Altair PollEx 2021

PCB Tutorials

altair.com

Intellectual Property Rights Notice

Copyrights, Trademarks, Trade Secrets, Patents & Third Party Software

Copyrights in the below are held by Altair Engineering, Inc., except where otherwise explicitly stated. This Intellectual Property Rights Notice is exemplary, not exhaustive.

> **Note:** Pre-release versions of Altair software are provided 'as is', without warranty of any kind. Usage of pre-release versions is strictly limited to non-production purposes.

Altair HyperWorks™ - The Platform for Innovation™

Altair AcuConsole™ ©2006-2021

Altair AcuSolve™ ©1997-2021

Altair Activate® ©1989-2021 (formerly solidThinking Activate)

Altair Compose® ©2007-2021 (formerly solidThinking Compose)

Altair ConnectMe™ ©2014-2021

Altair EDEM ©2005-2021 DEM Solutions Ltd, ©2019-2021 Altair Engineering Inc.

Altair ElectroFlo™ ©1992-2021

Altair Embed® ©1989-2021 (formerly solidThinking Embed)

- **Altair Embed SE** ©1989-2021 (formerly solidThinking Embed SE)
- **Altair Embed/Digital Power Designer** ©2012-2021
- **Altair Embed Viewer** ©1996-2021

Altair ESAComp™ ©1992-2021

Altair Feko™ ©1999-2014 Altair Development S.A. (Pty) Ltd., ©2014-2021 Altair Engineering Inc.

Altair Flux™ ©1983-2021

Altair FluxMotor™ ©2017-2021

Altair HyperCrash™ ©2001-2021

Altair HyperGraph™ ©1995-2021

Altair HyperLife™ ©1990-2021

Altair HyperMesh™ ©1990-2021

Altair HyperStudy™ ©1999-2021

Altair HyperView™ ©1999-2021

Altair HyperXtrude™ ©1999-2021

Altair Inspire™ ©2009-2021 including Altair Inspire Motion, Altair Inspire Structures, and Altair Inspire Print3D

Altair Inspire Cast ©2011-2021 (formerly Click2Cast)

Altair Inspire ElectroFlo ©1992-2021

Altair Packaged Solution Offerings (PSOs)

Altair PBSWorks™ - Accelerating Innovation in the Cloud™

Note:

Compute Manager™ ©2012-2017 is now part of **Altair Access Display**

Manager™ ©2013-2017 is now part of **Altair Access**

PBS Application Services™ ©2008-2017 is now part of **Altair Access**

PBS Analytics™ ©2008-2017 is now part of **Altair Control**

PBS Desktop™ ©2008-2012 is now part of **Altair Access**, specifically **Altair Access desktop**, which also has **Altair Access web** and **Altair Access mobile**

e-Compute™ ©2000-2010 was replaced by **"Compute Manager"** which is now **Altair Access**

Altair KnowledgeWorks™

Third party software licenses

AcuConsole contains material licensed from Intelligent Light (www.ilight.com) and used by permission.

Software Security Measures:

Altair Engineering Inc. and its subsidiaries and affiliates reserve the right to embed software security mechanisms in the Software for the purpose of detecting the installation and/or use of illegal copies of the Software. The Software may collect and transmit non-proprietary data about those illegal copies. Data collected will not include any customer data created by or used in connection with the Software and will not be provided to any third party, except as may be required by law or legal process or to enforce our rights with respect to the use of any illegal copies of the Software. By using the Software, each user consents to such detection and collection of data, as well as its transmission and use if an illegal copy of the Software is detected. No steps may be taken to avoid or detect the purpose of any such security mechanisms.

Technical Support

Altair provides comprehensive software support via web FAQs, tutorials, training classes, telephone, and e-mail.

Altair One Customer Portal

Altair One (https://altairone.com/) is Altair's customer portal giving you access to product downloads, a Knowledge Base, and customer support. We strongly recommend that all users create an Altair One account and use it as their primary means of requesting technical support.

Once your customer portal account is set up, you can directly get to your support page via this link: www.altair.com/customer-support/

Altair Training Classes

Altair's in-person, online, and self-paced trainings provide hands-on introduction to our products, focusing on overall functionality. Trainings are conducted at our corporate and regional offices or at your facility.

For more information please visit: https://learn.altair.com/

If you are interested in training at your facility, please contact your account manager for more details. If you do not know who your account manager is, please contact your local support office and they will connect you with your account manager.

Telephone and E-mail

If you are unable to contact Altair support via the customer portal, you may reach out to technical support via phone or e-mail. Use the following table as a reference to locate the support office for your region.

When contacting Altair support, please specify the product and version number you are using along with a detailed description of the problem. It is beneficial for the support engineer to know what type of workstation, operating system, RAM, and graphics board you have, so please include that in your communication.

Location	Telephone	E-mail
Australia	+61 649 413 7981	anzsupport@altair.com
Brazil	+55 113 884 0414	br_support@altair.com
Canada	+1 416 447 6463	support@altairengineering.ca
China	+86 400 619 6186	support@altair.com.cn
France	+33 141 33 0992	francesupport@altair.com
Germany	+49 703 162 0822	hwsupport@altair.de
Greece	+30 231 047 3311	eesupport@altair.com

Location	Telephone	E-mail
India	+91 806 629 4500 +1 800 425 0234 (toll free)	support@india.altair.com
Israel		israelsupport@altair.com
Italy	+39 800 905 595	support@altairengineering.it
Japan	+81 36 225 5830	support@altairjp.co.jp
Malaysia	+60 32 742 7890	aseansupport@altair.com
Mexico	+52 555 658 6808	mx-support@altair.com
New Zealand	+64 9 413 7981	anzsupport@altair.com
South Africa	+27 21 831 1500	support@altair.co.za
South Korea	+82 704 050 9200	support@altair.co.kr
Spain	+34 910 810 080	support-spain@altair.com
Sweden	+46 46 460 2828	support@altair.se
United Kingdom	+44 192 646 8600	support@uk.altair.com
United States	+1 248 614 2425	hwsupport@altair.com

If your company is being serviced by an Altair partner, you can find that information on our web site at https://www.altair.com/PartnerSearch/.

See www.altair.com for complete information on Altair, our team, and our products.

Contents

Conventions Used in this Guide

This guide uses the following conventions:

Bold Italic	All commands from the user interface. Options, menus, buttons, and dialog box names are bolded, but not italicized. Example: On the **Welcome** screen, click ***Next***.
`Courier`	The path of a program or folder; a web address; a file name or component; text that the user is expected to enter. Example: The default path is `C:\Program Files\Altair\2019\PollEx`

Questions regarding the document may be directed to PollEx team at PollEx_support_kr@altair.com.

PollEx PCB Tutorial

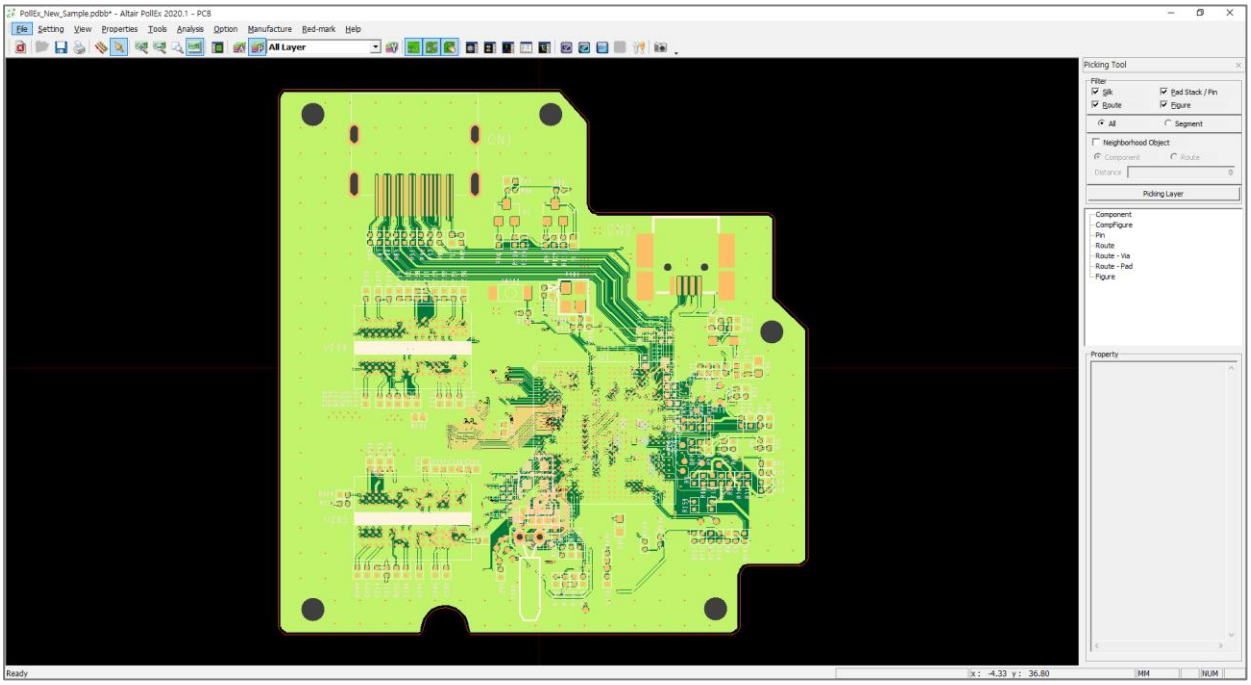
1. Open/Save PollEx PCB layout design file, PDBB

PollEx PCB is design browser(viewer) from various ECAD vendors. PollEx PCB can support CADENCE, Mentor Graphics, Zuken, Altium and different standard neutral files.

Click ***PollExPCB*** icon from the PollEx Launcher.

1.1. File Open

Open the file, `C:\Temp\Altair-PollEx\PollExPCB\PollEx_New_Sample.pdbb` using the main menu, **File-Open**.



1.2. Manual Open

Select menu, **Help – PollEx Manual**.
Or press the ***F1*** key while PollEx PCB is opened.

2. Look around PCB design

2.1. Display control

Display control is under **View** menu.
User can zoom-in/out, zoom window and zoom 1:1.

2.2. Layer display control

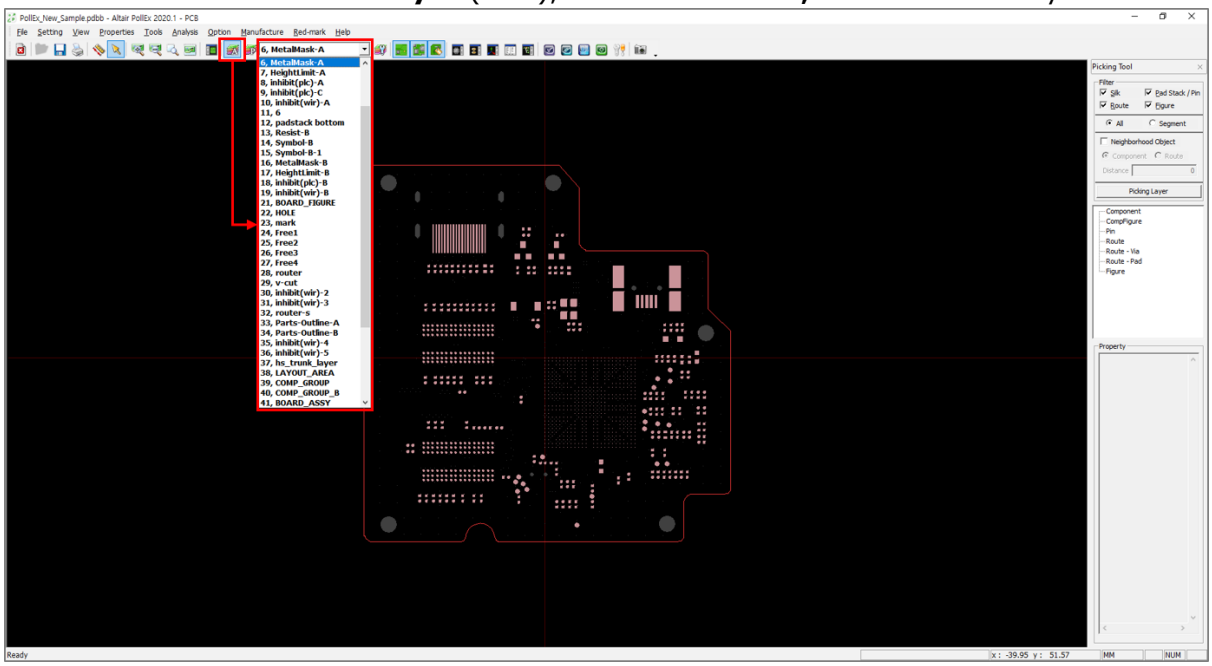
It is able to control the layer by Individual Layer function in PollEx PCB.

2.2.1. Layer Control

Select menu, **Setting - Layer** or click icons on the toolbar.

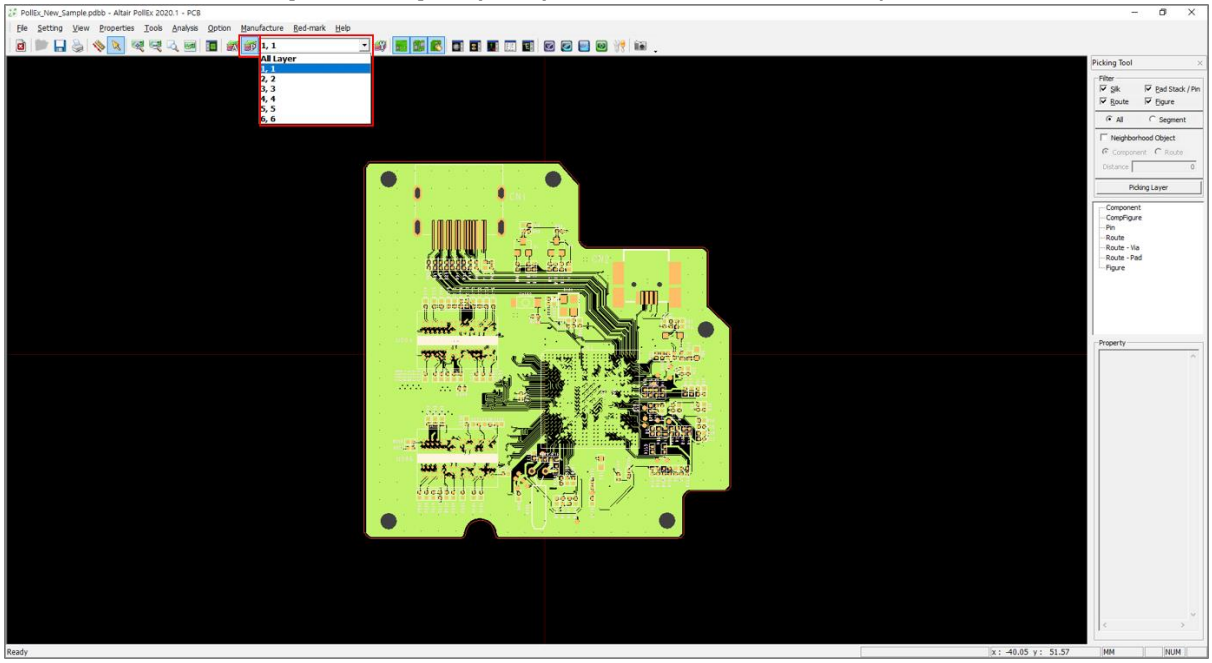
2.2.1.1. **Artwork Layer** ( ) or **Physical Layer** ( ) Control

Select the icon of **Artwork Layer** ( ), and then select **6, MetalMask-A** layer on the list.



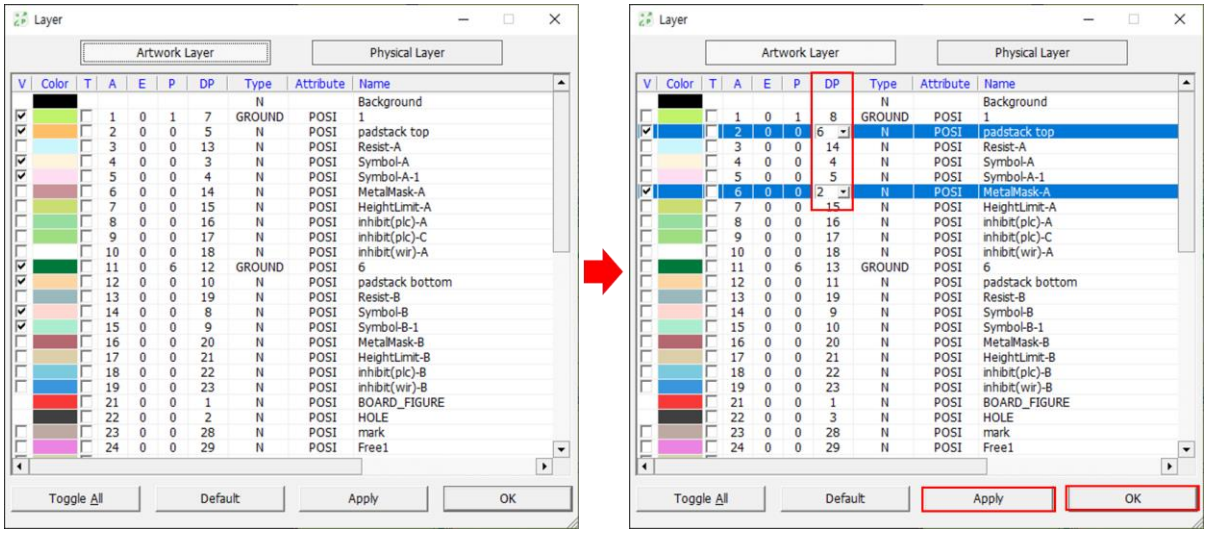
Shape data on MetalMask-A is displayed. As same way, it is able to display and see the layer by selecting the name on the list.

Select the icon of **Physical Layer** ( ), and then select **1, 1** layer on the list.

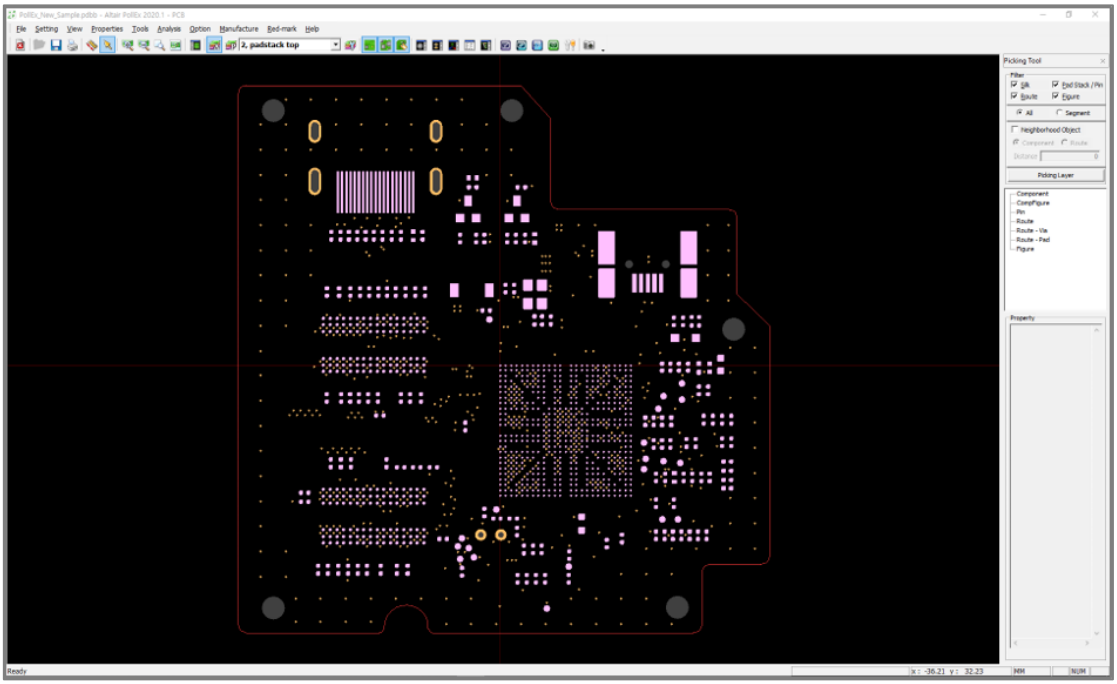


Combined layers as 1 (Top) is shown on the window.

2.2.1.2. To display the target layer(s)
Select menu, **Setting – Layer**.
Select a layer name of **padstack top** and **MeatalMask-A** on **Layer** dialog.
Then, change the number of MetalMask-A on DP (Display Priority) column to **2**.
Click ***Apply*** and ***OK*** button.



MetalMask-A is overlapped on **padstack top** layer.



3. Check out the property of PCB objects, nets, components and others

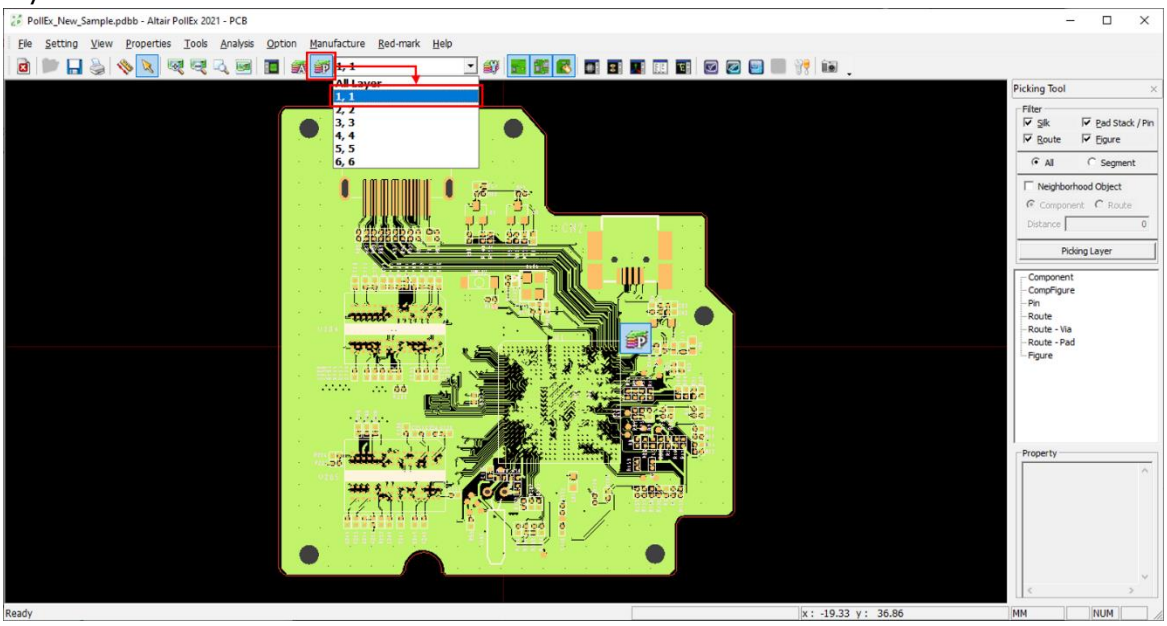
3.1. Finding objects properties using Picking Tool

Picking feature is used for checking the object information and property that included in the PCB design data.

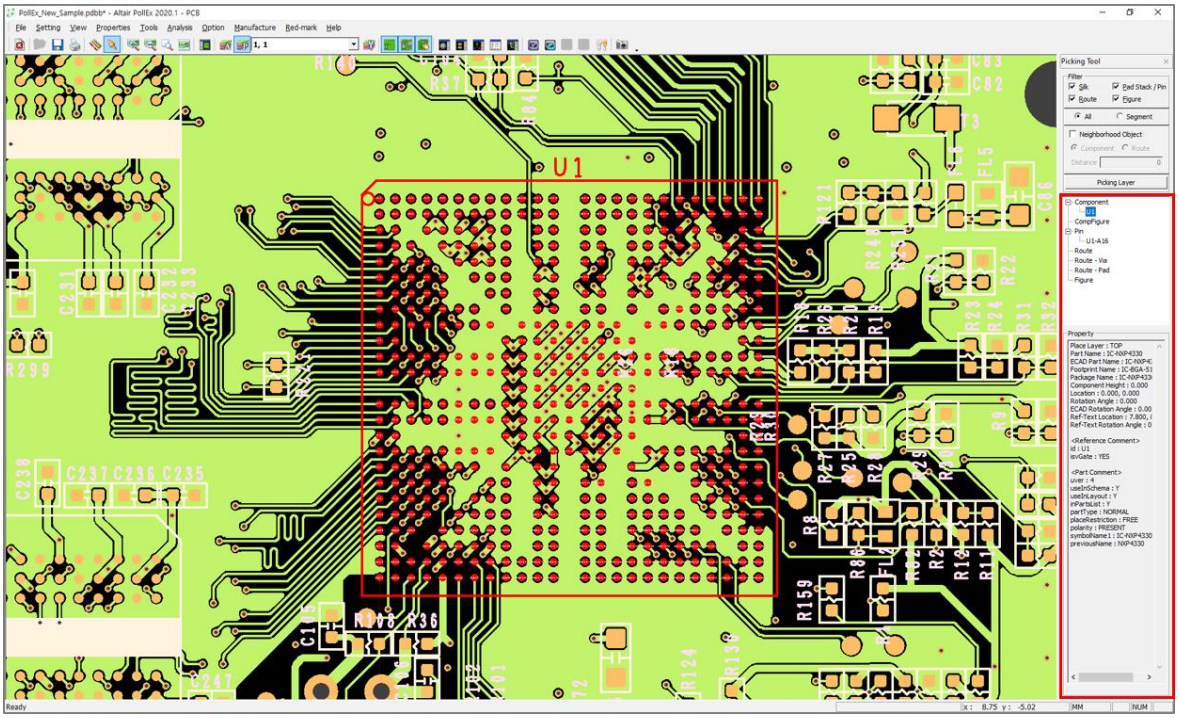
3.1.1. Run Picking Tool

Picking Tool is enabled as a default on the right side when initiated the PollEx PCB program.
If the Picking Tool is disabled, select menu Setting - Picking.

Change to the Physical layer by clicking the icon (), and click **1,1** layer which is the Top layer.



Click an object on the design to check the information.
The information of the object's name and property information is displayed on the right side

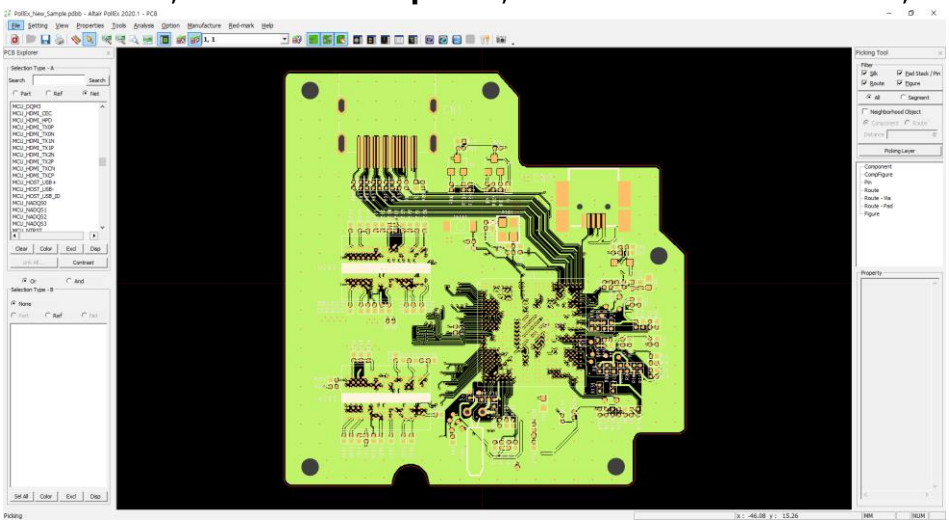


3.2. Finding objects on PCB design using PCB Explorer

PCB Explorer helps to find parts and nets easily in the design data.

3.2.1. Run PCB Explorer

Select menu, **Tools – PCB Explorer**, click icon  on the toolbar, or input ***Ctrl + F***.

