



# Altair PollEx 2021

## Gerber To PCB User Guide

# Intellectual Property Rights Notice

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

Licenses Altair Engineering Inc. Copyright © 1986-2021. All Rights Reserved.

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 Inspire Extrude Metal** ©1996-2021 (formerly Click2Extrude-Metal) **Altair**

**Inspire Extrude Polymer** ©1996-2021 (formerly Click2Extrude-Polymer) **Altair**

**Inspire Form** ©1998-2021 (formerly Click2Form)

**Altair Inspire Friction Stir Welding** ©1996-2021

**Altair Inspire Mold** ©2009-2021

**Altair Inspire Play** ©2009-2021

**Altair Inspire PolyFoam** ©2009-2021

**Altair Inspire Render** ©1993-2016 Solid Iris Technologies Software Development One PLLC,  
©2016-2021 Altair Engineering Inc (formerly Thea Studio)

**Altair Inspire Resin Transfer Molding** ©1990-2021

**Altair Inspire Studio** ©1993-2021 (formerly 'Evolve')

**Altair Manufacturing Solver™** ©2011-2021

**Altair Material Data Center** ©2019-2021

**Altair MotionSolve™** ©2002-2021

**Altair MotionView™** ©1993-2021

**Altair Multiscale Designer™** ©2011-2021

**Altair nanoFluidX™** ©2013-2018 Fluidyna GmbH, ©2018-2021 Altair Engineering Inc.

**Altair newFASANT** ©2010-2021

**Altair OptiStruct™** ©1996-2021

**Altair PolIEx** ©2003-2021 **Altair**

**Radioss™** ©1986-2021

**Altair Seam™** ©1985-2019 Cambridge Collaborative, Inc., ©2019-2021 Altair Engineering Inc.

**Altair SimLab™** ©2004-2021

**Altair SimSolid™** ©2015-2021

**Altair ultraFluidX™** ©2010-2018 Fluidyna GmbH, ©2018-2021 Altair Engineering Inc.

**Altair Virtual Wind Tunnel™** ©2012-2021

**Altair WinProp™** ©2000-2021

**Altair WRAP** ©1998-2021 WRAP International AB, ©2021 Altair Engineering AB

## **Altair Packaged Solution Offerings (PSOs)**

**Altair Automated Reporting Director™** ©2008-2021

**Altair GeoMechanics Director™** ©2011-2021

**Altair Impact Simulation Director™** ©2010-2021

**Altair Model Mesher Director™** ©2010-2021

**Altair NVH Director™** ©2010-2021

**Altair Squeak and Rattle Director™** ©2012-2021

**Altair Virtual Gauge Director™** ©2012-2021

**Altair Weight Analytics™** ©2013-2021

**Altair Weld Certification Director™** ©2014-2021

**Altair Multi-Disciplinary Optimization Director™** ©2012-2021

## **Altair PBSWorks™ - Accelerating Innovation in the Cloud™**

**Altair PBS Professional®** ©1994-2021

**Altair Control™** ©2008-2021; (formerly **PBS Control**)

**Altair Access™** ©2008-2021; (formerly **PBS Access**)

**Altair Accelerator™** ©1995-2021; (formerly **NetworkComputer**)

**Altair Accelerator™ Plus** ©1995-2021; (formerly **WorkloadXelerator**)

**Altair FlowTracer™** ©1995-2021; (formerly **FlowTracer**)

**Altair Allocator™** ©1995-2021; (formerly **LicenseAllocator**)

**Altair Monitor™** ©1995-2021; (formerly **LicenseMonitor**)

**Altair Hero™** ©1995-2021; (formerly **HERO**)

**Altair Software Asset Optimization (SAO)™** ©2007-2021



### **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™**

**Altair Knowledge Studio®** ©1994-2021 Angoss Software Corporation, ©2021 Altair Engineering Inc.

**Altair Knowledge Studio for Apache Spark** ©1994-2021 Angoss Software Corporation, ©2021 Altair Engineering Inc.

**Altair Knowledge Seeker™** ©1994-2021 Angoss Software Corporation, ©2021 Altair Engineering Inc.

**Altair Knowledge Hub™** ©2017-2021 Datawatch Corporation, ©2021 Altair Engineering Inc.

**Altair Monarch™** ©1996-2021 Datawatch Corporation, ©2021 Altair Engineering Inc. **Altair**

**Monarch Server** ©1996-2021 Datawatch Corporation, ©2021 Altair Engineering Inc. **Altair**

**Panopticon™** ©2004-2021 Datawatch Corporation, ©2021 Altair Engineering Inc.

### **Altair SmartWorks™**

**Altair SmartCore™** ©2011-2021 Altair Engineering Inc.

**Altair SmartEdge™** ©2011-2021 Altair Engineering Inc.

**Altair SmartSight™** ©2011-2021 Altair Engineering Inc.

**Altair One™** ©1994-2021

Altair intellectual property rights are protected under U.S. and international laws and treaties. Additionally, Altair software may be protected by patents or other intellectual property rights. All other marks are the property of their respective owners.

ALTAIR ENGINEERING INC. Proprietary and Confidential. Contains Trade Secret Information.

Not for use or disclosure outside of Altair and its licensed clients. Information contained in Altair software shall not be decompiled, disassembled, "unlocked", reverse translated, reverse engineered, or publicly displayed or publicly performed in any manner. Usage of the software is only as explicitly permitted in the end user software license agreement. Copyright notice does not imply publication.

### **Third party software licenses**

AcuConsole contains material licensed from Intelligent Light ([www.ilight.com](http://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/](http://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	<a href="mailto:anzsupport@altair.com">anzsupport@altair.com</a>
Brazil	+55 113 884 0414	<a href="mailto:br_support@altair.com">br_support@altair.com</a>
Canada	+1 416 447 6463	<a href="mailto:support@altairengineering.ca">support@altairengineering.ca</a>
China	+86 400 619 6186	<a href="mailto:support@altair.com.cn">support@altair.com.cn</a>
France	+33 141 33 0992	<a href="mailto:francesupport@altair.com">francesupport@altair.com</a>
Germany	+49 703 162 0822	<a href="mailto:hwsupport@altair.de">hwsupport@altair.de</a>
Greece	+30 231 047 3311	<a href="mailto:eesupport@altair.com">eesupport@altair.com</a>

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

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](http://www.altair.com) for complete information on Altair, our team, and our products.

