Quadcept Tutorial

Product Version 7.0.0

August 2014

About this tutorial

This tutorial is designed to provide the user with an understanding of the basic operation of Quadcept, fundamentals of electronic design and also to actually design a simple PCB from a schematic. The content covers basic Quadcept functions as well as convenient functions to shorten the time of electronic design.

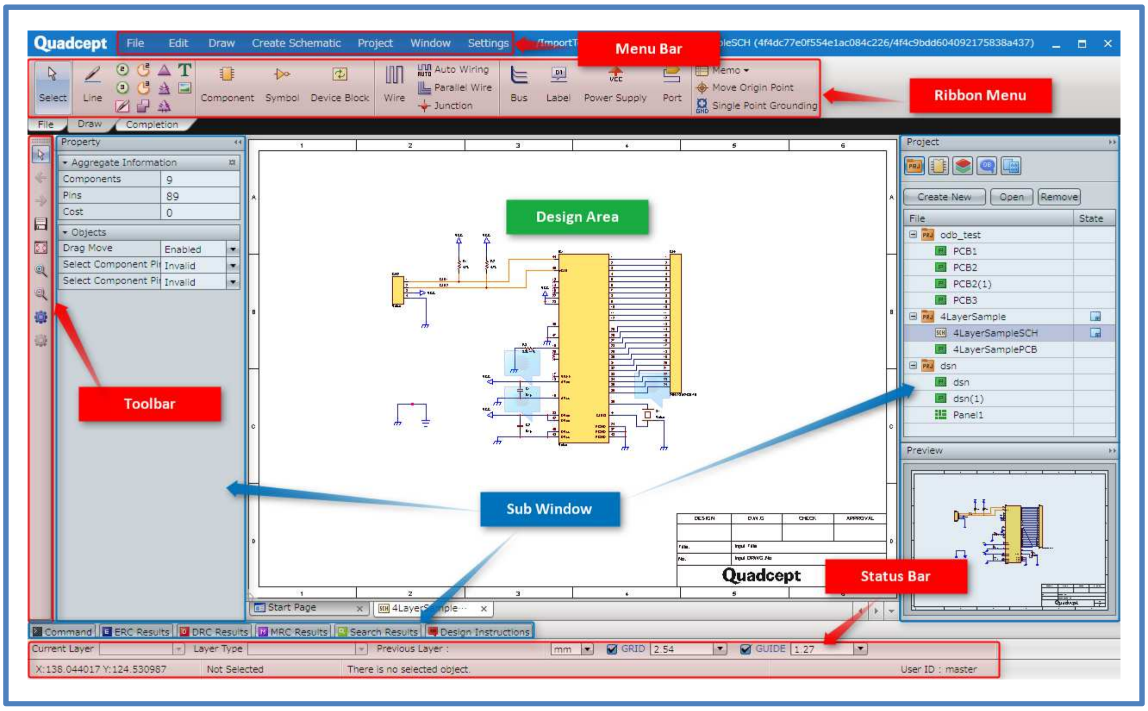
For more details, please refer to Quadcept online manual.
http://www.quadcept.com/en/manual/

Chapter 1 Getting Started

- Screen Layout
- Customizing Menu
- Using the Mouse
- Filter Function
- Stroke Function

Screen Layout

This is how the Quadcept screen looks like:



■Design Area

This is the main design window.

■Menu Bar

Various functions are available in the pull-down menu.

■Ribbon Menu

You can execute the function with a click on the icon in the ribbon menu.

Ribbon tabs including icons in the tabs are customizable.

■Toolbar

Icons can be grouped together.

Icons in the toolbar are customizable.

■Status Bar

Current design work status can be viewed or changed.

■Sub window

Various contents of such as properties, project or component can be viewed in the window, and also select show/hide as desired.

※The display position and shape of design area, sub window and toolbar can be easily changed or return them back to the default setting. For more information, please refer to our online manual.

Customizing Menu

When you change your CAD system to a new one, it is essential for designers that its operational feeling is the same as the familiar CAD.

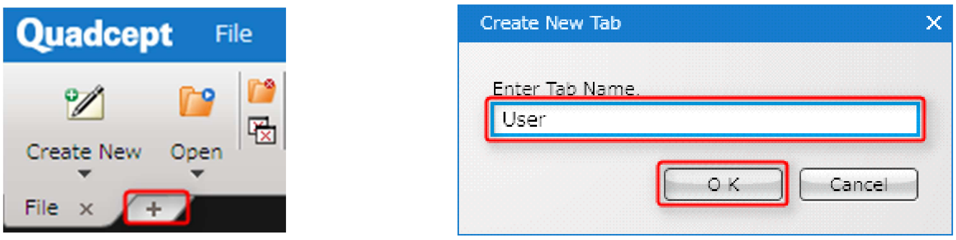
That’s all the more reason that Quadcept allows you to customize the ribbon menu, toolbar and shortcut keys. You can create your original easy-to-use menu to shorten work time.

Customizing Ribbon

You can add/delete ribbon tab or menus, as well as change the size of the icons.

Add a ribbon tab

1. Menu bar【Settings】→ Select【Customize Menu】
2. Click [+] tab that appears on the ribbon menu
3. The [Create New Tab] dialog appears. Enter the tab name and click【OK】.



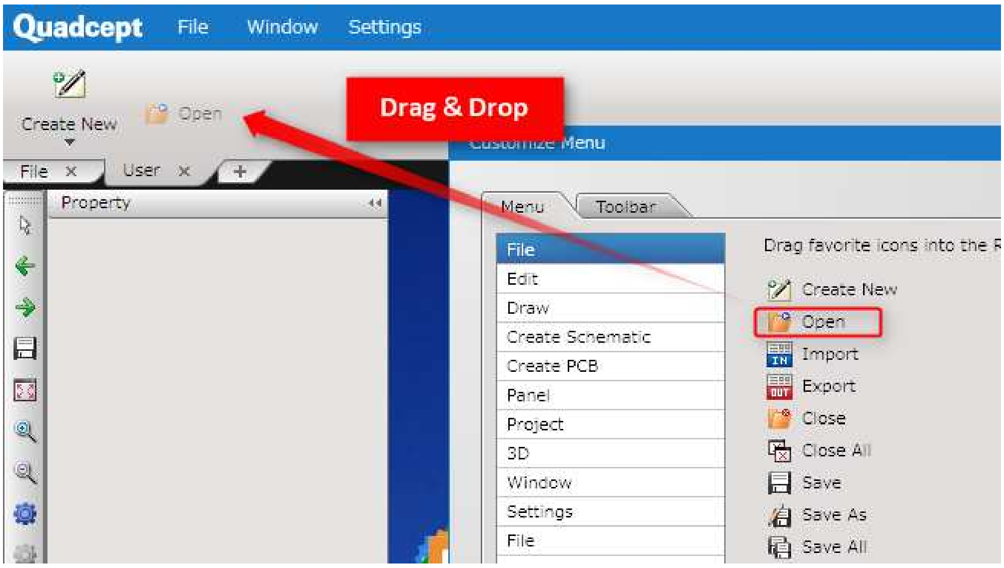
*To delete a ribbon tab:

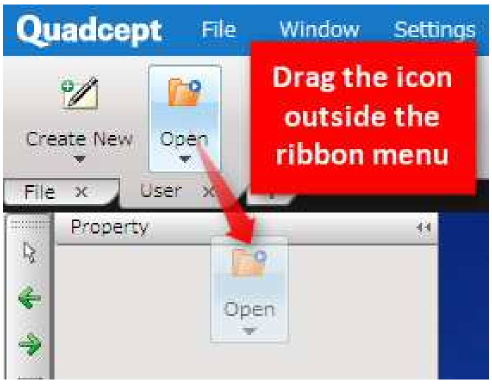
【Settings】→ Select【Customize Menu】

Click [x] that appears next to the tab

Add a menu

1. Menu bar【Settings】→ Select【Customize Menu】
2. Select the icon you want to add from the [Customize Menu] dialog. Drag and drop into the ribbon menu.





*To delete a menu:

Drag and drop the icon outside the ribbon menu.

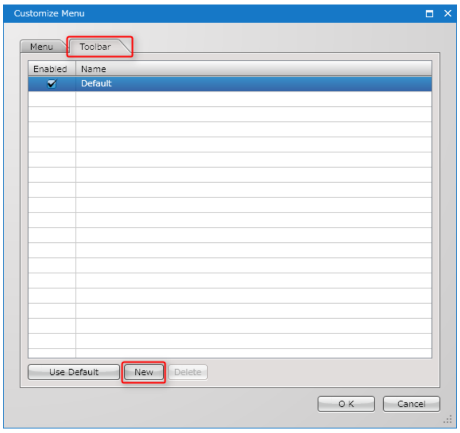
Customizing Toolbar

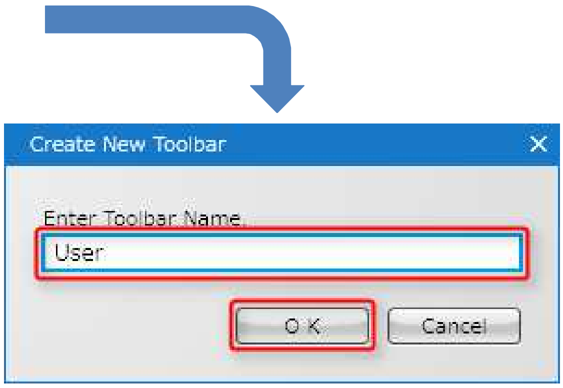
You can add/delete toolbars or menus, as well as move the toolbar to a convenient position.

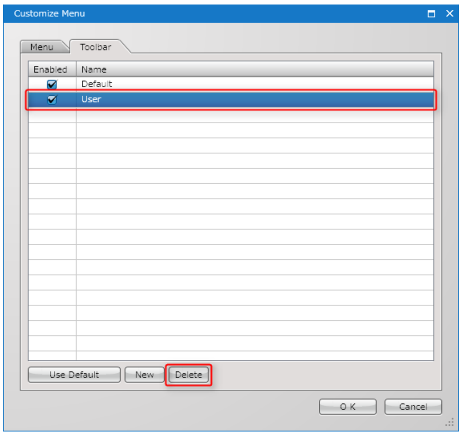
Add a toolbar

1. Menu bar【Settings】→ Select【Customize Menu】
2. The [Customize Menu] dialog appears. Select <Toolbar> tab and click【New】.
3. The [Create New Toolbar] dialog appears. Enter the toolbar name and click【OK】.

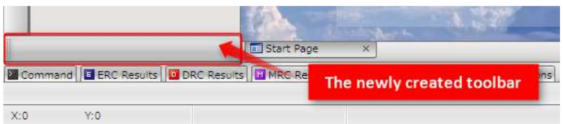
   The new name will be displayed in the [Toolbar] tab and the new toolbar appears around the bottom left corner where the default position is.

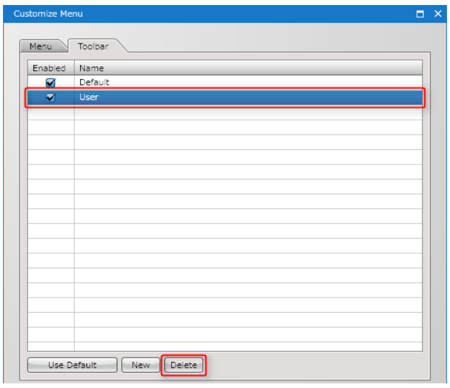






*When a new toolbar is created, you'll see the toolbar name in the [Toolbar] tab and the new toolbar appears around the bottom left corner.





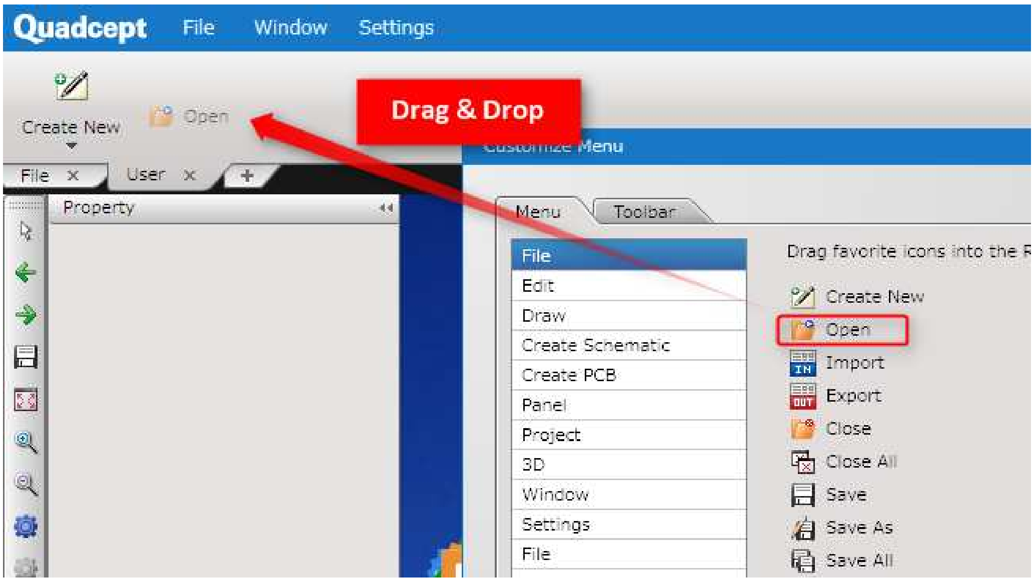
* To delete a toolbar:

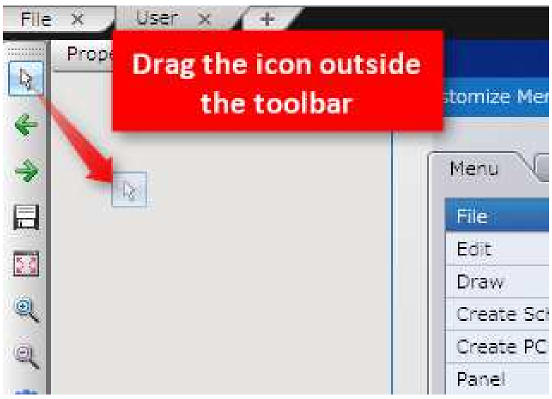
Menu【Settings】→ Select【Customize Menu】

Select the toolbar you want to delete in the <Toolbar> tab and click【Delete】.

Add a menu

1. Menu bar【Settings】→ Select【Customize Menu】
2. The [Customize Menu] dialog appears. You can drag and drop the icon you want to add in the toolbar.





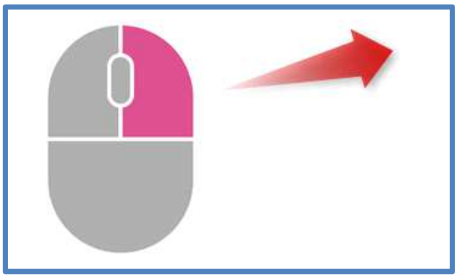
*To delete a menu:

Drag and drop the icon outside the toolbar.

Using the Mouse

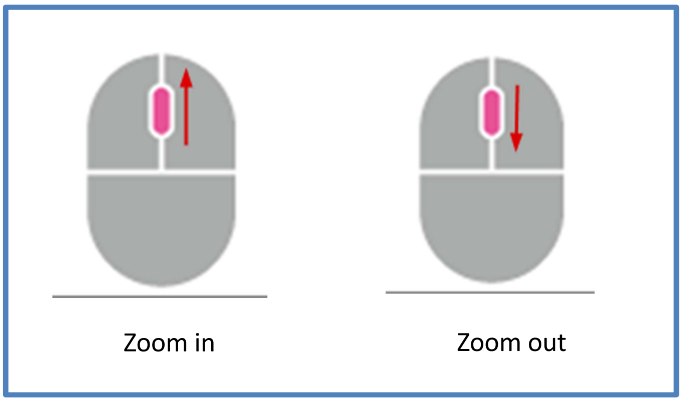
Scroll the design area

You can scroll the displayed area of the document in the design area. Click and keep 【the right mouse button held down while dragging the mouse】 to the desired direction.



