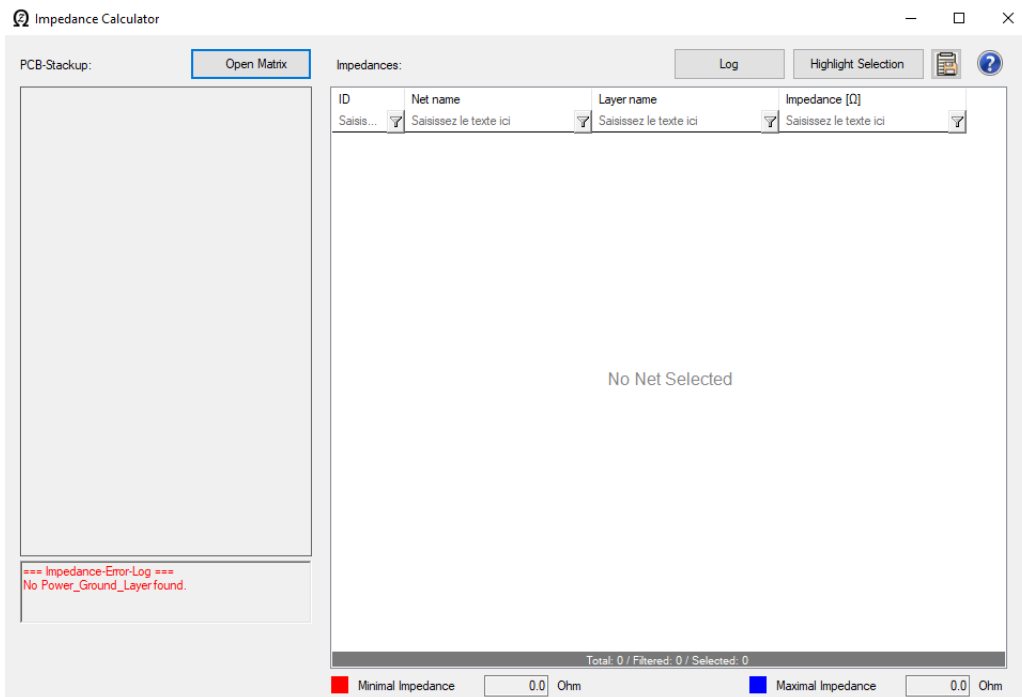


N.B: we can choose the positive net and the negative net to do the capacity between components. About here, we left it on GND.

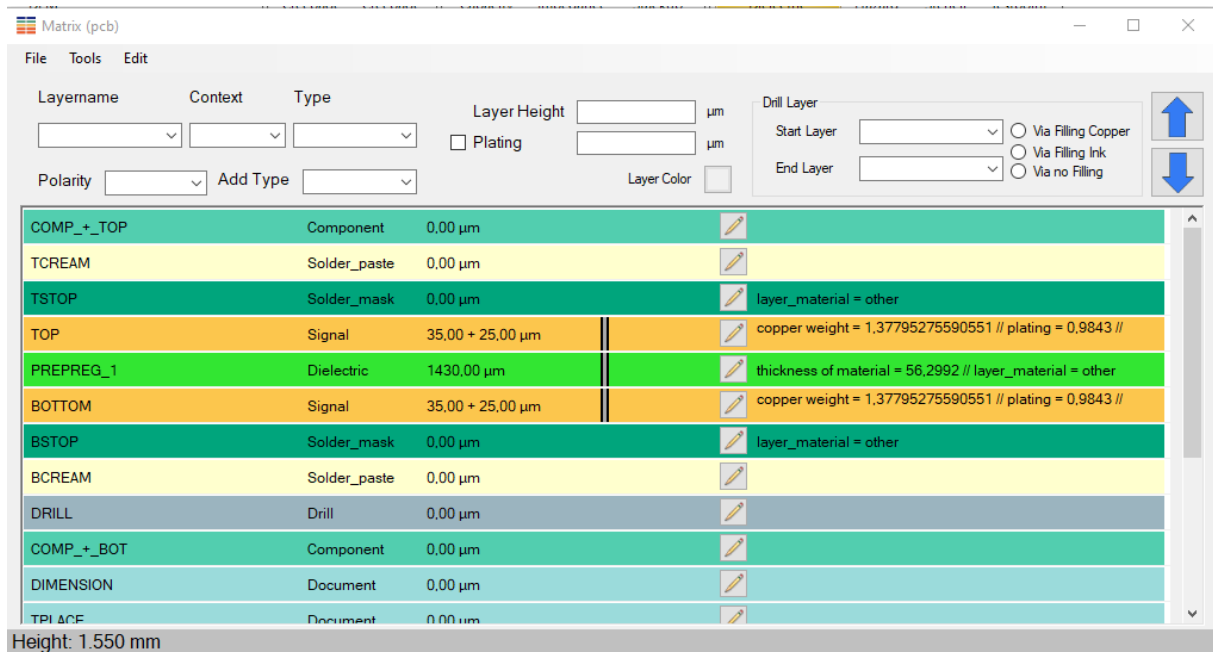
2. Impedance Calculator



Click on the button :



By opening the Matrix configuration when clicking on **Open Matrix**, you can check or modify the existing stackup definition with its parameters:



To carry out the approximation, you have to select a net at first. All values relevant for the calculation are then displayed at the bottom left. Click on the “**Log**” button for more detailed information and the result of the approximation. The table also shows the calculated values for the selected net(s)