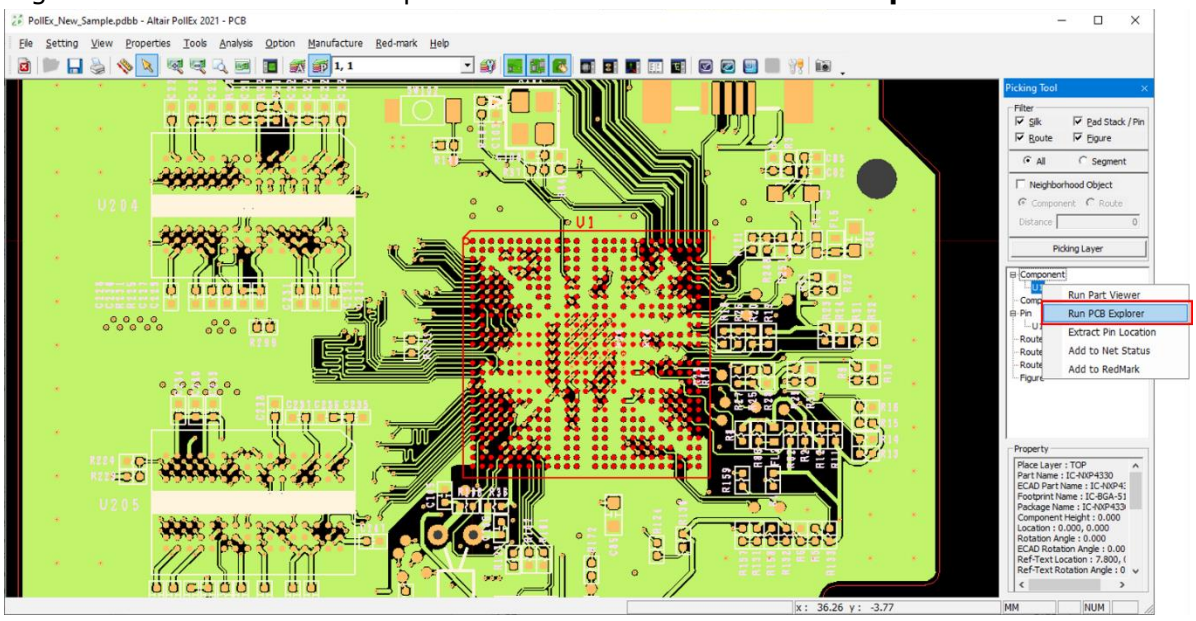


PCB Explore is enabled on the left side.
User can search name by selecting Part, Reference, or Net.
Using Selection Type –A and B, user can check the connected part, reference, and net name.

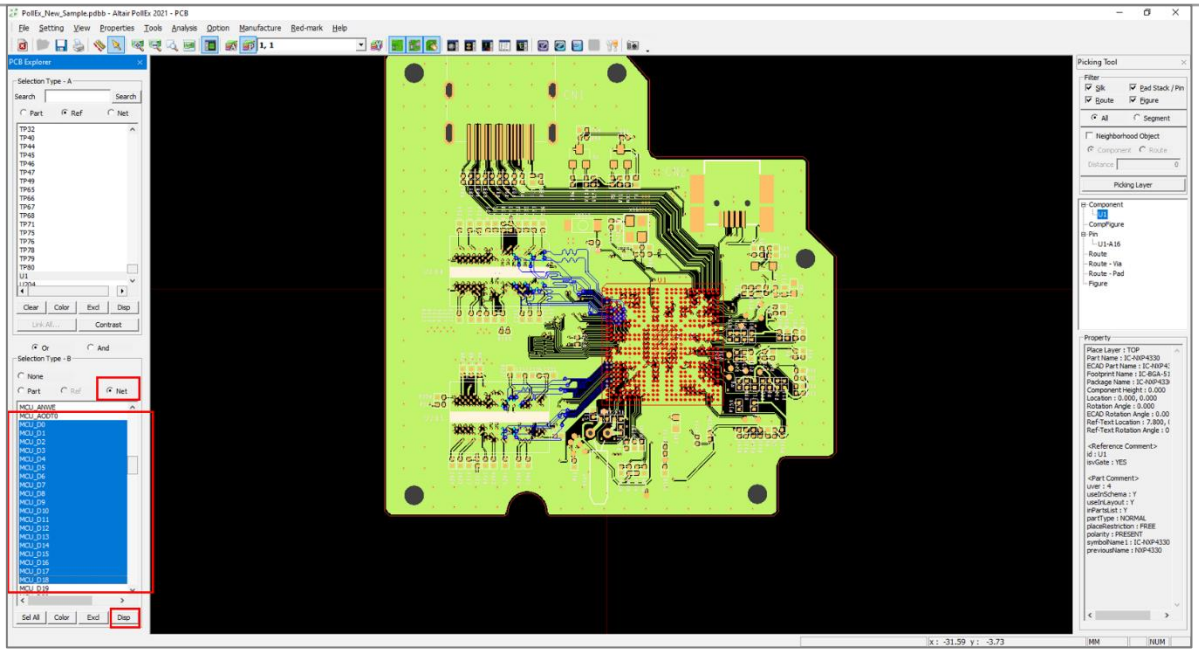
3.2.2. Connected information of part or net by using Picking Tool and PCB Explorer

3.2.2.1. Check Net connected to a component

In the design, select a component to check.
Right-mouse click on the component name and click **Run PCB Explorer** menu.



In this tutorial, **U1** is selected.
When the PCB Explorer is enabled, please select **Net** on Selection Type-B.
Select Net name and click ***Disp***, then it displays the selected nets with connected component.

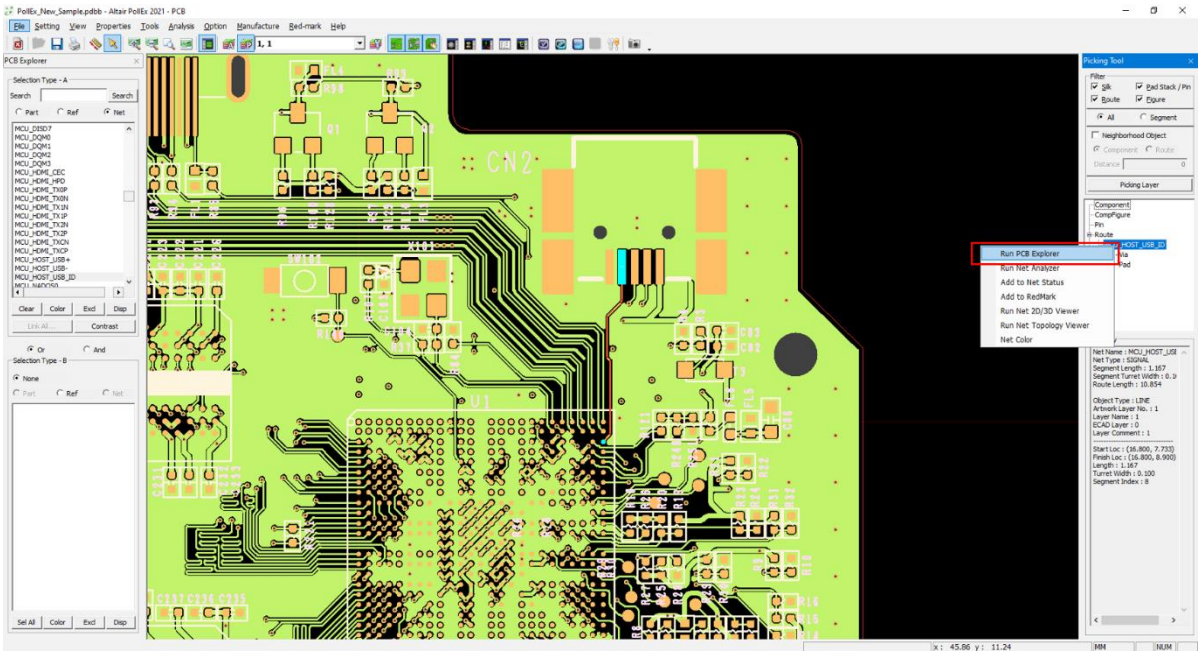


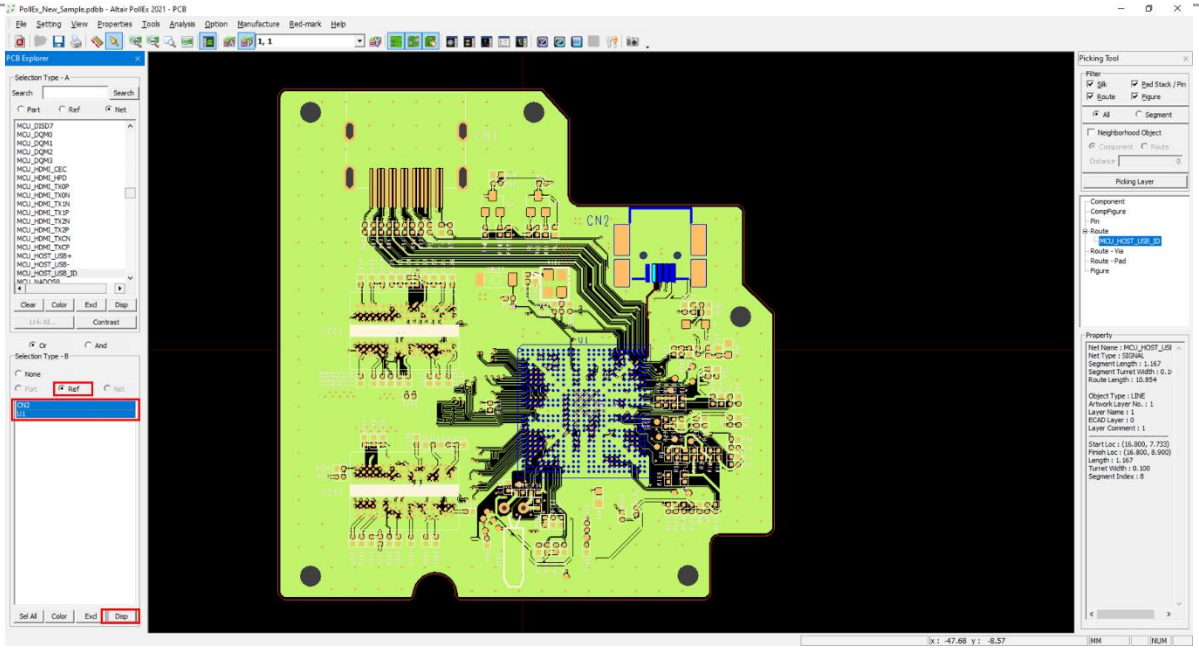
3.2.2.2. Check component connected to net

In the design, select a Net to check.

Select a certain net to check by clicking right-mouse button.

Then, right-mouse click on the Route name on **Picking Tool** dialog and click **Run PCB Explorer** menu.





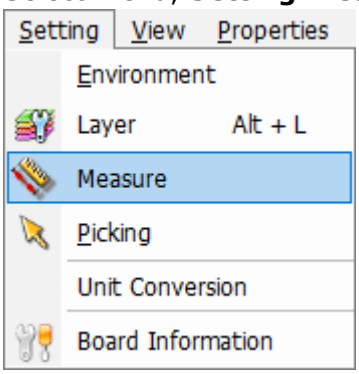
When the PCB Explorer is enabled, please select **Ref** on Selection Type-B.
Select Ref name and click ***Disp***, then it displays the selected nets with connected components.
Click ***Excl*** button to display only the selected objects, other objects are not displayed on main window.

3.3. Measure

It can measure easily the distance between object to object in PCB design.

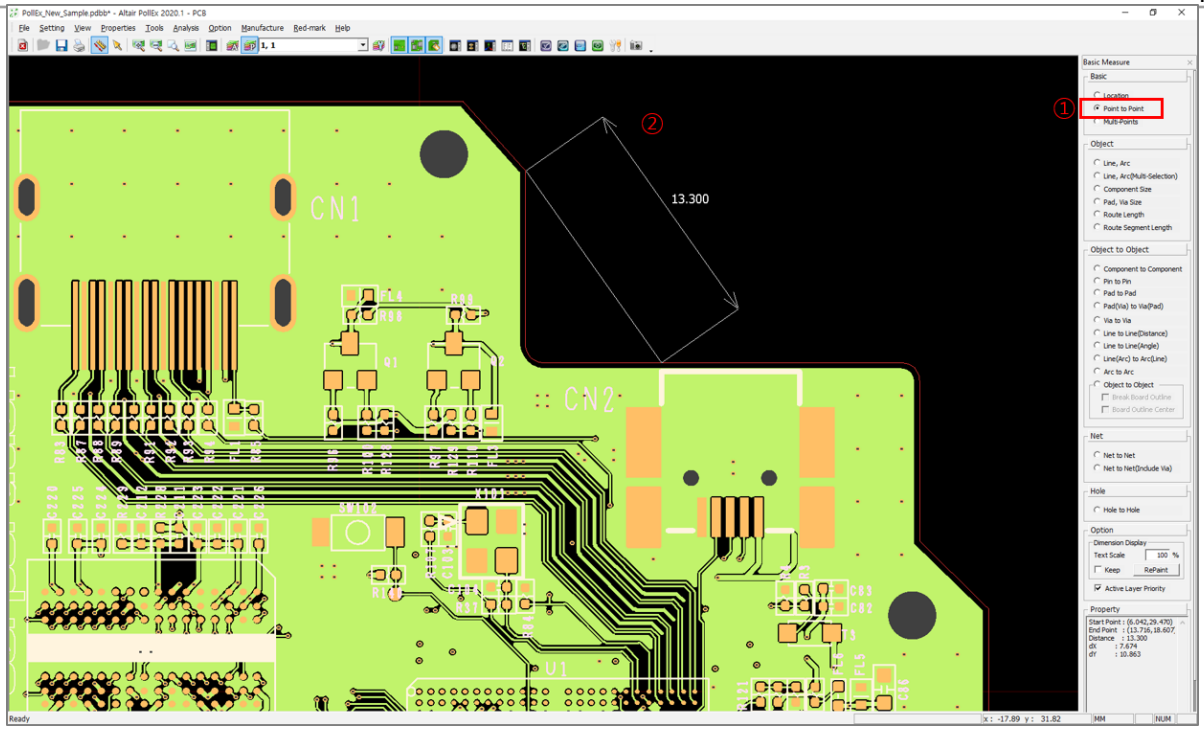
3.3.1. Run Measure Function

Select menu, **Setting-Measure** or click measure icon on the toolbar.

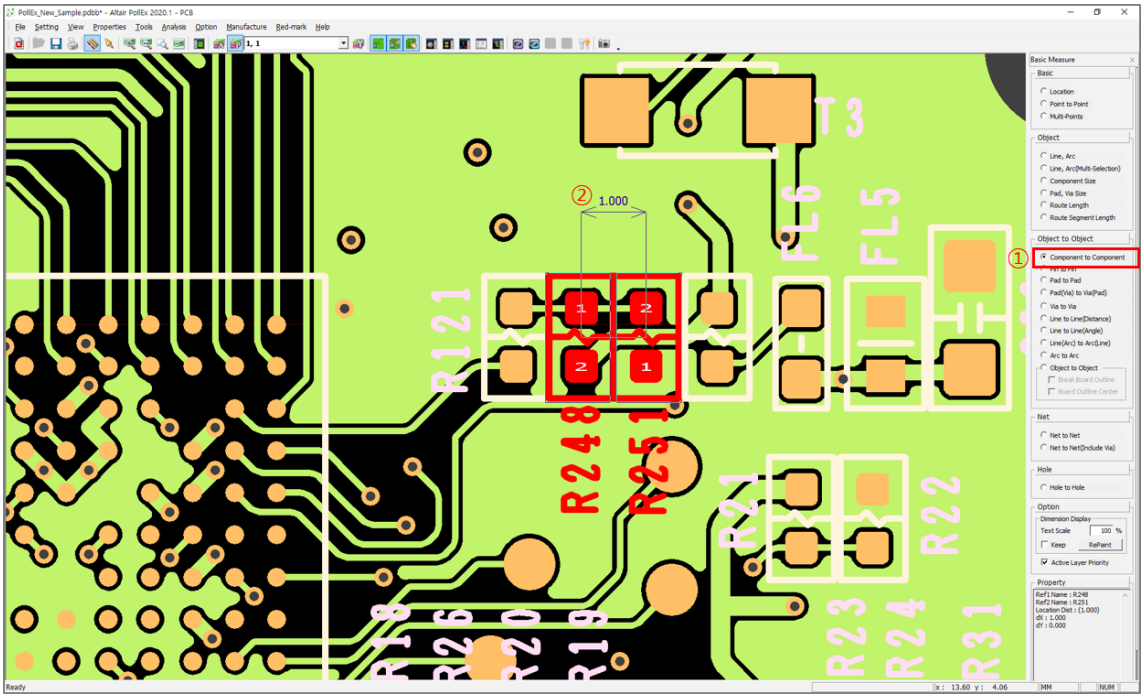


After running **Measure** function, it is displayed on the right-side of main window.

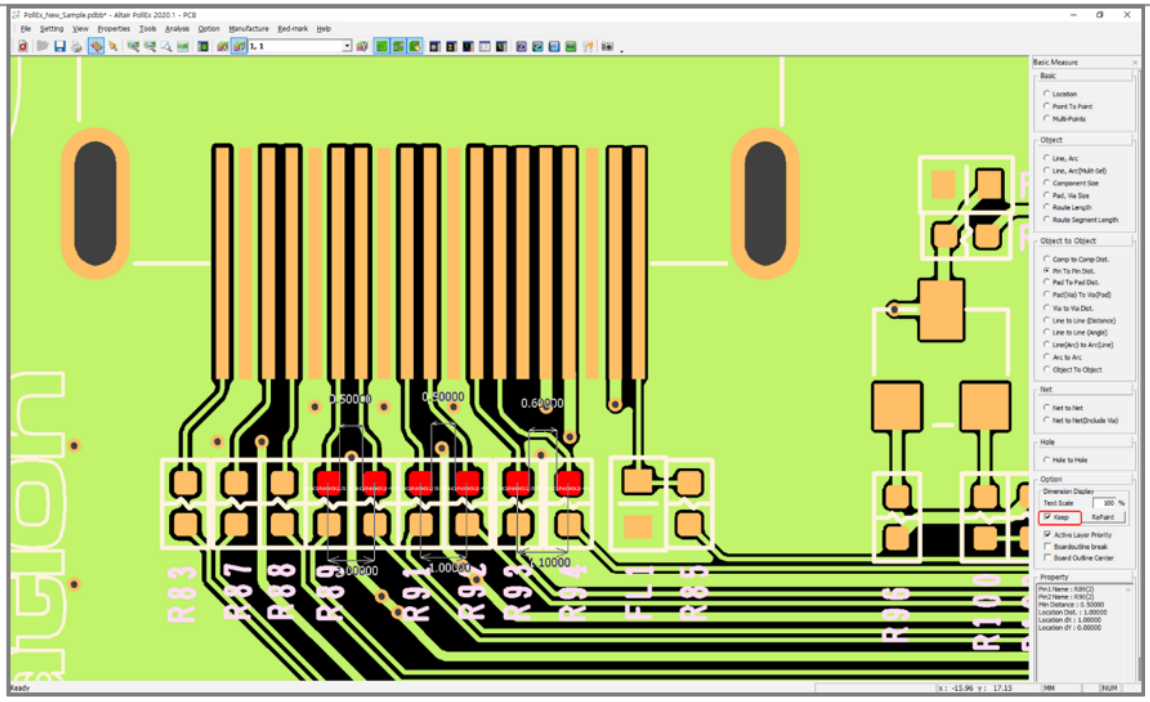
- Measure between location to location
  - Select **Point to Point** (①) on Basic of Measure menu, and then ② click two points by mouse clicking.



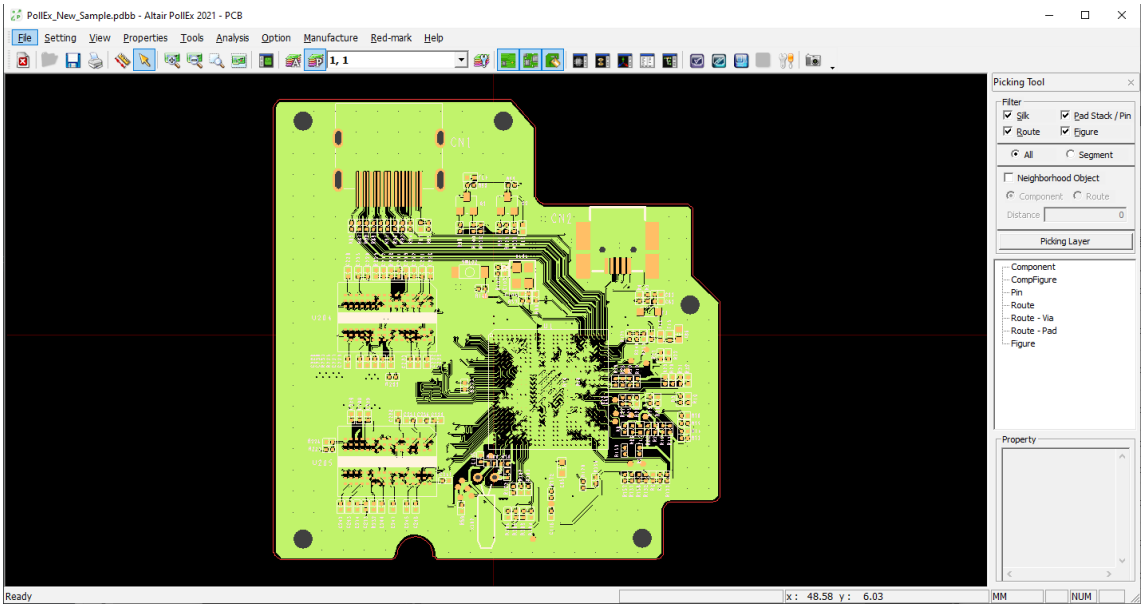
- Measure between component to component
  - Select **Component to Component** (①) distance on Object to Object of Measure menu, and then ②click two points by mouse clicking.



To maintain the measured results, select **Keep** on Option.



Click  icon and  icon from the toolbar.

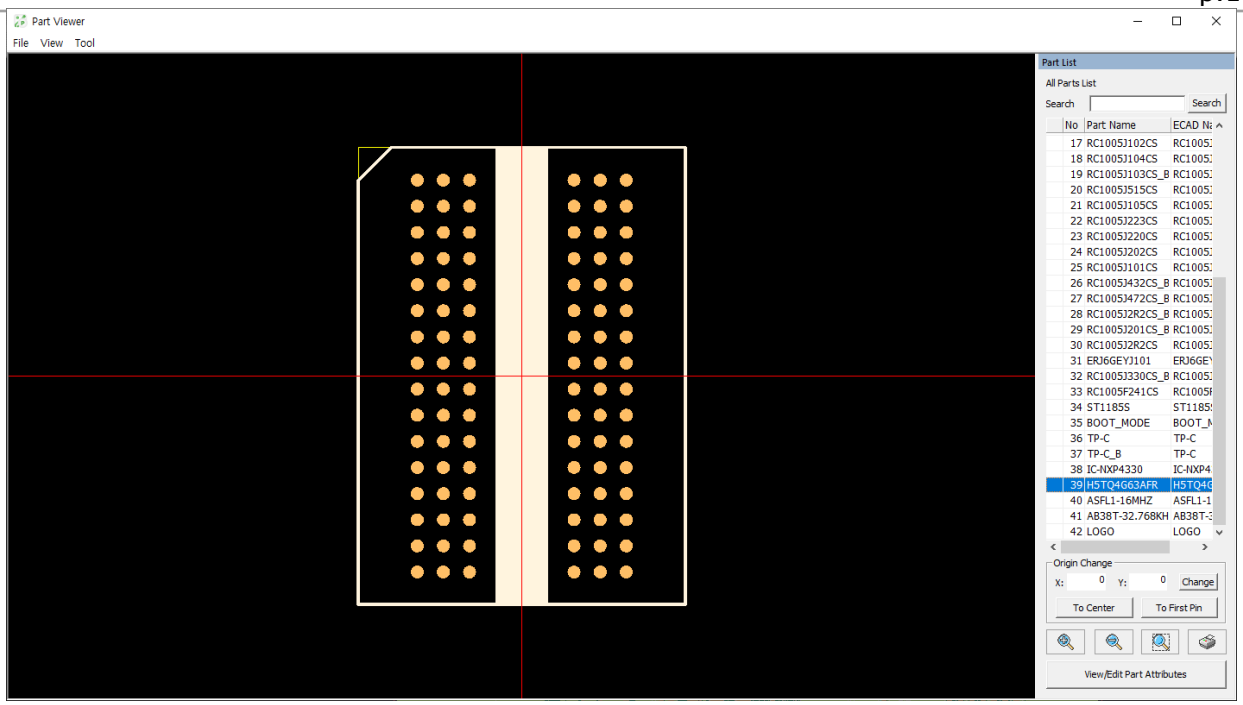


4. Various Object Viewers

4.1. Part Viewer

4.1.1. Execute PollEx PCB Part Viewer

Select menu **Option - Part Viewer** or click icon  from the toolbar.

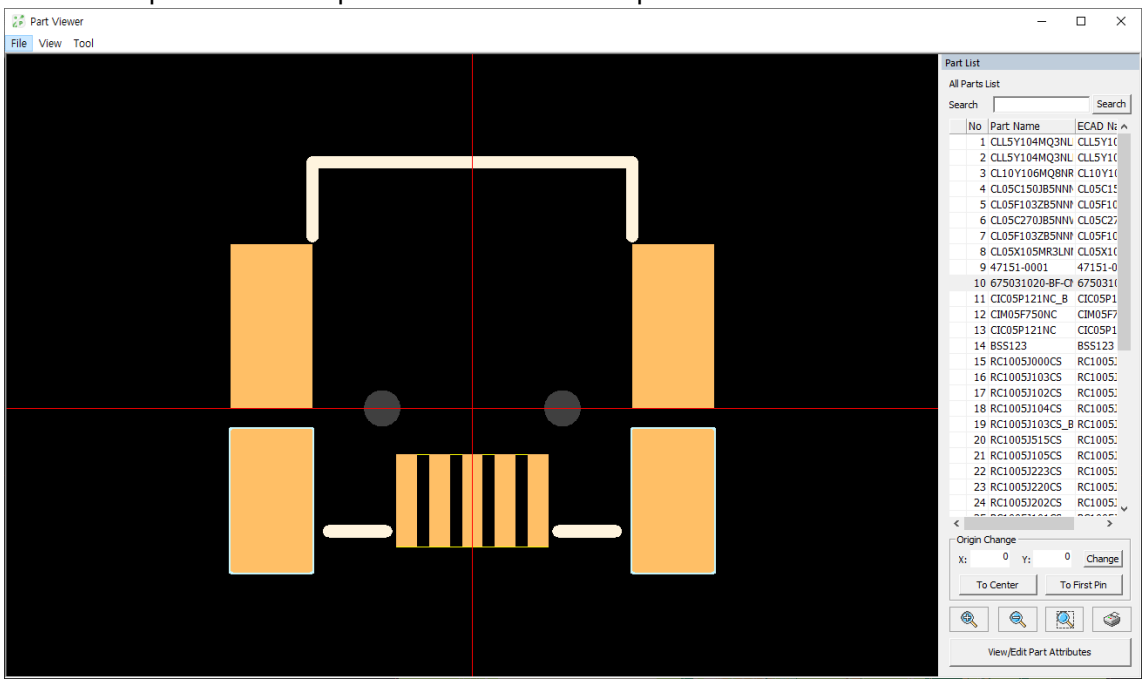


The **Part Viewer** window is activated in the center of the PollEx PCB, and the part is listed on the right-side of the window.

4.1.2. View Part Attribute

4.1.2.1. Check Part Shape

Select the part from the part list or search the part name in the search bar.