# Contents

<b>Intellectual Property Rights Notice .....</b>	<b>ii</b>
<b>Technical Support .....</b>	<b>vi</b>
<b>Conventions Used in this Guide .....</b>	<b>10</b>
<b>Gerber to PCB .....</b>	<b>11</b>
<b>ODB++ .....</b>	<b>11</b>
1. Launch ODB++ .....	11
2. ODB++ GUI .....	12
2.1. Save and Load for the environment setup.....	12
3. Layer Setting .....	13
3.1. Unit setup.....	13
3.2. Specify JIG Data.....	13
3.3. Component Layers.....	14
3.4. Layer Stack-up.....	14
3.5. Drill Layers .....	15
3.6. Board Layers.....	15
4. Make Component .....	16
4.1. Pin Name Order.....	16
4.2. Auto .....	17
4.3. Manual .....	17
4.4. Control by JIG Data .....	17
4.5. Display Layer Control .....	17
4.6. Reference List .....	17
5. Make Netlist .....	18
5.1. Verification by JIG Data.....	18
5.2. Write Nominal Resistance for by JIG Data .....	19
5.3. Signal Net Rule .....	19
5.4. Ground Net Rule.....	19
5.5. Power Net Rule.....	19
5.6. Dummy Net Rule .....	19
5.7. Remove 0 pin assigned nets.....	19
5.8. Maximum Number of Pins Connected to Net When Calculating Nominal Resistance .....	19
5.9. Generate Netlist .....	19
5.10. Exclude Mode .....	20
6. Result Information.....	20
6.1. Export to MS Excel.....	21
<b>Gerber + IPC-D-356 .....</b>	<b>22</b>
1. Launch Gerber + IPC-D-356 .....	22



---

2. Gerber + IPC-D-356 GUI.....	23
2.1. Save and Load for the environment setup.....	23
3. Layer Setting .....	24
3.1. Component Layers setup .....	24
3.2. Layer Stack-up setup .....	25
3.3. Drill Layers setup.....	25
4. IPC-D-356 .....	25
4.1. Load IPC-D-356 data .....	25
4.2. Generate Component .....	25
4.3. Match Location with Gerber.....	26
4.4. Generate Net Data .....	26
5. Result Information.....	27

## Conventions Used in this Guide

This guide uses the following conventions:

**Bold** All commands from the user interface. Options, menus, buttons, and dialog box names are bolded, but not italicized.

**Italic**

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](mailto:PollEx_support_kr@altair.com).

## Gerber to PCB

Gerber to PCB is a PCB generator from graphical Gerber or ODB++ file which is non-intelligent data. By importing graphical Gerber or ODB++ file, users can generate it into intelligent PCB design file through the Gerber to PCB.

Gerber to PCB has two features of **ODB++** and **Gerber + IPC-D-356**.


## ODB++

The **ODB++** converts a limited ODB++ design file that has only graphical data into PCB.

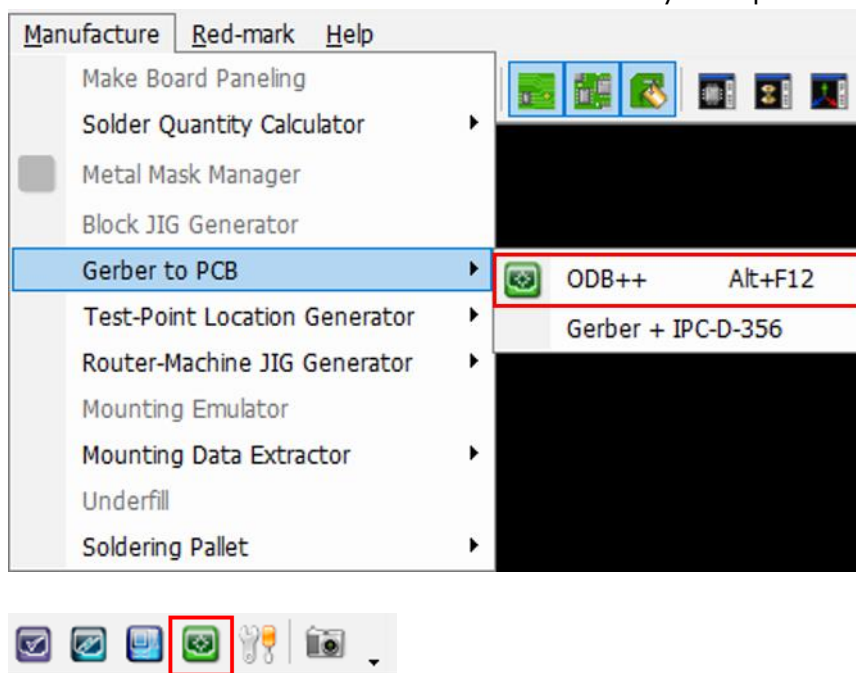
After converting as the intelligent PCB design, it can be used for the Bare Board Test (BBT), which checks electrical liabilities on the PCB.

### 1. Launch ODB++

**Gerber to PCB** is the optional module of PolIEx PCB. After launching the PolIEx PCB, Select the menu,

**Manufacture – Gerber to PCB – ODB++** or click the icon  from the menu bar.

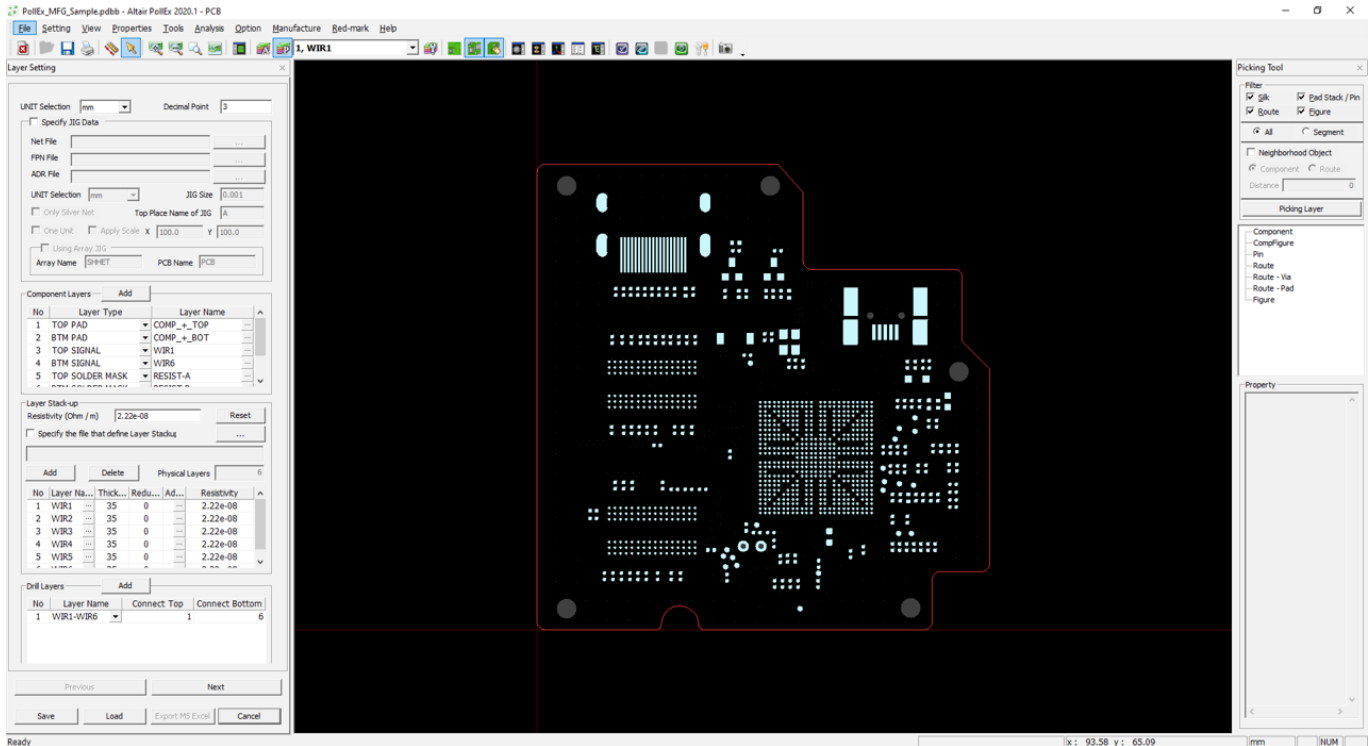
Please note that the menu will not be enabled if you import ODB++ data which includes net data.