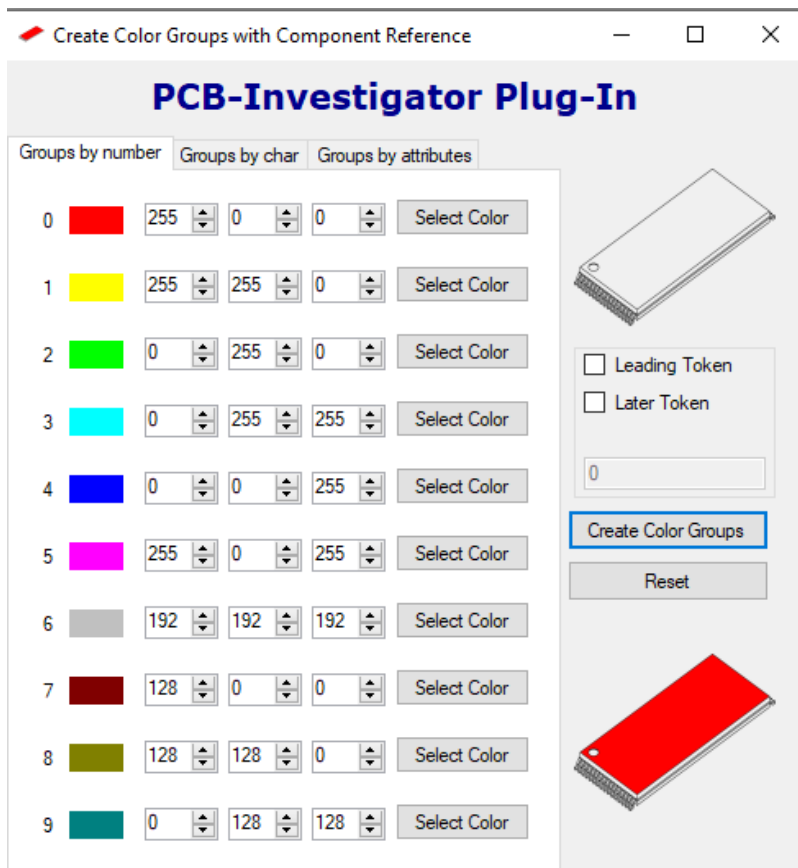
MODULE 3 : Net & Components



A. Color group

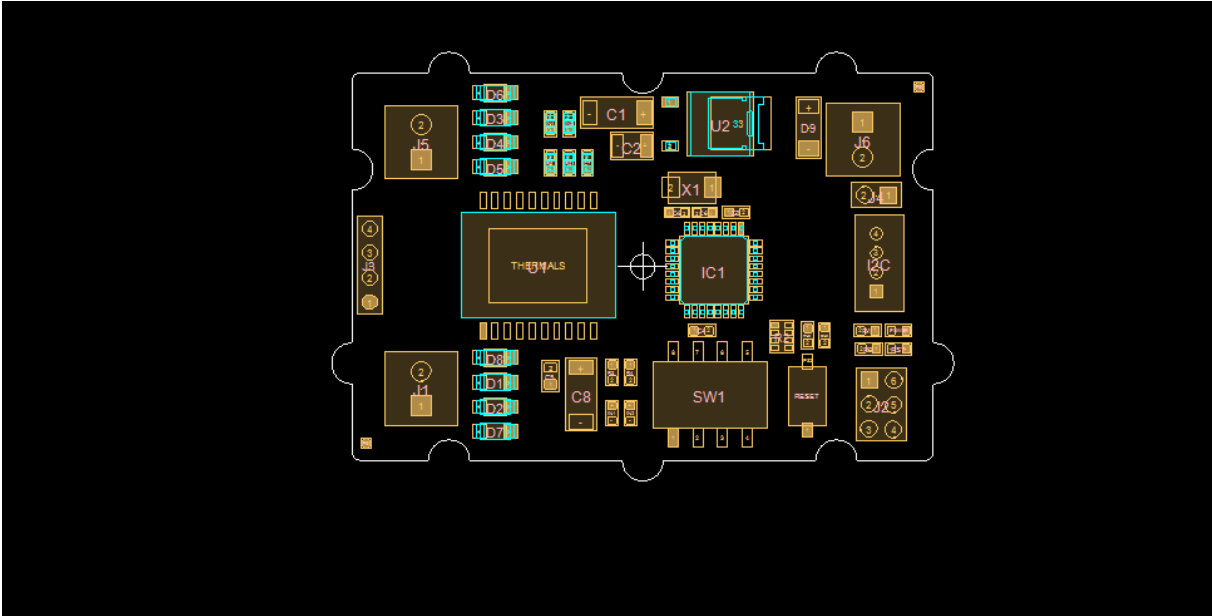
It depends on how the schematic is structured. Some companies use for example a leading number for schematic pages. There is Color by numbers which are represented by the numbers of the schematic sheets, or we can group them by attributes. If you have a specific color system, you can use special coloring tool engine.

To open the editing window, you need to go to **Developer** menu and select **Color Group** option in components section:

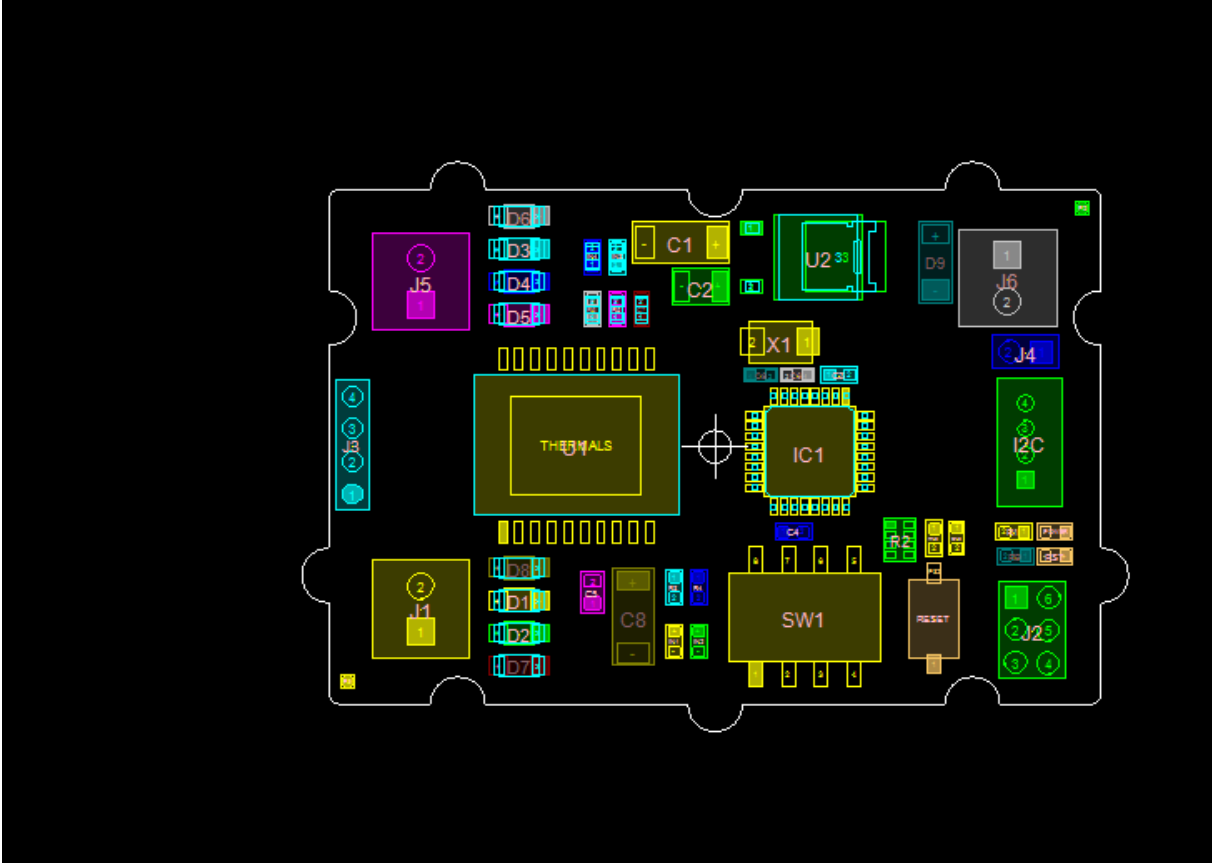


Click on the button **Create Color Groups** to see the difference:

- Without the color group:



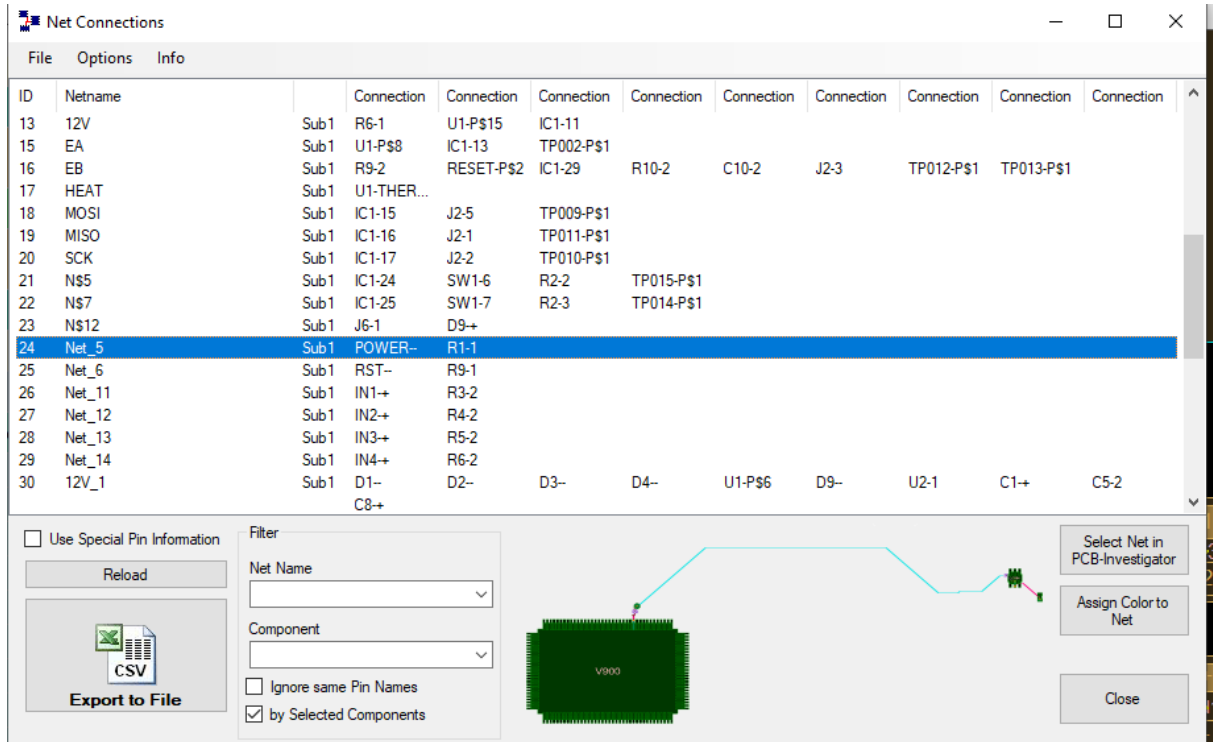
➤ With the color group:



Click on **Reset** to go back to the original colors.