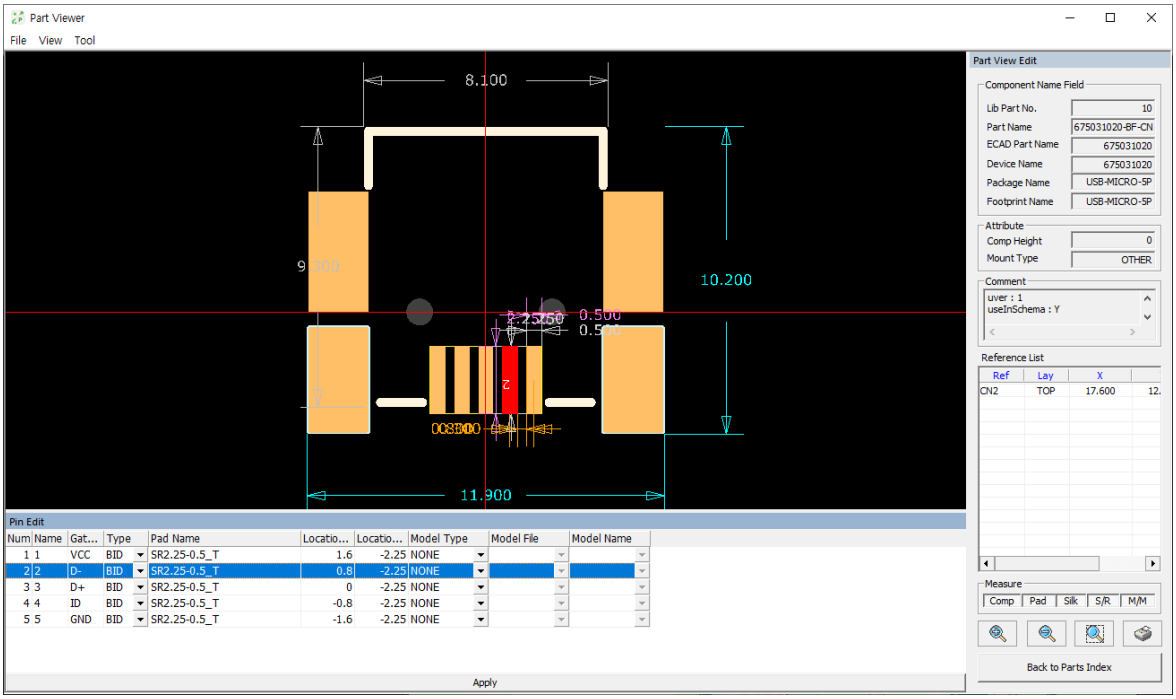


The selected part shape is displayed on the left side of the window.

The intersection of the red lines is the origin point of the part when created the library.

4.1.2.2. Check Pin Information of the Part

Click ***View/Edit Part Attributes*** button on the right-bottom side of the window after selected part from the list.



User can check the pin information of selected part as follows;

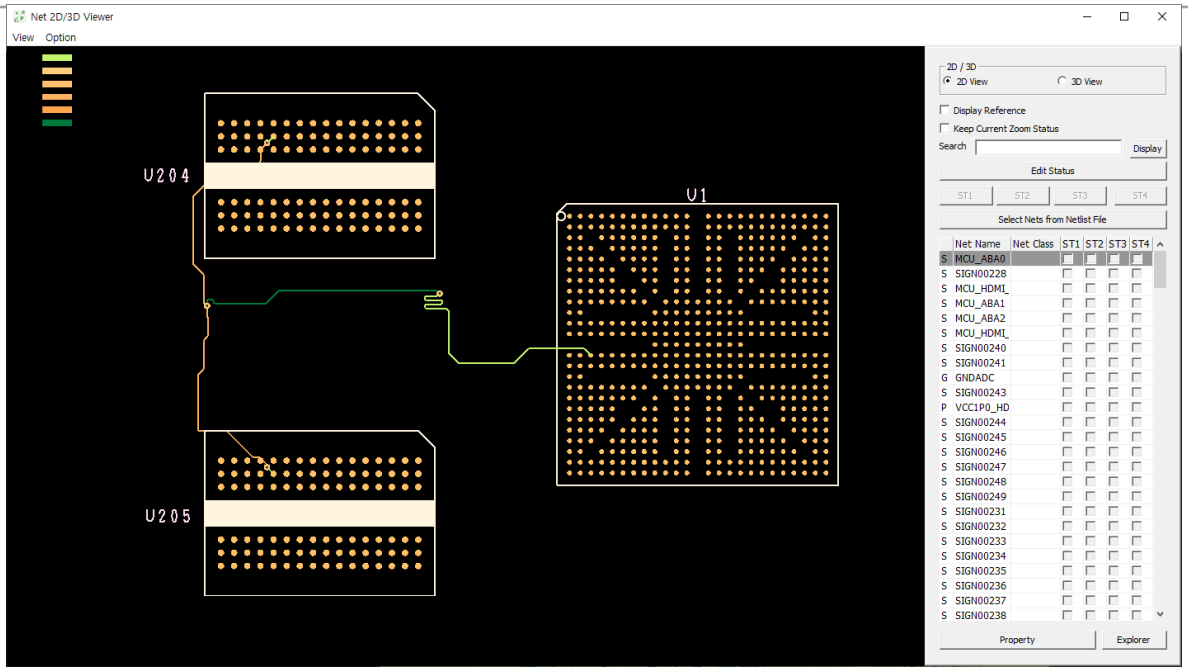
- Checking reference name list with the same part name, Layer location, and Layer coordinates.
- Checking dimension information with selection of Comp, Pad, Silk, S/R, M/M buttons.
- Checking the Pin name and location of the selected part.

4.2. Net 2D/3D Viewer

Net 2D/3D Viewer helps user to check nets connectivity and their 3-dimensional structures.

4.2.1. Open Net 2D/3D viewer

Press the menu, **Option – Net 2D/3D Viewer** or Click icon  from the toolbar.
Running the Net 2D/3D viewer, user can see 2D image with net lists and connected components as following picture.



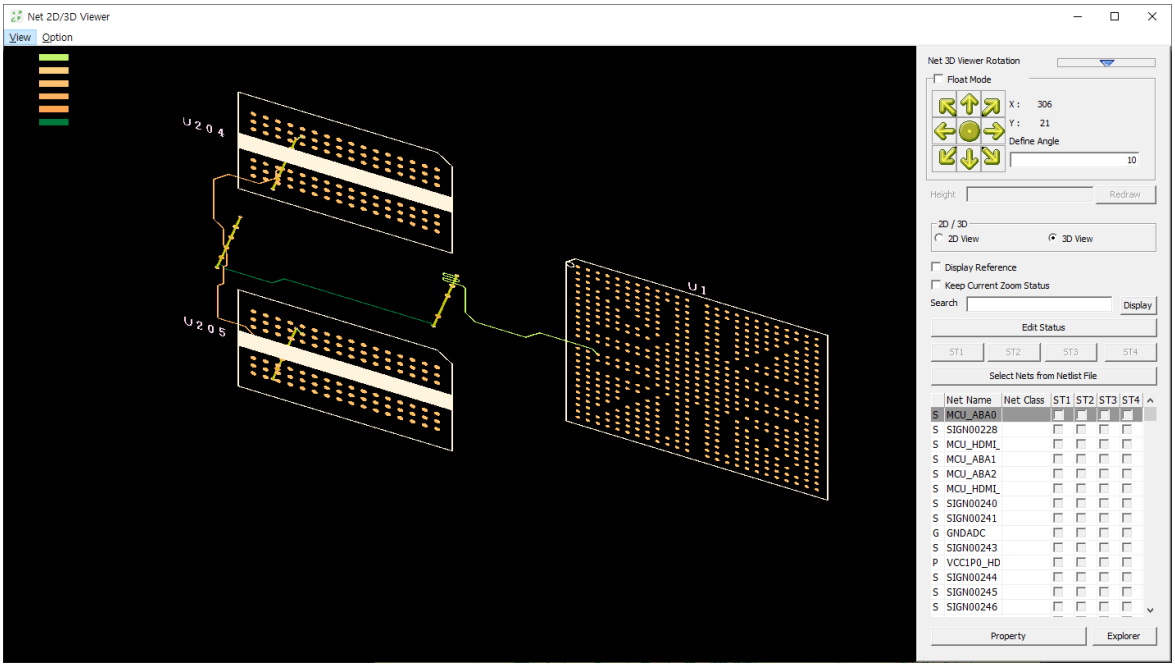
On the right side of the window, user can search net name or select from the netlists. If selected a net, user will see the selected net on the left side of the window. If user wants to check the net with the 3D, select **3D View** from **2D/3D** section on the top-right menu.

4.2.2. Check nets structure and their properties

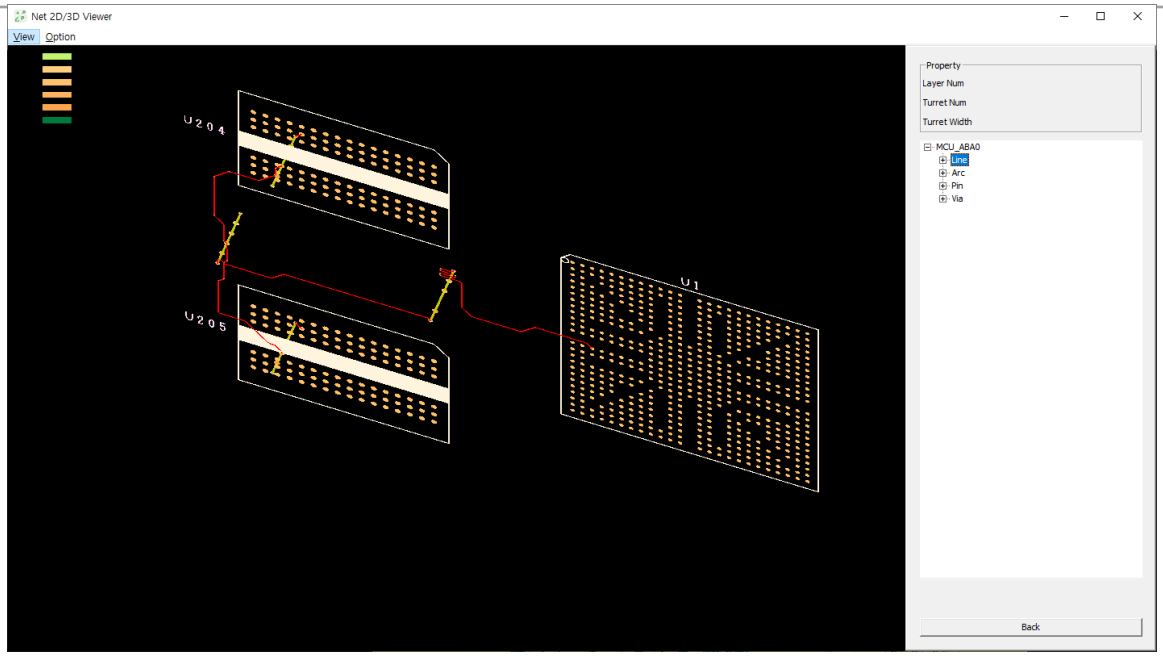
Click a net name ***MCU_ABA0*** from net list on right dialog window.

The arrow buttons under **Net 3D Viewer Rotation** menu can rotate the selected nets, or user can rotate it by moving mouse while holding the left-mouse click.

Using the mouse wheel, the nets can be zoom in or out.



Press the **Property** button menu at bottom of right dialog box.



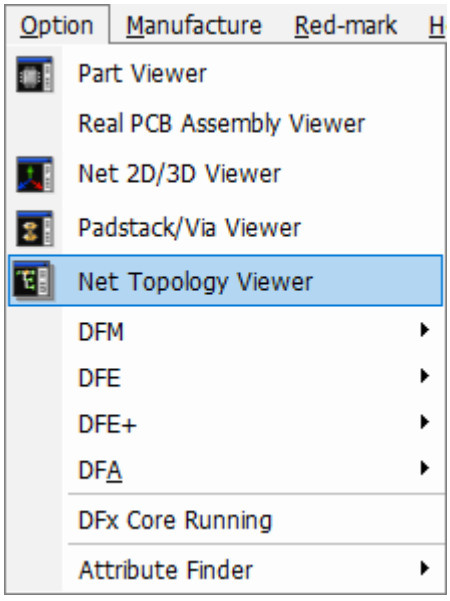
On property dialog, user can see the objects composing net. Clicking each object's item, user can see each structure in net routing pattern with highlight.

4.3. Net Topology Viewer

Net Topology Viewer is durable to check the net as topology structure (Driver-Load-Receiver) with the connected Via and branch. This menu also can display with different color per placed layer with net length and width.

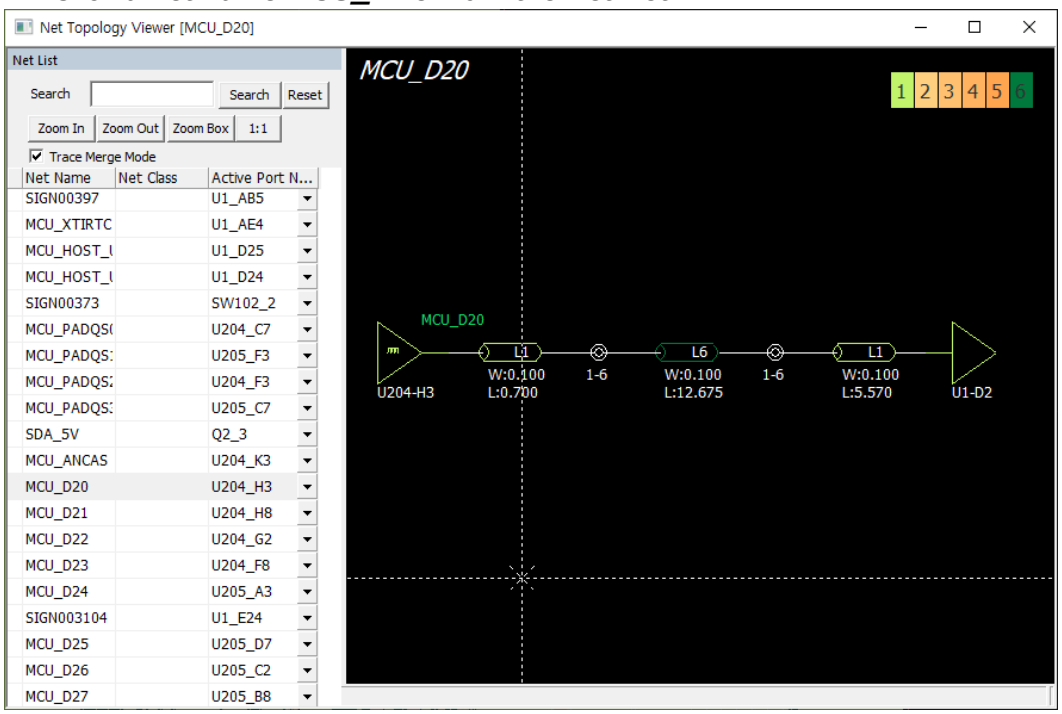
4.3.1. Run Net Topology Viewer

- Select menu, **Option-Net Topology Viewer** or Click icon on the toolbar.



Net Topology Viewer is shown as above.

- Click a net name ***MCU_D20*** from the Net List.



MCU_D20 is displayed as topology structure.

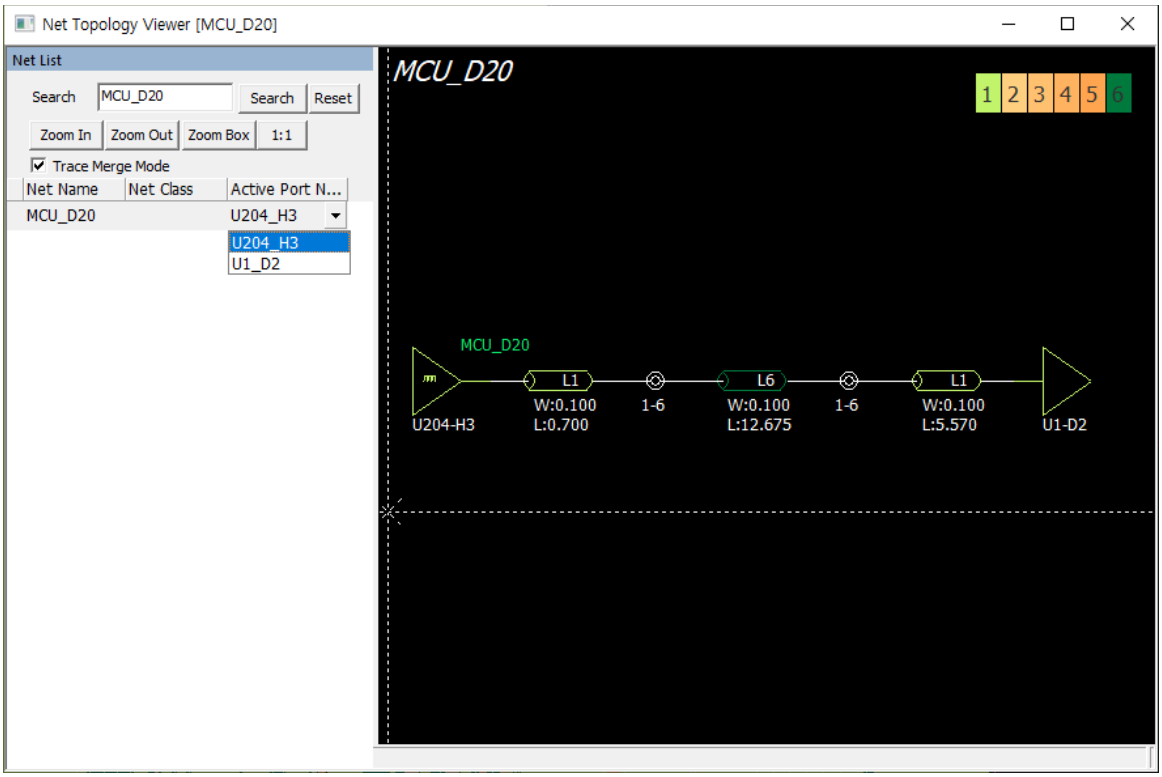
- 1 2 3 4 5 6: Placed layer and color
- Driver
- W: Net width
- L: Net length

4.3.2. Change the driver of certain net

It can change the driver component on **Active Port Name** menu.

Input **MCU_D20** in the search bar and then press Enter key.

Select **MCU_D20** on the list.
Then change the **Active Port Name** to **U1_D2**.



4.4. Real PCB Assembly Viewer

Real PCB Assembly Viewer can review design into 3D shapes and export its data into STEP format for linking to mechanical CAD system.

For more detailed information, please go to the **PollEx Real PCB Assembly Viewer Tutorial** section.

5. Documentation and Red-Mark, making comments on PCB

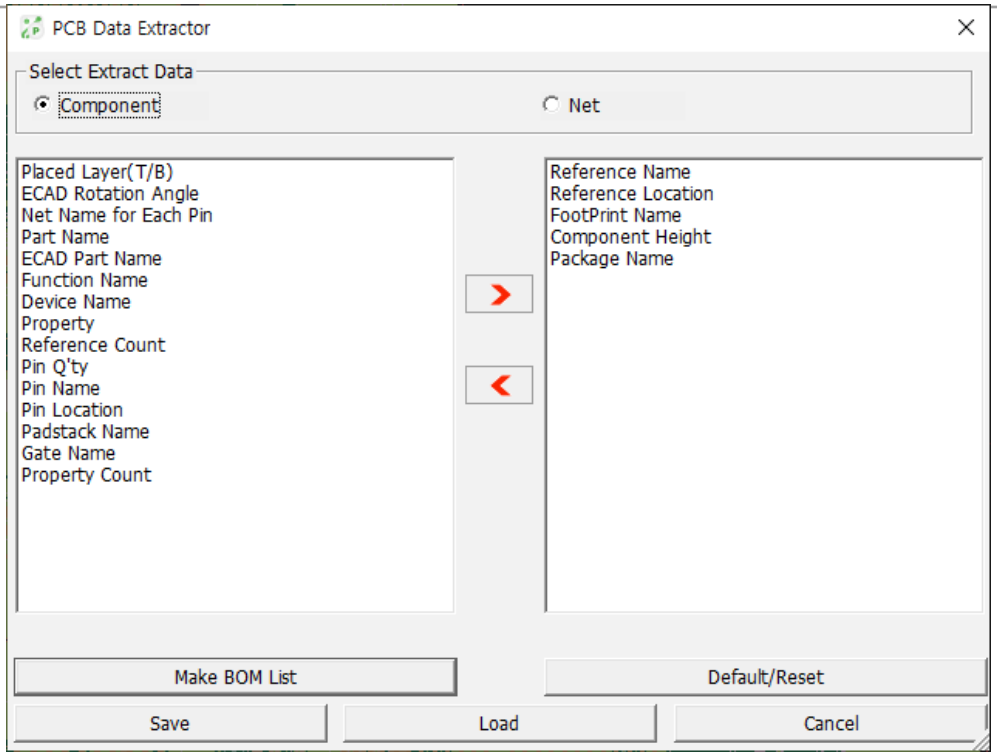
5.1. PCB Data Extractor

PCB Data Extractor on PollEx PCB helps to make document of components and nets information easily, and it also can export to MS Excel file.

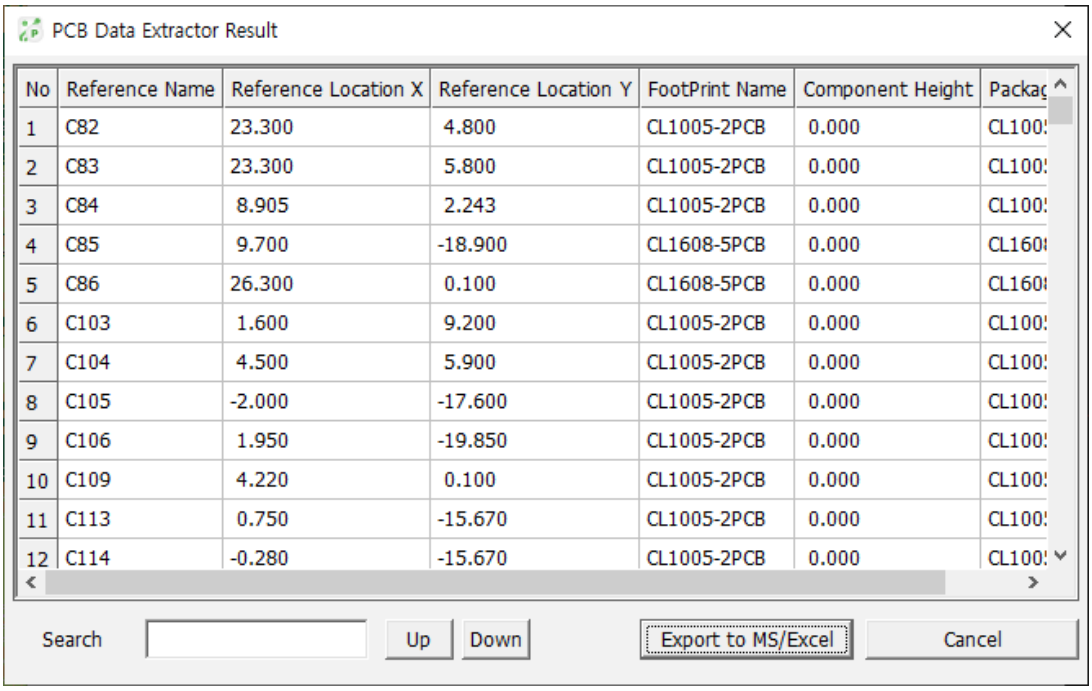
5.1.1. Execute PollEx PCB Data Extractor

Select menu **Tools – PCB Data Extractor** or click icon  from the toolbar.

To make the report of component list or net list, move items to be documented from left to right column by using red arrows.



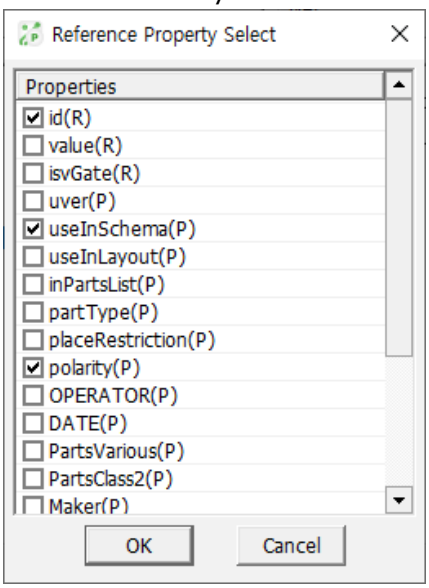
Click ***Make BOM List*** after user select items.



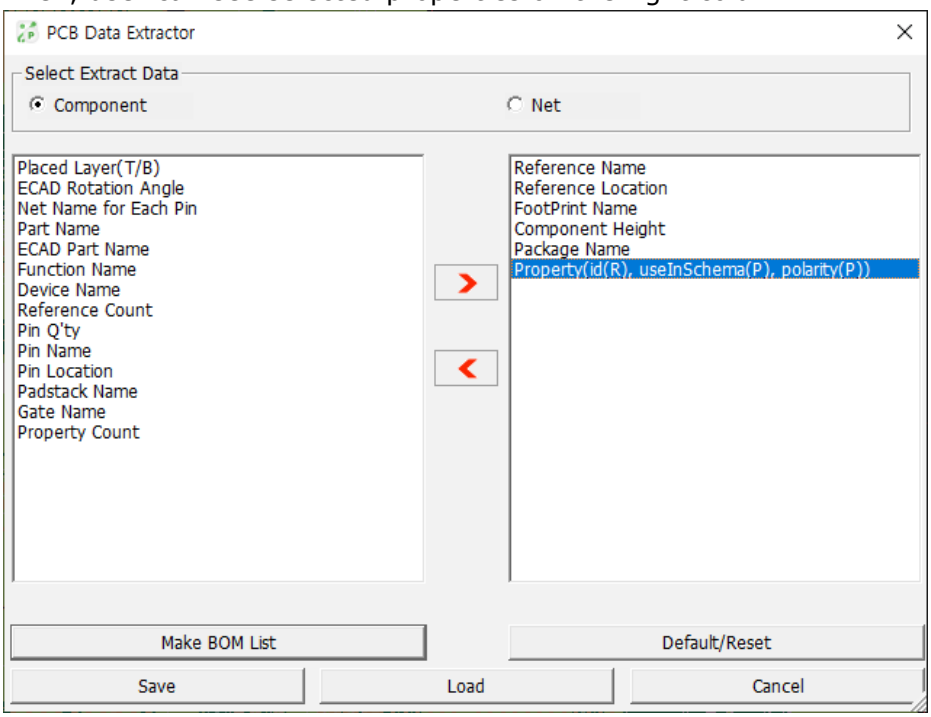
Click ***Export to MS/Excel*** button if user wants to export to MS Excel file.
Regarding the Net information (Net length, connected via, total via numbers, and so on), choose with Net and run the same way as Component. User can make a document and export to MS Excel file.

5.1.2. Exporting report with having reference property
When user select **Property** on the left column and try to move to the right, user will see a pop-up window of **Reference Property Select**.

Select necessary information to be included in the document and click OK.



Then, user can see selected properties on the right column.



5.1.3. Save the environment for making report

Select **Save** or **Load** button on bottom side of PCB Data Extractor's window.

It can save or load the environment as file(`*.pdt`) for using whenever user make the report.

Insert captured image into BOM file.

5.2. Net Analyzer

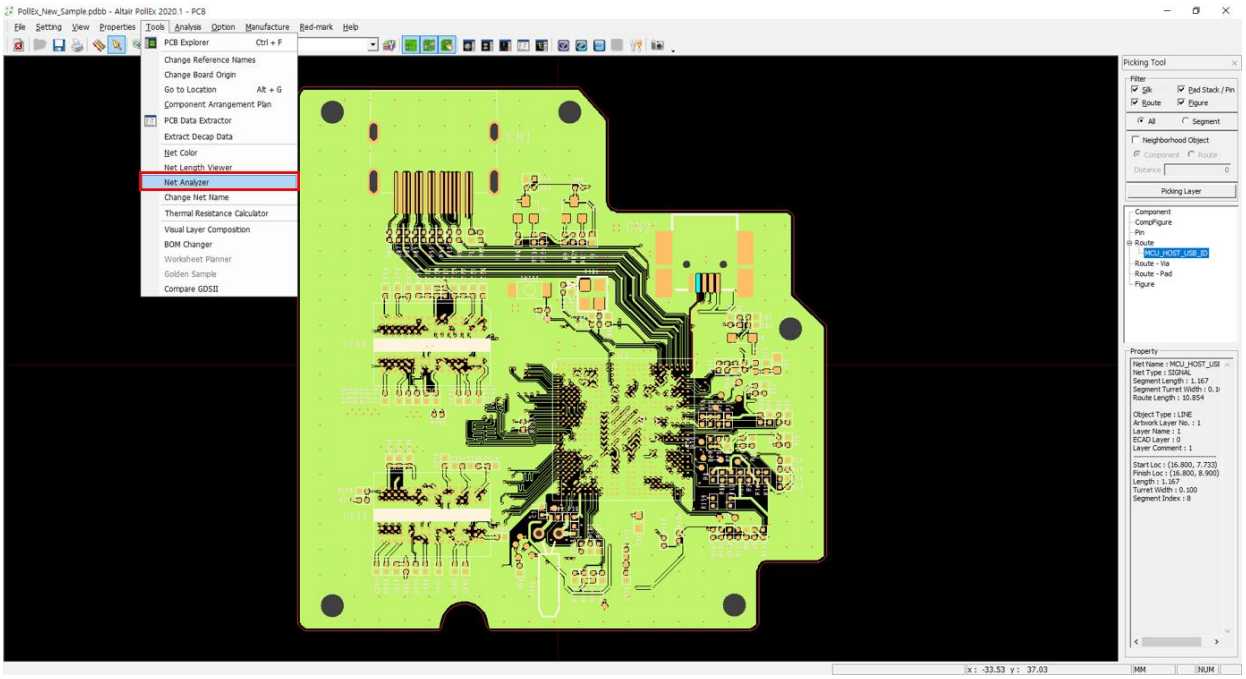
Net Analyzer is used for analyzing all net length information in the PCB by considering Layer, Branch, and Via location.