Zoom in/out the image, Fit to screen

On schematic, PCB and component creation screens, the display magnification of the document can be changed.



* There are three types of shortcuts for fit to screen;

1. Whole drawing : 【Keyboard [1] 】
2. Selected object : 【Keyboard [2] 】
3. Board outline : 【Keyboard [3] 】

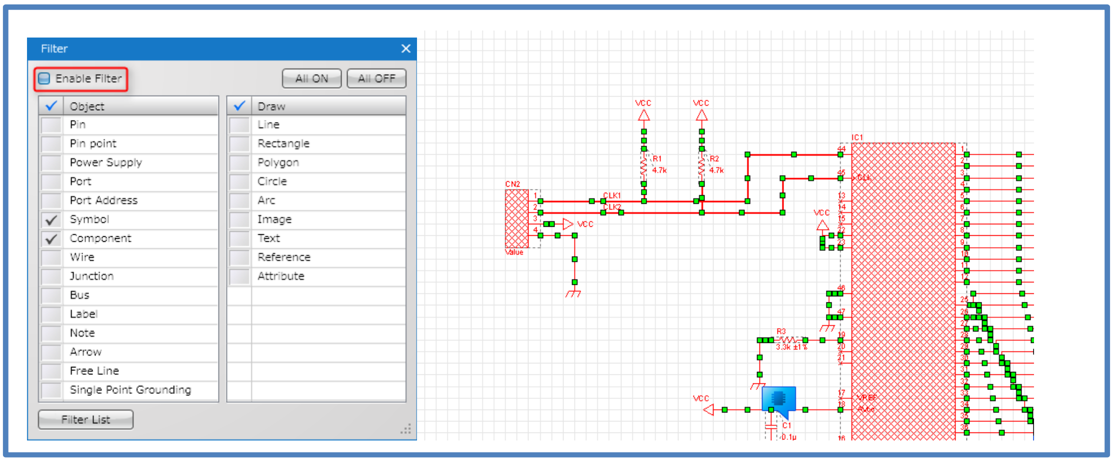
Filter Function

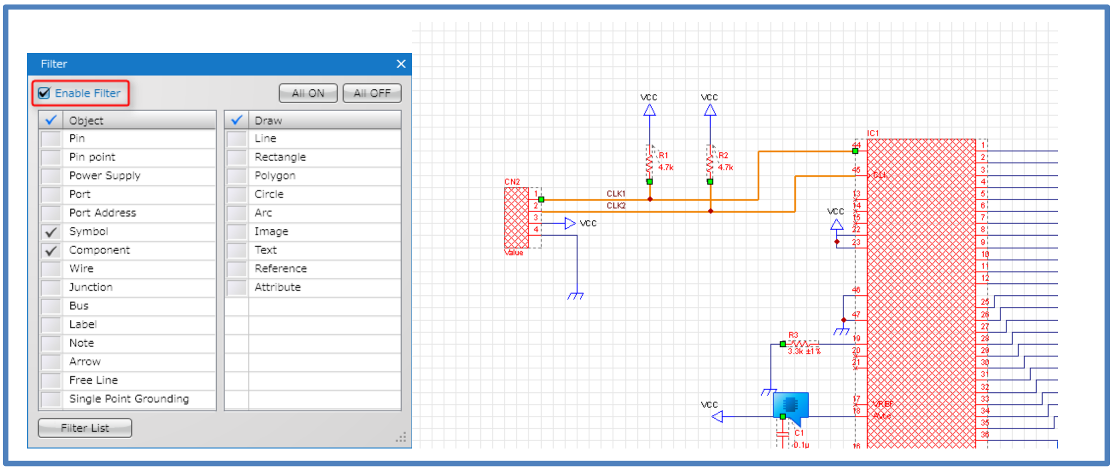
The filter function enables you to limit the selection of various objects (component, symbol, line, rectangle, wire etc.).

Doing Filter

The filter is enabled when [Enable Filter] around the top left corner is checked.

1. Menu Bar【Edit】→Select【Filter】
2. Check/Uncheck【Enable Filter】around the top left corner.
3. Choose the target objects.

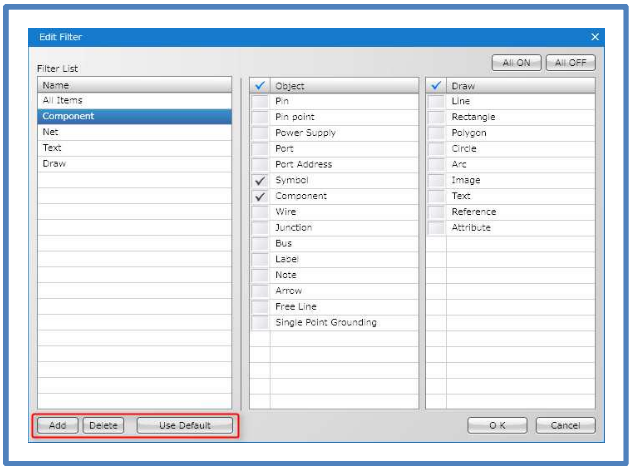


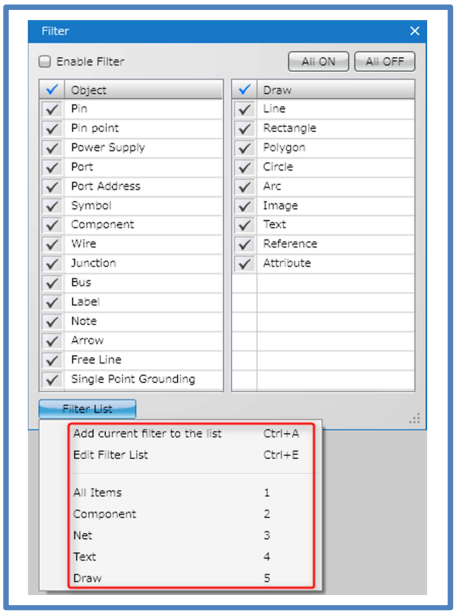


*The filter dialog can be closed while the filter is enabled so that it doesn't get in your way. However, you should be aware which objects are set as filter enabled when you close the dialog.

Setting a Filter Group

You can set a filter group by clicking [Filter List] around the bottom left corner and select [Edit Filter List]. Setting a group filter enables the use of keyboard shortcuts.



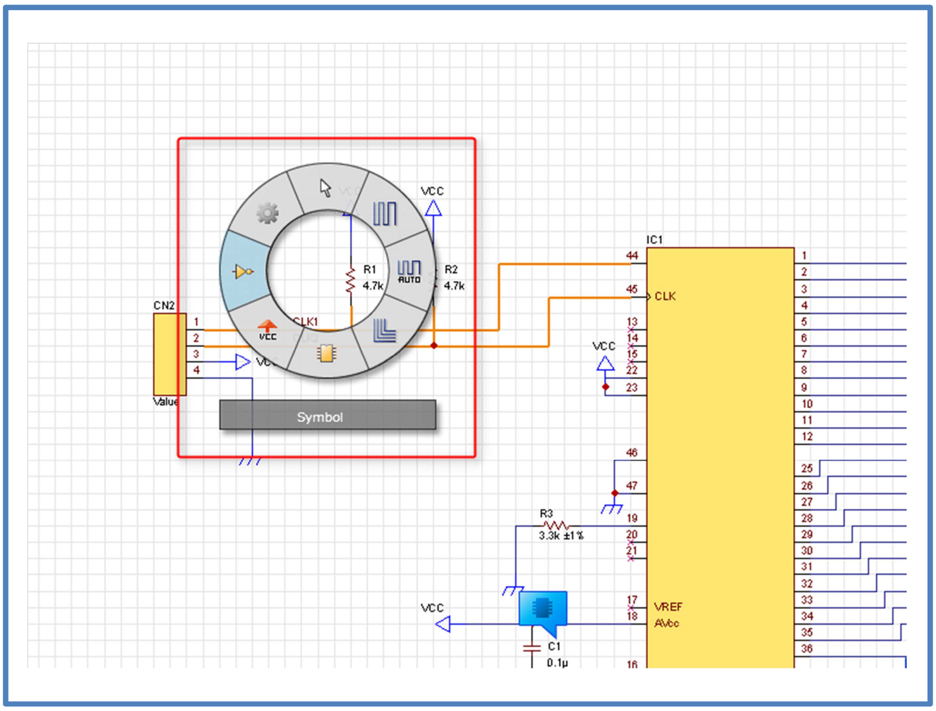


Stroke Function

The stroke command is the convenient Quadcept's original function.

You can register frequently used icons in the stroke command. The registered icons on the circle command appear by clicking the scroll button on the mouse, allowing you to instantly select the task you need while designing. By using the stroke command, you will be able to substantially reduce mouse movement.

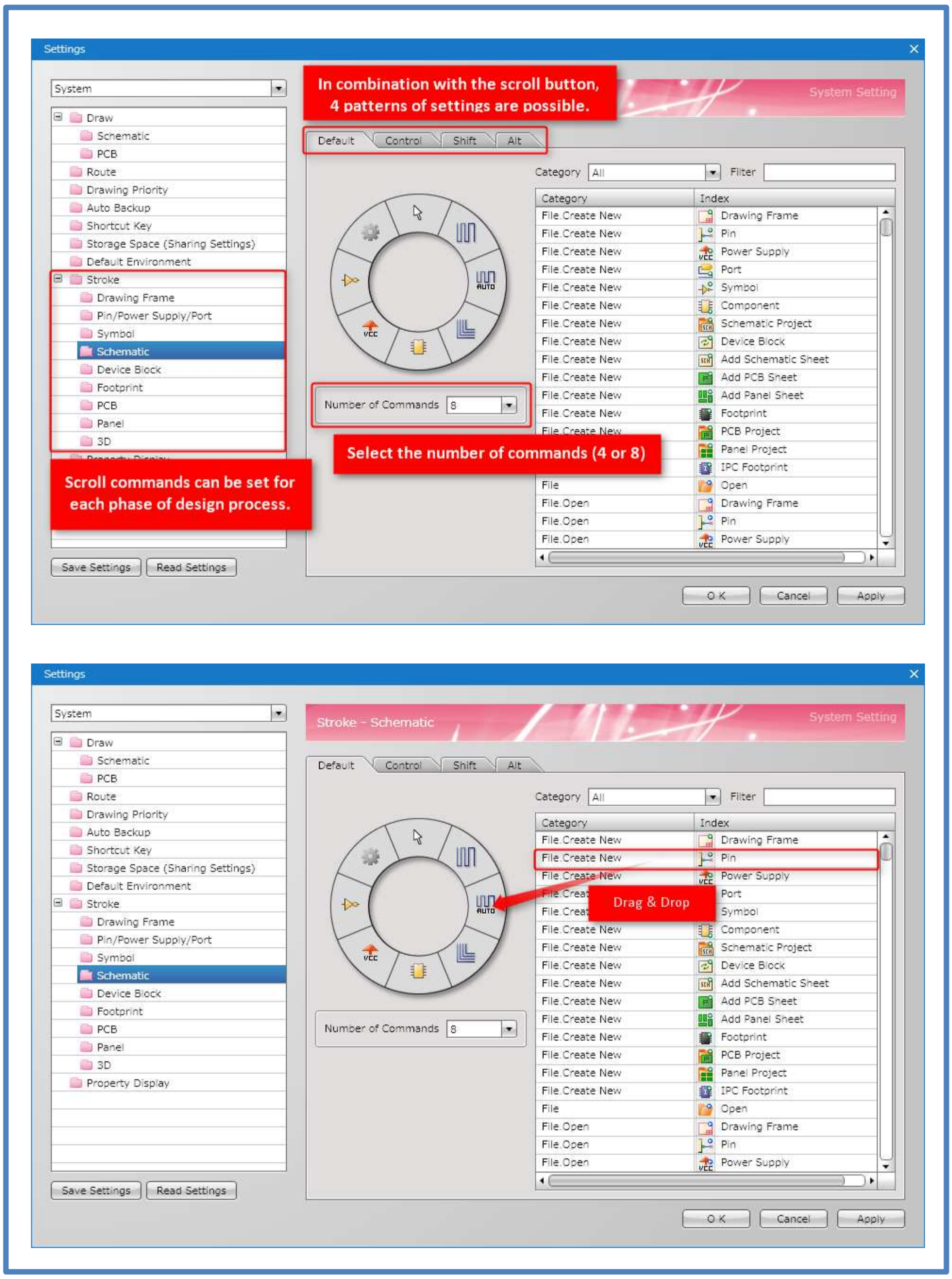
1. Press and held down the scroll button.
2. The stroke command appears. With the button pressed down, navigate the mouse pointer to the desired task command.
3. Navigate the mouse pointer onto the icon to select. (Release the scroll button to cancel.)



Customizing Stroke Menu

You can add or delete menus on the stroke function. Up to 32 commands can be set at each phase of design process using Ctrl, Shift and Alt keys.

1. Menu Bar【Settings】→ Select【Environment Settings】
2. Under the Stroke folder on the left, select the phase of design process you want to customize.
3. Select the tab you want to set.
4. Select the number of commands (4 or 8)
5. Select the icon from the right list and drag and drop it on the stroke command in the middle.



Chapter 2 Circuit Designer

- Schematic Design Flow
- Concept of Component
- Shape of Schematic Component
- Registering Component
- Creating Schematic

Schematic Design Flow

This flow describes the basic phases of the schematic design.

The objective is to output netlist.

■Schematic Design Flow

1. Creating Component (Pin + Symbol + Register Component)

First of all create component data required for the design.

You can make component by creating pins & symbols and registering the shapes & information.

2. Placing Component

Place components on the schematic.

It is asked to place component in a way the subsequent wiring task can be handled efficiently.

3. Wiring

This wiring information is output as netlist.

4. Verifying Schematics (ERC/DRC)

Perform the Electrical Rule Check (ERC) and Design Rule Check (DRC) .

5. Output (BOM, Netlist and Print)

Output the components used, their information and the netlist which contains the connection information from the completed schematic.

Print the schematic as you want.