B. Nets

We can go to **Net Connections**, and select a net to visualize it. To do so, Select Net-5 , then click on **Select Net n PCB-Investigator**:



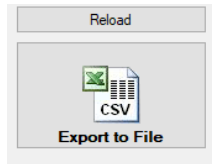
The net will be highlighted:



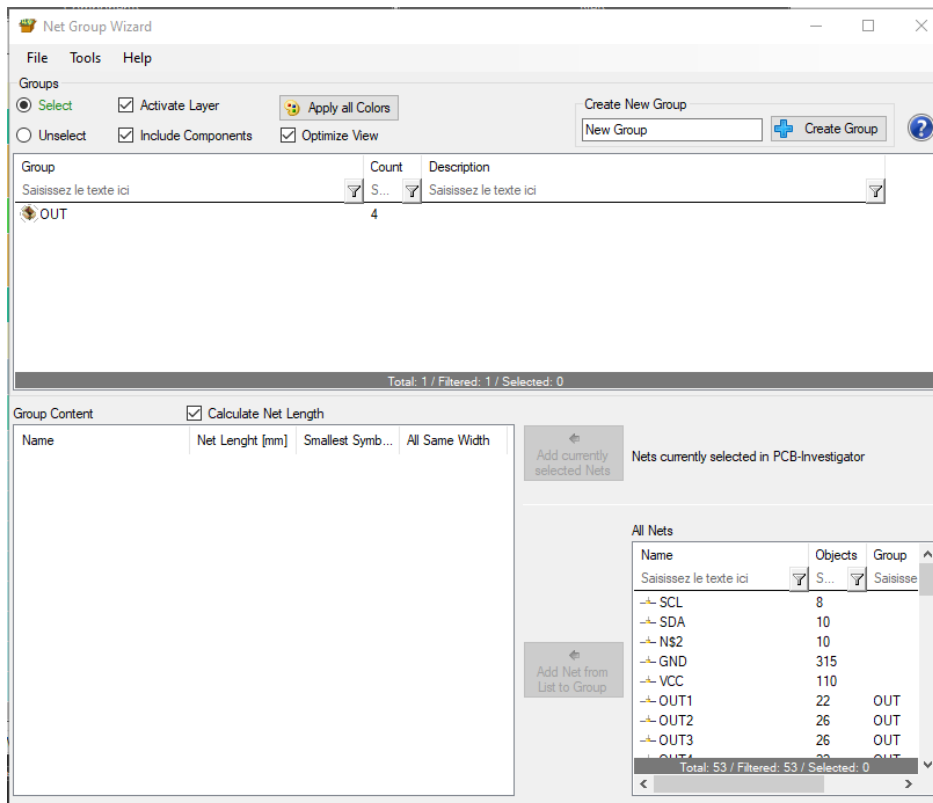
We can change the color of the highlighted net by clicking on **Assign Color to Net**:



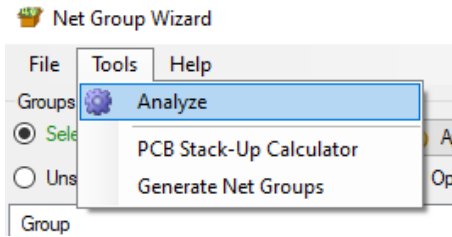
We can also export the net connection to an Excel file:



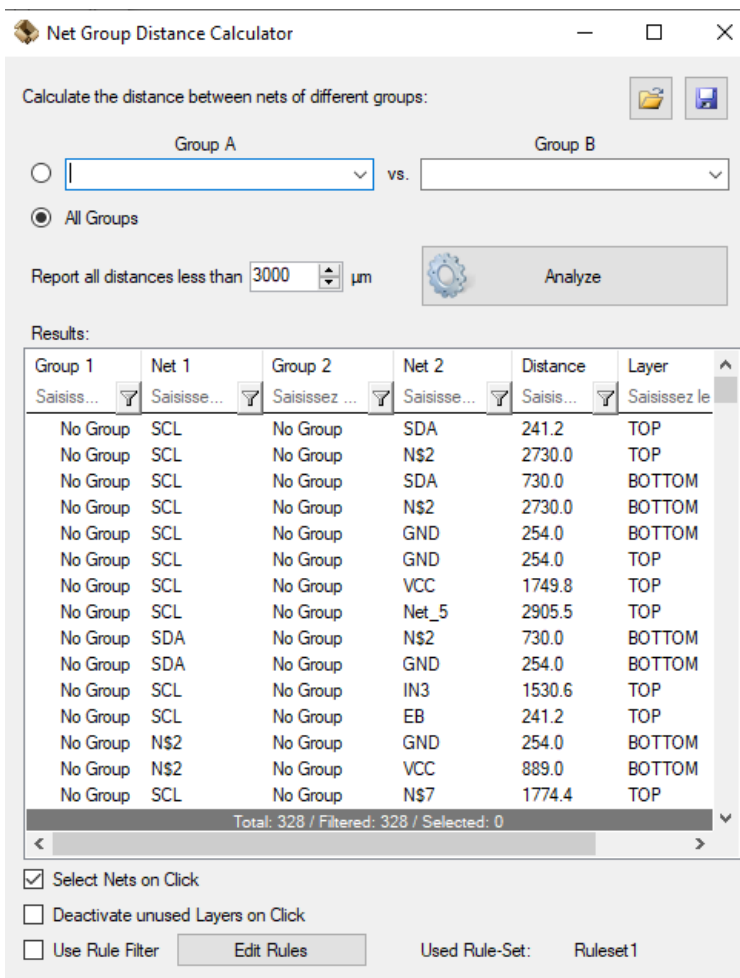
Close Net Connections windows. Click on **Net Groups**, where we can group nets, analyze them and compare the 2 groups:



To start the analysis, choose the sub-menu **Tools**, and select **Analyze**:

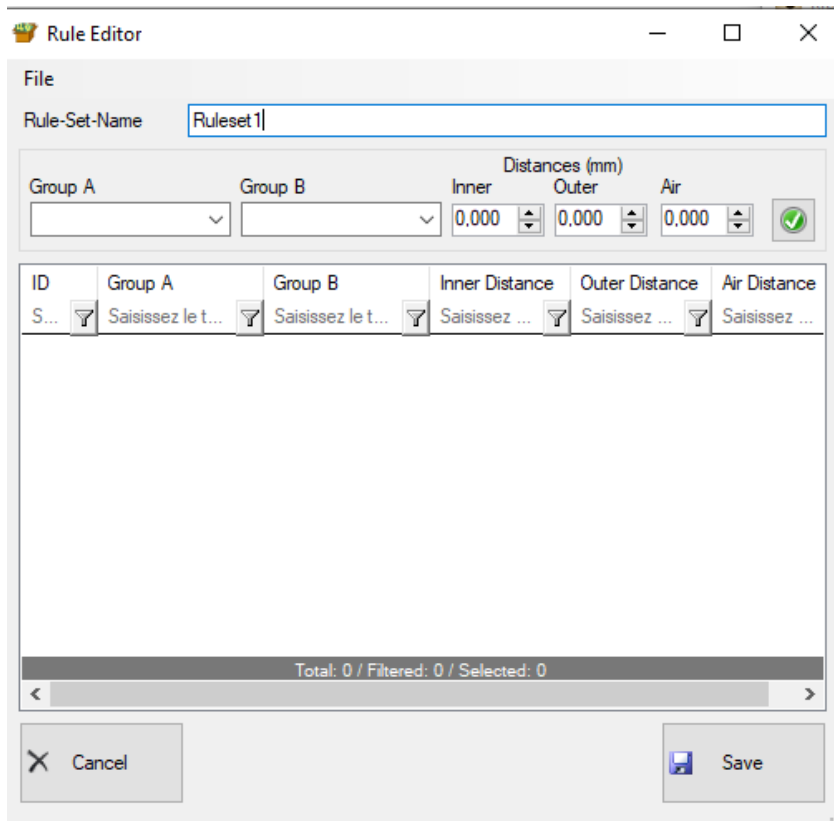


A new popup window appears, and with the default options, click on **Analyze**:

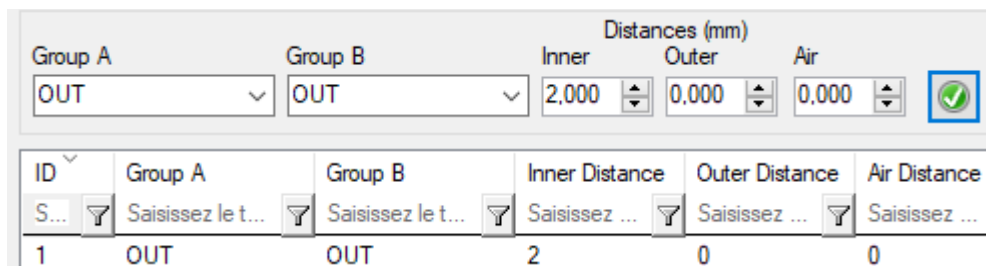


In this case, we apply the analysis to a all groups of nets. We can also compare two groups of nets by selecting Group A and Group B.

We can also use rule filter to compare. By clicking on the button **Edit Rules**:



We choose the group names and click on the green check button, then save:



To use this rule, we created, select the option **Use Rule Filter**:



Note that when you use the filter, a new section adds in the Results:

Results:

Status	Group 1	Net 1	Group 2	Net 2	Distance
No Group	GND	No Group	EB	254.0	
No Group	GND	No Group	MOSI	254.0	
No Group	GND	No Group	EB_1	782.8	
No Group	GND	No Group	EA	254.0	
No Group	GND	No Group	IN3	254.0	
No Group	GND	No Group	IN4	254.0	
No Group	SDA	No Group	IN3	1530.6	
No Group	GND	No Group	RST	254.0	
No Group	GND	No Group	12V	254.0	
No Group	SDA	No Group	EB	1041.2	
No Group	GND	No Group	12V_1	254.0	
No Group	GND	No Group	NS5	254.0	
No Group	GND	No Group	NS7	254.0	
No Group	GND	No Group	HEAT	482.6	
OUT	OUT1	OUT	OUT2	399.1	

Total: 327 / Filtered: 327 / Selected: 0

Select Nets on Click
 Deactivate unused Layers on Click
 Use Rule Filter Edit Rules Used Rule-Set: Ruleset1

To remind, without the filter:

Results:

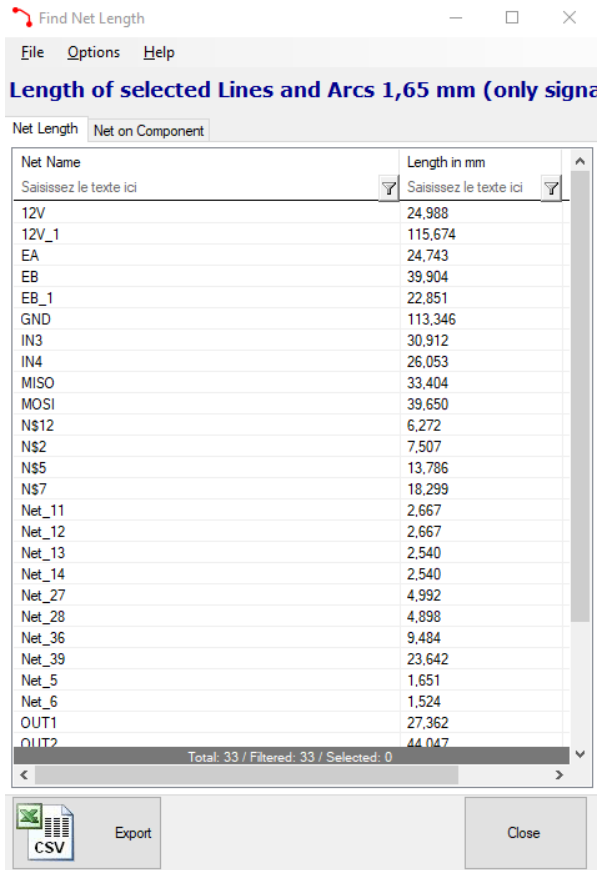
Group 1	Net 1	Group 2	Net 2	Distance	Layer
No Group	SCL	No Group	SDA	241.2	BOTTOM
No Group	SCL	No Group	NS2	2730.0	TOP
No Group	SCL	No Group	NS2	2730.0	BOTTOM
No Group	SCL	No Group	GND	254.0	BOTTOM
No Group	SCL	No Group	GND	254.0	TOP
No Group	SCL	No Group	VCC	1749.8	TOP
No Group	SCL	No Group	Net_5	2905.5	TOP
No Group	SDA	No Group	NS2	730.0	BOTTOM
No Group	SDA	No Group	GND	254.0	BOTTOM
No Group	SCL	No Group	IN3	1530.6	TOP
No Group	SCL	No Group	EB	241.2	TOP
No Group	NS2	No Group	GND	254.0	BOTTOM
No Group	NS2	No Group	VCC	889.0	BOTTOM
No Group	SCL	No Group	NS7	1774.4	TOP
No Group	SCL	No Group	Net_39	1041.2	TOP

Total: 327 / Filtered: 327 / Selected: 0

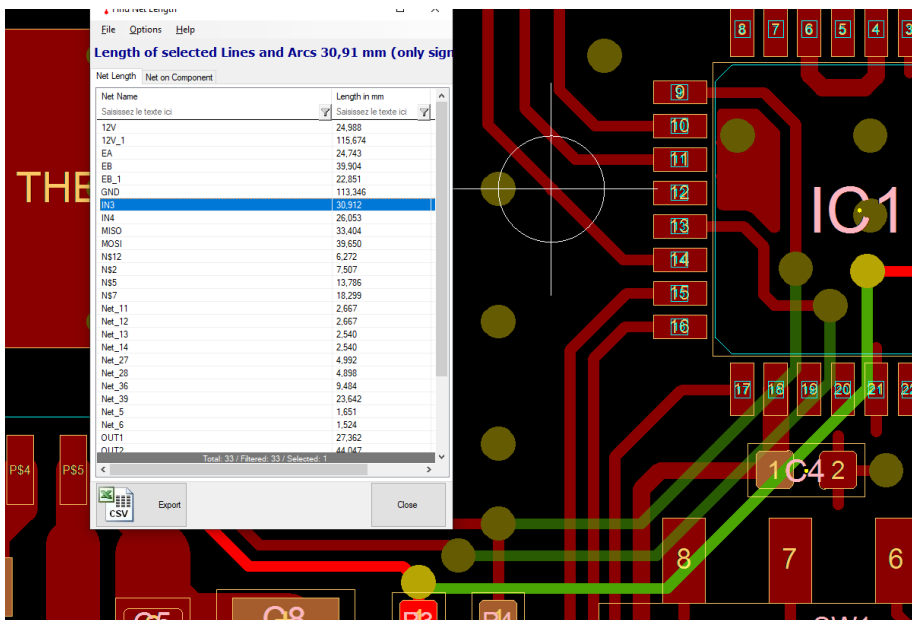
Select Nets on Click
 Deactivate unused Layers on Click
 Use Rule Filter Edit Rules Used Rule-Set: Ruleset1

3. Net Length

We can see the length of the nets and export the details:



When we double click on the net in the list, it highlights on the PCB:

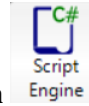


MODULE 4 : Extras



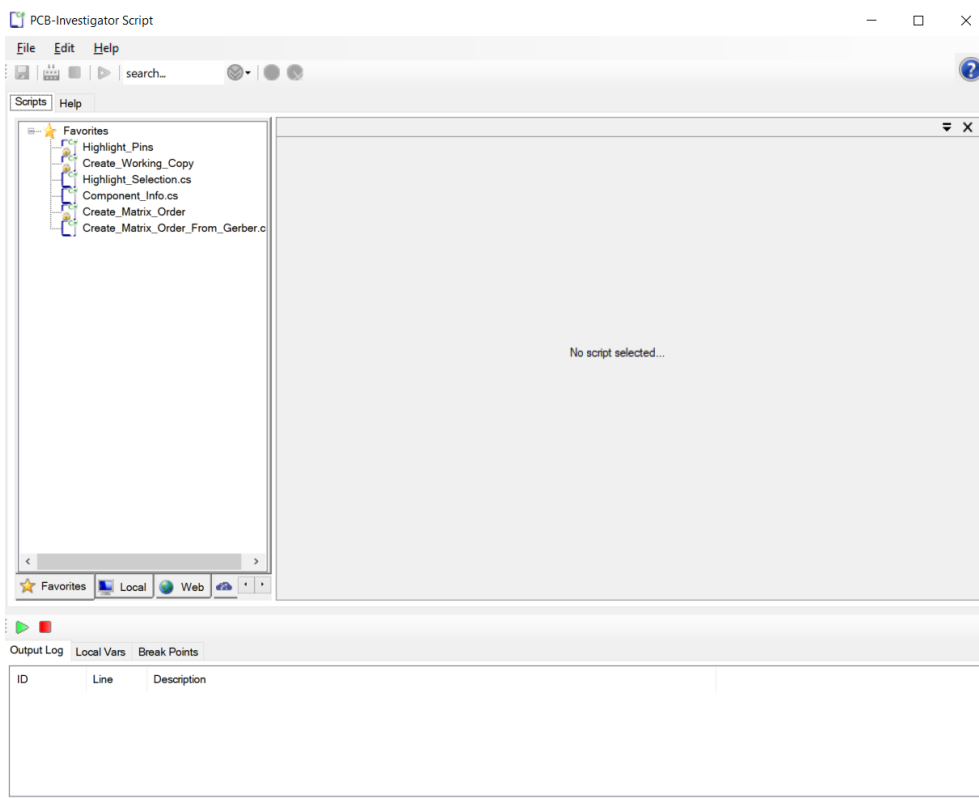
A. Script engine

The script engine allows you to integrate your own or existing scripts to customize PCB Investigator according to your needs.



From the menu **Extras**, click on the **Script Engine** button

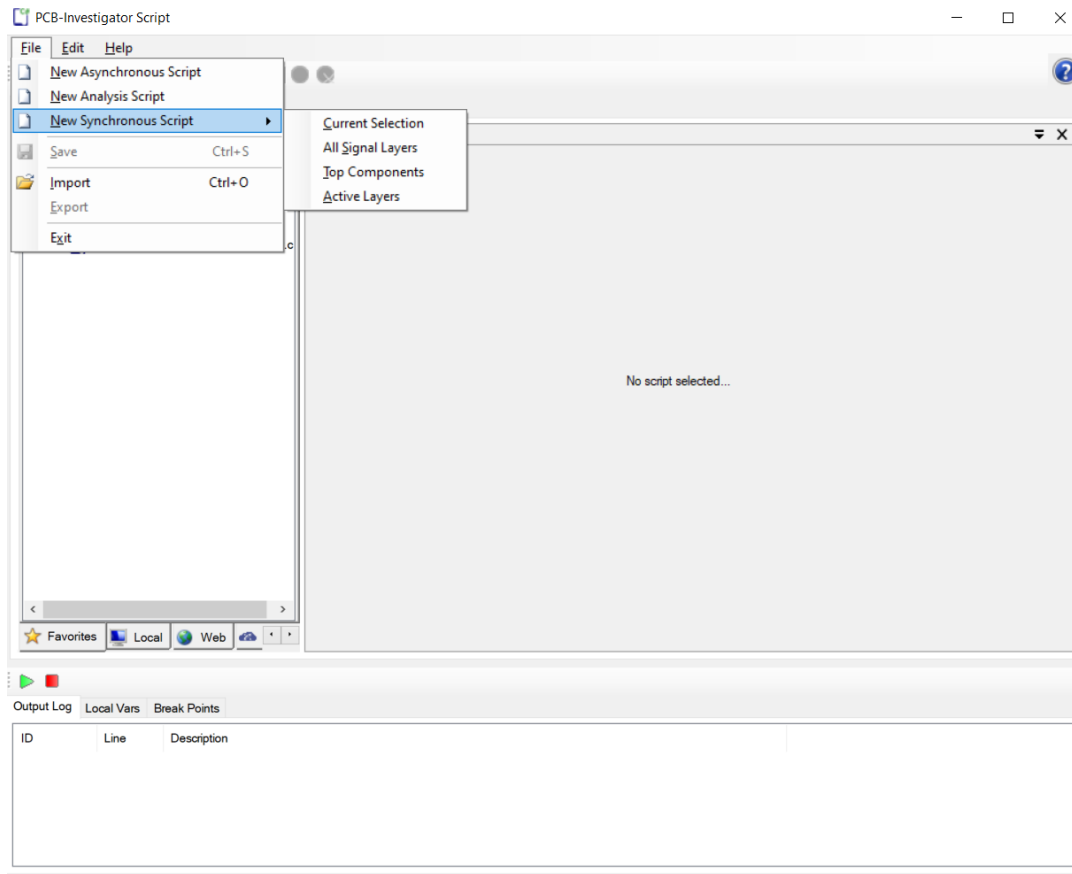
It allows you to write your own code in a script:



The favorites tab contains the scripts that are the most used. Every script within the **Favorites** can be placed on the PCB Investigator main ribbon menu as a button

To add scripts to the **Local** tab, there are two ways:

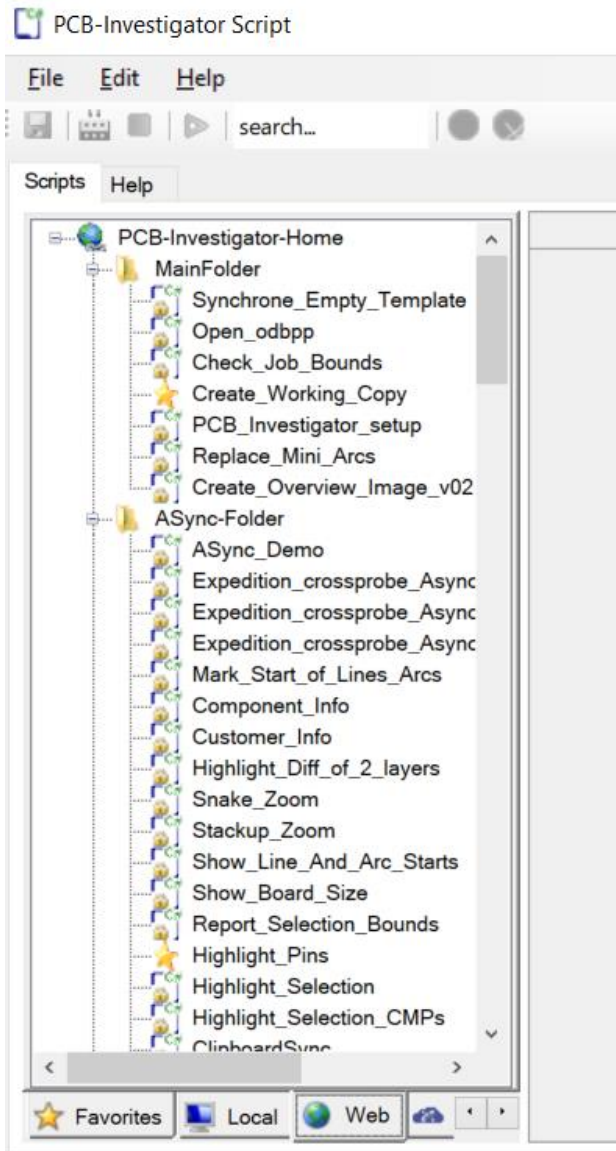
The first is creating your own script. You can add it by using the **File** tab at the bottom or by just right clicking on the white area in tab control:



- **Asynchronous Script:** Every asynchronous script runs if you stop it manually after starting. It allows you to run a script in the background until you do not need it anymore. An example of an asynchronous script is the "Highlighting" script. It makes all components of the design flash in one color temporary.
- **Analysis Script:** This type of scripts includes synchronous scripts. The difference to a usual synchronous script is that the variables of an analysis script will be managed by PCB Investigator itself.
- **Synchronous Script:** These scripts will run once until their job has been completed. There is no need to stop the script manually. You can choose between the current selection, all signal layers, top components and active layers.

The other way to add scripts are via **Web** tab where we can add an existing script.

Choose the tab **Web**:

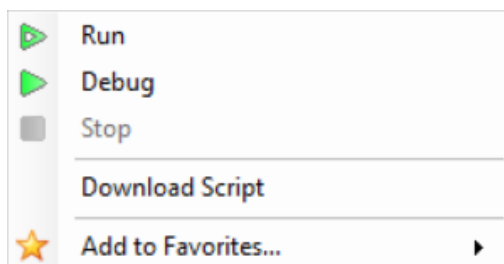


Select a script to open its code:

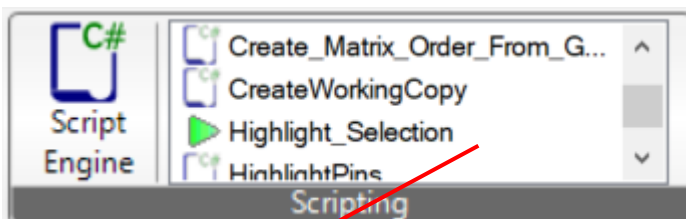
```

Highlight_Selection.cs
49 //fill code here code
50
51
52
53
54 IStep step = parent.GetCurrentStep();
55 bool onOff = false;
56 if (step == null)
57 {
58     MessageBox.Show("No job loaded.");
59     return;
60 }
61
62 while (true)
63 {
64     Thread.Sleep(1000);
65     foreach (IODBObject objSel in step.GetSelectedElements())
66     {
67         if (onOff)
68         {
69             parent.IColorsetting.DrawOnlySelectedElements = true;
70             objSel.ObjectColorTemporary(Color.White);
71         }
72         else
73             objSel.ObjectColorTemporary(Color.Empty);
74     }
75     onOff = !onOff;
76
77     parent.UpdateView();
    
```

If you want to add a “Web” script to your Local tab, right click on the selected script:



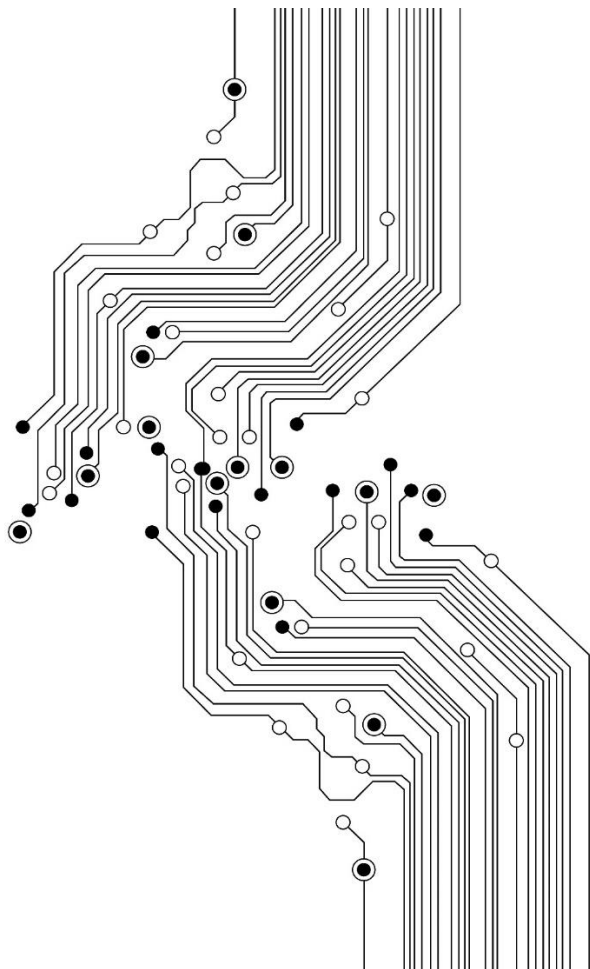
Select **Add to Favorites...** to have access to it easily in the ribbon:



Select one script, right click to Run.

B. Panelization

The panelization option allows you to check whether your panel meets the specified measurements.



EDA Expert

1 Avenue Paul Vaillant Couturier

94110 Arcueil, France

Tel : +33 (0) 1 58 07 00 79

Email : contact@eda-expert.com