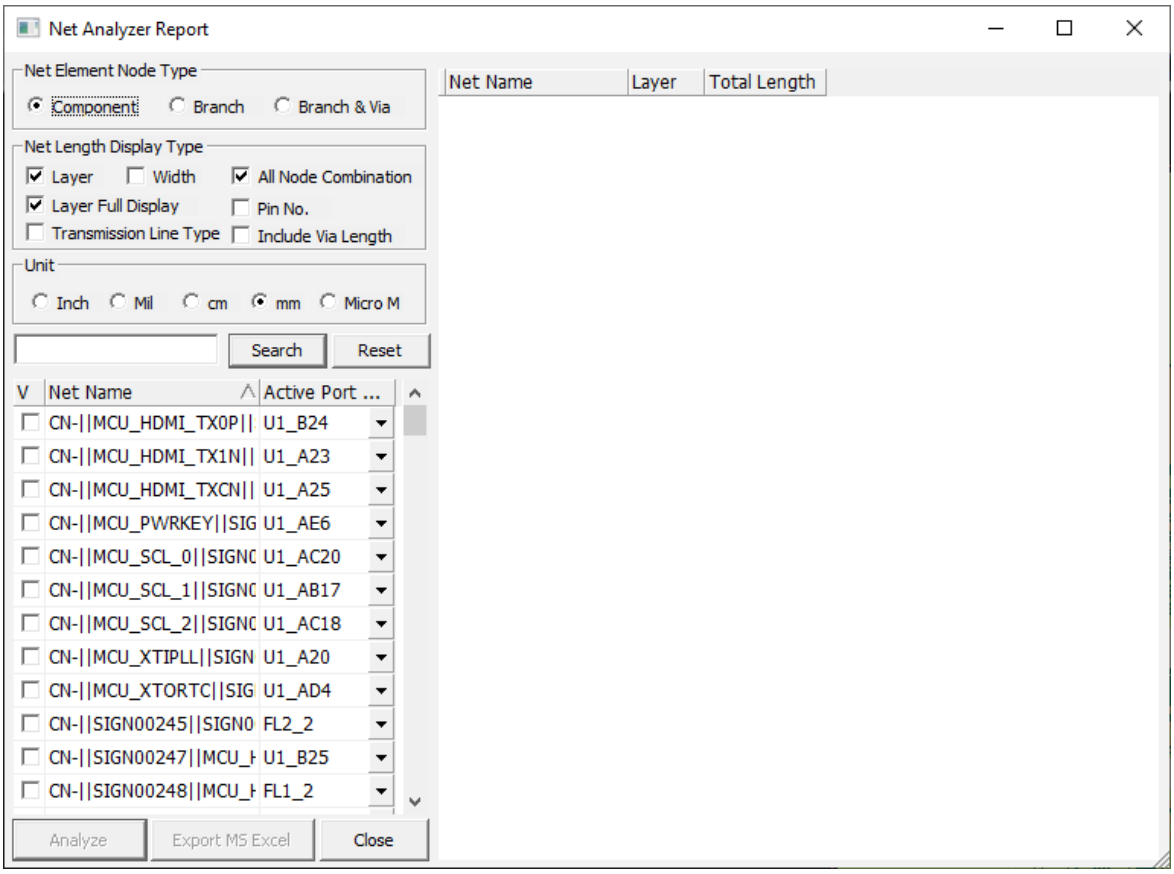
5.2.1. Open PollEx PCB design file

Open sample file in `C:\Temp\Altair-PollEx\PollExPCB\PollEx_New_Sample.pdbb`.

5.2.2. Open Net Analyzer

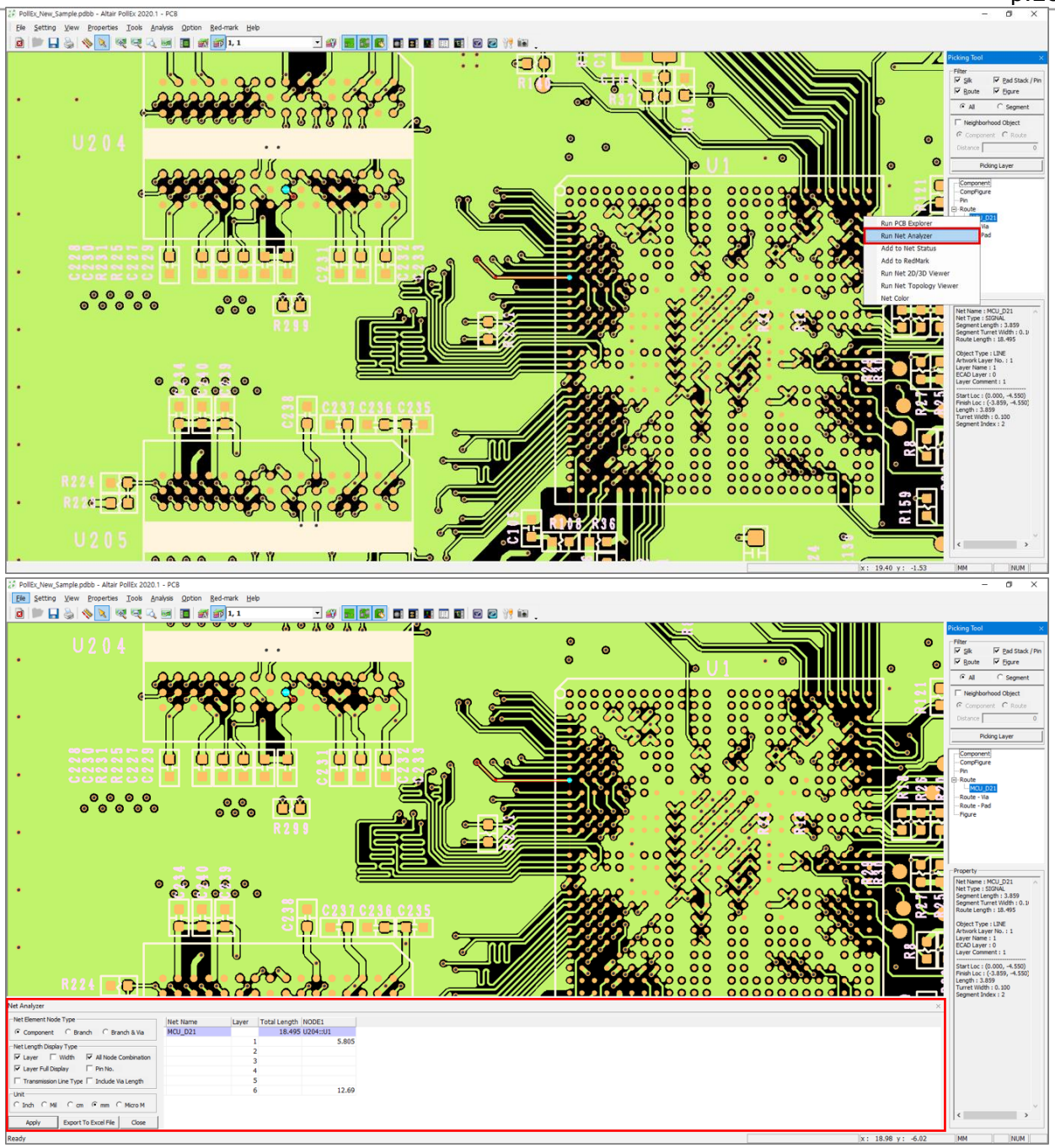
Select menu, **Tools – Net Analyzer**





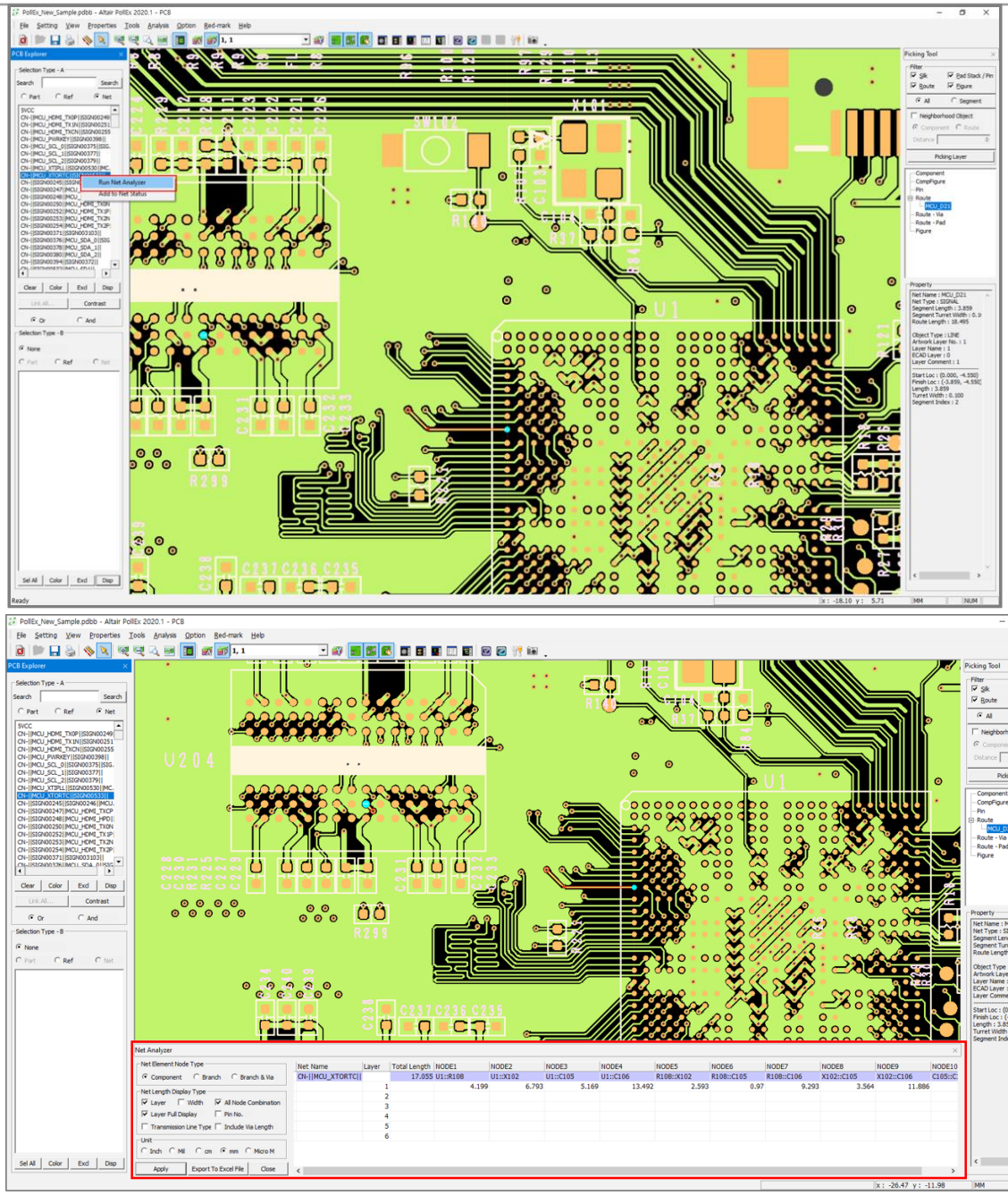
There are two other ways to run the Net Analyzer.

Select Net on the PCB design and right-mouse click of the Net on the right-side picking tool menu. Select menu **Run Net Analyzer**.



Net Analyzer window is shown on the bottom side.

When user run the PCB Explorer on the right side, user can select Net menu and select Net from the netlists. Then, run the **Net Analyzer** with right-mouse click.



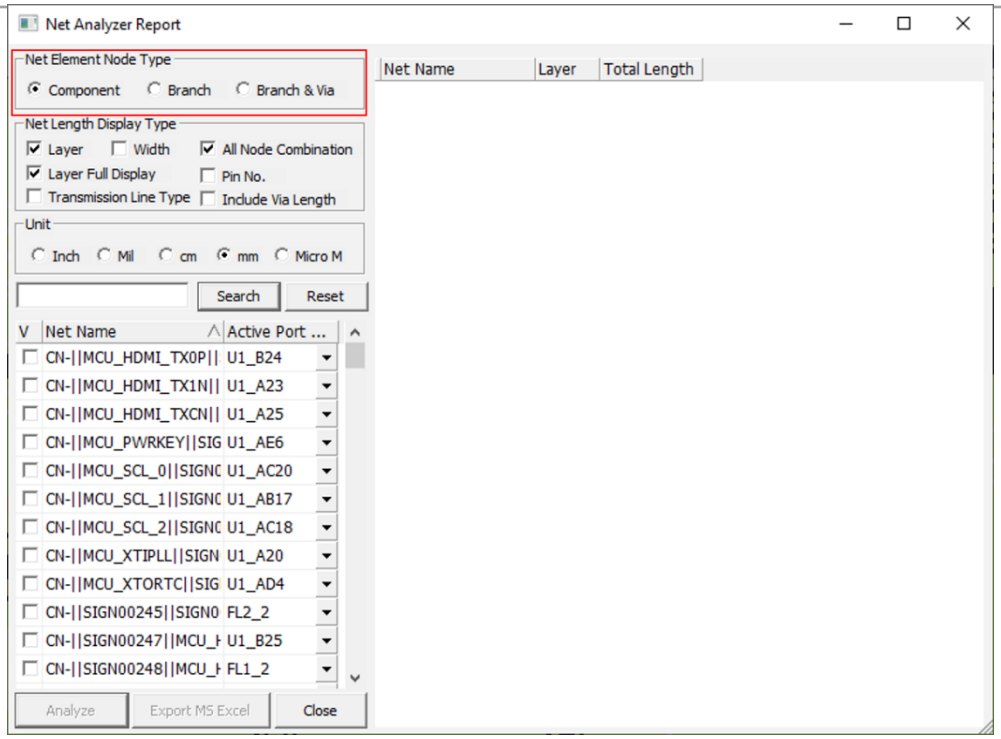
Net Analyzer window is shown on the bottom same as the previous method.
On this tutorial, we explain the Net Analyzer based on running Tools menu.

5.2.3. Net Element Node Type Selection

Select menu, **Tools – Net Analyzer**

Select a type from Component, Branch, and Branch & Via.

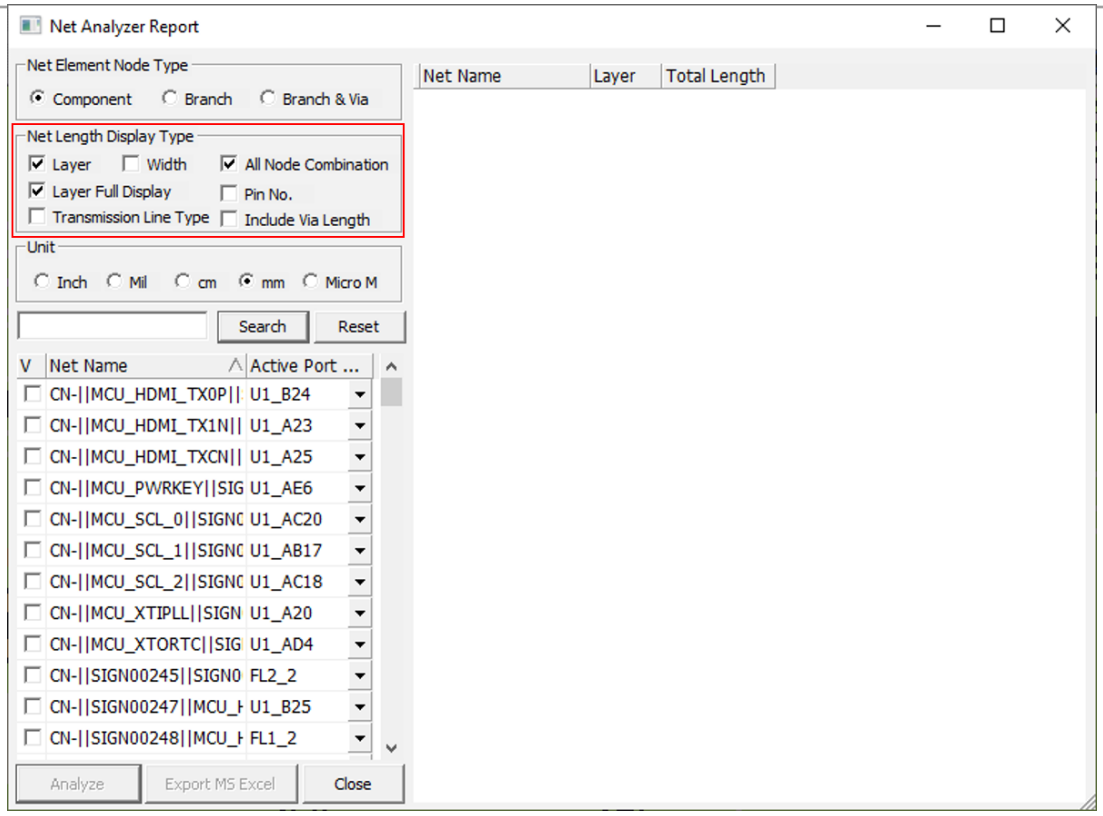
a. Component is net length among components.
b. Branch is net length among components and including length of branched points.
c. Branch & Via is net length among components, branching points, and vias.



5.2.4. Net Length Display Type Selection

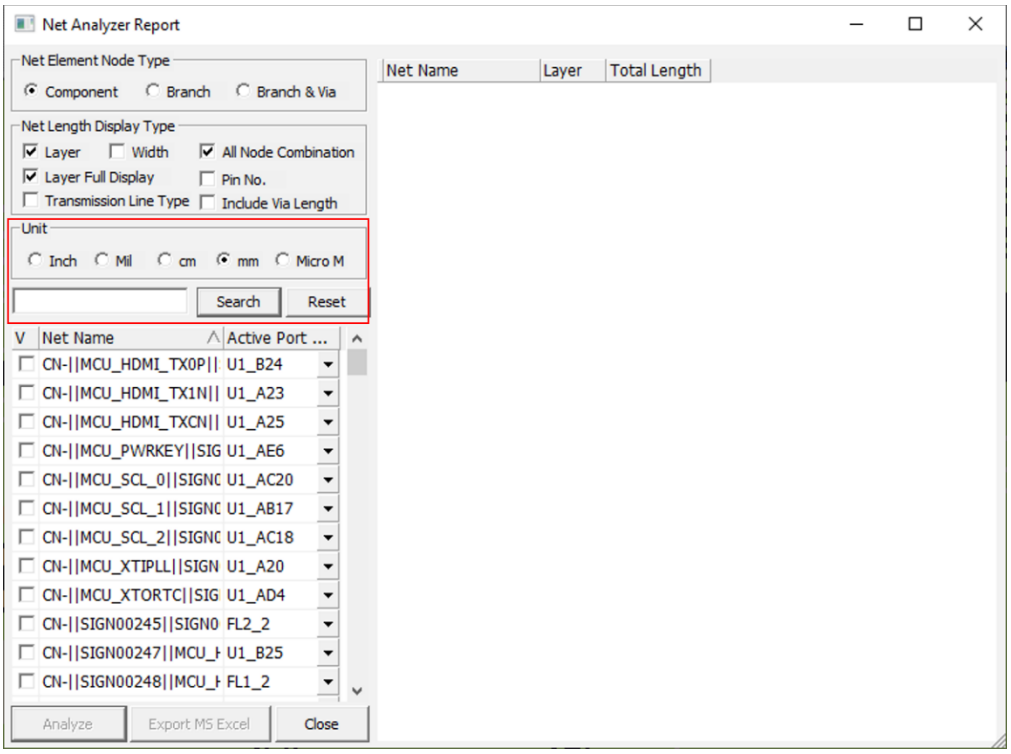
Select an item to be display (At least, 1 or more items must be selected).

a. Layer: Display net length by a layer.
b. Width: Display routing pattern's width
c. All Node Combination: Display all combination of nodes
d. Layer Full Display: Display stack-up information and net length by a layer
e. Pin No: Display component's pin number
f. Transmission Line Type: Display lengths of micro strip line and strip line
g. Include Via Length: Display via information


Net Analyzer Report
Net Element Node Type
Component
Branch
Branch & Via
Net Length Display Type
Layer
Width
All Node Combination
Layer Full Display
Pin No.
Transmission Line Type
Include Via Length
Unit
Inch
Mil
cm
mm
Micro M
Search
Reset
V
Net Name
Active Port ...
CN-||MCU_HDMI_TX0P|| U1_B24
CN-||MCU_HDMI_TX1N|| U1_A23
CN-||MCU_HDMI_TXCN|| U1_A25
CN-||MCU_PWRKEY||SIG U1_AE6
CN-||MCU_SCL_0||SIGNC U1_AC20
CN-||MCU_SCL_1||SIGNC U1_AB17
CN-||MCU_SCL_2||SIGNC U1_AC18
CN-||MCU_XTIPLL||SIGN U1_A20
CN-||MCU_XTORTC||SIG U1_AD4
CN-||SIGN00245||SIGN0 FL2_2
CN-||SIGN00247||MCU_I U1_B25
CN-||SIGN00248||MCU_I FL1_2
Analyze
Export MS Excel
Close
Net Name
Layer
Total Length

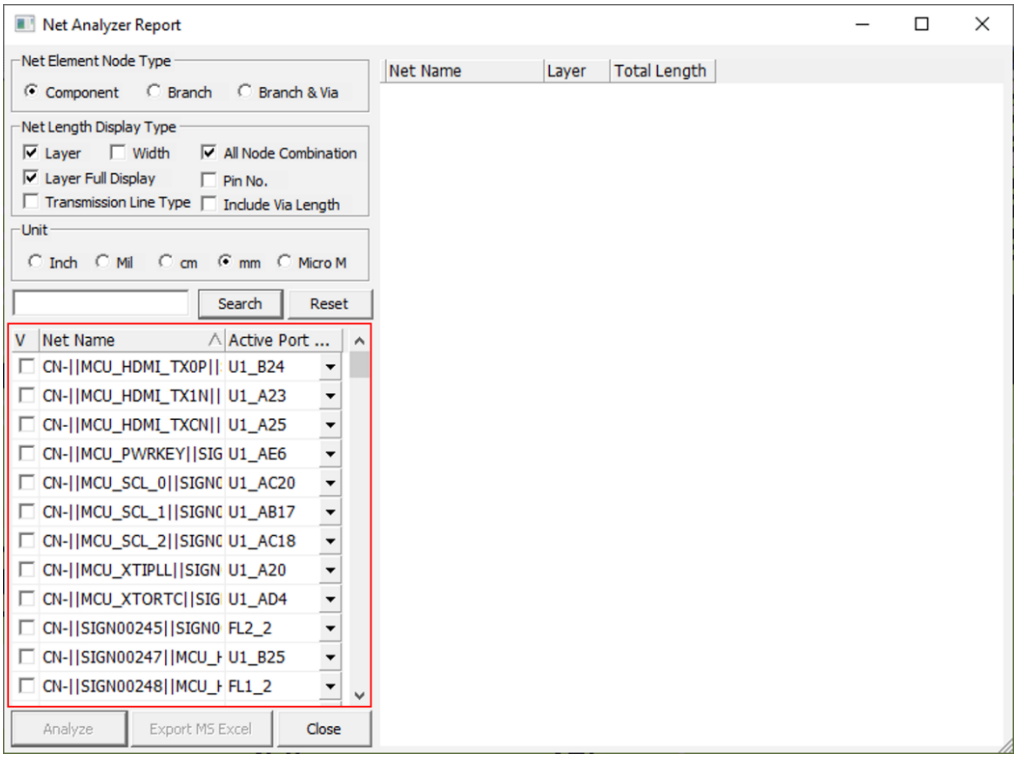
5.2.5. Unit Selection

Select display unit among Inch, Mil, cm, mm, and Micro M.



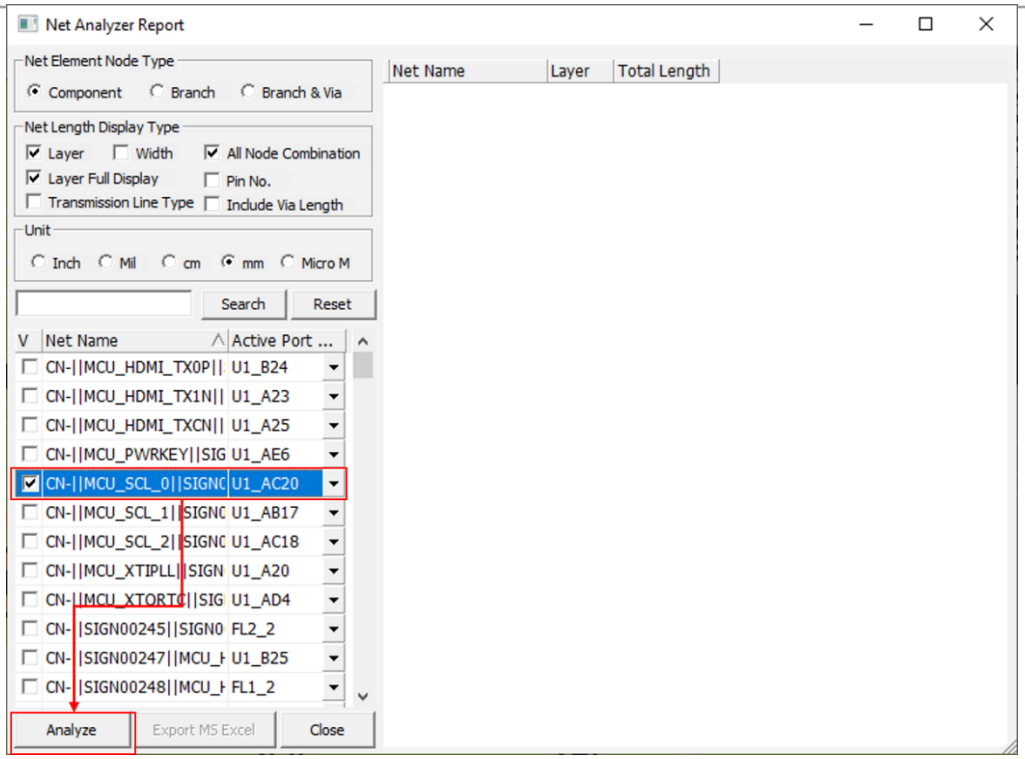
5.2.6. Net Search

Search Net Name to be displayed from the lists.