## 2. ODB++ GUI

To make the intelligent PCB design data, users will have four different processes, **Layer Setting**, **Make Component**, **Make Netlist**, and **Result Information**. These dialogs are displayed on the left-side of the main window. By clicking **Previous** or **Next** button, users can move to different dialog.

On the right-side of the main window, the **Picking Tool** dialog is displayed. For more detailed information, please refer to the **PolliEx PCB** manual.



### 2.1. Save and Load for the environment setup

On the bottom side of each dialog, users can save or load the environment setup.



- a. **Save:** The current user setting environment can be saved as \*.pgt file.
- b. **Load:** Users can load the pre-defined setting environment.
- c. **Export MS Excel:** After generating PCB design, users can see the result in **Result Information** tab. Users can export the result to MS Excel by using this button.

### 3. Layer Setting

To generate a PCB design file using ODB++ data, it is required to properly load the necessary files for the layers. When the required data are imported, users need to match the data to create the PCB design file.

The screenshot shows the 'Layer Setting' dialog box with several sections highlighted by red boxes and numbered annotations:

- 3.1** and **3.2** point to the top section containing 'UNIT Selection' (set to mm) and 'Decimal Point' (set to 3).
- 3.2** also points to the 'Specify JIG Data' section, which includes fields for 'Net File', 'FPN File', 'ADR File', 'UNIT Selection' (mm), 'JIG Size' (0.001), checkboxes for 'Only Silver Net', 'One Unit', 'Apply Scale' (X: 100.0, Y: 100.0), 'Using Array JIG', and text boxes for 'Array Name' (SHHET) and 'PCB Name' (PCB).
- 3.3** points to the 'Component Layers' section, which contains a table with columns 'No', 'Layer Type', and 'Layer Name'. The table lists layers 4 through 8: BTM SIGNAL (WIR6), TOP SOLDER MASK (RESIST-A), BTM SOLDER MASK (RESIST-B), TOP METAL MASK (METALMASK-A), and BTM METAL MASK (METALMASK-B).
- 3.4** points to the 'Layer Stack-up' section, which includes 'Resistivity (Ohm / m)' (2.22e-08), a 'Reset' button, a checkbox for 'Specify the file that define Layer Stackup', and a table for 'Physical Layers' (6 layers) with columns 'No', 'Layer Name', 'Thick...', 'Redu...', 'Ad...', and 'Resistivity'.
- 3.5** points to the 'Drill Layers' section, which contains a table with columns 'No', 'Layer Name', 'Connect Top', and 'Connect Bottom'. It shows layer 1 (WIR1-WIR6) with 'Connect Top' set to 1 and 'Connect Bottom' set to 6.
- 3.6** points to the 'Board Layers' section, which contains a table with columns 'No', 'Layer Type', 'Layer Name', and 'Target Layer'.

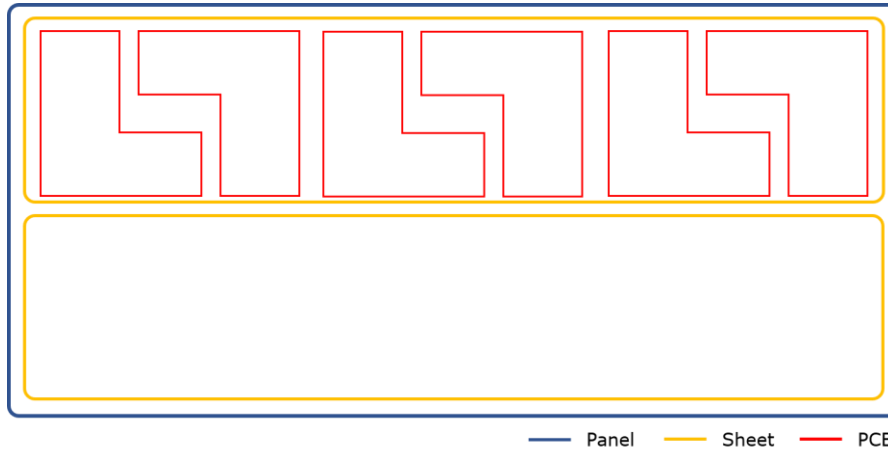
#### 3.1. Unit setup

- **UNIT Selection:** Select a unit.
- **Decimal Point:** Set the decimal point for the coordinates of part and net generation.

#### 3.2. Specify JIG Data

- **Net/ FPN/ ADR File:** These files are exported from the Bare Board Tester (BBT) with the pin number, pin name, net name, and location.
- **UNIT Selection:** Select the unit used in the Net, FPN, and ADR files.
- **JIG Size:** Enter the JIG pin size to be applied.

- Only Silver Net: When this option is checked, checks only 4W pins.
- Top Place Name of JIG: Specify the top placed JIG pin's name in the ADR file.
- One Unit: Check this option when the imported design is a single board.
- Apply Scale: Enter the contraction rate of the produced board.
- Using Array JIG: Specify the Array Name and PCB Name of the imported design. Refer to the name definition for each board as follows.