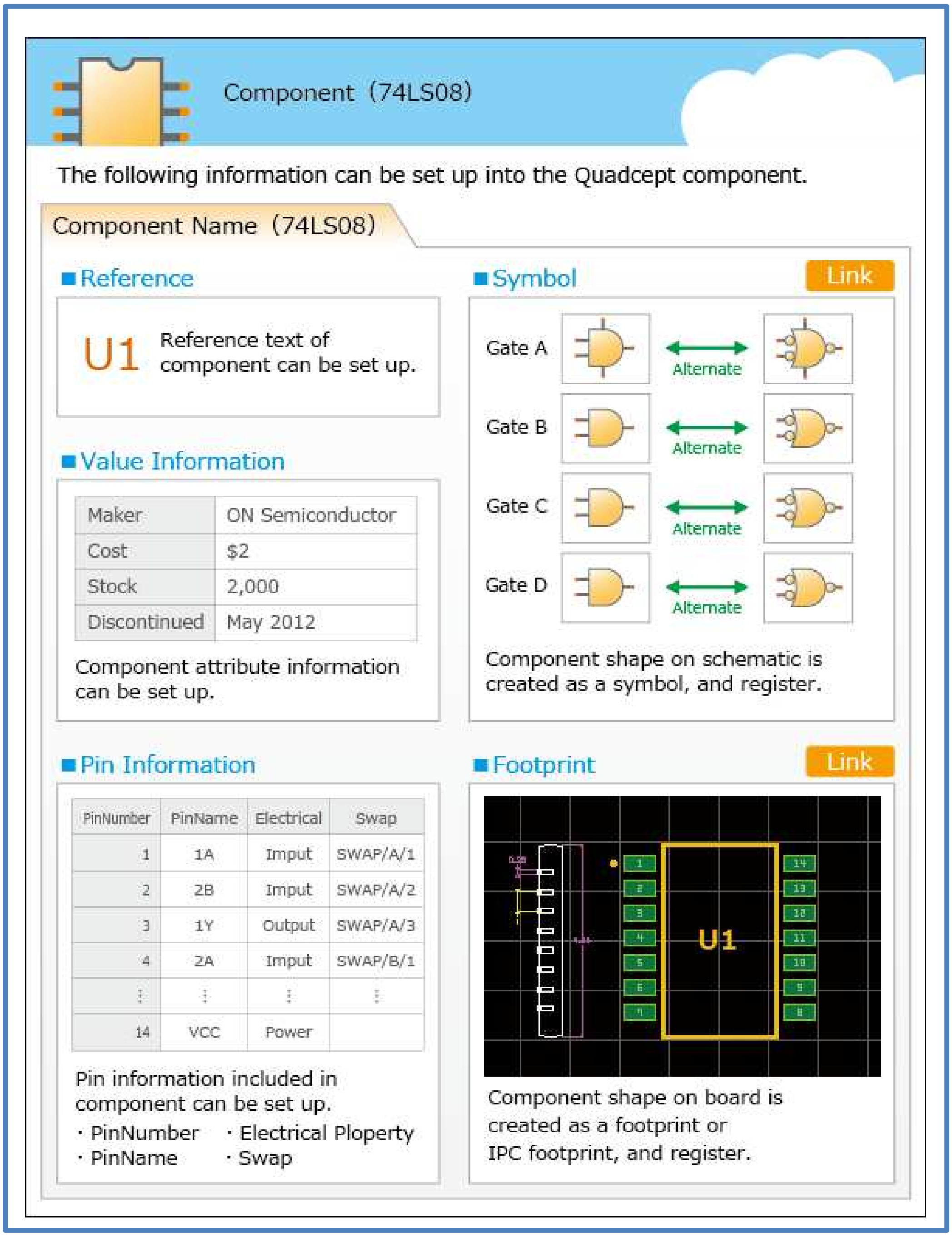
Concept of Component

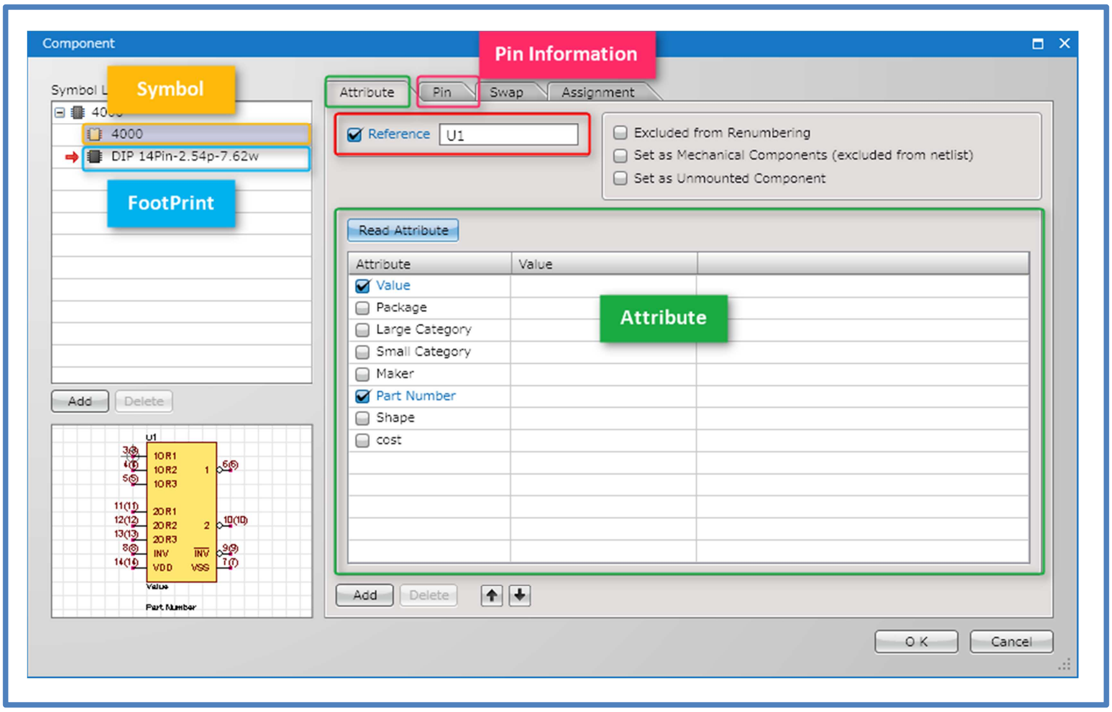
The shape of component, its internal attributes and other information are defined and registered as Quadcept component. Firstly, create the symbol (shape of schematic component) and footprint (shape of PCB component). Then, register them with links to the attributes and other information.

What is a component?

It consists of

Reference + Attribute information + Pin information + Shape on the schematic (symbol) + Shape on the PCB (footprint)




Component
Pin Information
Symbol
FootPrint
Attribute
Pin
Swap
Assignment
4000
DIP 14Pin-2.54p-7.62w
Reference
U1
Excluded from Renumbering
Set as Mechanical Components (excluded from netlist)
Set as Unmounted Component
Read Attribute
Attribute
Value
Value
Package
Large Category
Small Category
Maker
Part Number
Shape
cost
Add
Delete
O K
Cancel

Shape of Schematic Component

Create symbols (shapes of component) to place on the schematic.

A symbol (shape) can be shared among components. All you need to do is register it as a component. When you want to change the shape of the symbol, all the components linked to the same symbol can be changed all at once.

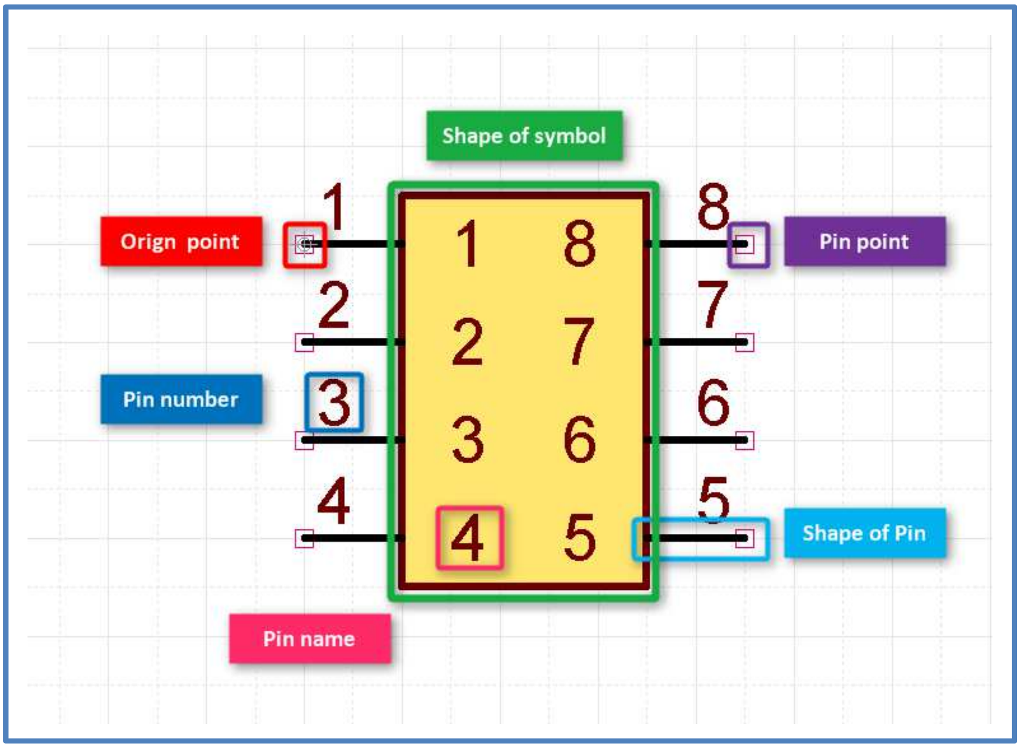
*Pins are needed to create a symbol so we have prepared various pins for you in the [Sample] folder. You can save time using those ready-made pins as you don't need to create a new one from scratch.

*Sample pins : Ribbon <Draw> → Select【Pin】→ [Sample]

Creating Symbol

Create a symbol (shape) of component to place on the schematic.

1. Open a sheet to create the new symbol
   Ribbon <File> → Select【Create New】→【Symbol】
2. Create the shape of symbol
   Ribbon <Draw> → Use【Line】or【Rectangle】or【Circle】etc.
3. Place pins
   Ribbon <Draw>→ Select【Pin】and place pins
4. Set origin point
   Ribbon <Draw> → Select【Move Origin Point】and place
5. Save

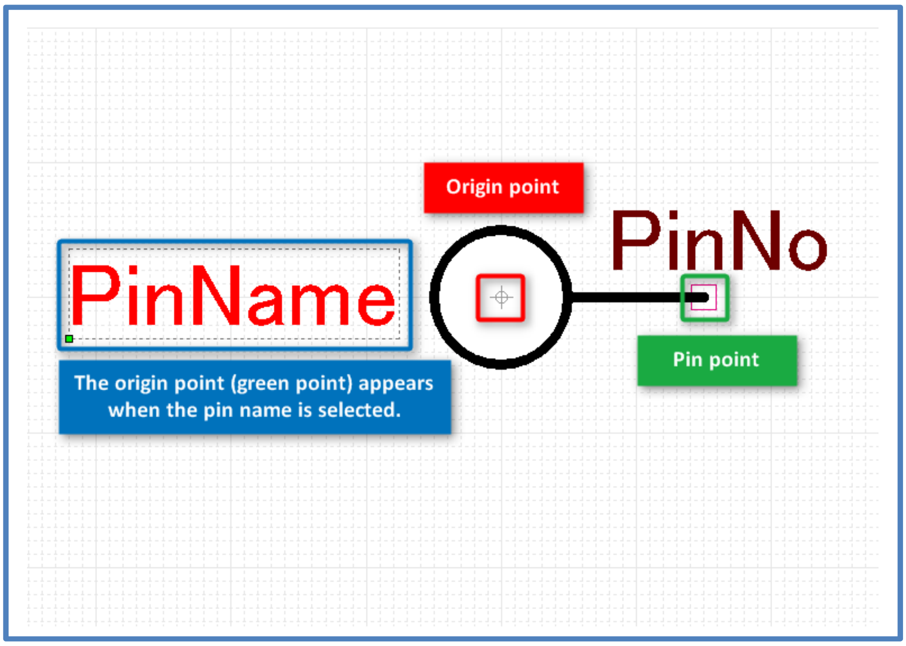


Pin

You can create your own pins as well as use the ready-made pins in the [Sample] folder.

*Sample pins : Ribbon <Draw> → Select【Pin】 → [Sample]

1. Open a sheet to create the new pin
   Ribbon <File> →【Create New】 →【Other】 →Select【Pin】
2. Create the pin shape
   Ribbon <Draw> → Select【Line】 or【2 Point Circle】 etc. and create the shape
   *The square in red is a pinpoint where wire is connected to on the schematic. Its default position is at the origin point.
3. Set the origin point
   Ribbon <Draw> → Select【Move Origin Point】 and place it
   *This origin point is the reference point when moving pins. It is easier to handle when placed on the opposite pin point.
4. Place its pin name and pin number
   When selected, its origin point (a green point) appears. You can drag/move and place objects using this as the reference point.
5. Save



Rotation and Mirroring

Pin and text can be placed in the direction you want by rotation and mirroring features.

*You can also use the rotation feature when placing components on schematic or footprints on PCB.

Rotation (in 90 degree increments)

When the object is in a movable state:

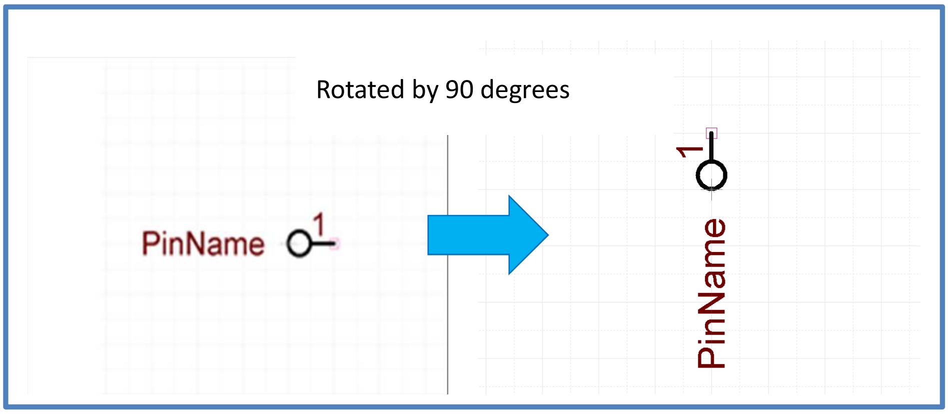
Method 1. 【Right click】→【Move/Rotate/Mirror】→【Rotate】

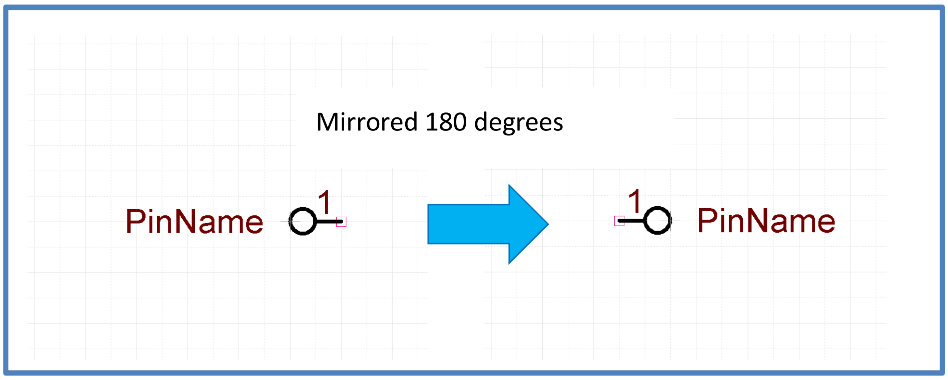
Method 2. Press【R】key to rotate Mirroring

When the object is in a movable state:

Method 1. 【Right click】→【Move/Rotate/Mirror】→【Mirror】

Method 2. Press【M】key to mirror

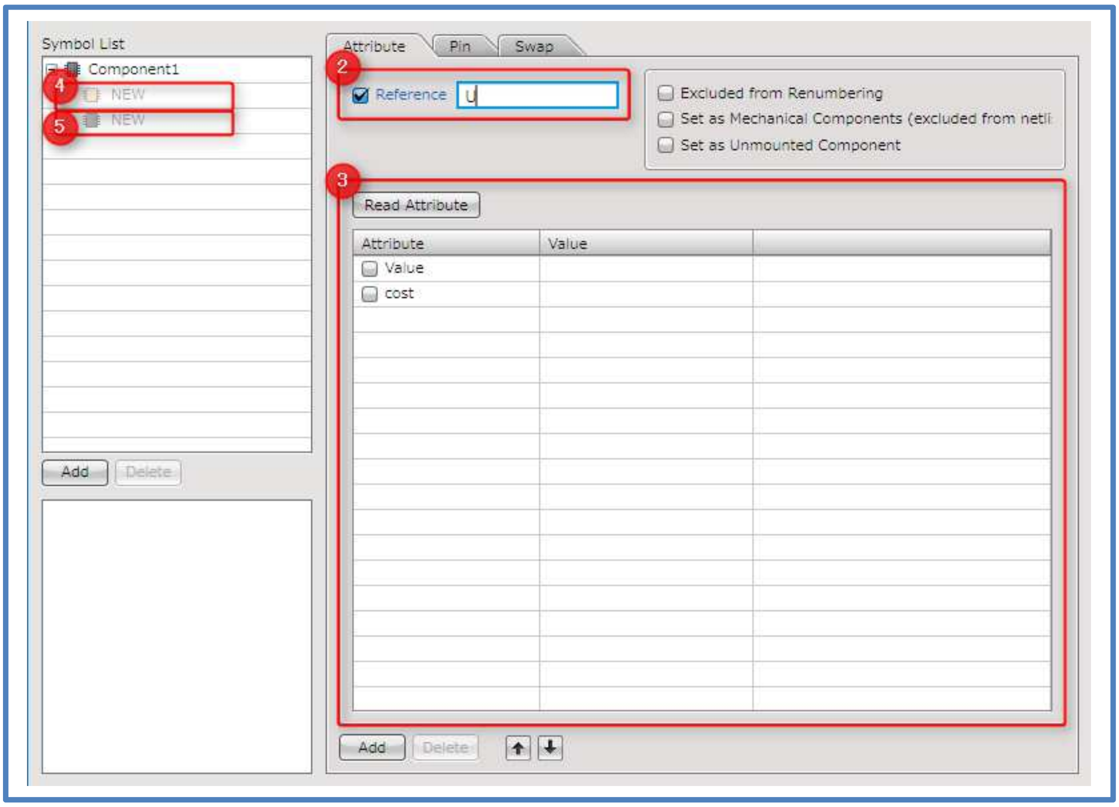




Registering Component

Create components to place on schematic/PCB drawings; register the shapes of schematic (symbol) and PCB components (footprint) as well as their internal attributes and other information.