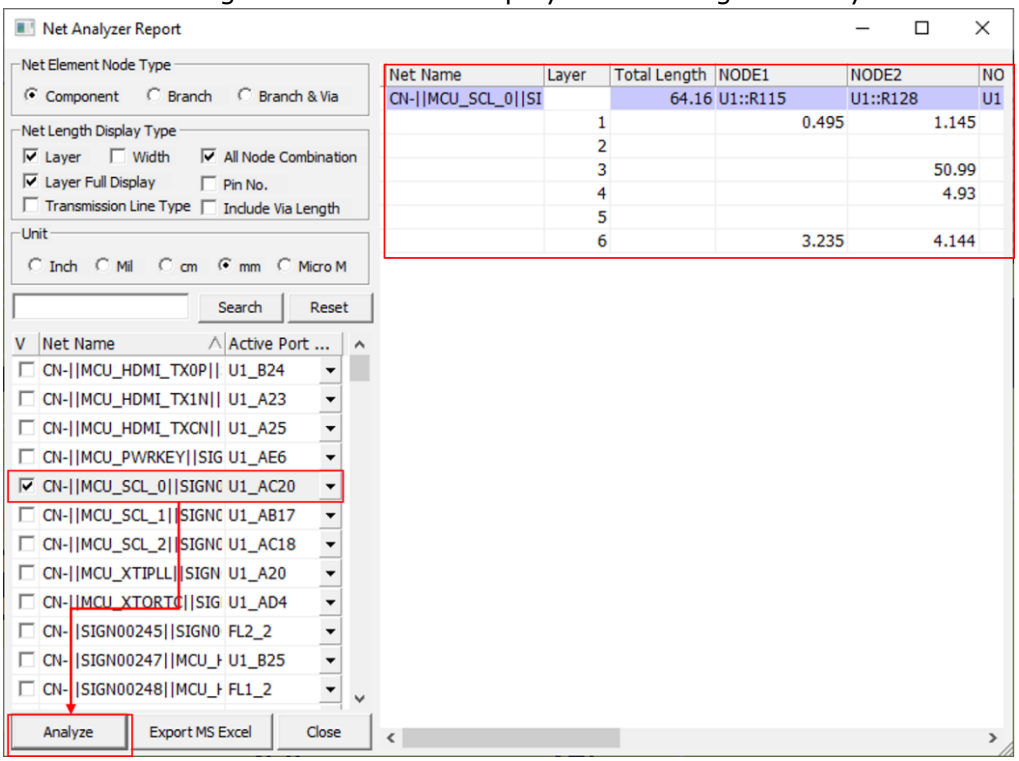
5.2.7. Net Check

Check the box on the V column to be displayed and click ***Analyze*** button.



5.2.8. Display

Selected net length information is displayed on the right side by a net name.



5.2.9. Excel Export

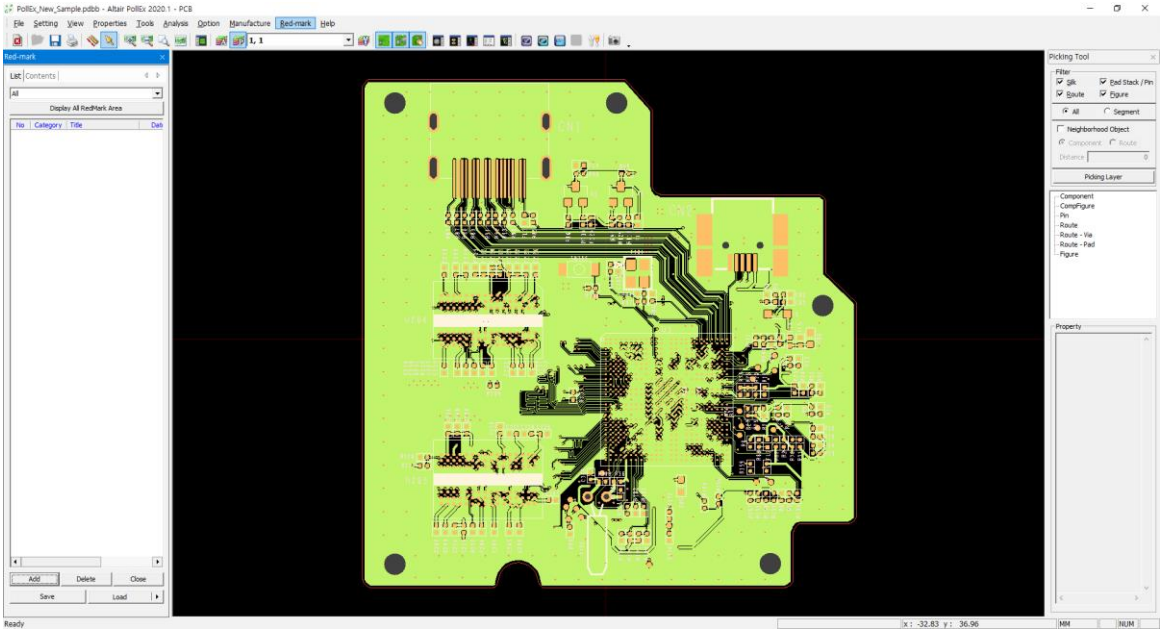
Click ***Export MS Excel*** button to make a report.

Close the PCB Explorer, Net Analyzer windows, then click  icon from the toolbar.

5.3. Red-Mark

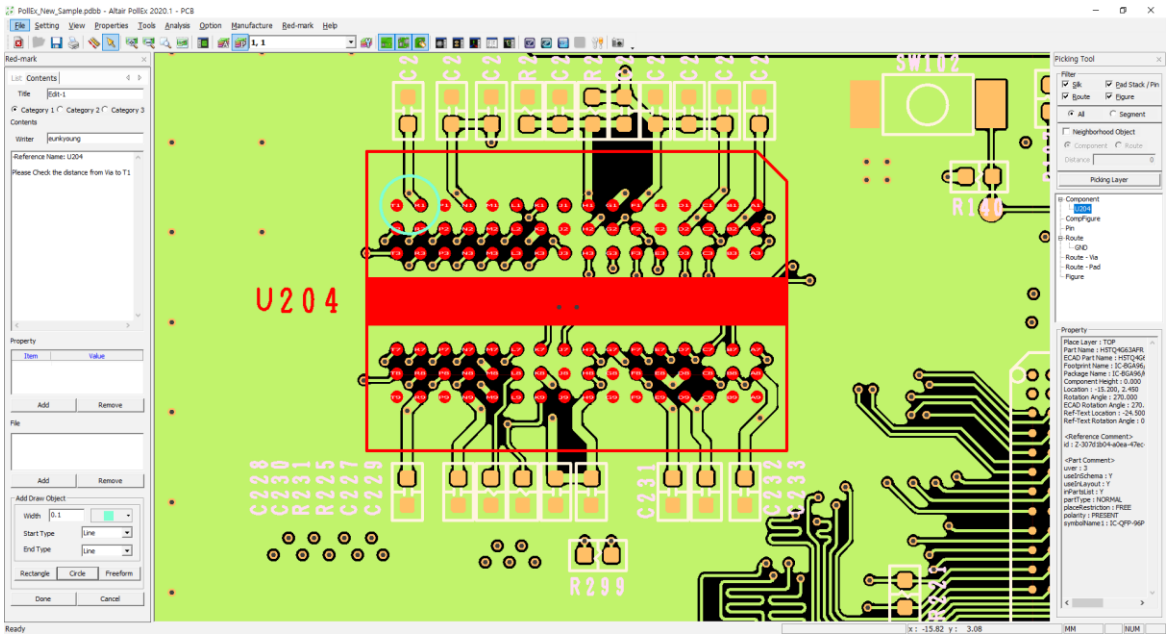
5.3.1. Execute Red-Mark

Select menu, **Red-mark – Red-mark**.



5.3.2. Make a comment

Click ***Add*** button on the bottom of the Red Mark window.



Input Title and Contents

Select area to make a comment by using Add Draw Object.

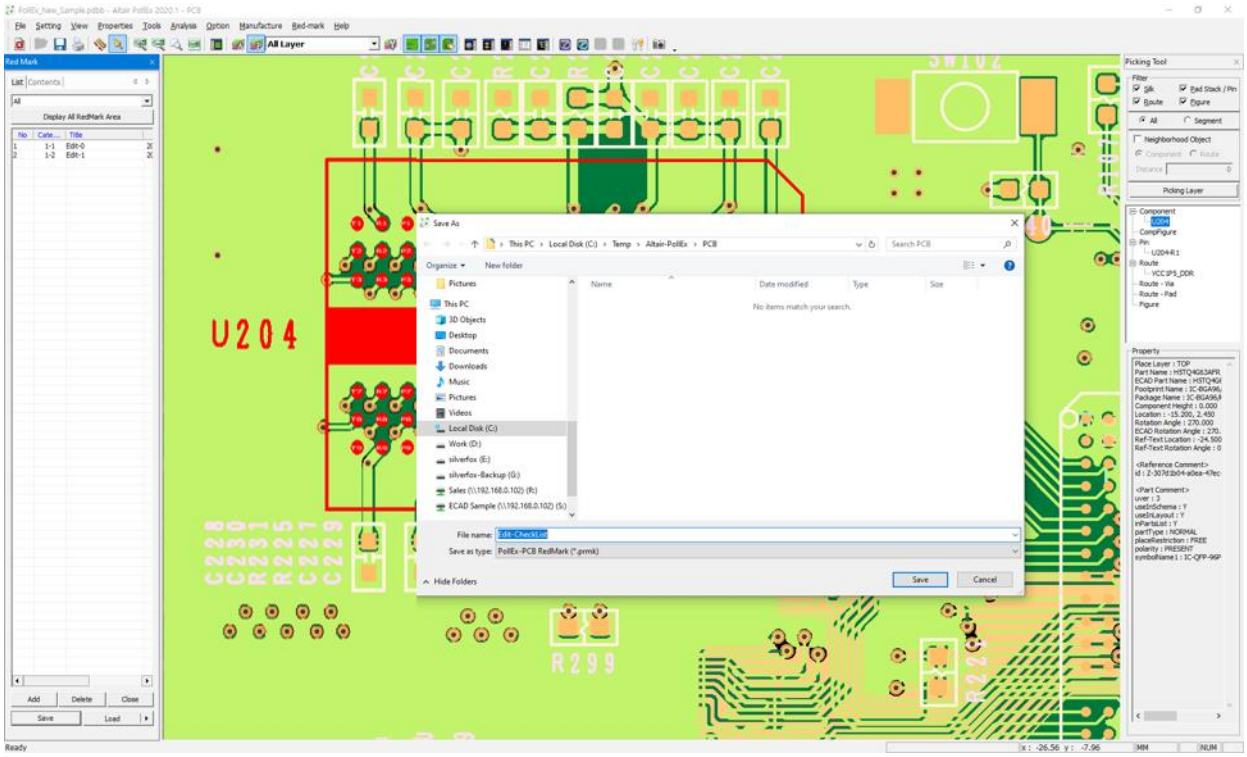
When selected the area, user can use Rectangle, Circle, or Freeform with line color and width.

Click Done when finished the input, then user can see the comment on the list.

5.3.3. Save

User can save file either `*.prmk` or `*.pdbb` format. Saved file can be shared with other users to check the comments.

5.3.3.1. Save with prmk file format



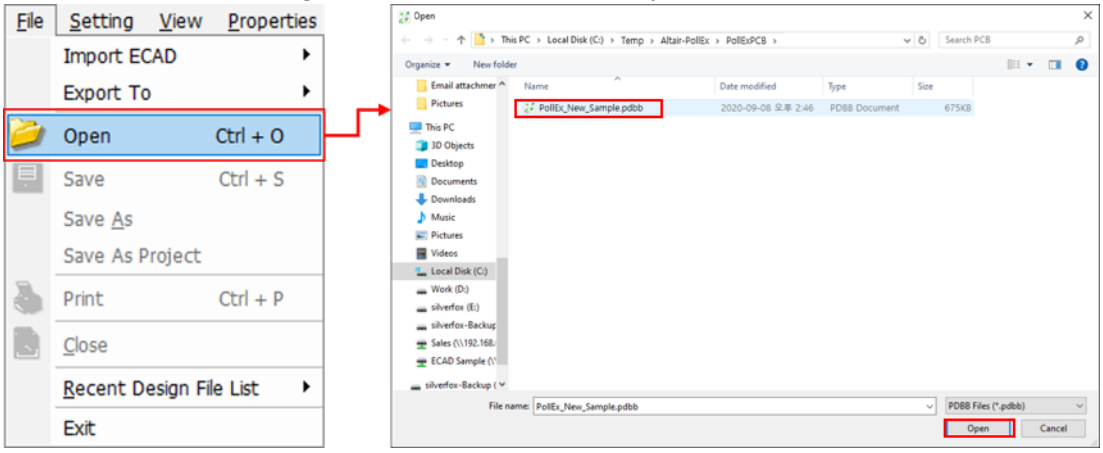
Click ***Save*** button on the bottom side.
Select file location and input file name, then click ***Save***.

5.3.3.2. Save with pdbb file format
Click menu, **File – Save As**.
Select file location and input file name, then click ***Save***.

5.3.4. Load Redmark File

5.3.4.1. Load prmk file
Click ***Load*** button on the bottom side of Red Mark menu (If the program is disabled, go to menu **Red-mark – Red-mark**).
Select file to load.

5.3.4.2. Load pdbb file
Select menu, **File – Open**, and select a file to open.



6. Useful functions for engineers

6.1. Component Arrangement Plan

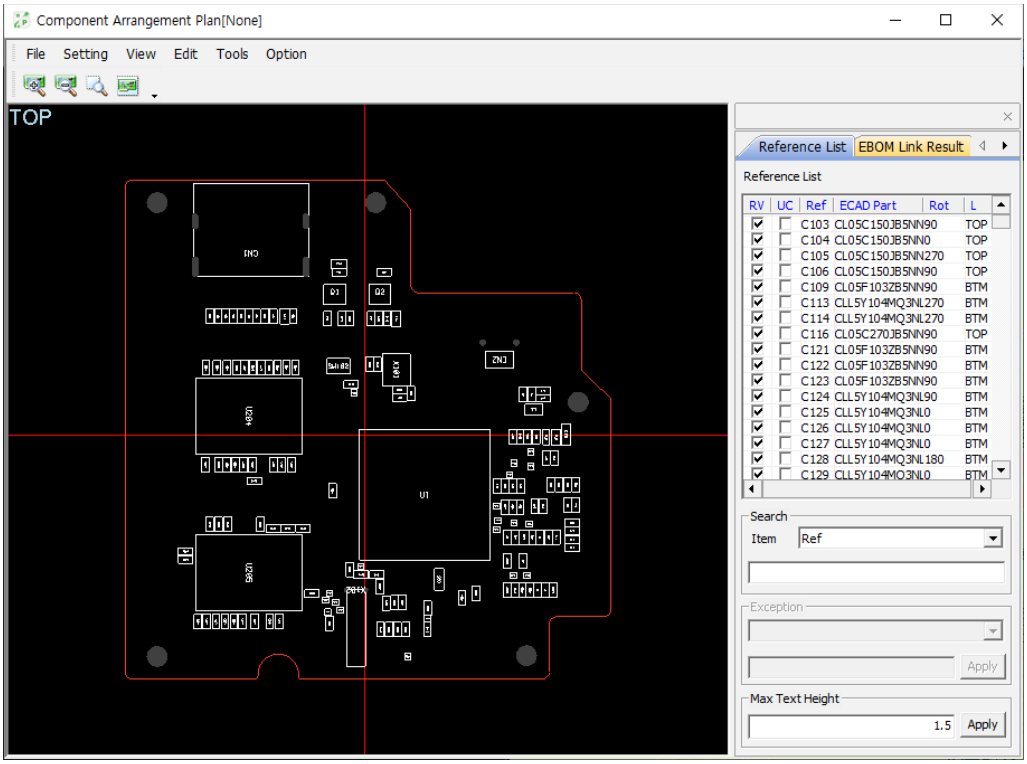
Using PollEx PCB's feature, **Component Arrangement Plan**, user can create work-plan sheet. And using created format, also user can apply this format to other design to make another work-plan sheet.

6.1.1. Open PollEx PCB design file

Open sample file, `PollEx_New_Sample.pdbb` file in path, `C:\Temp\Altair-PollEx\PollExPCB` using menu, **File-Open**.

6.1.2. Launch tool, **Component Arrangement Plan**

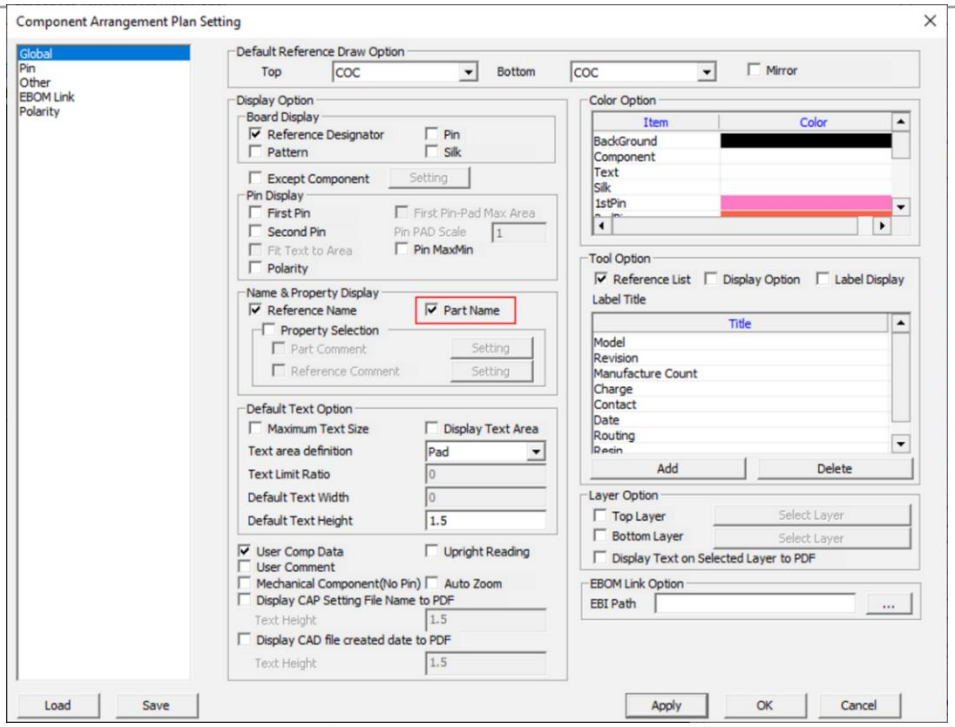
Select menu, **Tools – Component Arrangement Plan**.



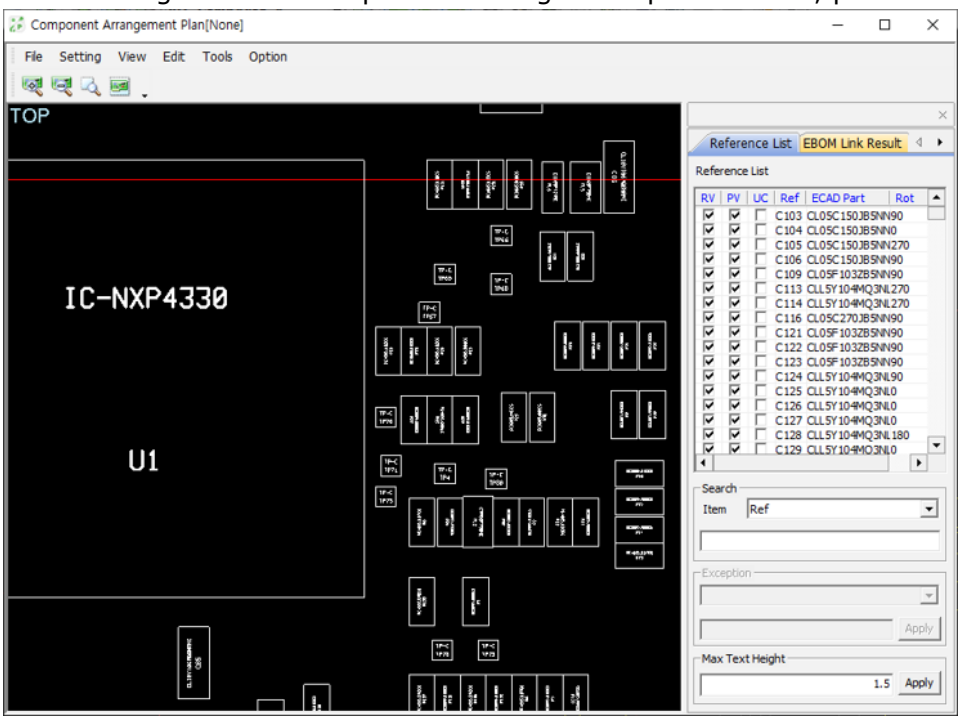
Above picture is launched **Component Arrangement Plan** window. At initial status, tool shows components outline and reference names.

6.1.3. Adding part name or values

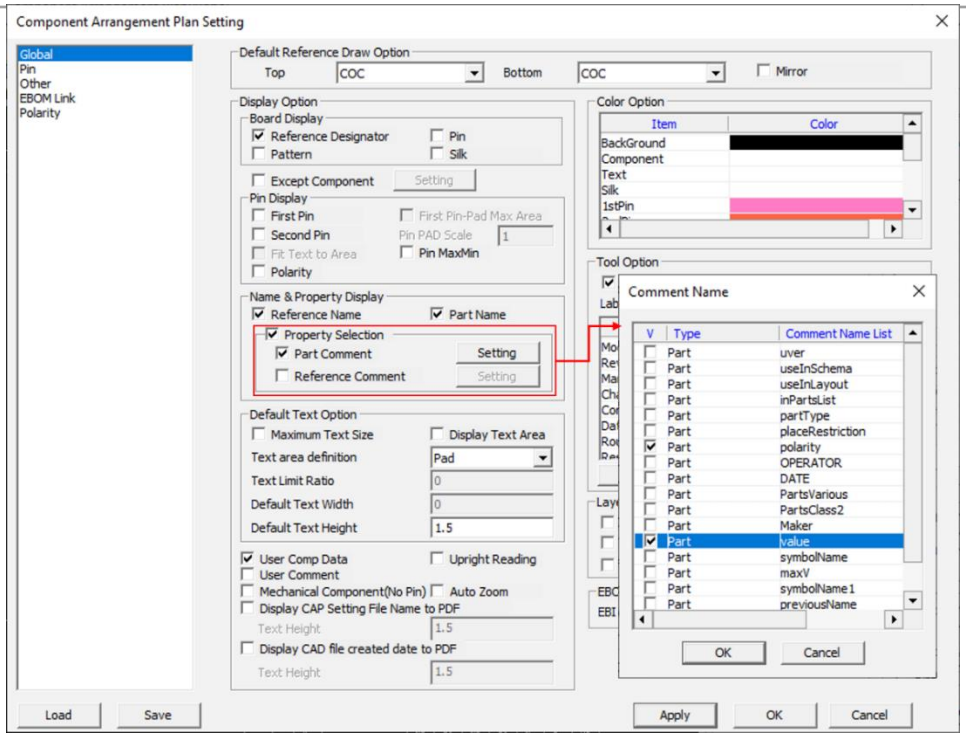
Select the menu, **Setting – CAP Data Setting**. At setting dialog box, select **Part Name** and press the button ***OK***.



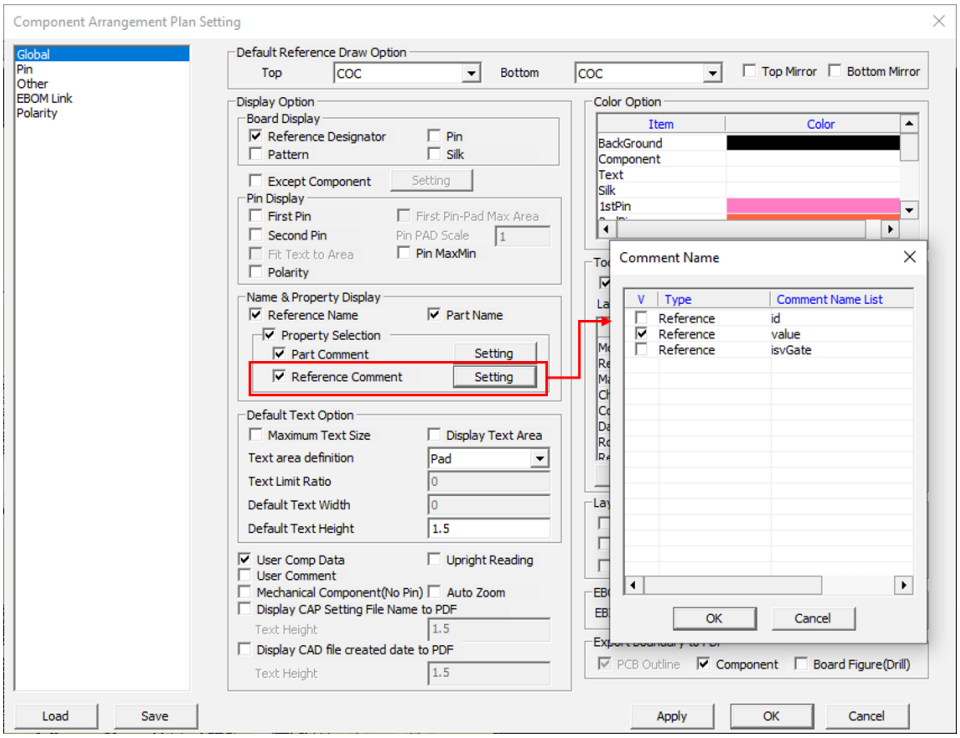
When user go back to component arrangement plan window, part name will be displayed.

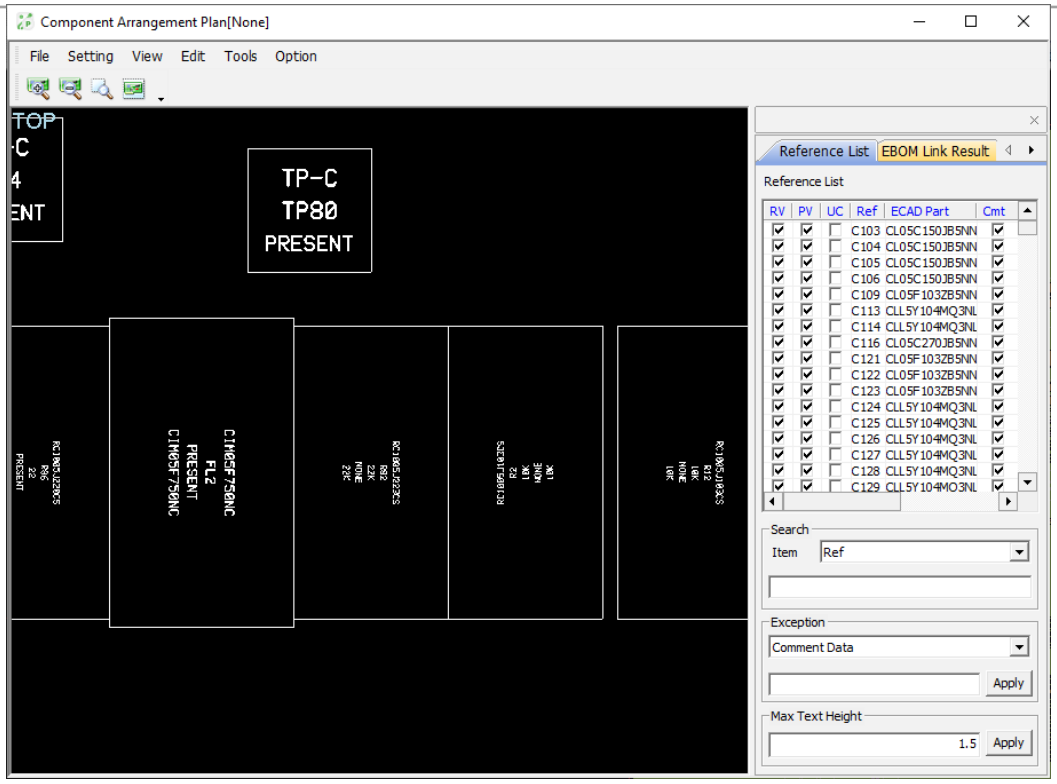


Using the menu, **Setting – CAP Data Setting**, add components value on display.