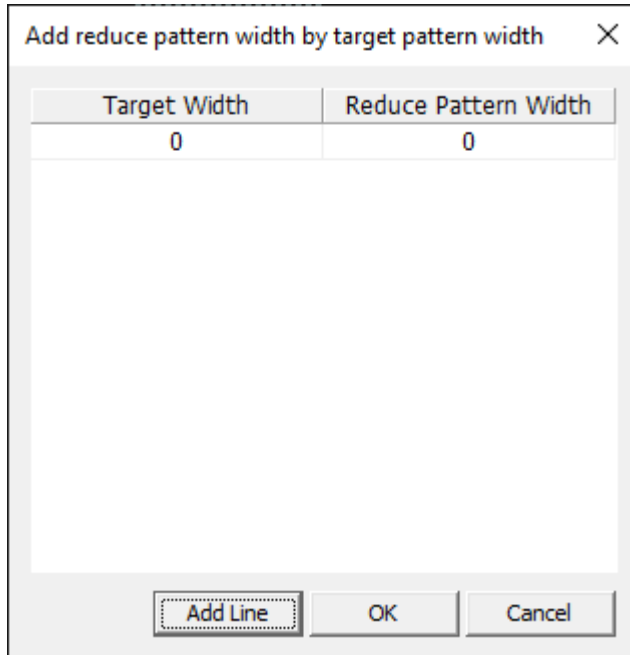
### 3.3. Component Layers

- **ADD:** By clicking **Add** button, users can add the line.
- **Layer Type:** Specify the layer type, such as Signal, PAD, Solder Mask, and Metal Mask per Top and Bottom.
- **Layer Name:** Specify the layer of imported Gerber of ODB++ data to match with the **Layer Type**.

### 3.4. Layer Stack-up

- **Resistivity (Ohm/m):** Enter the resistance value of the material.
- **Reset:** Initialize the **Layer Stack-up** settings.
- **Specify the file that define Layer Stackup:** Import the Excel format file of the layer stack up information.
- **Add:** By clicking **Add** button, users can add a line.
- **Delete:** By clicking **Delete** button, users can delete a line.
- **Physical Layers:** Number of the total physical layers.
- **Layer Name:** Select the layer stacking order as the actual design physical layer.
- **Thickness:** Specify the thickness of the physical layer.
- **Reduce Pattern Width:** Enter the reduced width of the produced route. This input value is reflected in the nominal resistance calculation.

- **Add Reduce Pattern Width:** Specify the reduced pattern width per a target pattern width. This input value is reflected in the nominal resistance calculation.



Target Width	Reduce Pattern Width
0	0

Add Line OK Cancel

- **Resistivity:** Enter the resistance value per a layer. This input value is reflected in the nominal resistance calculation.

### 3.5. Drill Layers

- **Add:** By clicking **Add** button, users can add a line.
- **Layer Name:** Select the drill layer used in the original design data.
- **Connect Top:** Set the start drill layer number.
- **Connect Bottom:** Set the end drill layer number.

### 3.6. Board Layers

- **Add:** By clicking **Add** button, users can add the line.
- **Layer Type:** Select the layer type either Board Outline or Etch.
- **Layer Name:** Select the layer name corresponding to the layer type.
- **Target Layer:** Select a layer to apply the board layer set in the **Layer Type**.

## 4. Make Component

After completing the layer setting, users can create components either automatically or manually.

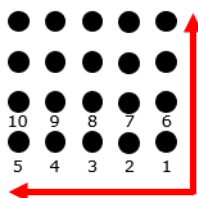
The screenshot shows the 'Make Component' dialog box with the following sections highlighted:

- 4.1 Pin Name Order:** Includes radio buttons for Numerical, Alpha-Numerical, and Numerical-Alpha. Below them are dropdowns for 'Order' (Right Bottom) and 'Left Bottom' with a right-pointing arrow between them.
- 4.2 Auto:** Includes input fields for 'Prefix for TOP' and 'Prefix for BOTTOM', and a 'Generate Component' button.
- 4.3 Manual:** Includes a 'Reference Name Prefix' input field, a checkbox for 'Use Manual Reference Name', and buttons for 'Select Component', 'Register Library Component', 'Scan Same Geometry', and 'Reset Selection'.
- 4.4 Control by JIG Data:** Includes a 'Move Rotate of Components' section with buttons for Rotate 90, Mirror, JIG, and Target, and an 'Apply' button. It also has a 'Repeated Components Remove' section with a checkbox for 'Automatic' and input fields for '1st' and '2nd' components, with an 'Apply' button.
- 4.5 Display Layer Control:** Includes radio buttons for TOP and BOTTOM. Under TOP are checkboxes for TOP PAD, TOP SOLDER, and TOP SIGNAL. Under BOTTOM are checkboxes for BOTTOM PAD, BOTTOM SOLDER, and BOTTOM SIGNAL.
- 4.6 Reference List:** Includes a table with columns 'Reference Name', 'Layer', 'Location', and 'Pin Count'. Below the table are 'Remove Component' and 'Reset Component' buttons.

### 4.1. Pin Name Order

For creating parts, specify the way of assigning the pin number.

- Numerical: Assign the pin number with a number (ex. 1, 2, 3...).
- Alpha-Numerical: Assign the pin number with an alphabet and a number (ex. A1, A2, A3...).
- Numerical-Alpha: Assign the pin number with a number and an alphabet (ex. 1A, 2A, 3A...).
- Order: Specify the direction of assigning pin number. For example, if users select from Right Bottom to Left Bottom, the pins are assigned like below.





#### 4.2. Auto

The menu creates the parts automatically. Users need to specify the prefix for the Top and Bottom. When clicked the **Generate Component** button, this menu automatically creates the parts for the top and bottom and created parts are listed in the **Reference List**.

#### 4.3. Manual

This menu creates the parts manually.

- Reference Name Prefix: Specify the prefix of the parts to be applied.
- Use Manual Reference Name: When this option is checked, users can manually define the reference name for the selected pins instead of the prefix defined in the **Reference Name Prefix**.
- Select Component: Select pins to create as a component with the **Reference Name Prefix**.
- Scan Same Geometry: Find the same components which defined in the **Select Component**.
- Register Library Component: When clicked this option, register the searched parts into library.
- Reset Selection: Reset the selected pins.

#### 4.4. Control by JIG Data

This menu is activated when the user specifies the menu of **Specify JIG Data** in the **Layer Setting**.

- **Move Rotate of Components**: Move or rotate the position of the JIG read from the JIG data.  
Click the **JIG** button and select a pin to be moved or rotated.  
Click the **Target** button and select a location to be moved or manually input the location with X and Y coordinates.
- **Repeated Components Remove**: This menu deletes the duplicated JIG data.