1. Open a sheet to create the new component
   Ribbon <File> → 【Create New】 → Select 【Component】
2. Set Reference
3. Input attributes
   Click 【Read Attributes】 and read in from <Attribute Category> or click 【Add】 and select the attribute and enter the value in the <Value> field directly
4. Register the symbol (shape of schematic component)
   Double click and open [Select Component] dialog and select the symbol
5. Register the footprint (shape of PCB component)
   Double click and open [Select Component] dialog and select the symbol
6. Save
   Ribbon <File> → 【Save】 or 【Save as】 Select a directory and save it with a name



Attribute

You can set the specification, maker or other necessary information.

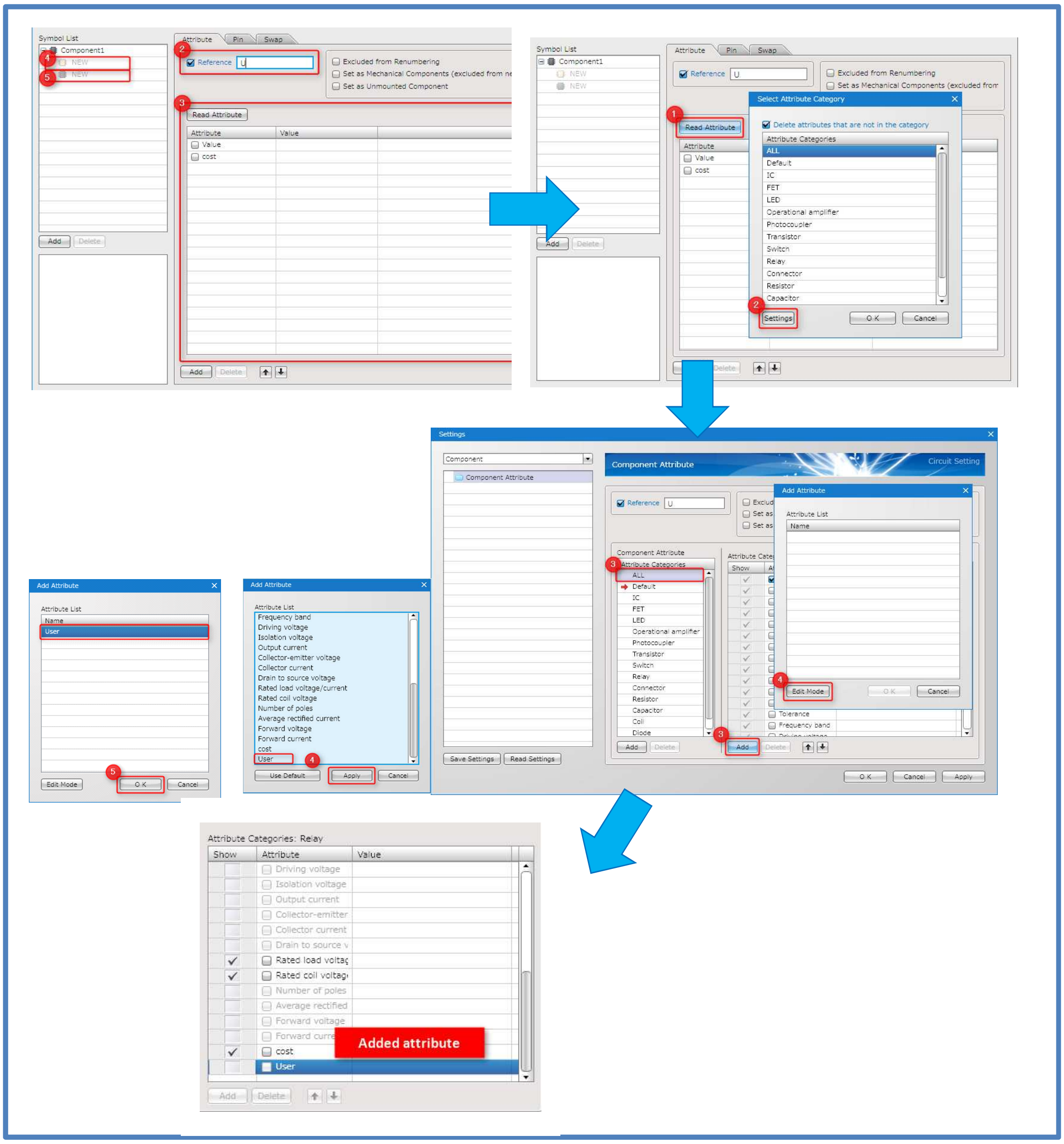
By checking the boxes on the left, you can show/hide the attribute on the schematic.

Input by setting attribute in the Attribute Category

You can create and manage attributes efficiently by setting them in the Attribute Category in advance. Some attributes, based on each component characteristics, are already registered as default settings.

How to add/set attributes

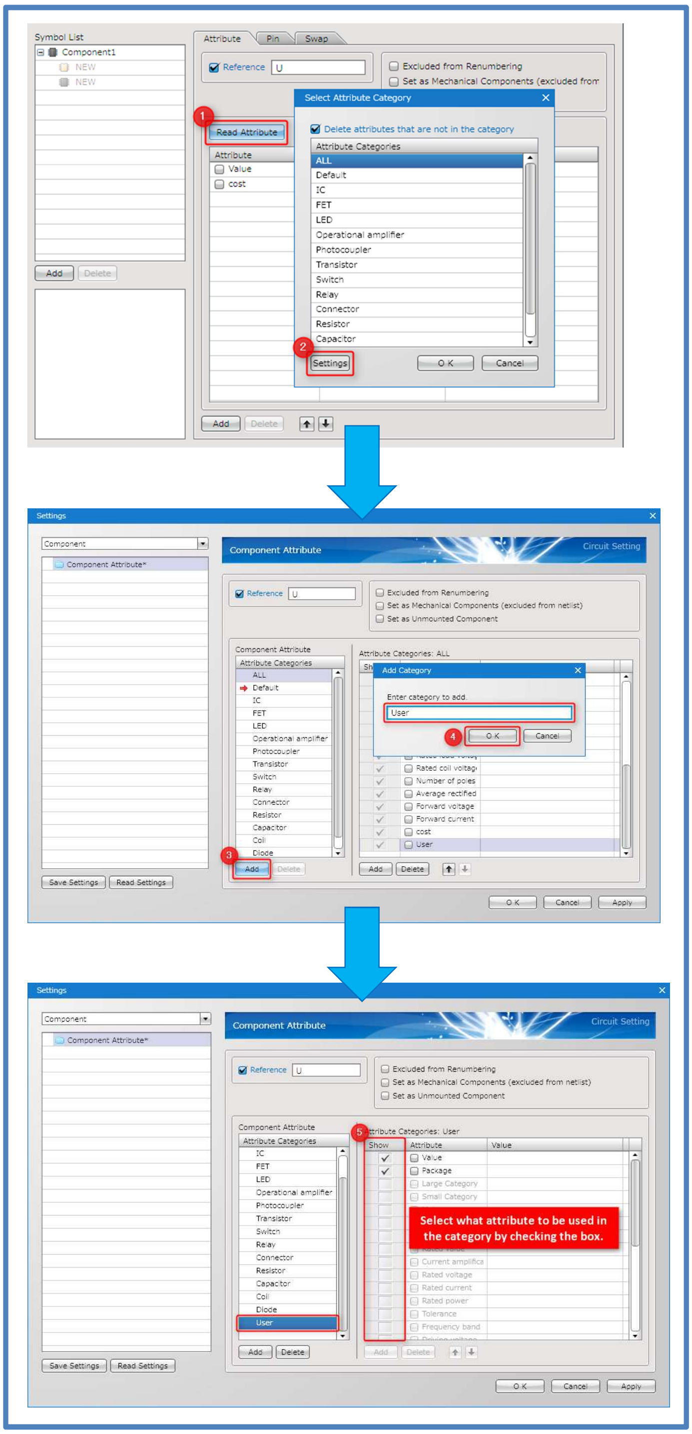
1. Click【Read Attribute】
2. The [Select Attribute Category] dialog will open. Click【Setting】
3. While "ALL" in the <Component Attribute> is being selected, Click【Add】at the bottom of <Attribute Categories>
4. The [Add Attribute] dialog will open. Click【Edit Mode】
   Enter the attribute in the [Attribute List] directly and click【Enter】key
5. What you entered will be displayed in the [Add Attribute] dialog Select the attribute and click【OK】
6. The attribute is added in the [Attribute Categories]
7. Click【Apply】



How to add/set attribute categories

1. Click【Read Attribute】
2. The [Select Attribute Category] dialog will open. Click【Setting】
3. Click【Add】 at the bottom of [Component Attribute]
4. The [Add Category] dialog will open Enter a new category name and click【OK】
5. The new category is added in the Attribute Categories under the [Component Attribute]

*You can select what attributes to be used in the new category by checking the [Show] field of the [Attribute Categories] on the right



Creating Schematic

This section describes the schematic design flow.

■Step 1. Creating a New Schematic

On Quadcept, a schematic (as well as PCB) always has a parent folder called Project. You can save multiple schematics in the same project so that the BOM and netlist can be output per project.

The opened project content is shown in the <Project> window on the right (default position).

Open a new schematic project sheet:

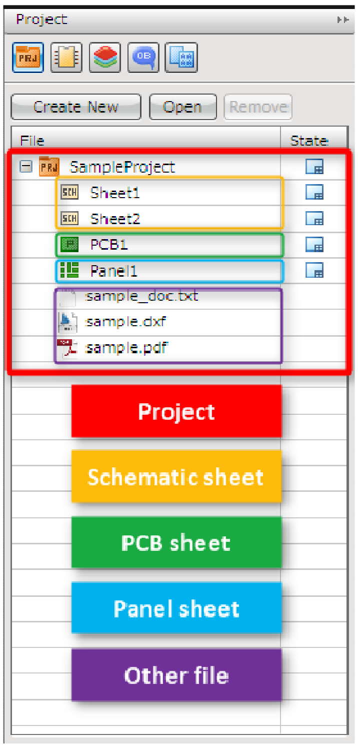
Ribbon <File> → Click【Create New】→【Schematic Project】

*Drawing frame is needed to create a schematic. We've prepared the frames of various sizes to choose from. You can save time using those ready-made frames as you don't need to create a new one from scratch. We'll use default pins on this training as well.

*On how to create a drawing frame, please refer to [Create Drawing Frame] in the online manual.

*Stored schematic and PCB in the project folder are paired; therefore the same reference cannot be used twice in the same project.

You can also check what has been modified in the designs.



*What you can do with the project

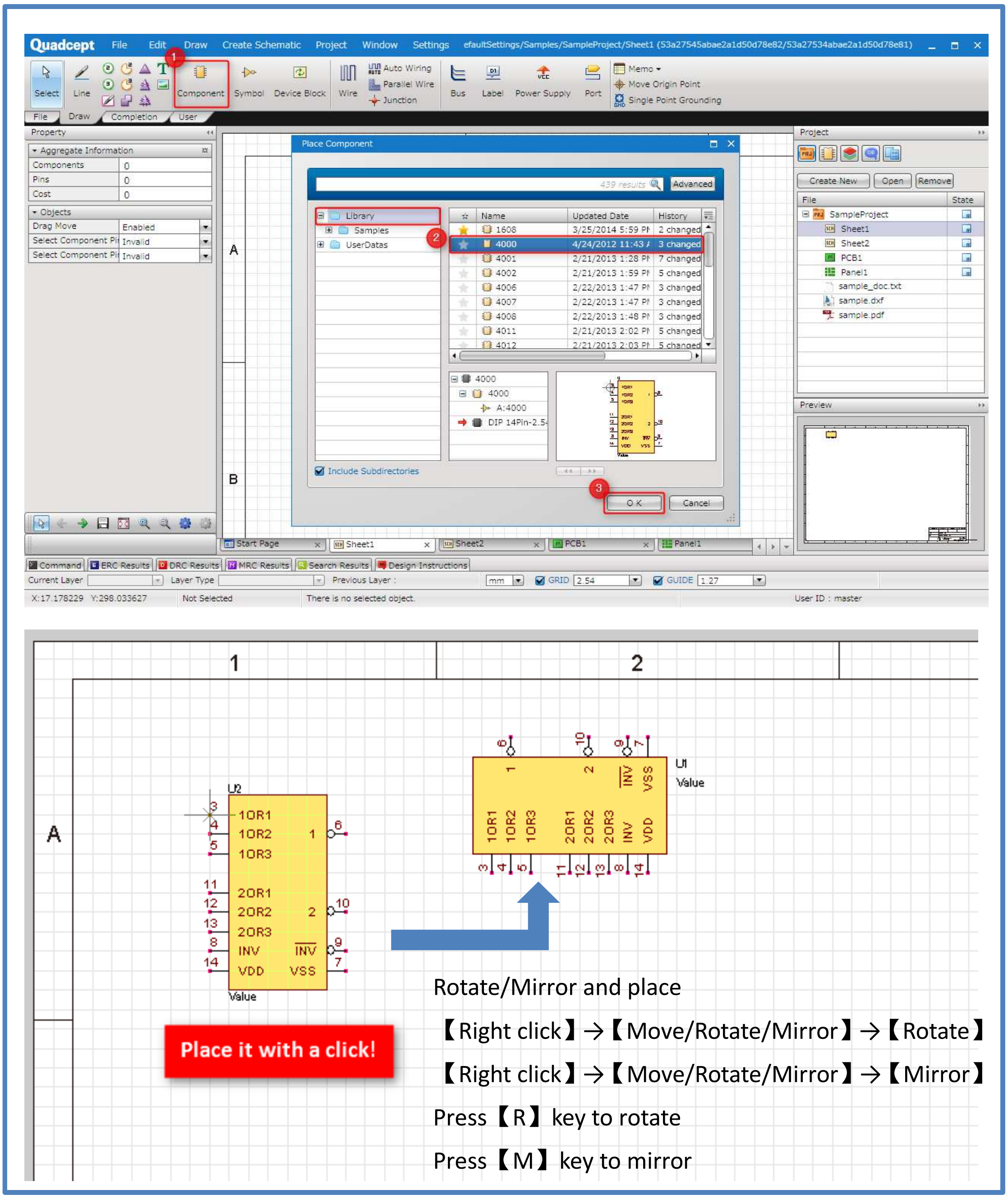
1. Managingmultiple schematics and PCB data (Netlist output)
2. Cross Probe of schematic and PCB (Annotation and transfer)
3. Storing related files (Netlist, BOM, etc.z)
4. File management and print per project

■Step 2. Placing Component

Place components on the schematic in the following way.