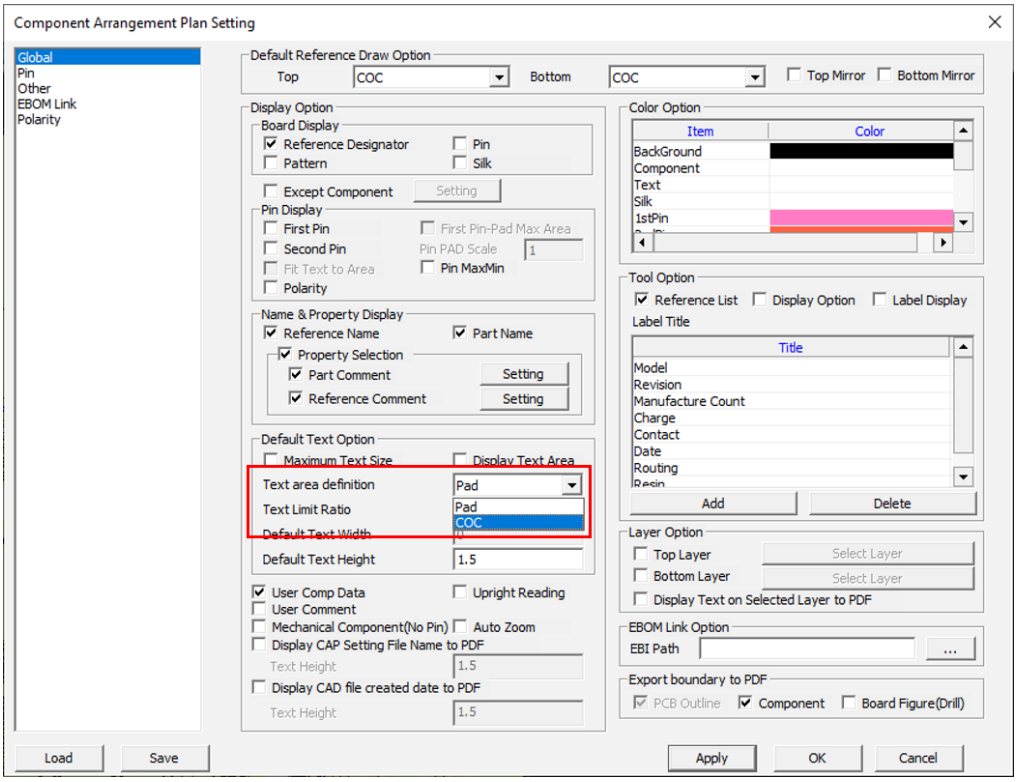


At setting dialog window, selecting **Part Comment** in **Name and Property Display** section and pressing **Setting** menu button will launch **Component Name** dialog window. Here user can select **value**. (If there is no value in part property, user can select value in **Reference Comment** as below picture.)

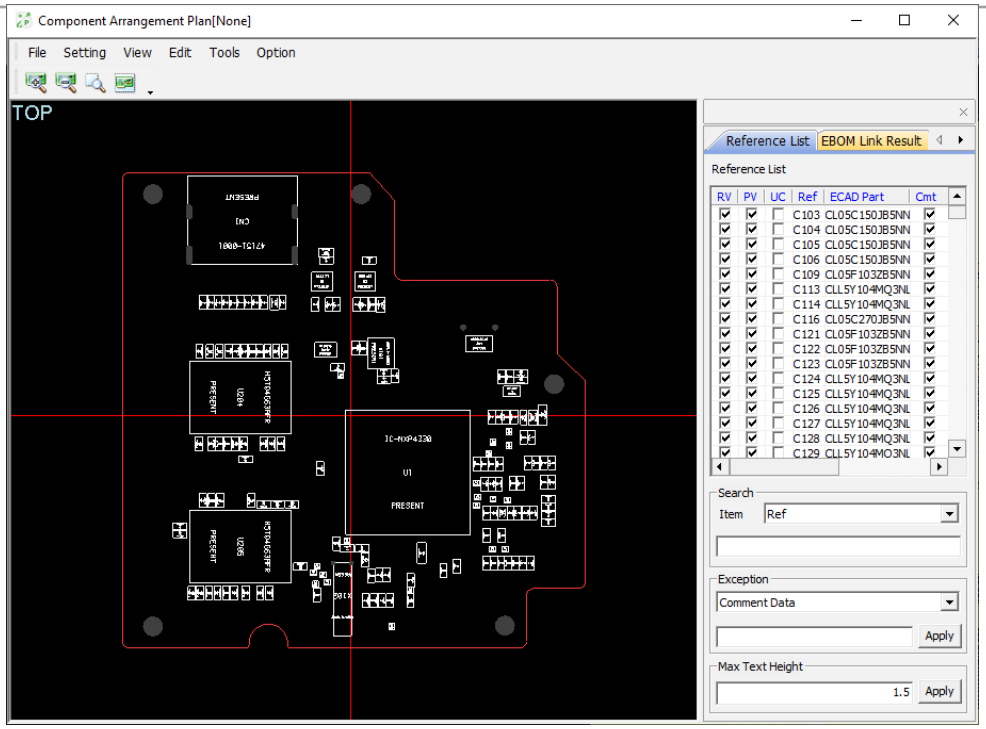




To make the text size for reference name and value, change the **Text Area Definition** from **Pad** to **COC** as below picture.

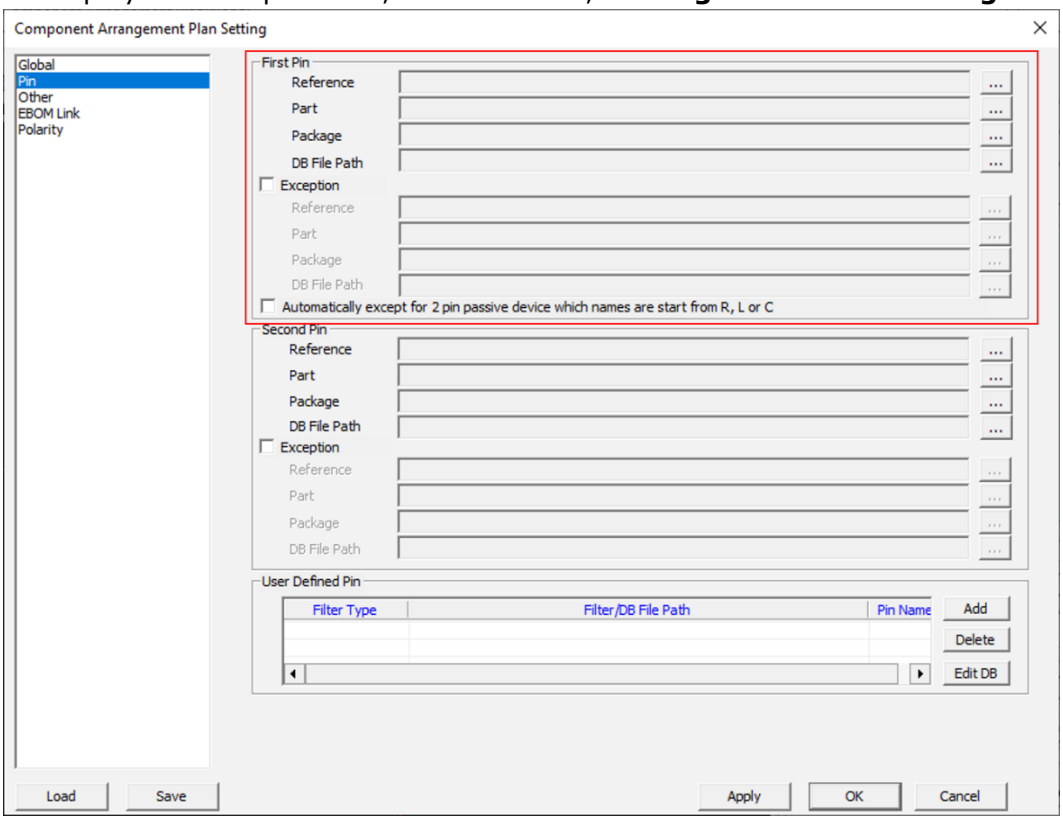


Text for reference name and value with COC based will be displayed with maximum size.

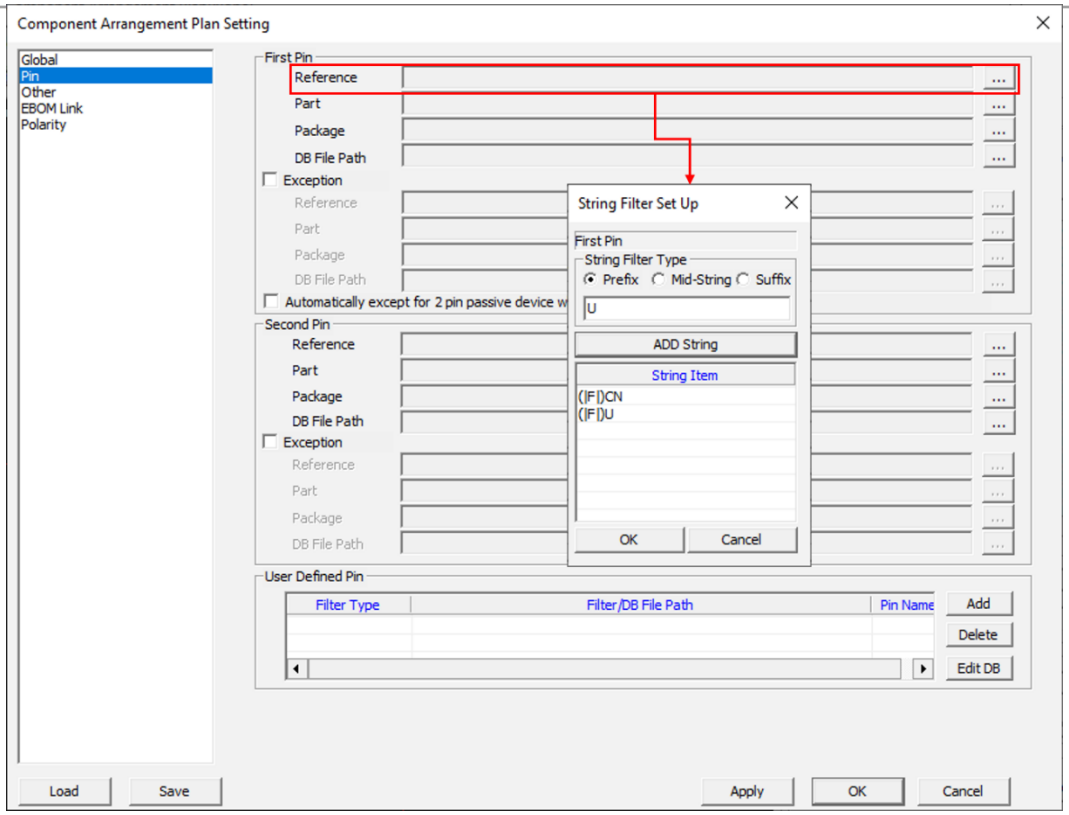


6.1.4. Showing 1(first) pin location

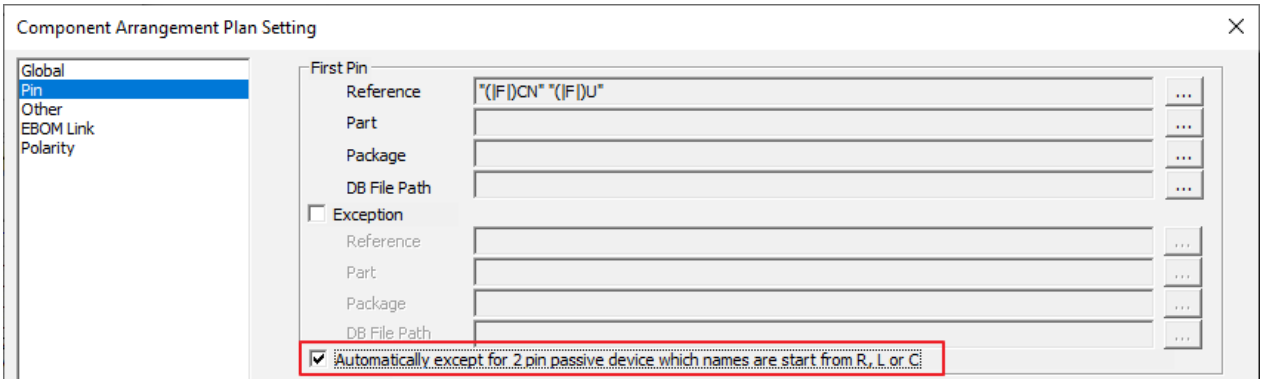
To display the first pin mark, use the menu, **Setting – CAP data setting** and select **Pin**.



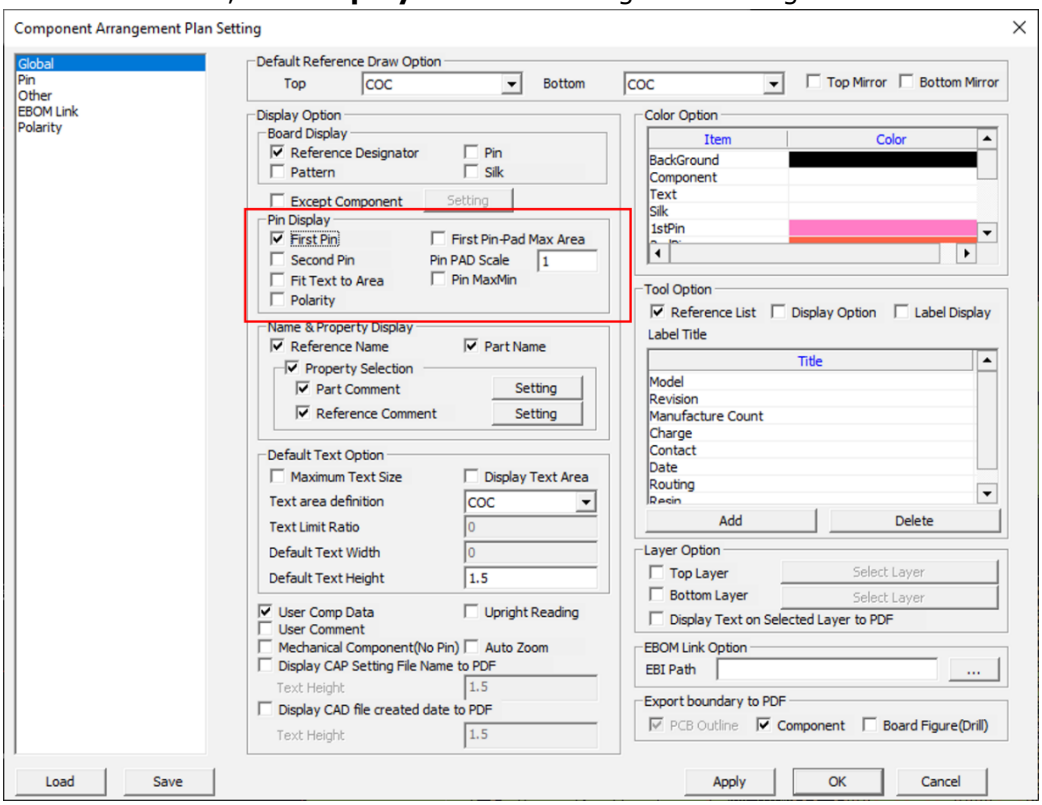
Press button, **Reference** and input 'U' and 'CN' for reference prefix recognition.
It means that references which names are starting with 'U' and 'CN' will be displayed 1st pin's location.



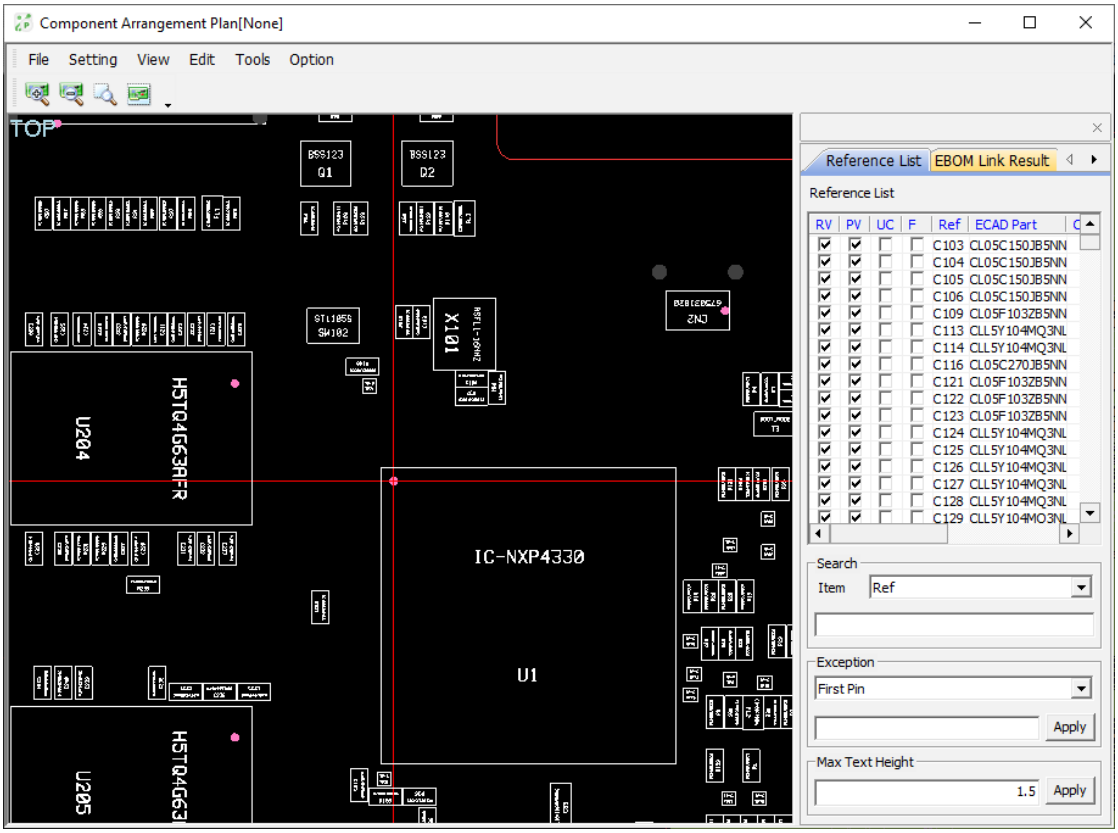
Check box menu, **Automatically except for 2 pin passive device which names are start from R, L or C** as the below picture.
It means 2 pin components which reference name starting with 'R, L, C' will not be display 1st pin's location.



Check box menu, **Pin Display – First Pin** in global setting window.



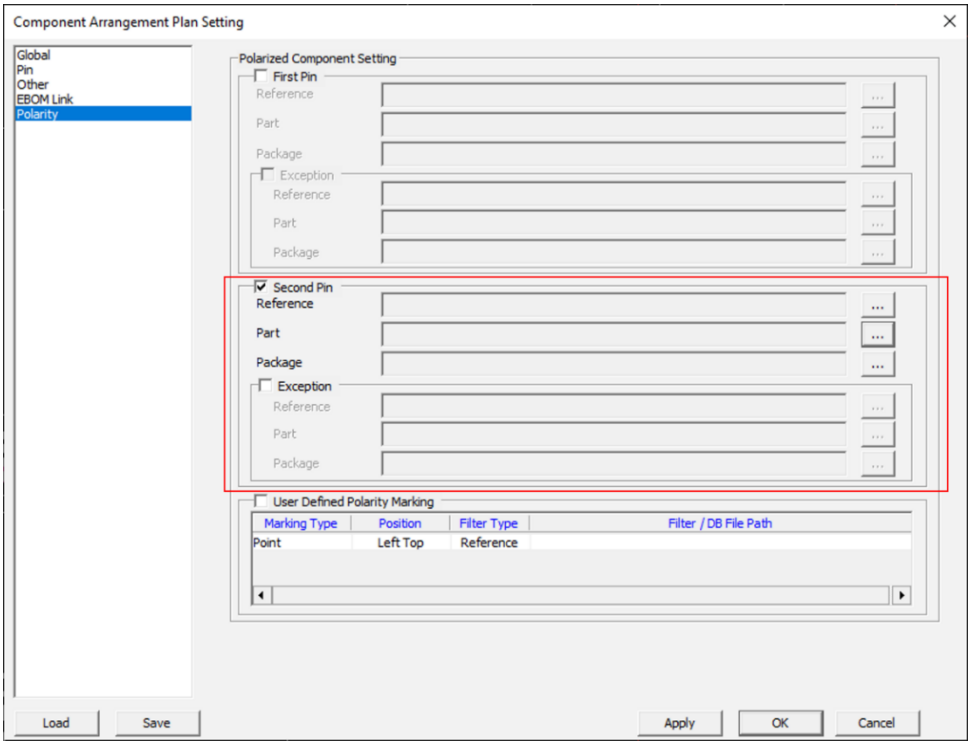
As the below picture, references which names are starting with 'U' and 'CN' will be shown with their 1st pin location.



6.1.5. Display component polarity

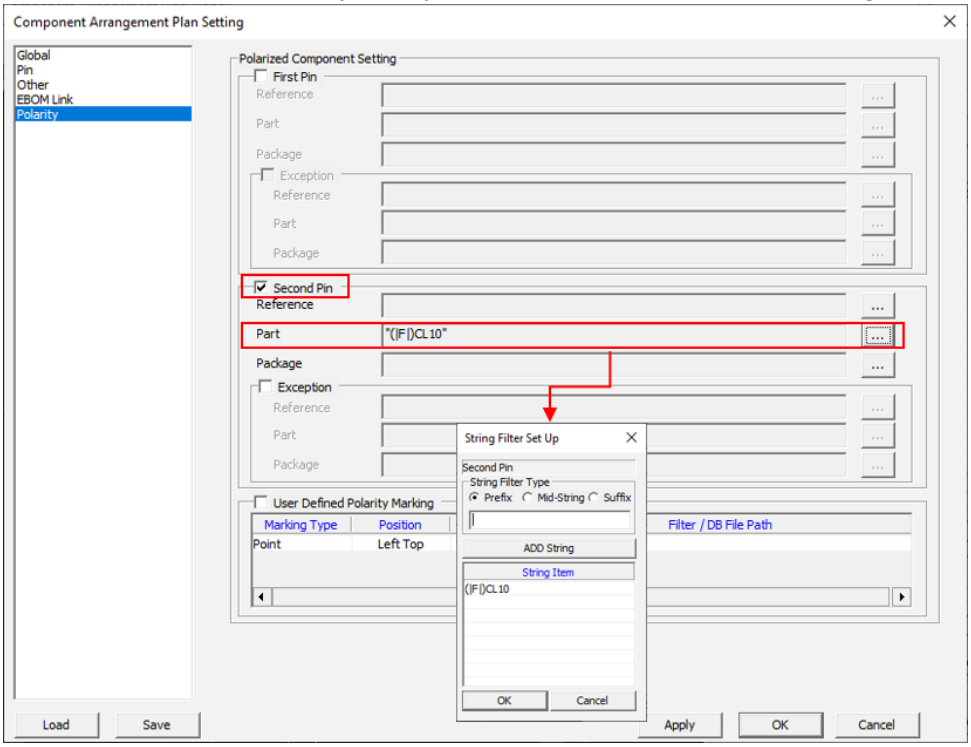
For component having polarity like capacitor, user can display their polarity.

Select **Polarity** in menu, **Setting – CAP Data Setting**.

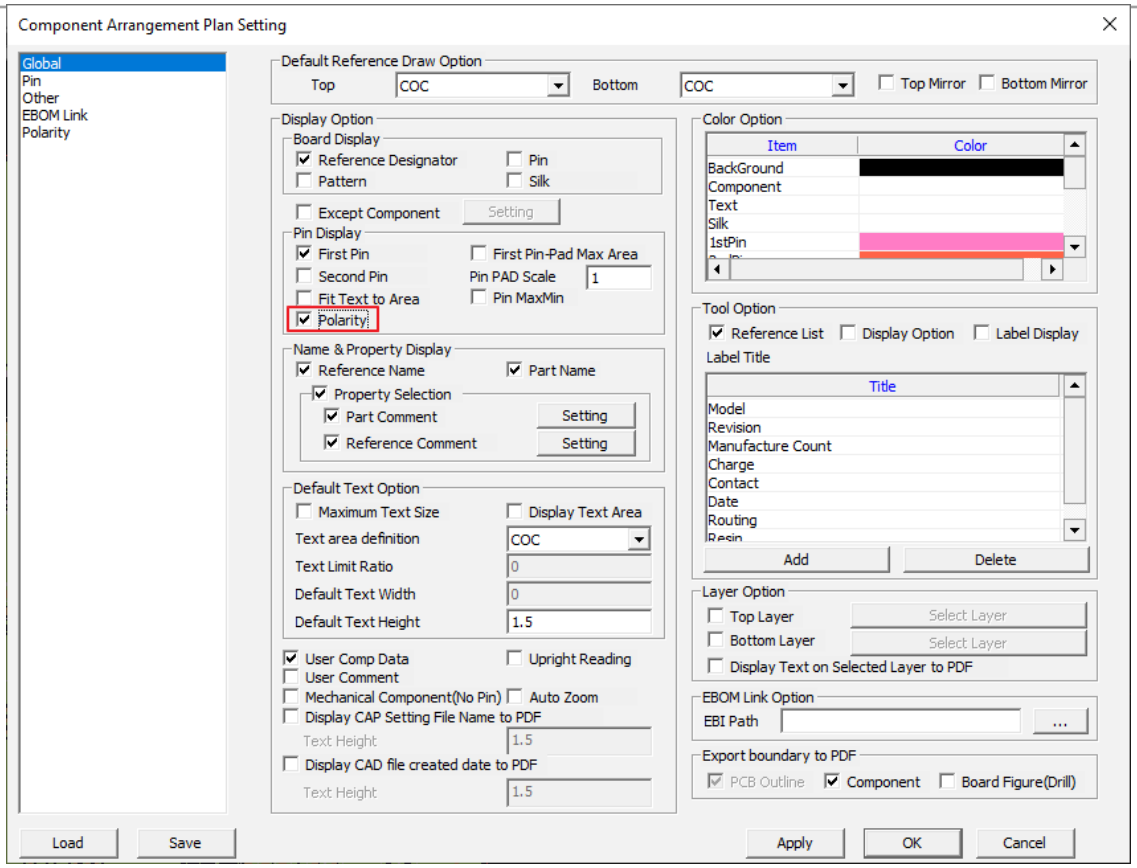


Using button in **Part** section on **Second Pin**, input 'CL10' for reference prefix.

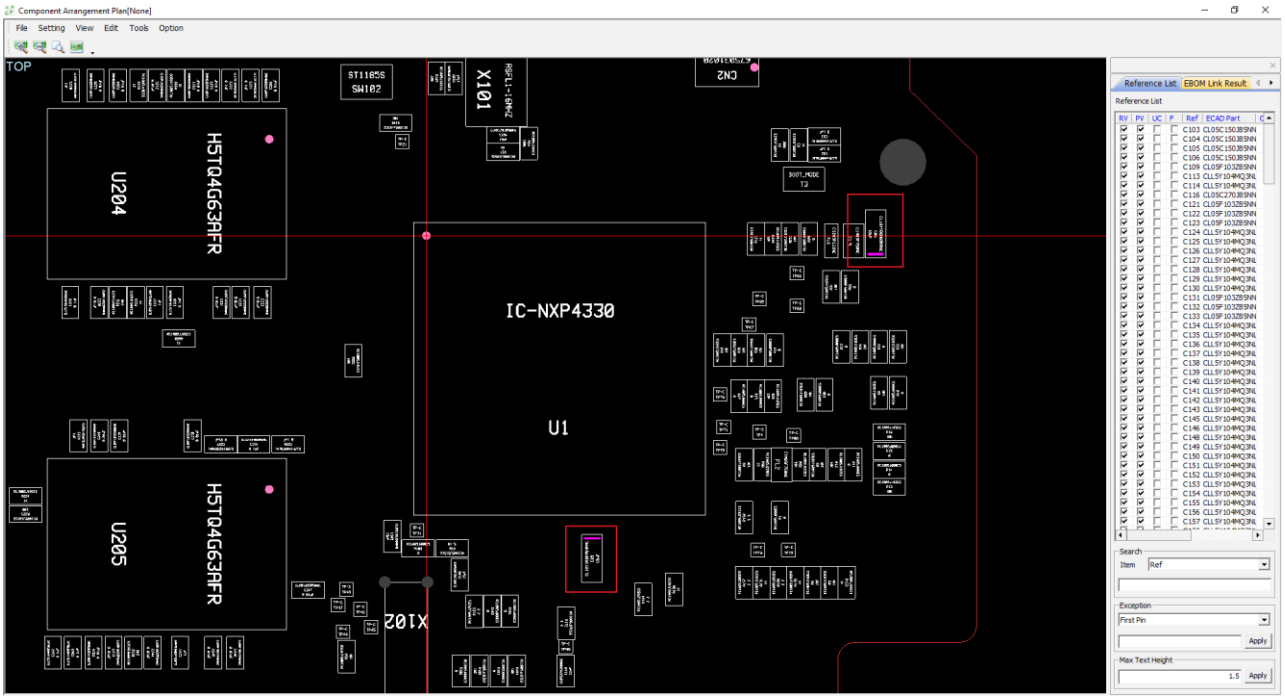
It is for selection electrolytic capacitors which names are starting with 'CL10'.