Control by JIG Data

**Move Rotate of Components**

Rotate 90 Mirror

JIG 0.000 0.000

Target 0.000 0.000

0.000 0.000

Apply

**Repeated Components Remove**

☐ Automatic

1st 0.000 0.000

2nd 0.000 0.000

0.000 0.000

Apply

#### 4.5. Display Layer Control

Select the display layer of PAD, Solder, and Signal for the Top and Bottom.

Display Layer Control

☒ TOP ☐ BOTTOM

☒ TOP PAD ☒ BOTTOM PAD

☒ TOP SOLDER ☒ BOTTOM SOLDER

☒ TOP SIGNAL ☒ BOTTOM SIGNAL

#### 4.6. Reference List

Created parts by automatically or manually are listed in this table.

- Remove Component: Remove the selected components.

- Reset Component: Reset all created components.

## 5. Make Netlist

The next step after part creation is that the user can create the routing nets using vias, pins, and patterns. In the **Make Netlist** menu, routing nets can be generated.

The screenshot shows the 'Make Netlist' dialog box with the following sections and callouts:

- 5.1** Verification by JIG Data
  - ☐ Exist Pin of JIG Data
  - ☐ Exist Net of JIG Data
- 5.2** Write Nominal Resistance for by JIG Data
  - ☐ Only Silver Point(4W)
- 5.3** Signal Net Rule
  - Net Name Prefix: [ ]
- 5.4** Ground Net Rule
  - ☐ Ground Net Rule
  - Net Name Prefix: [ ]
  - Pin Count >: [10]
- 5.5** Power Net Rule
  - ☐ Power Net Rule
  - Net Name Prefix: [ ]
  - Min Width >: [3]
- 5.6** Dummy Net Rule
  - ☐ Dummy Net Rule
  - Net Name Prefix: [ ]
- 5.7** Remove 0 pin assigned nets
  - ☐ Remove 0 pin assigned nets
- 5.8** Maximum Number of Pins Connected to Net When Calculating Nominal Resistance
  - ☒ Maximum Number of Pins Connected to Net When Calculating Nominal Resistance: [65]
- 5.9** Generate Netlist
  - [Generate Netlist]

Below the rules section, there is a checkbox for 'Exclude Mode' and a table with the following headers: Net Name, Net Type, Ref-Pin, Pin Count. At the bottom are buttons for 'Remove Netlist' and 'Reset Netlist'.

### 5.1. Verification by JIG Data

This menu is activated when the **Specify JIG Data** in the **Layer Setting** is selected. This function is to find differences by comparing JIG data and the design data.

Verification by JIG Data

☐ Exist Pin of JIG Data ☐ Exist Net of JIG Data

### 5.2. Write Nominal Resistance for by JIG Data

Calculate the nominal resistance value and save the value in the NET file which is defined in the **Specify JIG Data** of the **Layer Setting**. When the **Only Silver Point (4W)** is checked, only 4W pins of calculated nominal resistance value are saved in the NET file.

Write Nominal Resistance for by JIG Data

☐ Only Silver Point(4W)

### 5.3. Signal Net Rule

Specify the common signal net prefix.

### 5.4. Ground Net Rule

Specify the ground net prefix. Nets having more connected pins than the specified **Pin Count** are recognized as the ground net.

### 5.5. Power Net Rule

Specify the power net prefix. Nets having larger width than the specified **Min Width** are recognized as the power net.



**Note:** When the **Signal**, **Ground**, and **Power** nets are specified, the ground net condition is searched and set first, then set the power net. The remaining nets are classified as the common signal nets.

### 5.6. Dummy Net Rule

Specify the dummy net prefix which number of connected pin counts are less than two.

### 5.7. Remove 0 pin assigned nets

When this option is checked, nets having zero connected pin will not be created as a net.

### 5.8. Maximum Number of Pins Connected to Net When Calculating Nominal Resistance

This option is to exclude nets having more connected pins than the user specified pin counts for the nominal resistance calculation.

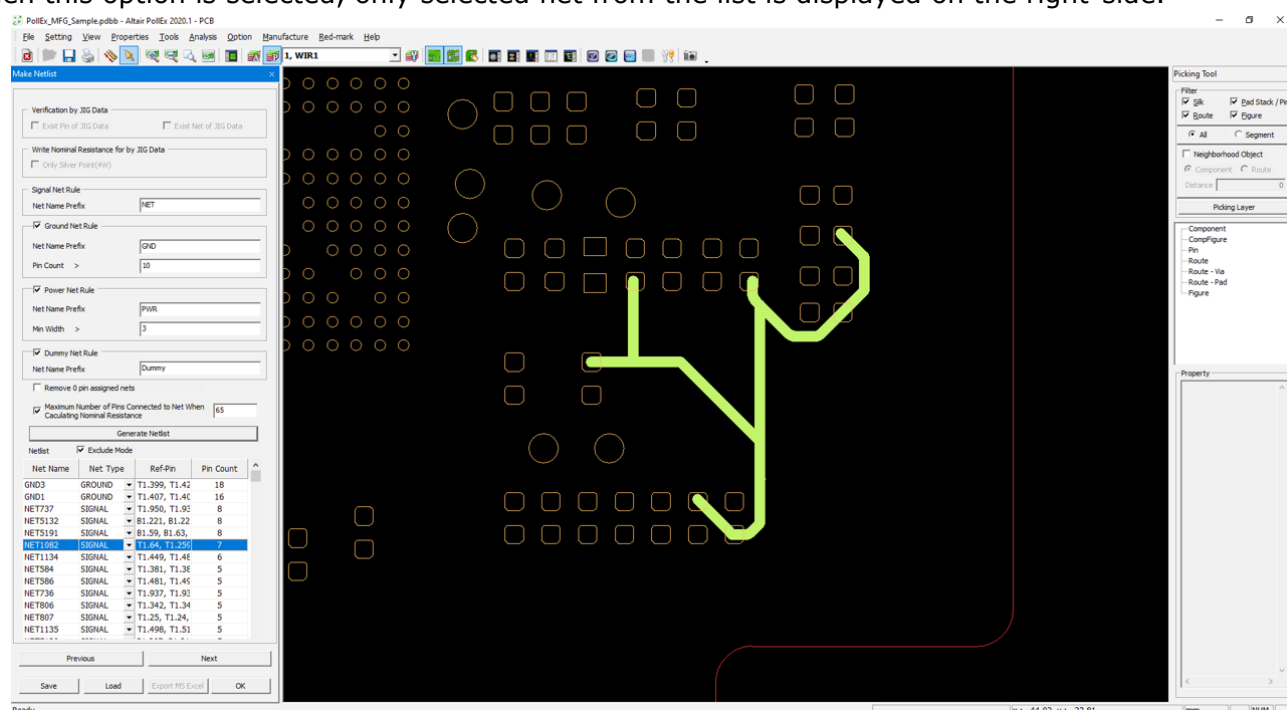
### 5.9. Generate Netlist

When clicked the **Generate Netlist** button, nets are generated with the specified options. The generated nets contain the information of net name, net type, reference pin, and pin count.