1. Ribbon <Draw> → Select【Component】
2. Select the directory and choose the component you want to place
3. Select the component with【double click】or click【OK】
4. The component is attached to the mouse pointer. Click anywhere on the sheet to place it. You can continuously place the same component with a click, or press【ESC】button to exit

   *[Power] is also placed in the same manner.



■Step 3-a. Wiring

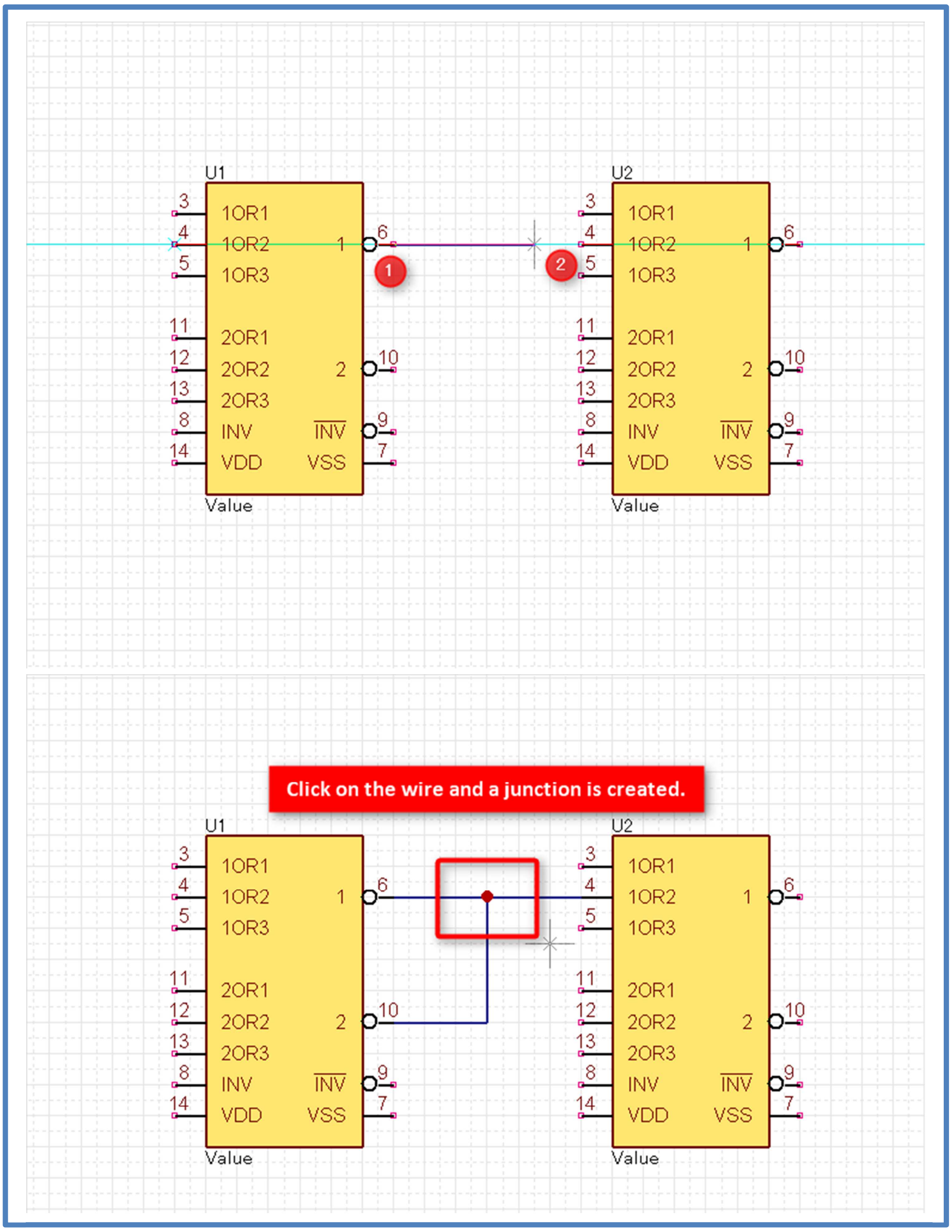
Connect pins of components. This task is important as the connection information will be output as the netlist.

1. Ribbon <Draw> → Select【Wire】
2. Click the pinpoint to start connecting
3. Click the pinpoint to finish connecting

   *The pinpoints will disappear when pinpoints are connected correctly.

   *You can start wiring at places other than pinpoints.

   Double click to exit the wiring mode when you don't connect the wire to a pinpoint.

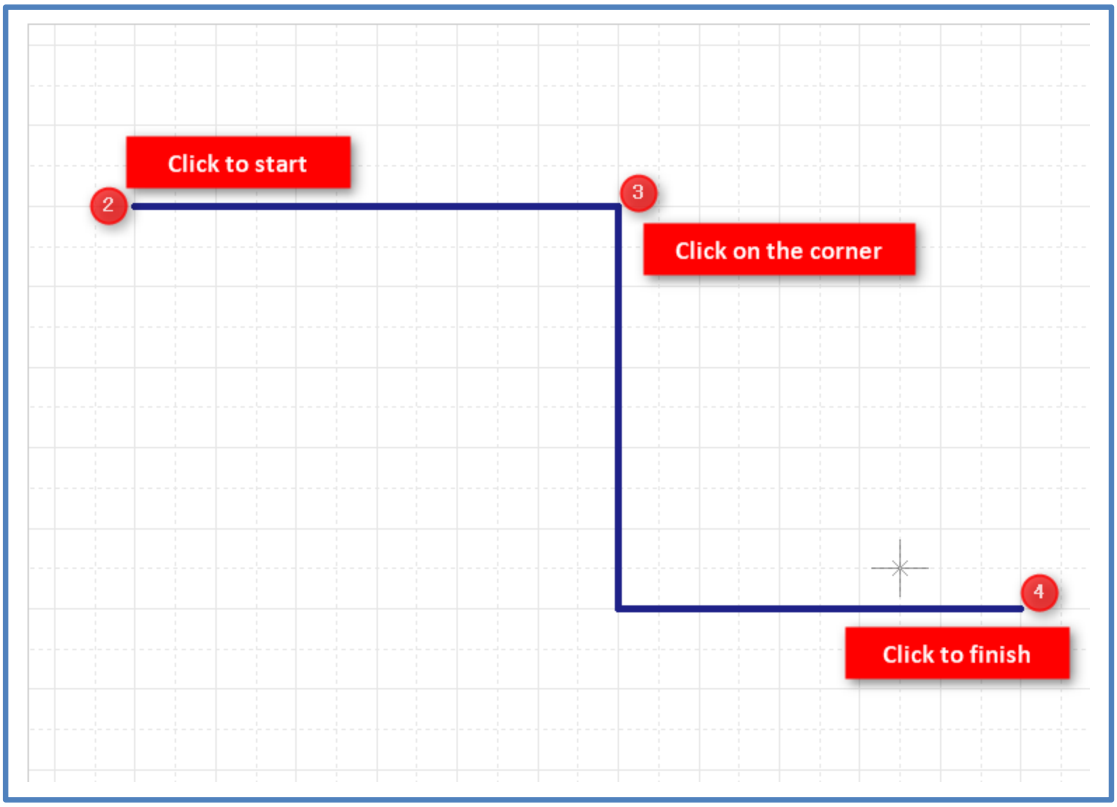


How to draw wire/line

This section describes various ways to draw [Line] to create shape or [Wire].

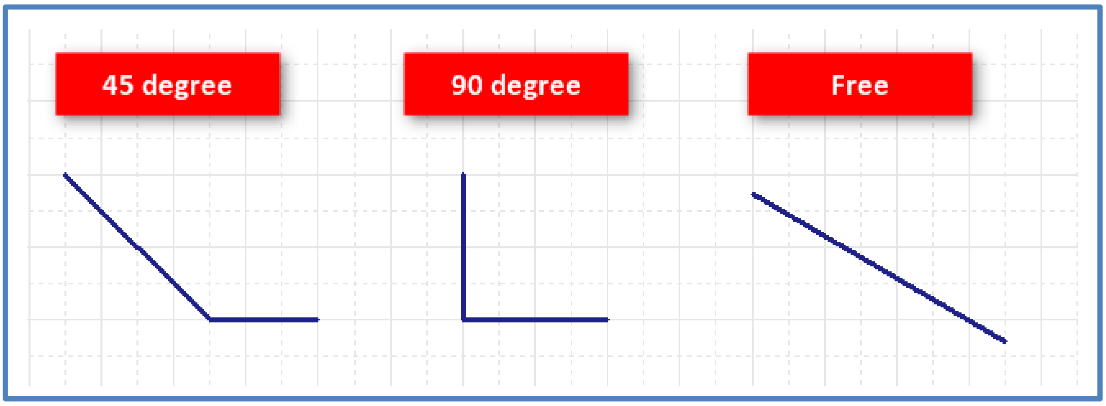
Drawing a corner

1. Ribbon <Draw> → Select【Wire】or【Line】
2. Click where you want to start
3. Click on the corner
4. Click where you want to finish (Double click when in the Line mode)



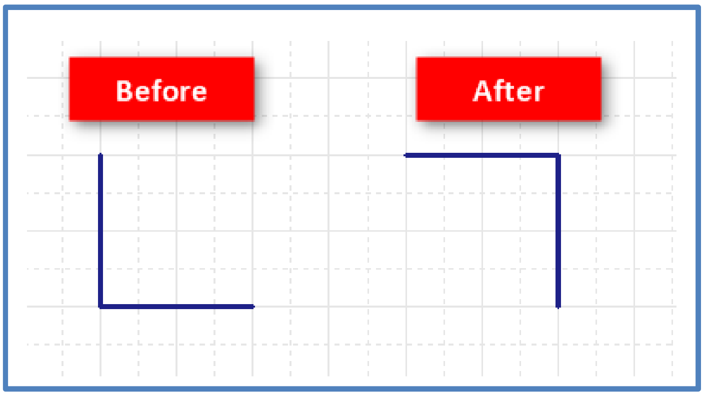
How to change the bending angle

【Right click】→ Select【Change the Bending Angle】or press【S】key
*You can check the bending angle in the [Property] window.



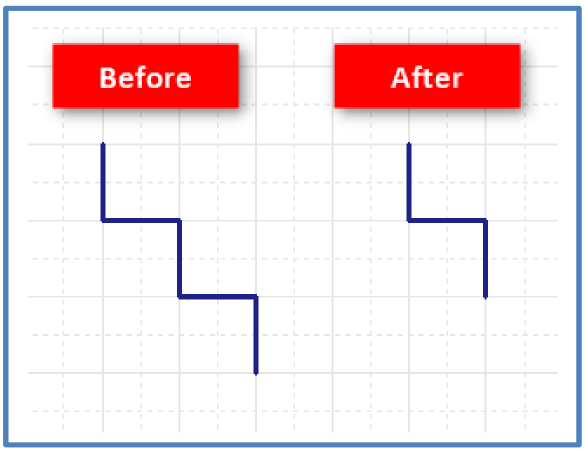
How to switch angle

【Right click】→ Select【Switch Angle】or press【X】key



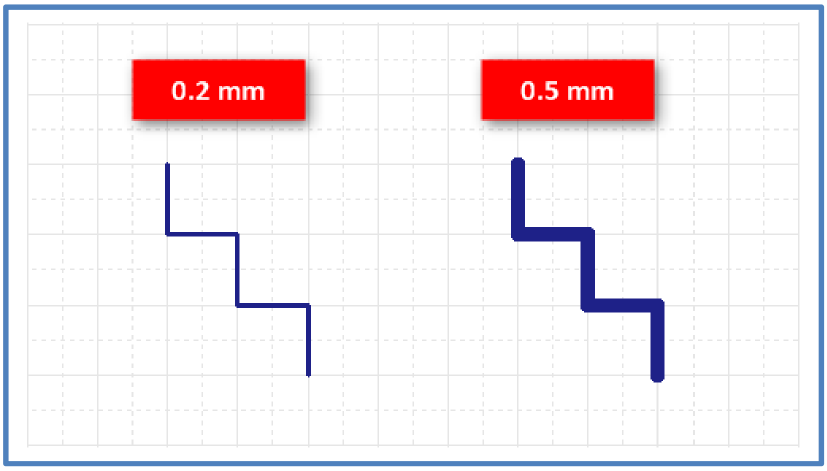
Return to a previous state

【Right click】→ Select【Pushback】or press【Back Space】key



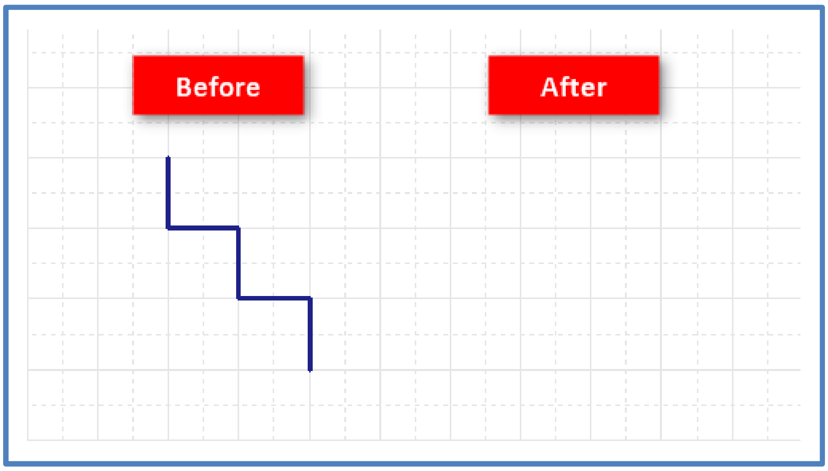
Edit line width

【Right click】→ Select【Edit Line Width】or press【W】key and enter the desired line width



Cancel drawing

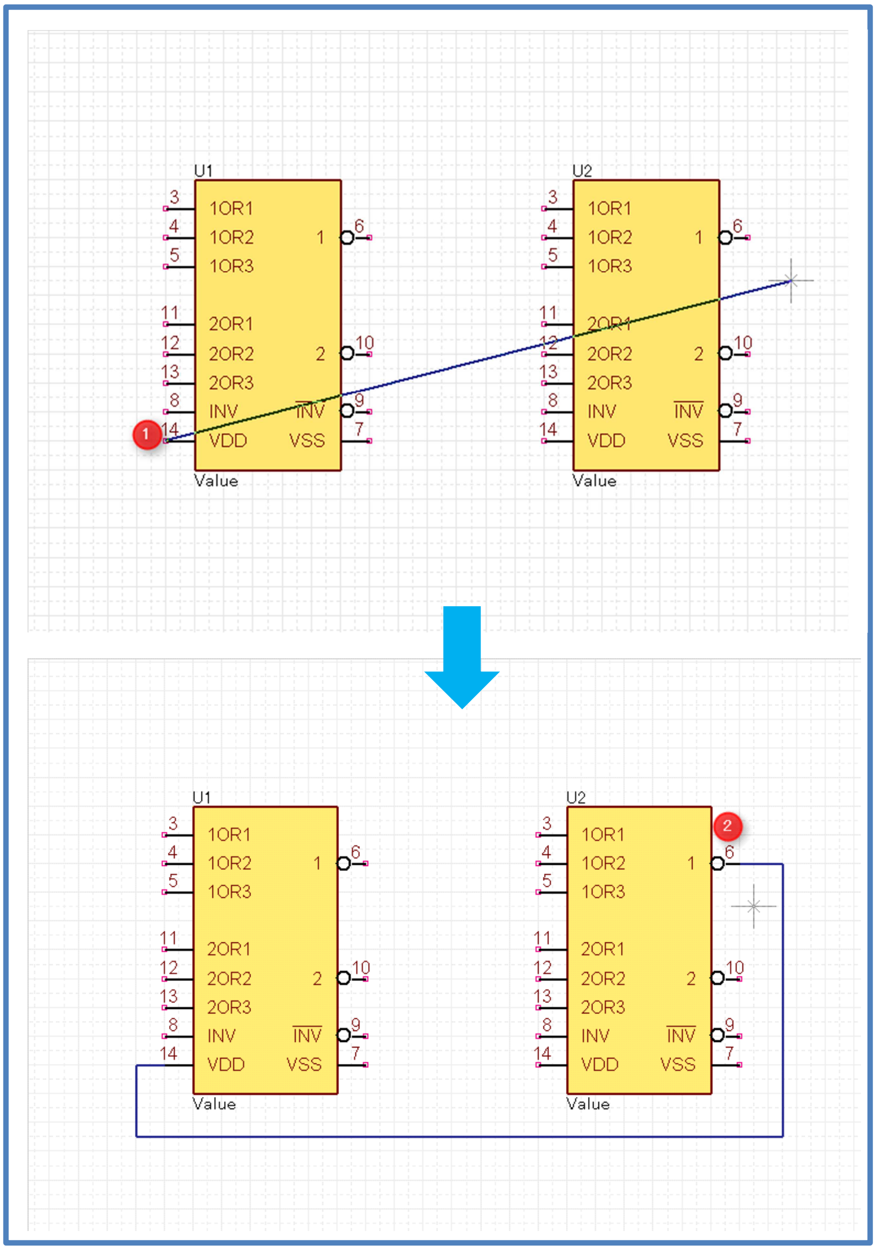
【Right click】→【Cancel】 or press【Escape】key



■Step 3-b. Auto Wiring

By specifying a start and finish point, it connects the two points automatically getting around obstacles such as other components and wires.