Check **Polarity** in **Pin Display** section.

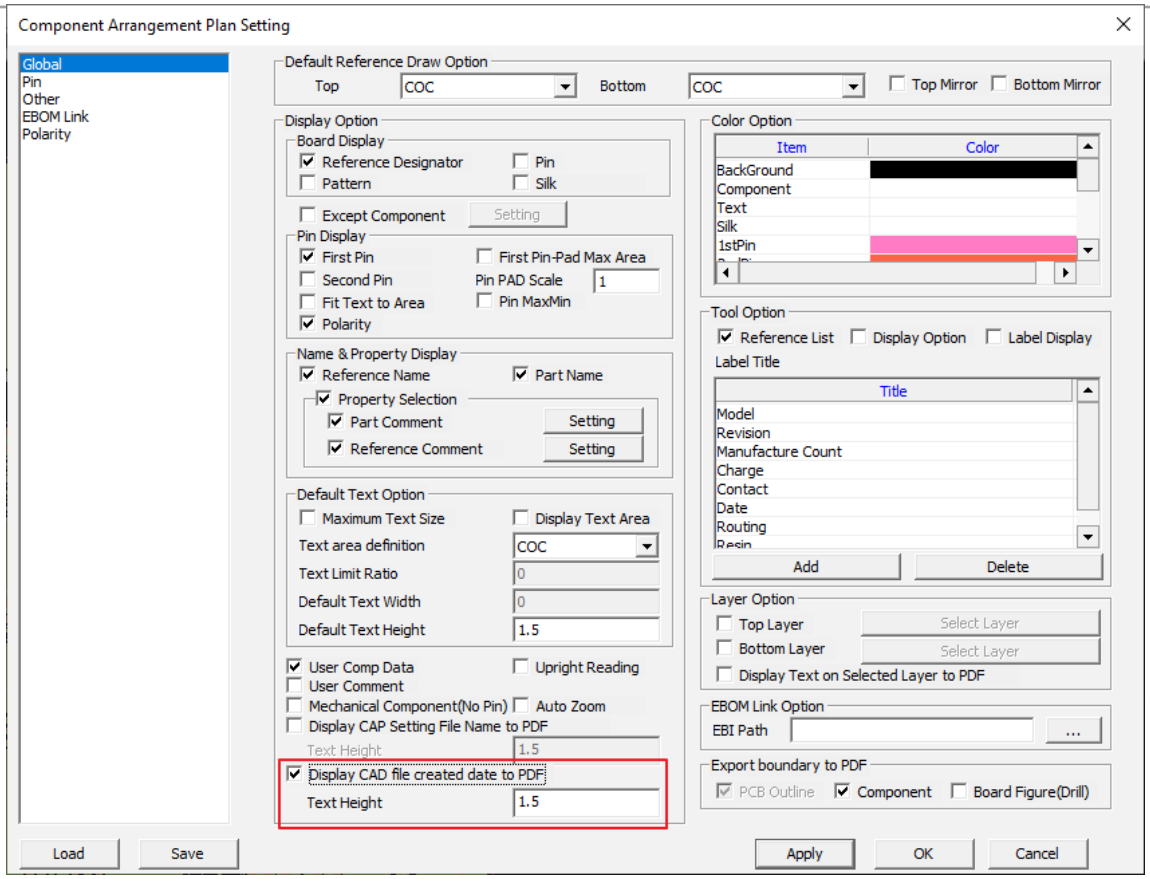


As the below picture, components which part names are starting with 'CL10' will be their polarity mark.

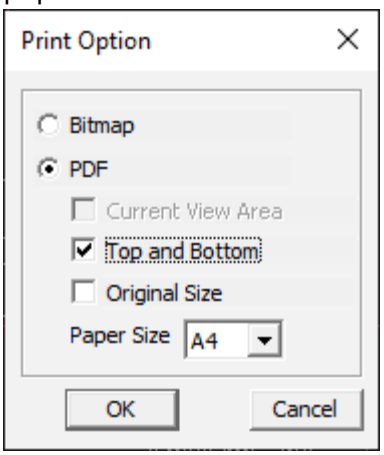


6.1.6. Output completed work-sheet

Check **Display CAD file created date to PDF** in global setting dialog window.
This is for showing design's created date.

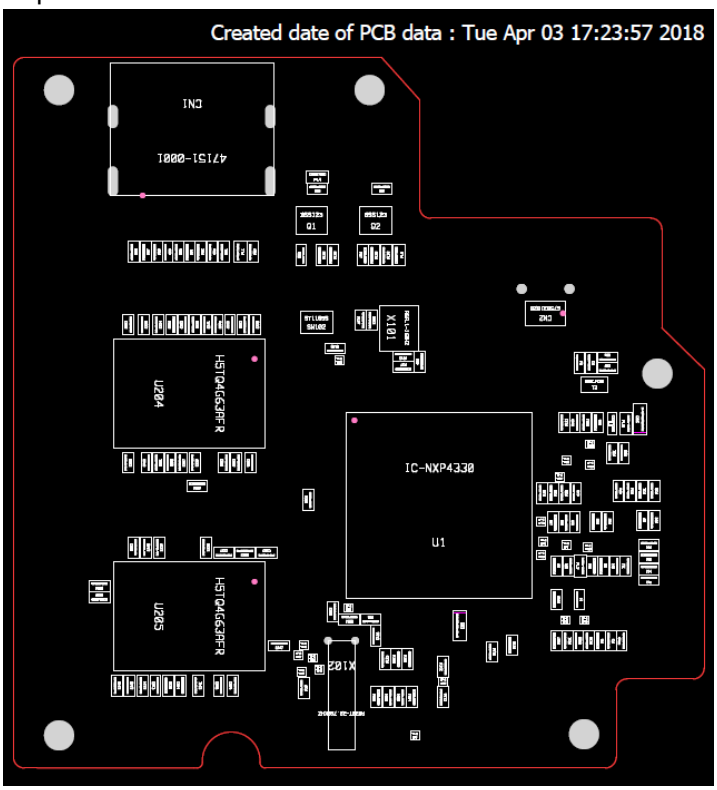


After selecting **File – Print** in **Print Option** and check layer for **Top and Bottom**. Also select paper size as 'A4' as the below picture.

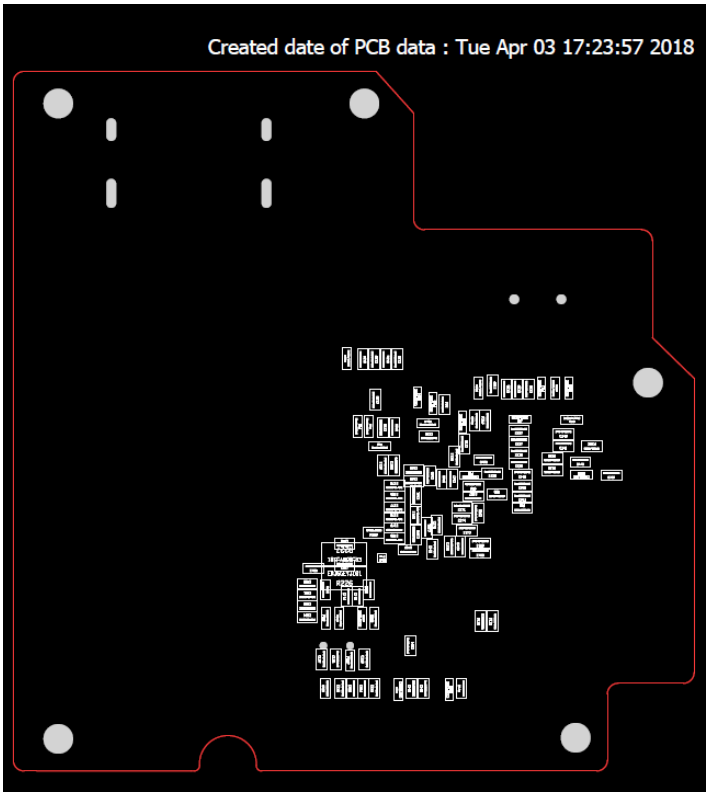


Press the menu button, **OK** and specify the file name to be saved.
Work-sheet for top and bottom will be outputted into PDF file

Top side work-sheet.



Bottom side work-sheet.



6.1.7. Setting file for work-sheet saving and loading

Select the menu, **File – Save** and input filename.

File's extension is `*.capb` and setting file's saved contents will be used for same design or other design. To re-use saved file, user can load saved setting file, `*.capb`.

Select the menu, **File – Load** (`*.capb`) to load saved setting file.

File's extension is `*.capb` and setting file(s) is(are) displayed for selecting.

7. Restricted PDBB

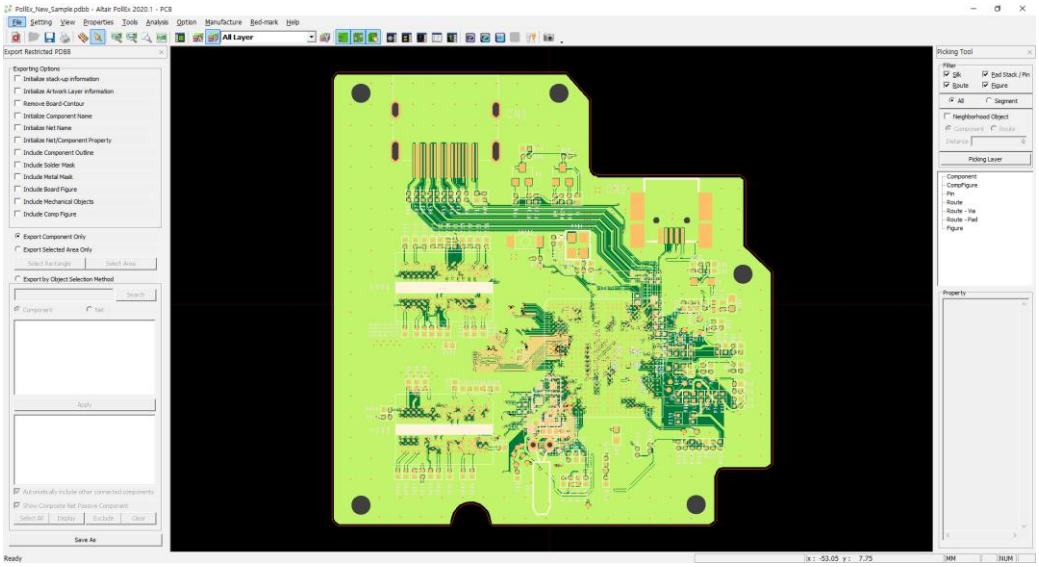
PollEx PCB is viewer but for some special purpose, user can change its design. Partially remove or hide some objects on design.

If user wants to share the PCB layout design with restricted objects or information such as certain components, nets etc. **Restricted PDBB** helps to save as edited PDBB.

7.1. Run Restricted PDBB

Select menu **File - Export To - Restricted PDBB** in PollEx PCB.

Export Restricted PDBB menu is shown on the left side of main window.

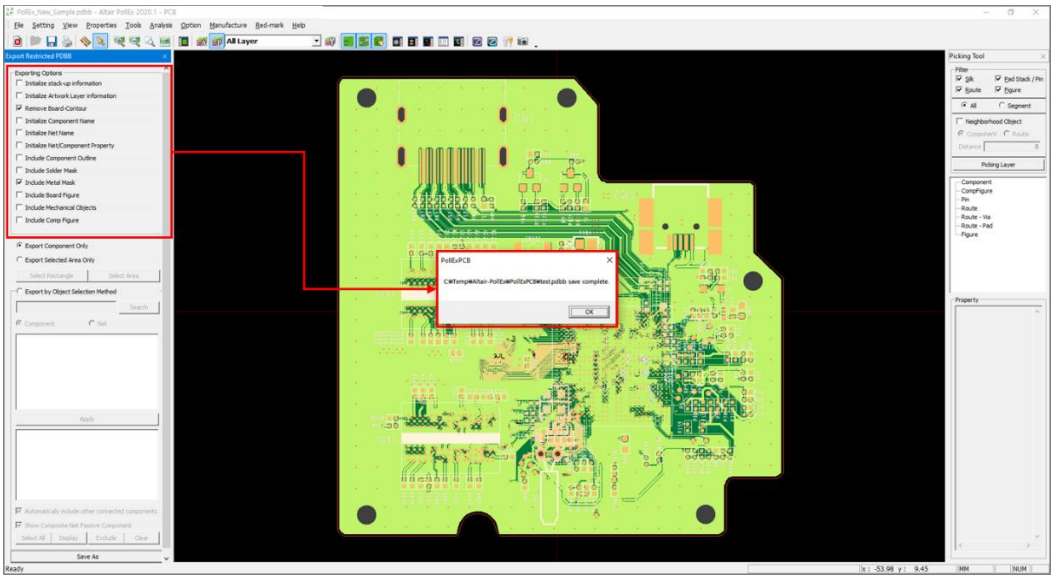


7.2. Save as PDBB file with giving conditions

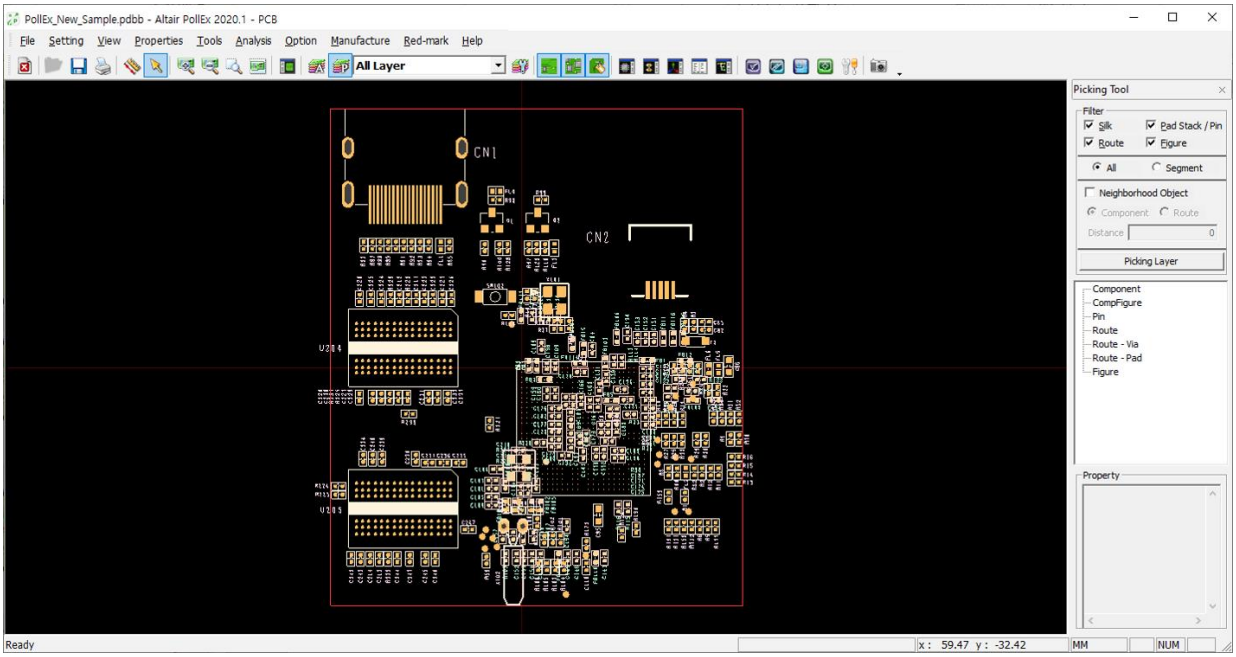
Select the following conditions in **Exporting Options** menu.

a. Remove Board-Contour
b. Include Metal Mask

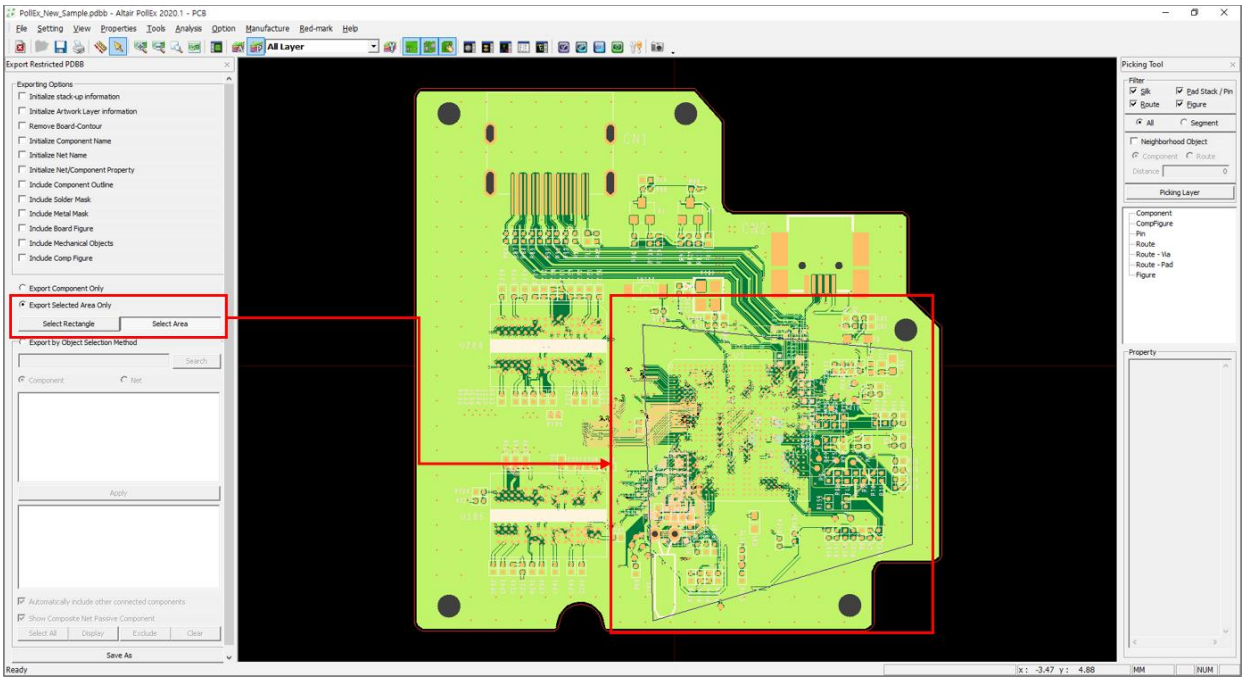
Save As to save new PDBB file.



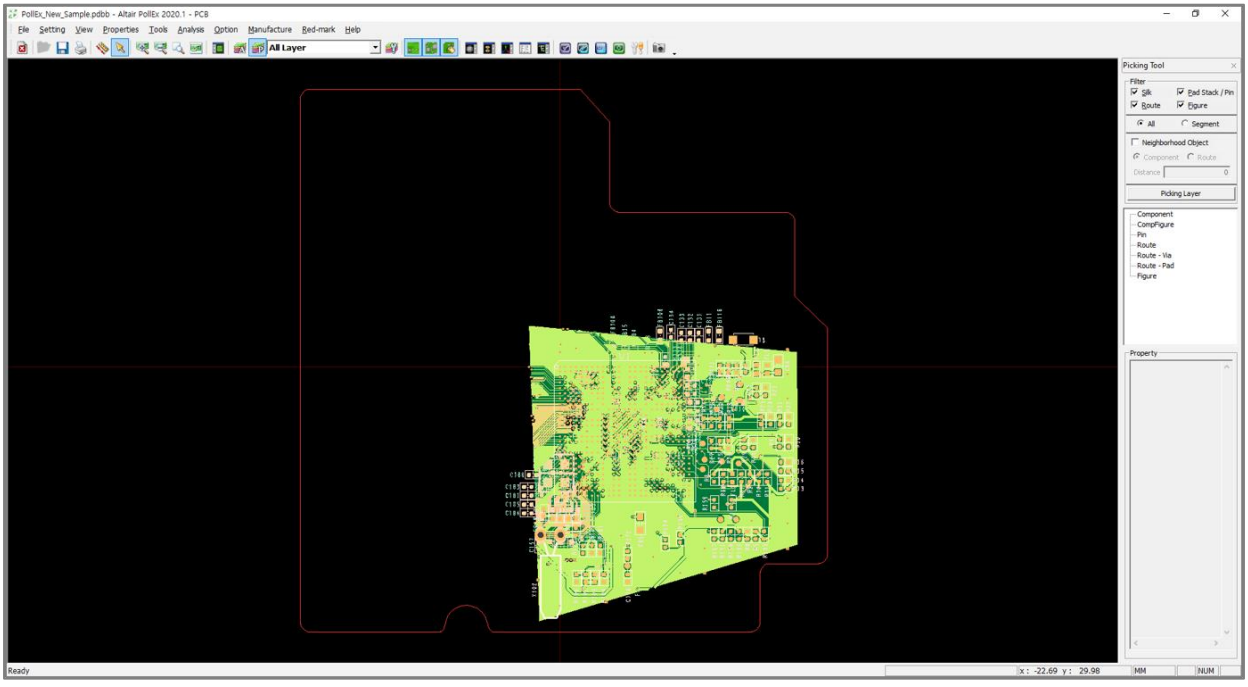
Exported Restricted PDBB file is saved with selected conditions.



Select **Export Selected Area Only** and then click ***Select Area*** button.
Select the area to save with mouse click and press ***Enter*** button.



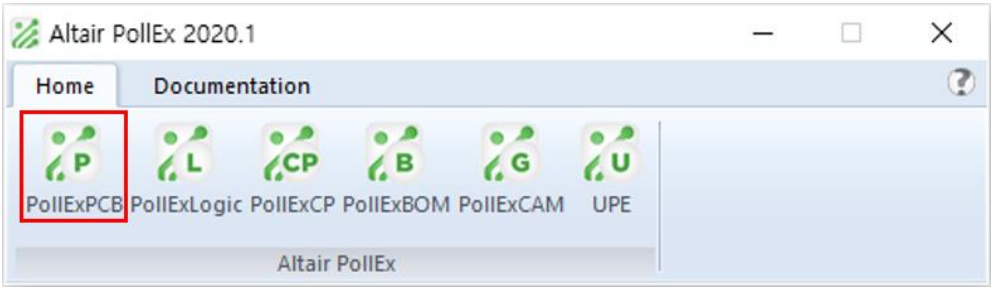
Click ***Save As*** button on the bottom side to save the design with user defined area.



PollEx Real PCB Assembly Viewer Tutorial

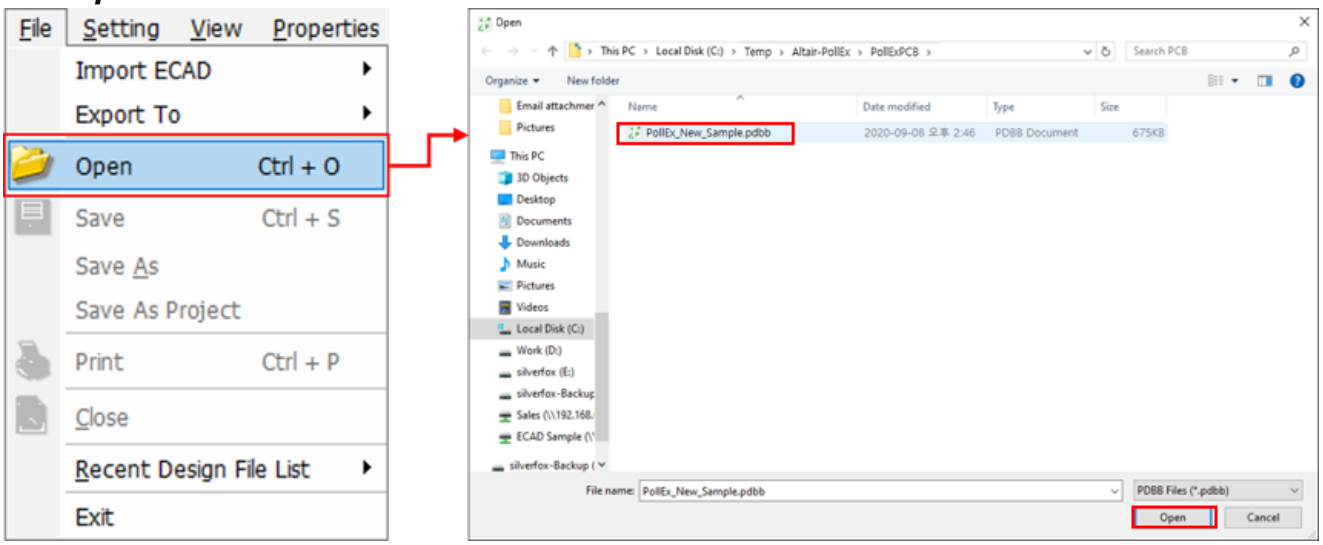
Using PollEx PCB's function, **Real PCB Assembly Viewer**, user can review design into 3D shapes and export its data into STEP format for linking to mechanical CAD system.

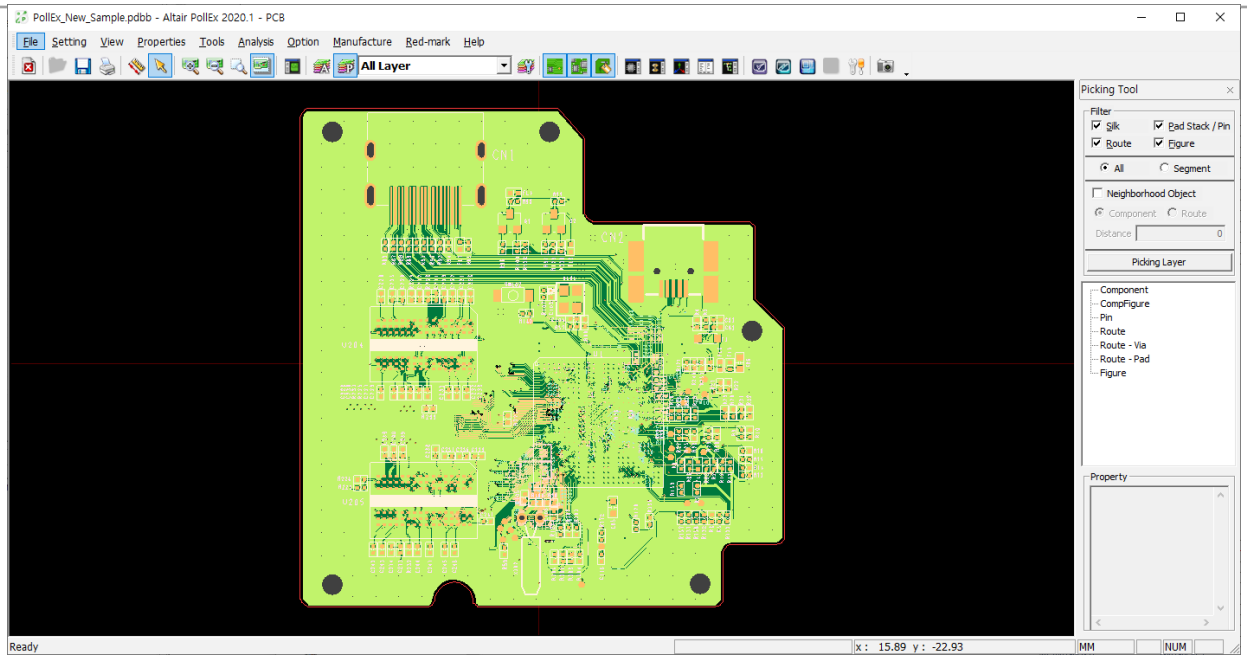
1. Launch PollEx PCB



2. Open PollEx PCB layout design file

Open the file, `C:\Temp\Altair-PollEx\PollExPCB\PollEx_New_Sample.pdbb` by using the main menu, ***File-Open***.