Net Name	Net Type	Ref-Pin	Pin Count
GND3	GROUND	T1.399, T1.421, T1	18
GND1	GROUND	T1.407, T1.406, T1	16
NET737	SIGNAL	T1.950, T1.932, T1	8
NET1172	SIGNAL	B1.220, B1.221, B1	8
NET1231	SIGNAL	B1.59, B1.63, B1.51	8
NET1078	SIGNAL	T1.64, T1.259, T1.	7
NET1122	SIGNAL	T1.449, T1.480, T1	6
NET584	SIGNAL	T1.381, T1.382, T1	5
NET585	SIGNAL	T1.481, T1.497, T1	5
NET736	SIGNAL	T1.937, T1.935, T1	5
NET806	SIGNAL	T1.342, T1.341, T1	5

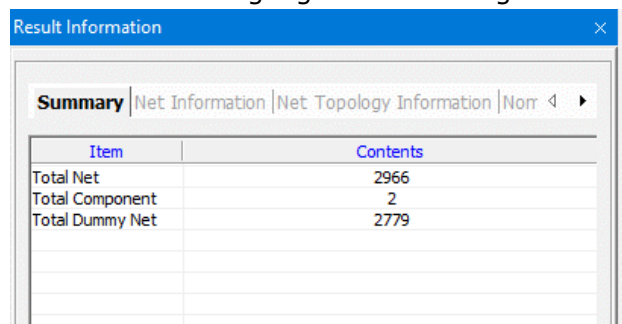
### 5.10. Exclude Mode

When this option is selected, only selected net from the list is displayed on the right-side.



## 6. Result Information

Once the PCB design generation is completed, users can review the results of four different information of Net, Net Topology, Summary, and Nominal Resistance. On each tab, selected net from the lists will be focused and highlighted on the right-side of window.



## 6.1. Export to MS Excel

The result information can be exported to MS Excel by clicking the **Export MS Excel** button.

Save
Load
Export MS Excel
OK

	A	B	C	D	E	F	G	H	I
1	Design File Name : Pollex_MFG_Sample								
2									
3	No	Net Name	Reference.Pin	X	Y	Place Layer			
4	1	Dummy1	T1.1013	19.900	52.700	TOP			
5	2	Dummy3	T1.967	53.900	46.100	TOP			
6	3	Dummy5	T1.965	47.400	41.850	TOP			
7	4	Dummy7	T1.969	35.500	46.700	TOP			
8	5	Dummy9	T1.820	32.050	32.000	TOP			
9	6	Dummy11	T1.818	33.350	32.000	TOP			
10	7	Dummy13	T1.817	34.000	32.000	TOP			
11	8	Dummy15	T1.815	35.300	32.000	TOP			
12	9	Dummy17	T1.812	37.250	32.000	TOP			
13	10	Dummy19	T1.809	39.850	32.000	TOP			
14	11	Dummy21	T1.808	40.500	32.000	TOP			
15	12	Dummy23	T1.807	41.150	32.000	TOP			
16	13	Dummy25	T1.804	43.100	32.000	TOP			
17	14	Dummy27	T1.297	35.950	19.000	TOP			
18	15	Dummy29	T1.295	37.900	19.000	TOP			
19	16	Dummy31	T1.291	41.800	19.000	TOP			
20	17	Dummy33	T1.292	40.500	19.000	TOP			
21	18	Dummy35	T1.288	43.750	19.000	TOP			
22	19	Dummy37	T1.287	45.050	19.000	TOP			
23	20	Dummy39	T1.284	47.000	19.000	TOP			
24	21	Dummy41	T1.278	35.300	18.350	TOP			
25	22	Dummy43	T1.274	38.550	18.350	TOP			
26	23	Dummy45	T1.275	37.900	18.350	TOP			
27	24	Dummy47	T1.268	43.750	18.350	TOP			
28	25	Dummy49	T1.267	44.400	18.350	TOP			
29	26	Dummy51	T1.264	47.000	18.350	TOP			
30	27	Dummy53	T1.247	36.600	17.700	TOP			
31	28	Dummy55	T1.244	39.850	17.700	TOP			
32	29	Dummy57	T1.243	40.500	17.700	TOP			
33	30	Dummy59	T1.236	46.350	17.700	TOP			
34	31	Dummy61	T1.235	47.000	17.700	TOP			
35	32	Dummy63	T1.215	32.050	17.050	TOP			
36	33	Dummy65	T1.211	34.650	17.050	TOP			
37	34	Dummy67	T1.207	37.250	17.050	TOP			
38	35	Dummy69	T1.204	39.850	17.050	TOP			
39	36	Dummy71	T1.203	40.500	17.050	TOP			

◀ ▶
Net Information
Net Topology Information
Summary
Nominal Resistance

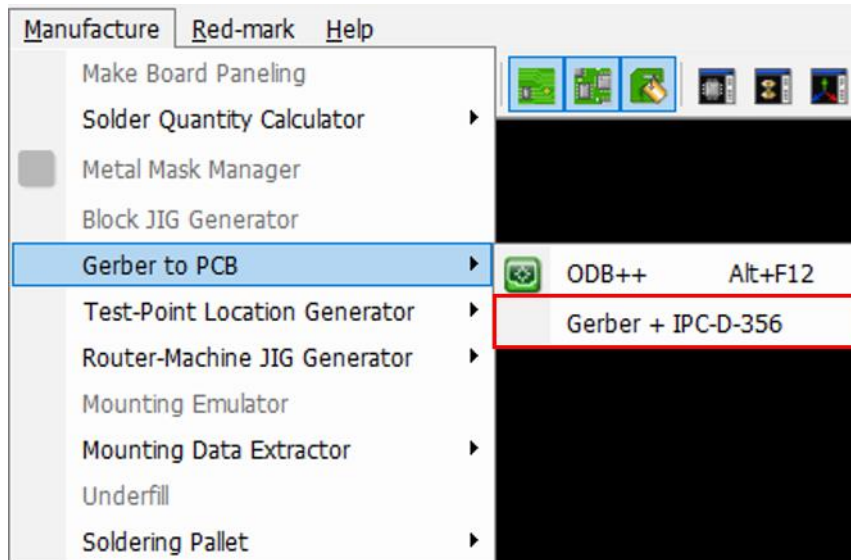
# Gerber + IPC-D-356

The **Gerber + IPC-D-356** converts a Gerber design file that has only graphical data into PCB utilizing the IPC-D-356 data by reverse engineering.