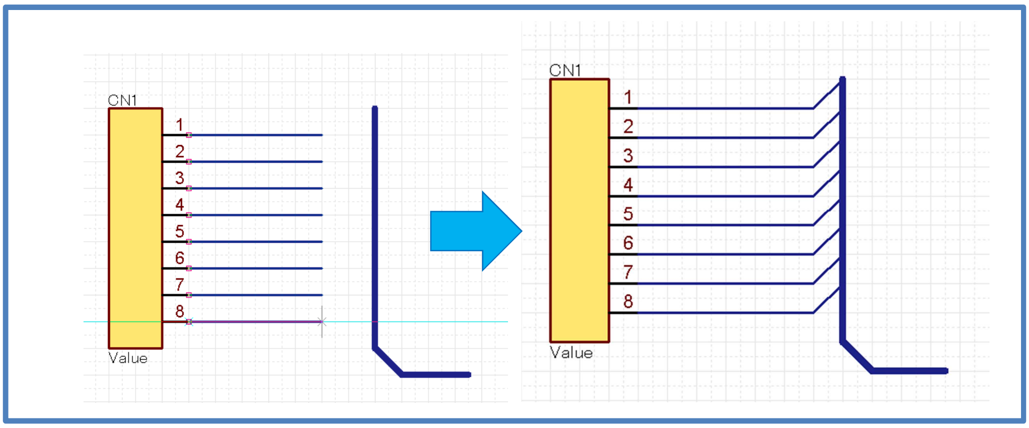
1. Ribbon <Draw> → Select【Auto Wiring】
2. Start wiring by clicking the start pinpoint
3. Finish wiring by clicking the end pinpoint



■**Step 3-c. Bus/Parallel Wiring**

Connect multiple wires in a bundle.

1. Ribbon <Draw> → Select【Bus】to create bus routing
2. Ribbon <Draw> → Select【Parallel Wiring】and select multiple pinpoints
3. by mouse drag to start.
4. Connect multiple wires to the bus routing
5. When the wires are connected to the bus routing, rippers are automatically created.

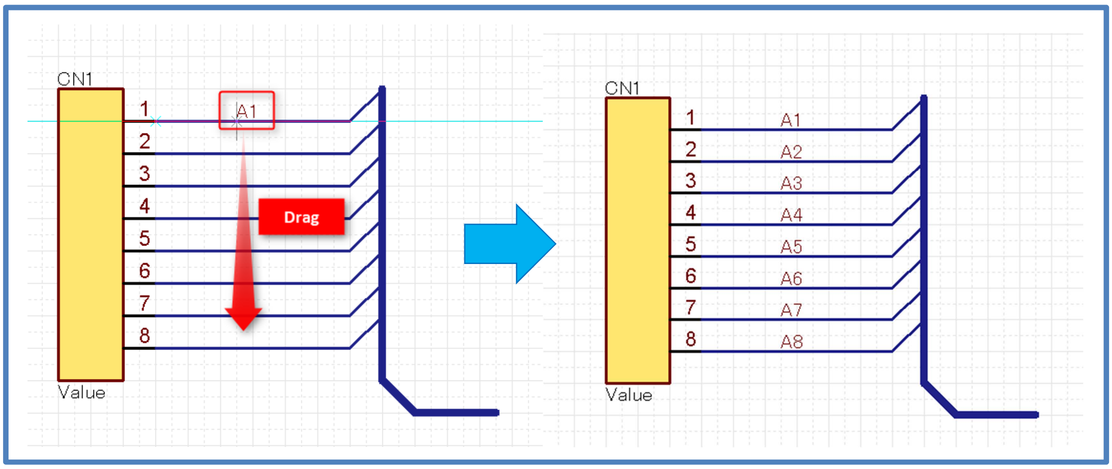


■Step 3-d. Placing Label

Label the wires to identify where they are connected to.

1. Ribbon <Draw> → Select【Label】
2. Enter the label name and click【OK】
3. Click on the wire you want to place the label

   *By mouse drag, you can sequentially number the wires incremented the last number of the label name by one.



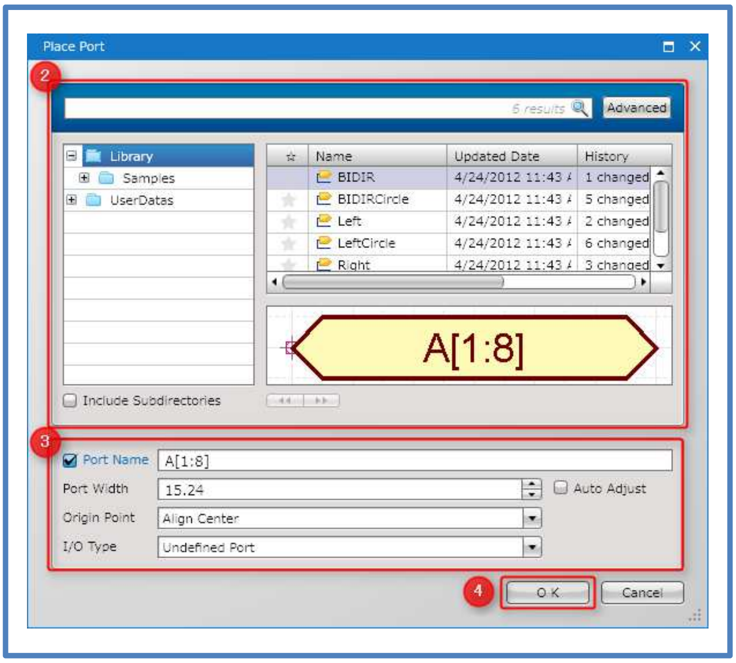
■Step 3-e. Placing Port

You can connect a wire to a pin on another sheet by placing the port on the schematic.

Port is like a warp point to connect the wire when using multiple schematics.

*We have also prepared various default port shapes. You can save time using those ready-made ports as you don't need to create a new one from scratch.

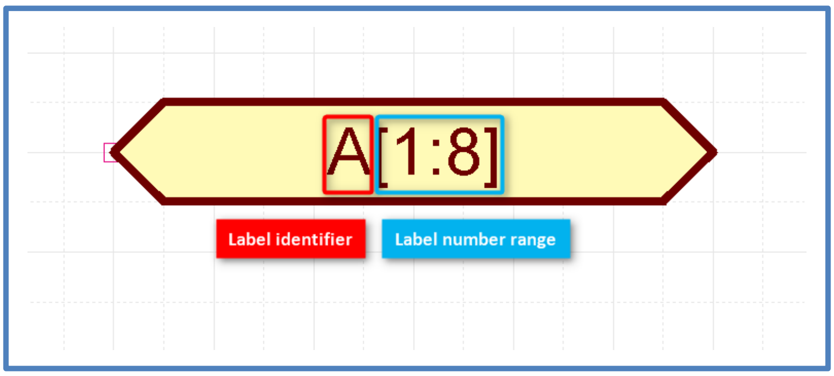
1. Ribbon <Draw> → Select【Port】
2. Select the directory in which the port is stored and choose the port
3. Specify [Port Name], [Port Width], [Origin Point] and [I/O Type]
4. 【Double click】 or click【OK】
5. The port is attached to the mouse pointer. Click anywhere on the sheet to place it



*Setting Bus Rule

Bus rule consists of label identifier, label number range and operator.

This will enable you to connect the bus wire to another sheet.

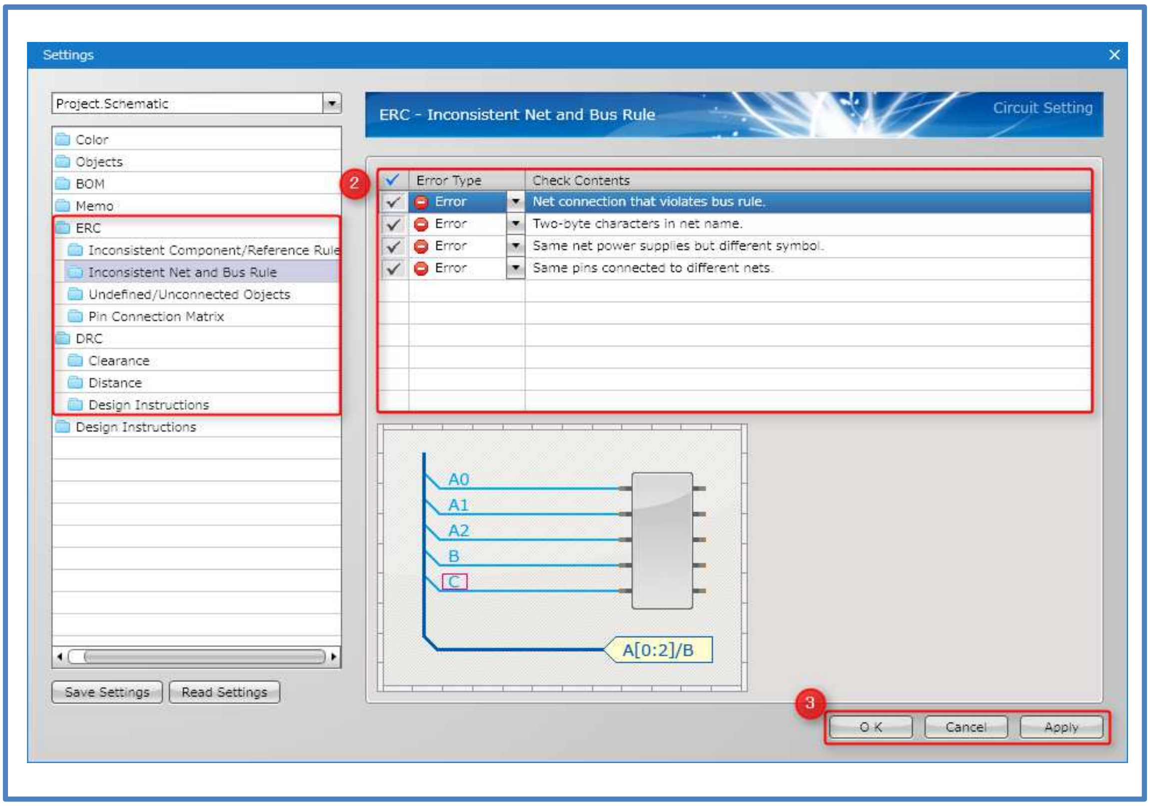


■**Step 4. Verifying Schematics (ERC/DRC)**

Verify schematics before output.

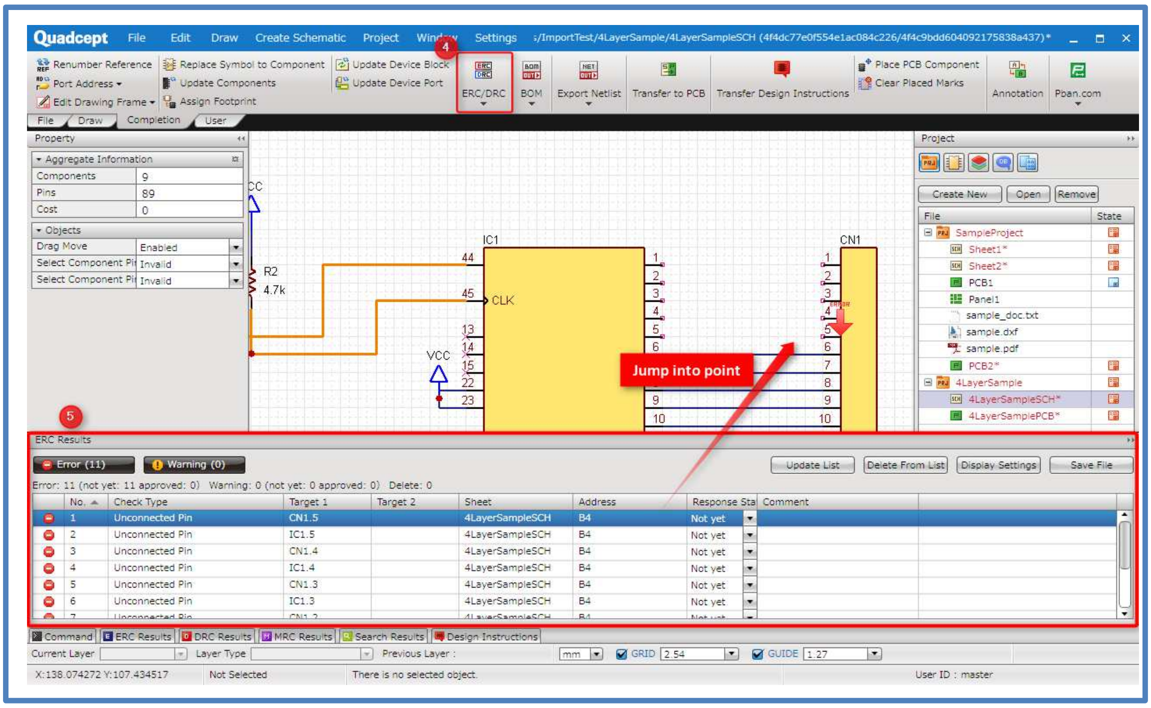
1. Ribbon <Completion> → Select【ERC/DRC】→【ERC/DRC Settings】
2. Set each item for the rule checks
3. Select【Apply】and click【OK】
4. Ribbon < Completion > → Select【ERC/DRC】→【Run ERC/DRC】
5. Check the results of [ERC]/[DRC]. When there are errors, correct them and repeat the above procedure until all errors are corrected.