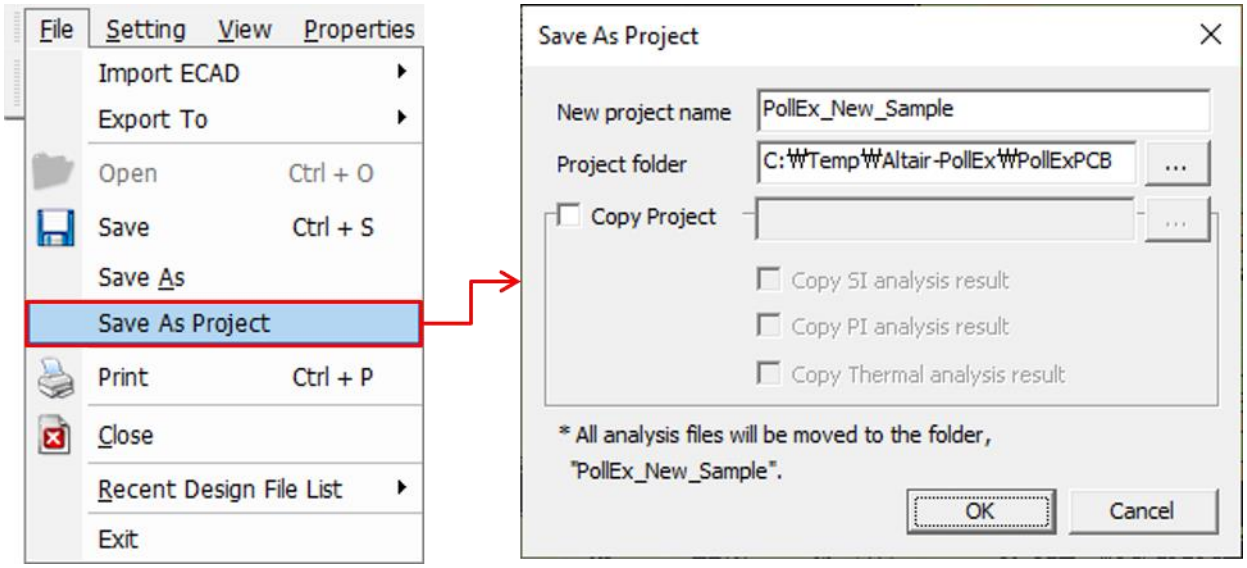


Note: Please refer to the PollEx PCB manual how to use the PollEx PCB viewer.

3. Save as Project

PollEx PCB operates on a design project database which contains entire data of a PCB design including the materials, parts, physical layout, analysis models and analysis result data. With the use of unified design project database, this application can be commonly used by multiple engineering disciplines.

File - Save as Project menu is used to create a new PollEx project directory from the scratch. Upon selecting the menu, the sample PCB design name is displayed as the **New project name** as shown below.



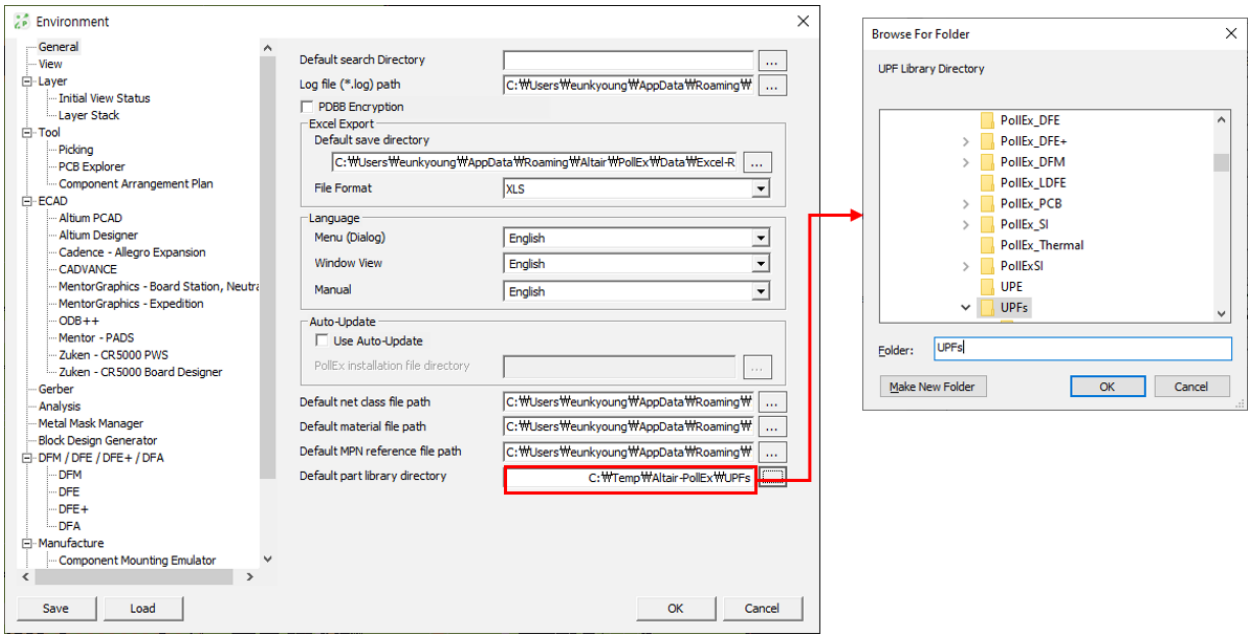
4. Link components

Select menu ***Setting – Environment***.

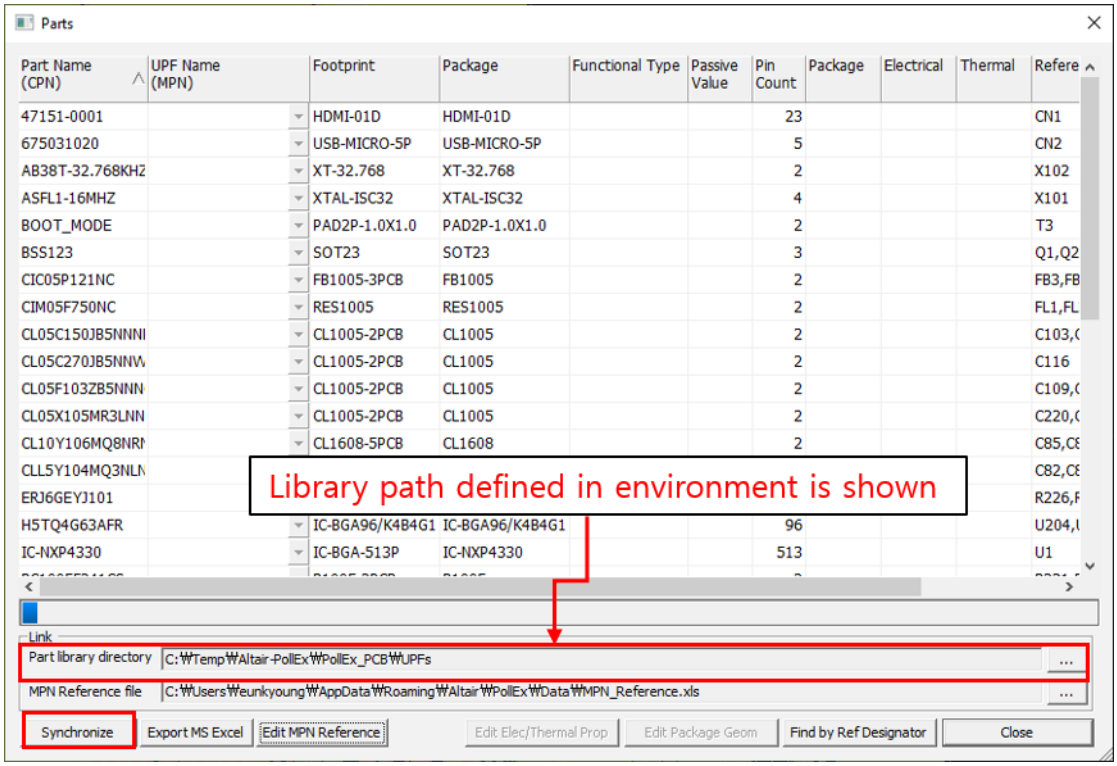
User can set the environment for importing 3D part libraries. Define the default 3D part librarian path in the **Default part library directory**.

Define the unified part library directory which is in following location.

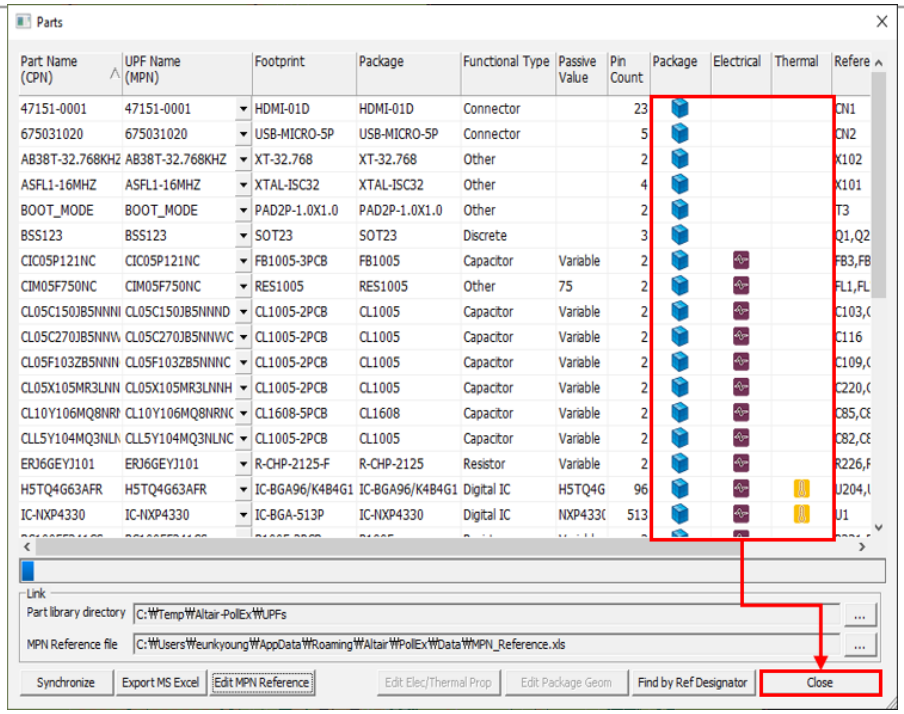
`C:\Temp\Altair-PollEx\PollExPCB\UPFs`



After selecting **Properties - Parts**, press the button menu ***Synchronize***.



Below picture show parts' linked status.



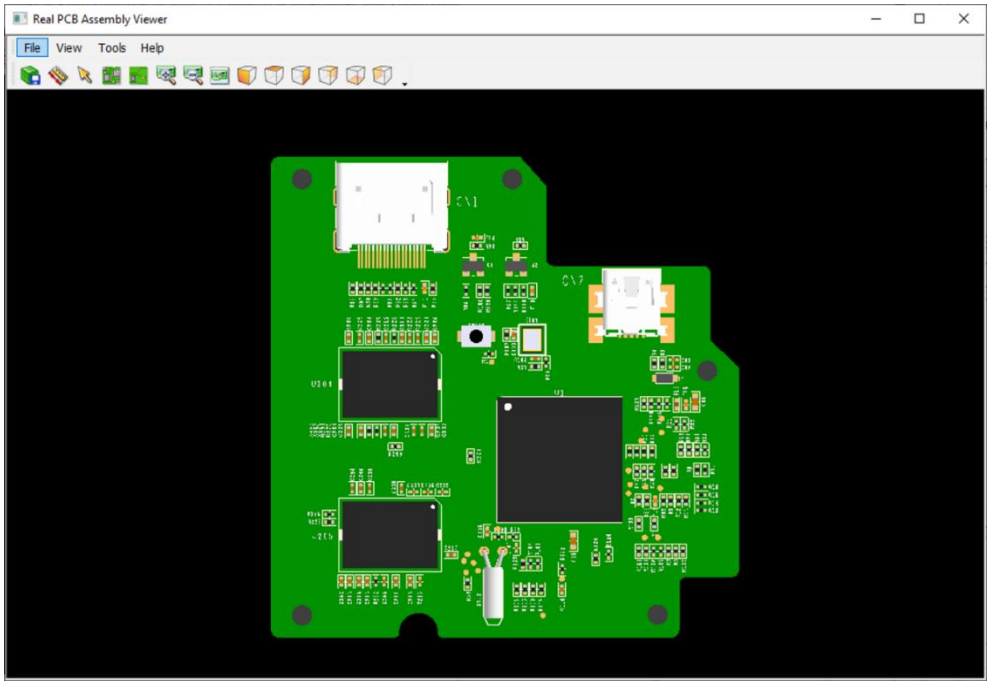
: Linked to 3D packages

: Linked to electrical buffer model

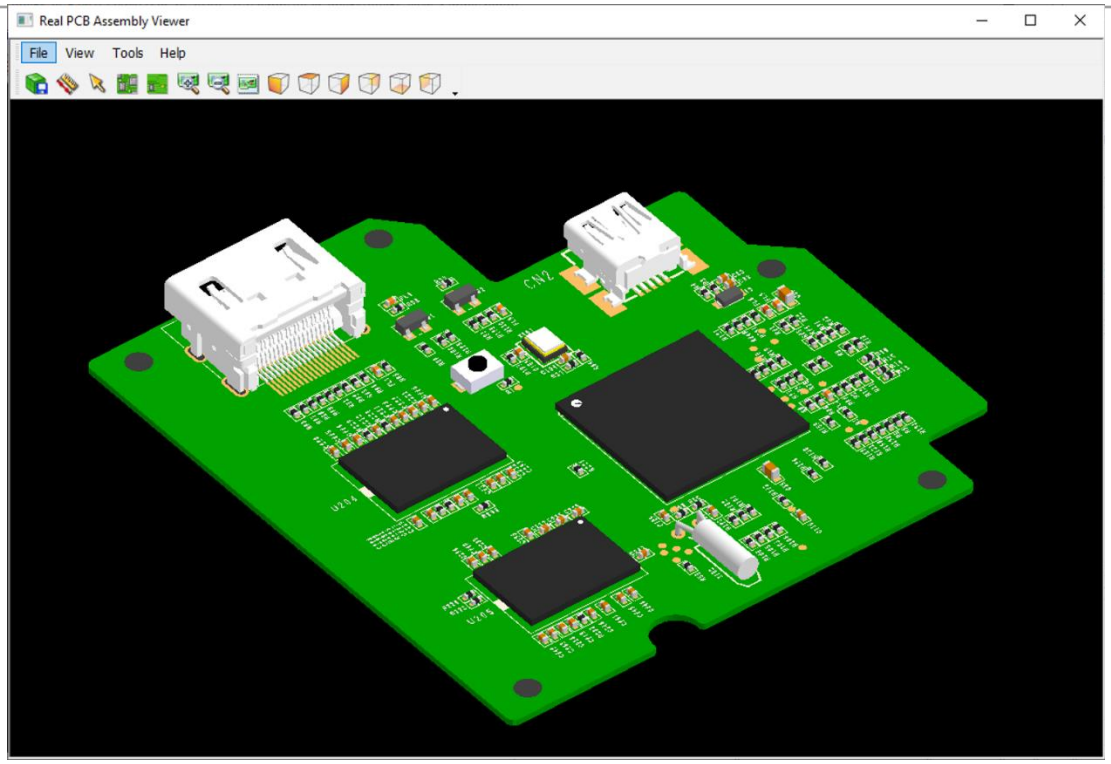
: Linked to package thermal libraries

5. Run Real PCB Assembly Viewer

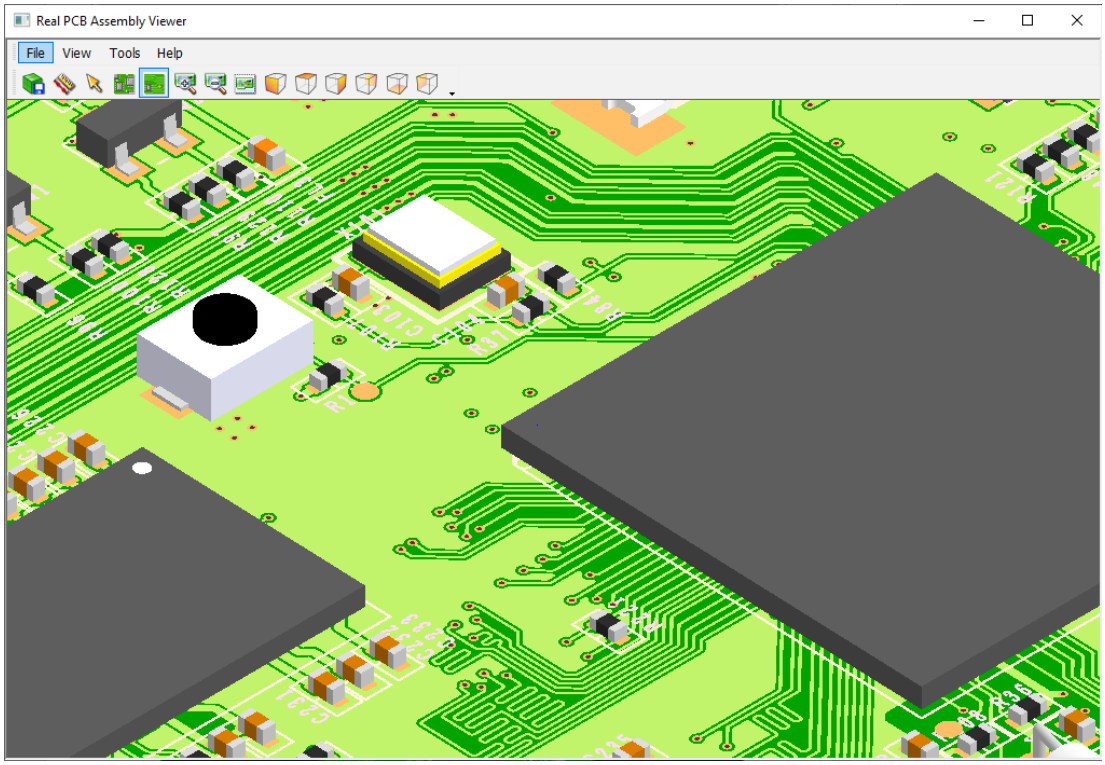
Execute the menu, Option – Real PCB Assembly Viewer.



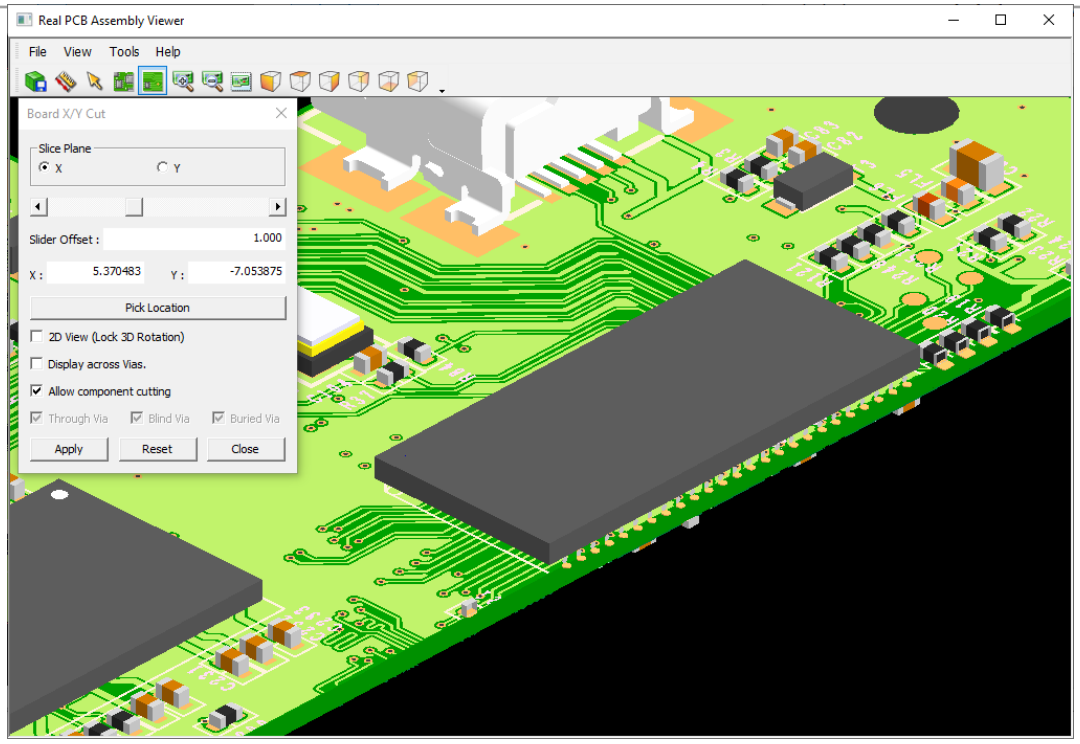
While pressing mouse left button, move the mouse cursor to various location or to rotate the design.



Select menu, **View - Route**.
User can see the design's routing status.

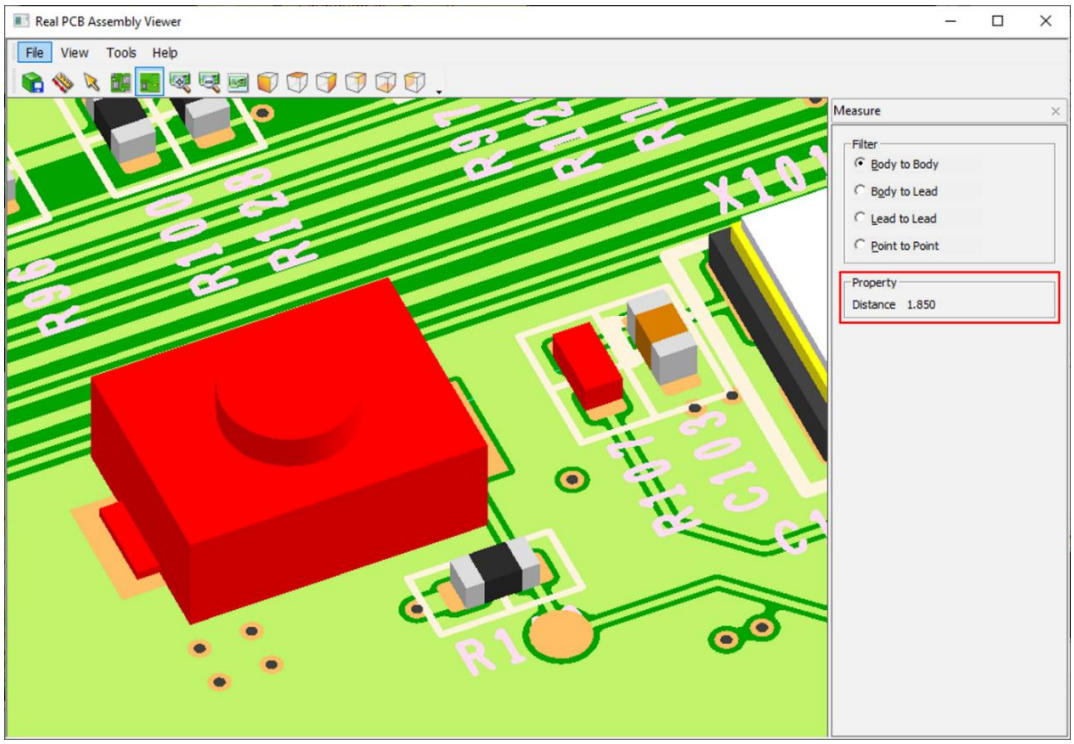


Use the menu, **Tools - Board X/Y Cut**.
User can see the cutting edge of design along X or Y axis.



6. Measuring between components

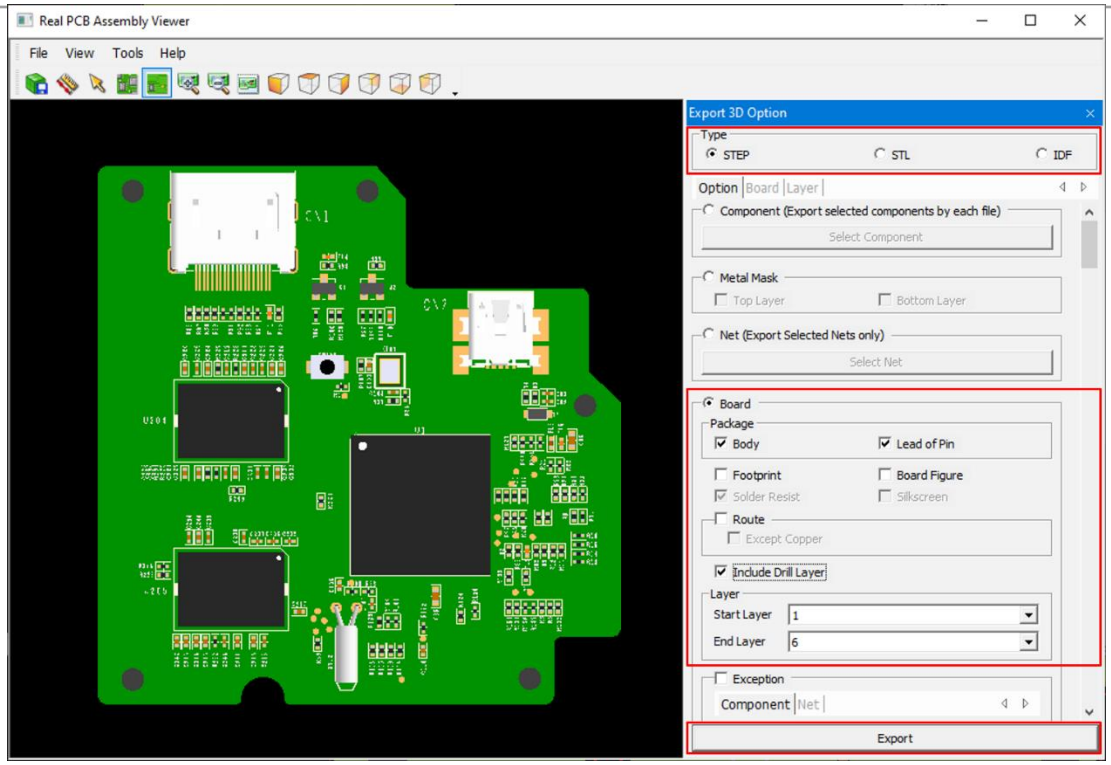
Use the menu, **Tool - Measure** or icon  to measure distance between components.



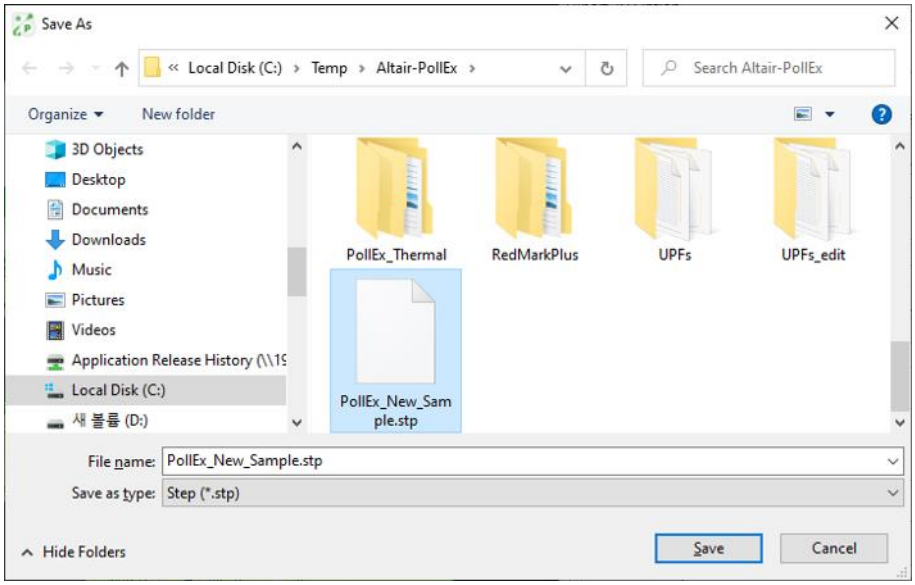
7. Export mechanical data format

Select the menu, **File – Export to 3D Data**.

PollEx PCB can export its design into mechanical standard format, STEP, STL or IDF.



Select the button, **Export** and specify saving file path.



At mechanical CAD system user can read exported STEP file.