## 1. Launch Gerber + IPC-D-356

**Gerber to PCB** is the optional module of Pollex PCB. After launching the Pollex PCB, Select the menu, **Manufacture – Gerber to PCB – IPC-D-356** menu.

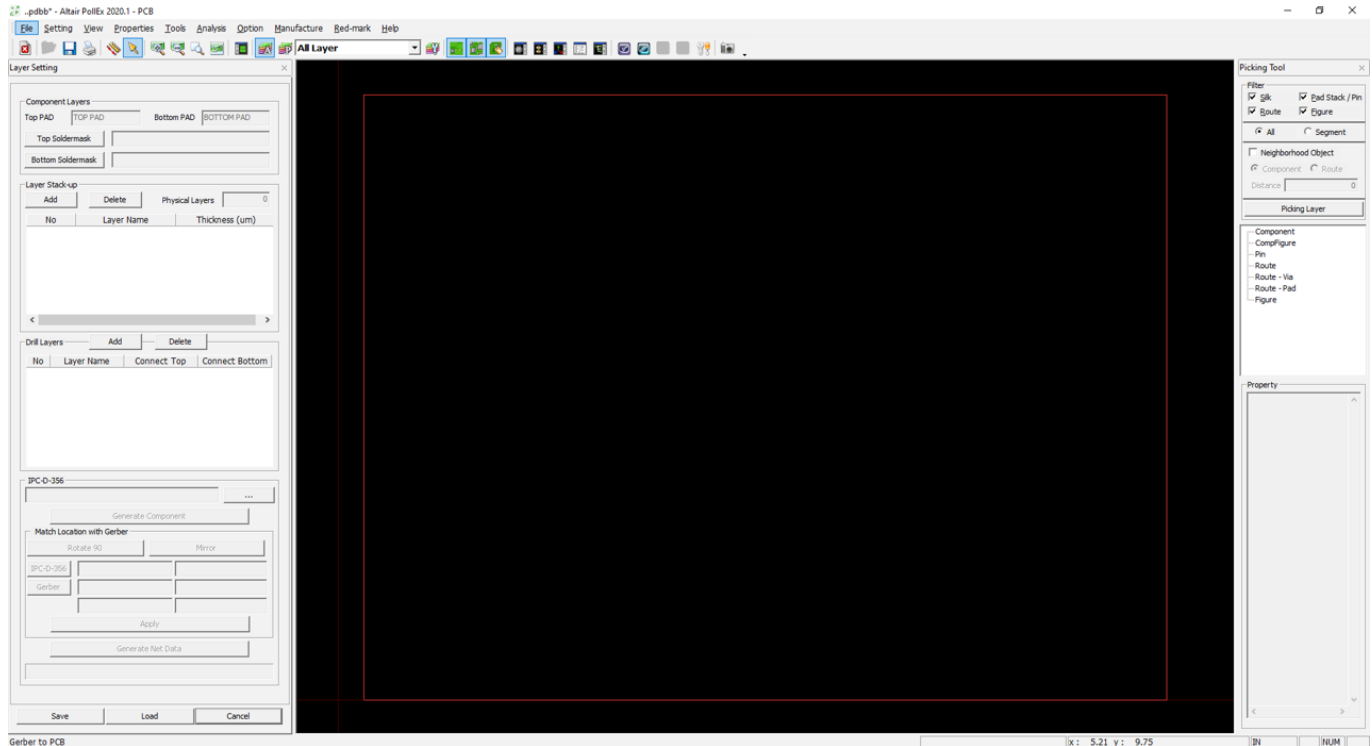
Please note that the menu will not be enabled if you import other than Gerber files.



## 2. Gerber + IPC-D-356 GUI

To make the intelligent PCB design data, users need to set layers of component, drill, and stack-up. Then, users need to import a IPC-D-356 data and to match the design origin between Gerber and IPC data.

On the right-side of the main window, the **Picking Tool** dialog is displayed. For more detailed information, please refer to the **PolliEx PCB** manual.



### 2.1. Save and Load for the environment setup

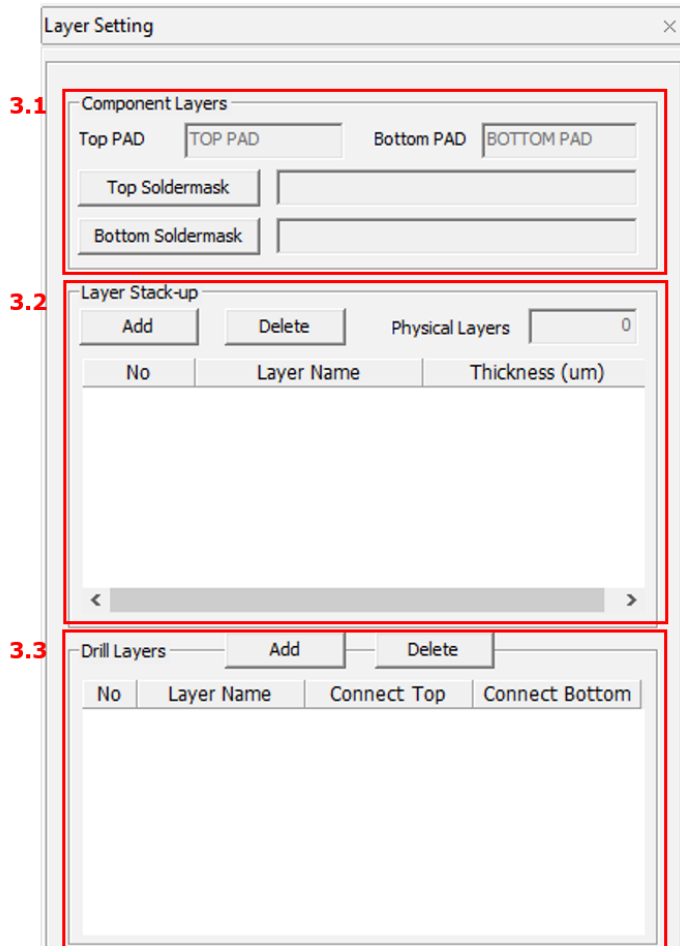
On the bottom side, users can save or load the environment settings.



- a. **Save:** The current user setting environment can be saved as \*.dpce file.
- b. **Load:** Users can load the pre-defined setting environment.