ERC/DRC settings



ERC/DRC Results

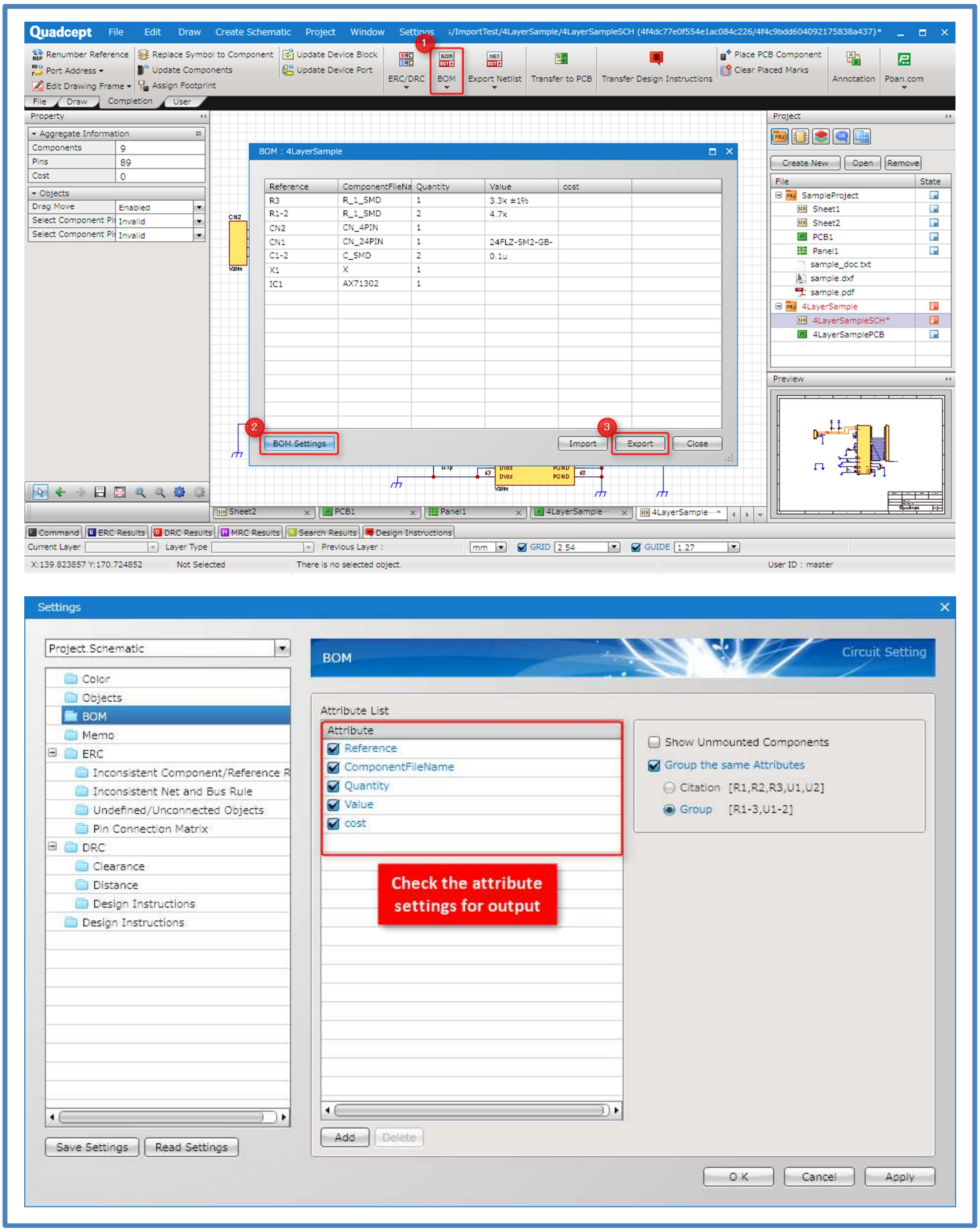
ERC/DRC correct error points after verifying.（double click to jump into that point）



■Step 5. Export BOM

You can export the component information used in the schematic and create the BOM which contains the component and attribute list.

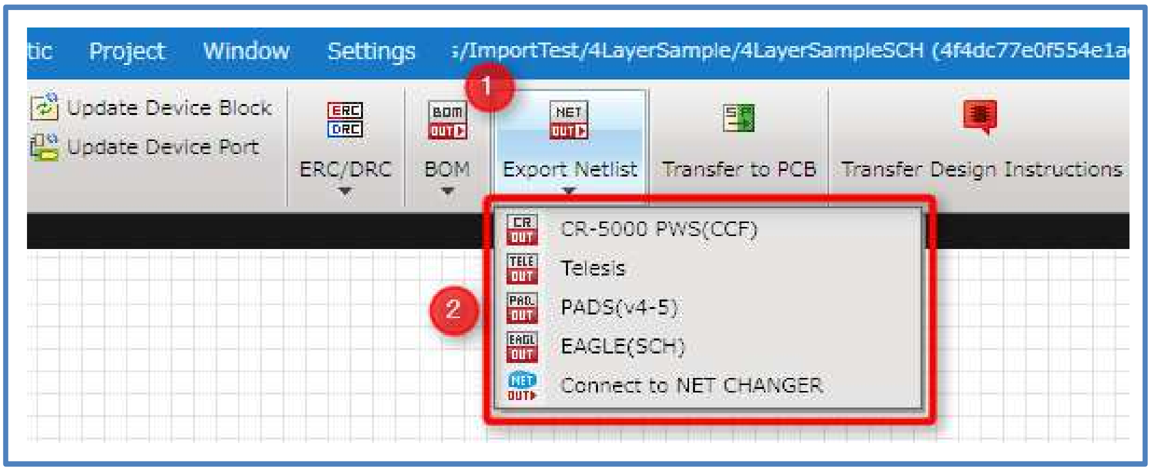
1. Ribbon < Completion > → Select【BOM】
2. Select【BOM Settings】and set what you want to output
3. Select【Export】and save dialog appear
4. Select where to save, name the file and click【OK】



■**Step 6. Export Netlist**

Output the netlist (connection information) from the schematic.

1. Ribbon < Completion > → Select【Export Netlist】
2. Select output format
   (Currently available formats are CR-5000 PWS(CCF), Telesis, PADS(v4-5) ,EAGLE and Connect to NET CHANGER)
3. Select where to save, name the file and click【Save】



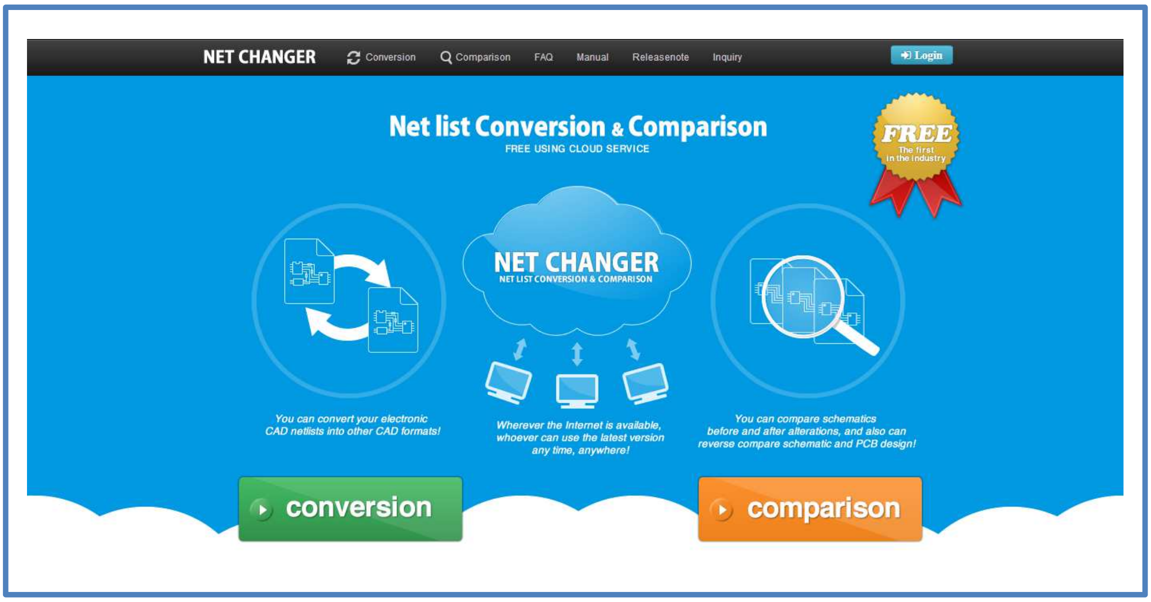
***Connect to NET CHANGER**

Make the netlist by connecting to NET CHANGER. Export of the netlist files in a compatible format with NET CHANGER is possible.

About NET CHANGER

NET CHANGER is the tool that Quadcept Inc. provides free of charge. It is easy to use and allows you to convert or compare netlists on the web!

https://netlist.quadcept.com/

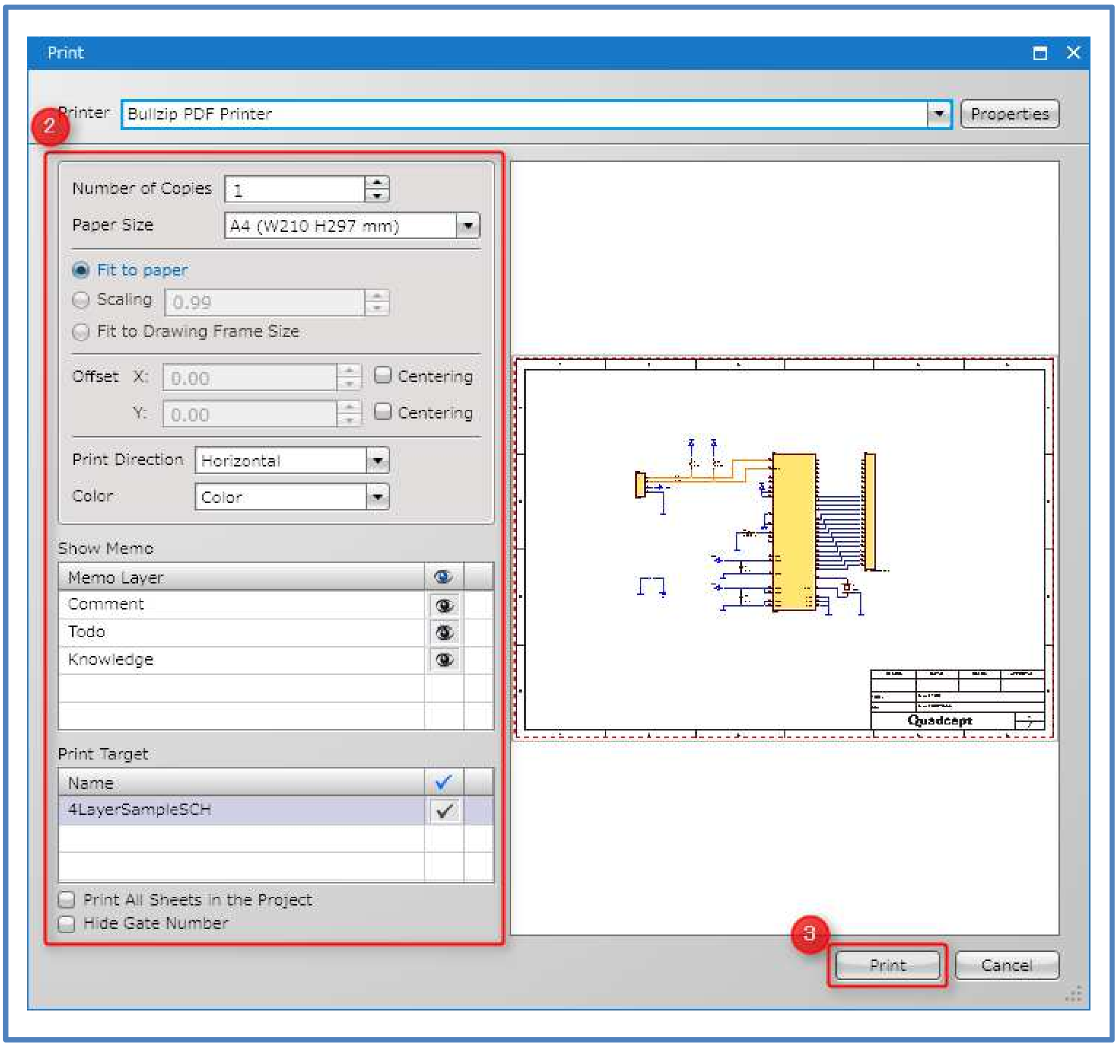


■Step 7.　Print Schematic

Print the schematic.

1. Ribbon <File> → Select【Print】
2. Check the print settings in the [Print] dialog
3. Click【Print】

   There are several ways to print out. You can print the design area as is (normal print) or print per project as well as batch print various output/print.



Chapter 3 PCB Designer

- Creating Shape of PCB Component (Footprint)
- Creating Shape of PCB Component (Footprint)
- Creating Shape of PCB Component (IPC Footprint)
- Registering Component
- Creating PCB

PCB Design Flow

This flow describes the phases of the PCB design.

The objective is to output CAM (gerber data).

■PCB Design Flow

1. Create Component (Footprint + Registering Component)

First, prepare component data for the design.

To register a component, it needs to have a pad and shape.

2. Annotation

Quadcept allows you to annotate from Schematic to PCB.

3. Various Settings (Layer Settings, Design Rule Settings)

Various settings are required to design.

4. Input Board Outline and Place Component

Place components in the board outline.

It is asked to place components in a way the subsequent routing task will be handled efficiently.

5. Routing and Creating Plane

Drawing the route and creating plane.

6．DRC/MRC

Perform DRC（Design Rule Check）and MRC（Manufacturing Rule Check）

7．CAM Output

Export Gerber data or desired data from the completed PCB.

Creating Shape of PCB Component (Footprint)

Create component's footprint to place on the PCB.

1. Open a new sheet to create a footprint

   Ribbon <File> → 【Create New】 → Select 【Footprint】

2. Place a pad

   Ribbon <Draw> → Select 【Pad】 to place

3. Input silk

   Press [L] key to set the current layer as [Silk] (or【Right click】→ Select【Change Layer】and set as [Silk]

   Ribbon <Draw> → Use 【Line】 or 【Rectangle】 or 【Circle】

4. Adjust the Reference position

   Move the reference [U] by mouse drag

5. Set the origin point

   Ribbon <Draw> → Select【Move Origin Point】and click on the desire point or while selecting the object 【Right click】 → 【Move Origin Point to Center】