### 3. Layer Setting

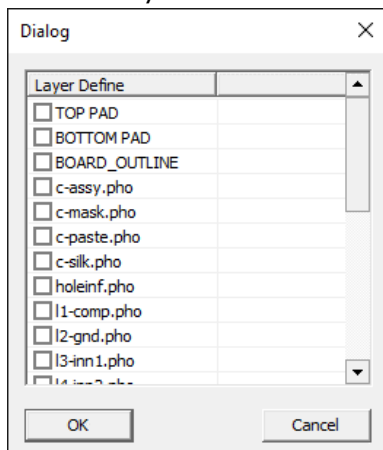
To generate a PCB design file using Gerber files, it is required to properly load the necessary files for the layers. When the data are imported into Pollex PCB, users need to define layers of the Gerber files. When imported Gerber files into Pollex PCB, physical layers are not defined. Imported Gerber files can be viewed in Layer menu. For more details, please refer **Setting - 2.1 Layer** section in Pollex PCB manual.



#### 3.1. Component Layers setup

Top and Bottom PADs are automatically detected.

- **Top Soldermask:** Select the soldermask for the Top layer. When clicked the icon, you can select a layer from the dialog.



- **Bottom Soldermask:** Select the soldermask for the Bottom layer.



### 3.2. Layer Stack-up setup

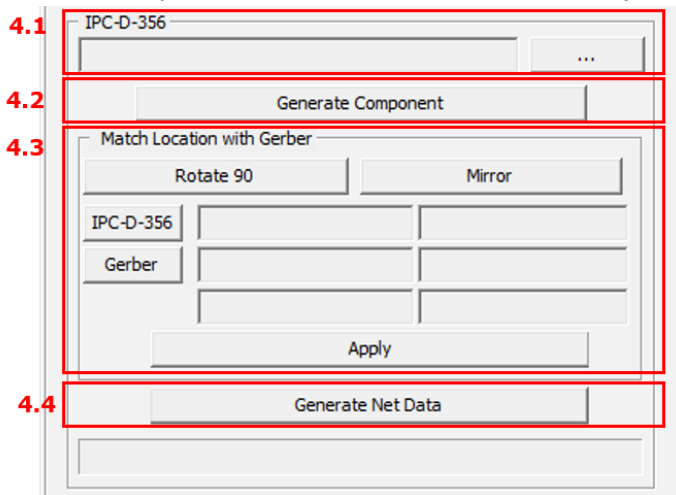
- **Add:** By clicking **Add** button, users can add a line.
- **Delete:** By clicking **Delete** button, users can delete a line.
- **Physical Layers:** Number of the total physical layers.
- **Layer Name:** Select the layer stacking order as the actual design physical layer.
- **Thickness:** Specify the thickness of the physical layer.

### 3.3. Drill Layers setup

- **Add:** By clicking **Add** button, users can add a line.
- **Layer Name:** Select the drill layer used in the original design data.
- **Connect Top:** Set the start drill layer number.
- **Connect Bottom:** Set the end drill layer number.

## 4. IPC-D-356

Next step after defined the layer settings is to combine the IPC-D-356 data to generate component and net data. Imported IPC data should be moved (if not matched together).

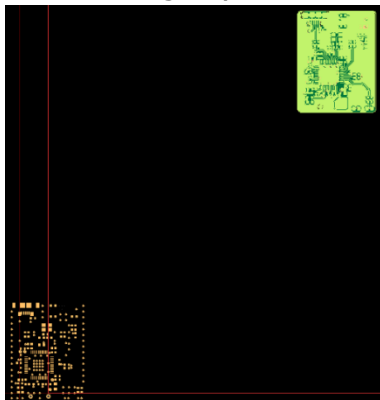


#### 4.1. Load IPC-D-356 data

Select an IPC-D-356 file by clicking the browse button. The supported format is IPC-D-356, IPC-D-356-A, and IPC-D-356-B.

#### 4.2. Generate Component

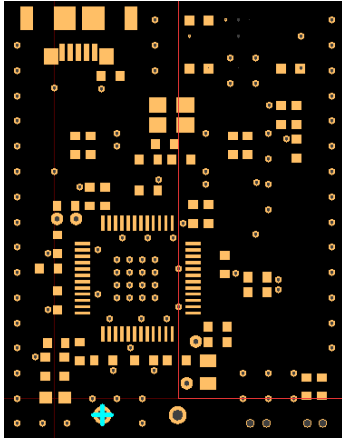
When clicked the **Generate Component** button, pads are displayed on the screen (left bottom on the below figure).



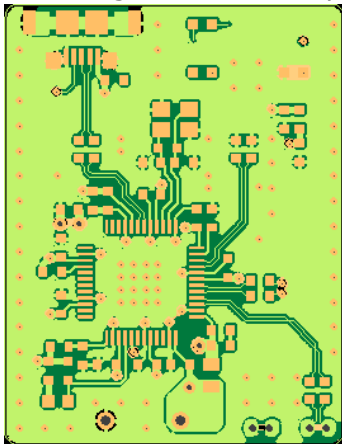
#### 4.3. Match Location with Gerber

For mismatched data, the IPC data should be moved, rotated, or mirrored.

- **Rotate90**: When clicked the button, the IPC data is rotated 90 degrees counterclockwise.
- **Mirror**: When clicked the button, the IPC data is flipped horizontally.
- **IPC-D-356**: Select a reference pin in the IPC data after clicked the **IPC-D-356** button. The selected pin is cross marked. The X and Y coordinates are automatically entered on the table.



- **Gerber**: Select a reference pin in the Gerber data after clicked the **Gerber** button. The X and Y coordinates are automatically entered on the table.
- **Apply**: When clicked the **Apply** button, the IPC data is moved to the Gerber data by matching the reference pin of both data.

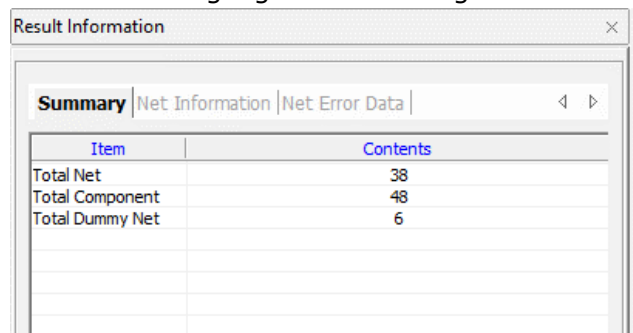


#### 4.4. Generate Net Data

This menu is to generate nets to convert as the intelligent PCB data. When clicked the **Generate Net Data** button, the progress bar is displayed the net generating status in the PCB design.

## 5. Result Information

Once the PCB design generation is completed, users can review the results of three different information of Summary, Net Information, and Net Error Data. On each tab, selected net from the lists will be focused and highlighted on the right-side of window for your review.



Result Information	
Summary   Net Information   Net Error Data	
Item	Contents
Total Net	38
Total Component	48
Total Dummy Net	6