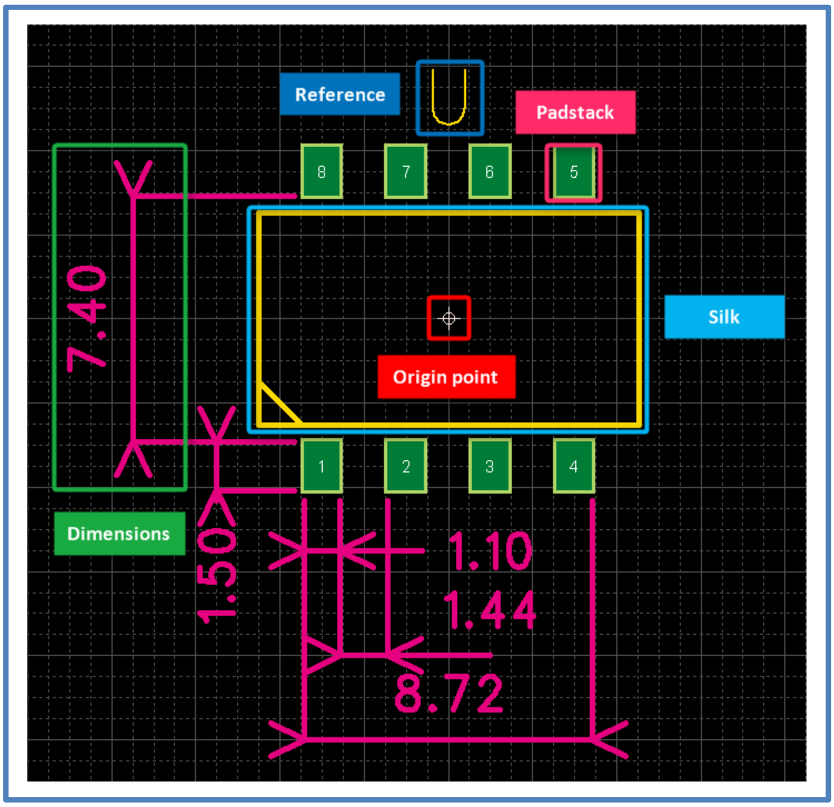
6. Input dimensions

   【Create PCB】 → Select 【Create Component Dimension Automatically】

7. Save

   Ribbon <File> → Select 【Save】 or 【Save as】

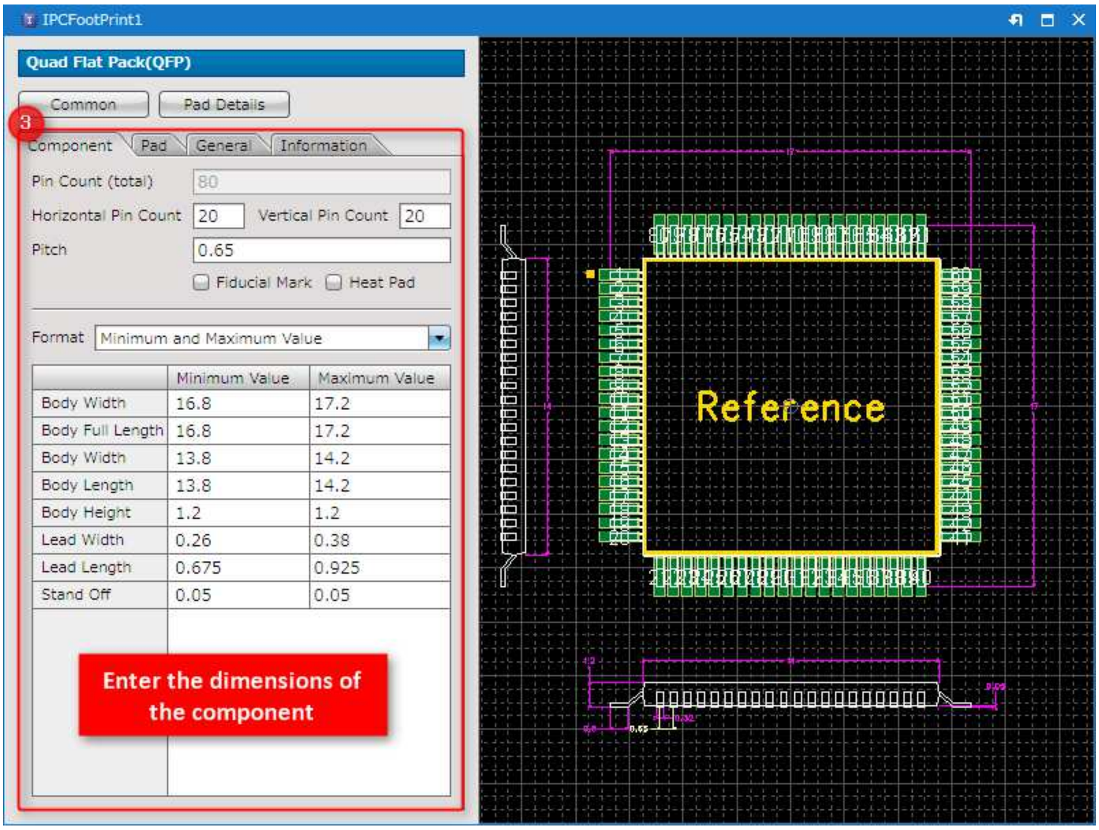
   Select a directory and save it with a name.



Creating Shape of PCB Component (IPC Footprint)

Automatically create component's footprint (shape) according to IPC standard.

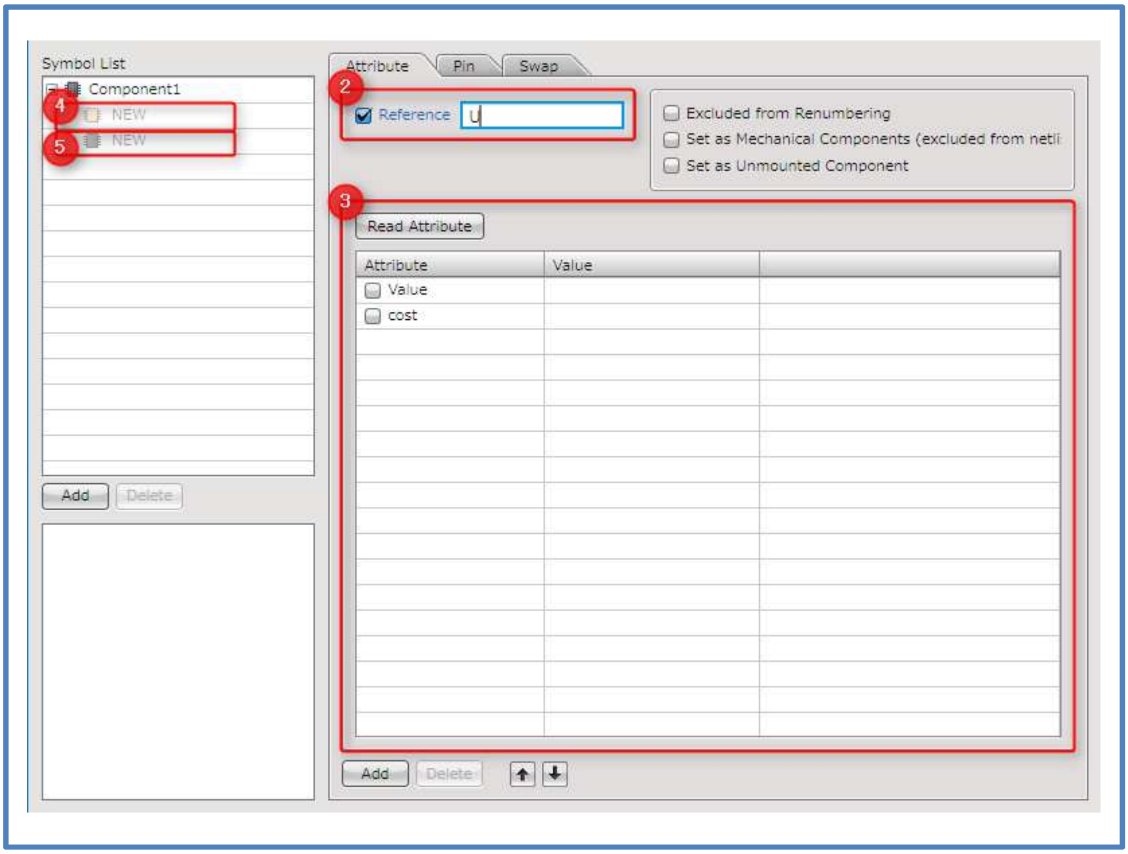
1. Open a new sheet to create IPC footprint
   Ribbon <File> → 【Create New】 → Select【IPC Footprint】
2. Select [Category] and click【OK】
3. Set various settings
   Check your settings with the preview
4. Save
   Ribbon <File> → Select【Save】 or【Save as】
   Select a directory and save it with a name.



Registering Component

Create component to place on design drawing (P.20), and register shapes of PCB component (footprint).

1. Open a new sheet to create a component
   Ribbon <File> → 【Create New】 → Select 【Component】
2. Set Reference
3. Input attributes
4. Register Symbol (shape of schematic component)
   Double click and open the [Select Component] dialog and select a symbol
5. Register Footprint (shape of PCB component)
   Double click and open [Select Component] dialog and select a symbol
6. Save
   Ribbon <File> → Select 【Save】 or 【Save as】
   Select a directory and save it with a name.



Creating PCB

This section describes the flow of creating PCB.

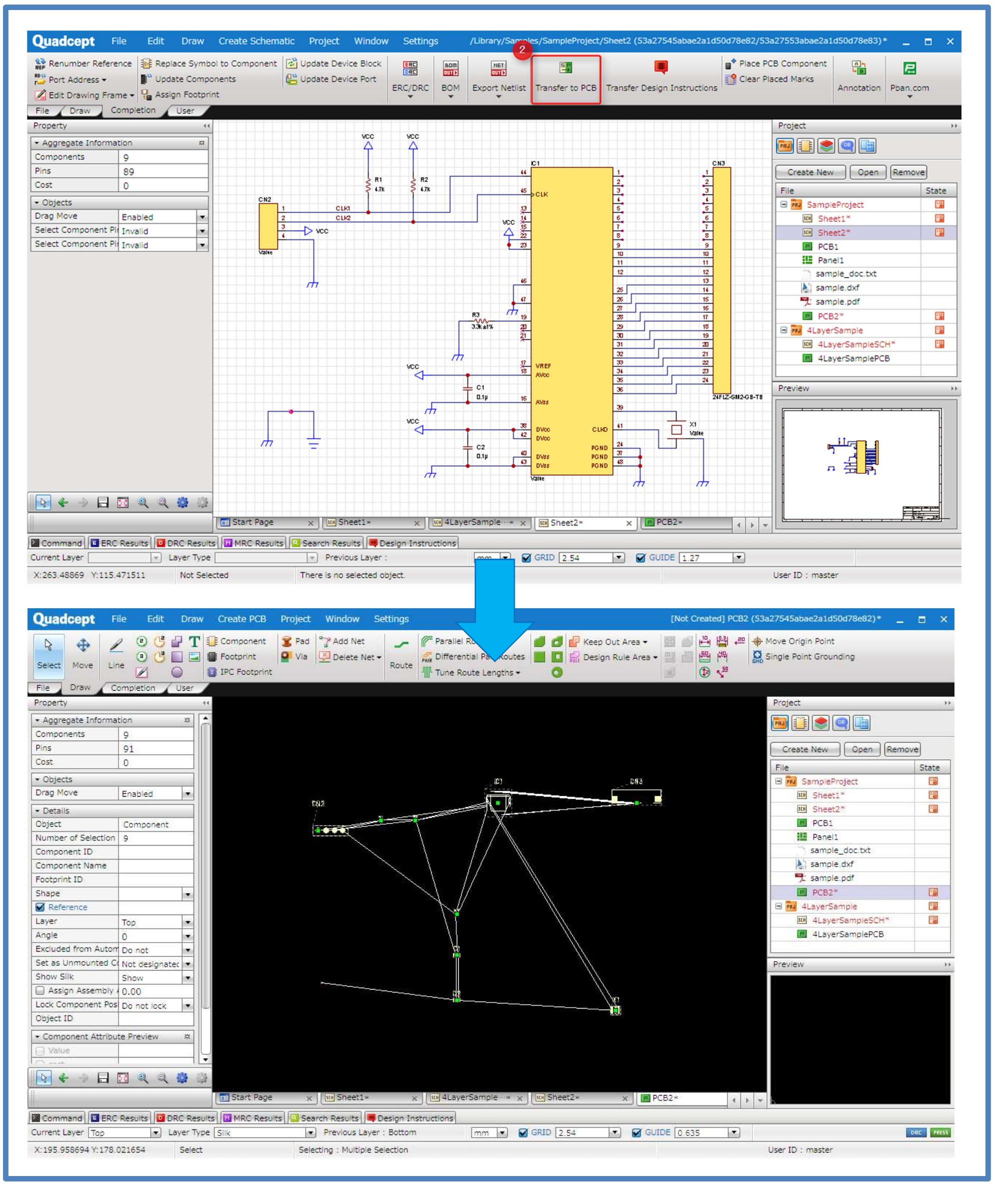
■Step 1. Annotation

Quadcept allows you to cross probe from Schematic to PCB.

Schematic netlists created with other CADs can also be read by the Quadcept PCB Designer.

1-a. When designed the schematic on Quadcept

1. Open the schematic you want to transfer
2. Ribbon < Completion > → Select【Transfer to PCB】
3. The PCB sheet is automatically generated. PCB components are placed based on the schematic and connected automatically based on the netlist information.



1-b. When designed the schematic with other CAD

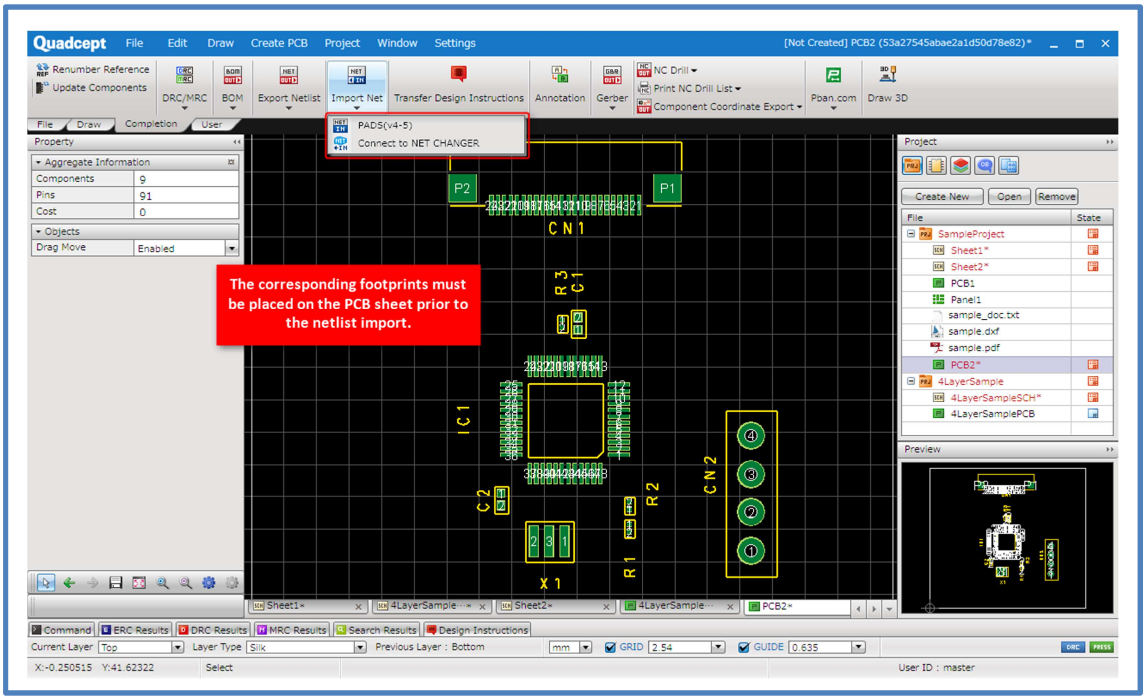
The netlist to be read must be in Pads format. The footprints must be placed on the PCB design sheet prior to the netlist import.

1. Ribbon <File> → Select【Create New】→【PCB Project】
2. Ribbon <Draw> → Select the corresponding footprint shapes from【Component】,【Footprint】or【IPC】and place them

   *Place the components in accordance with the netlist information to match the Reference and pin number.
3. Ribbon <Completion> → Select 【Import Net】→【PADS(v4-5)】or【Connect to NET CHANGER】
4. Select the netlist in Pads format and click【OK】

   The netlist information will be read to the placed components.

   When an error occurs, the error message will appear. Resolve the issue or if it is not a problem, click【Continue】to complete the import.



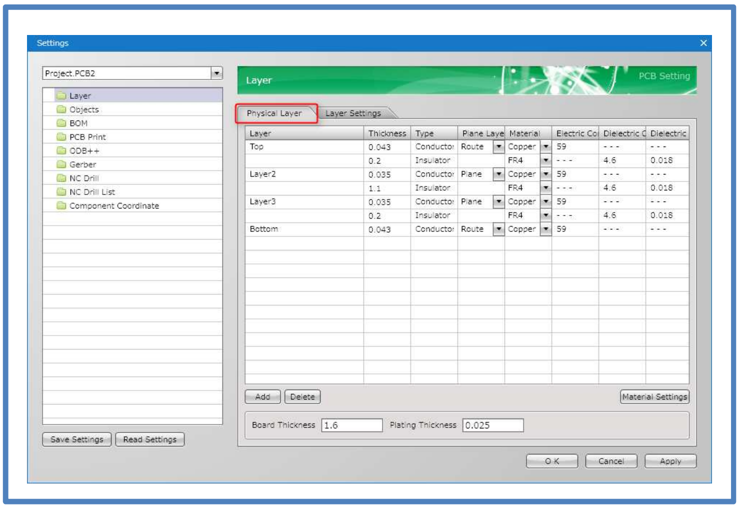
■Step 2. Layer Settings

First, board layers need to be set before proceeding with PCB design.

1. Open a PCB sheet, and select【Settings】→【Settings】→【Layer】
2. Set each layer in the【Physical Layer】and【Layer Settings】and Click【Apply】→【OK】

Physical Layer Setting Screen

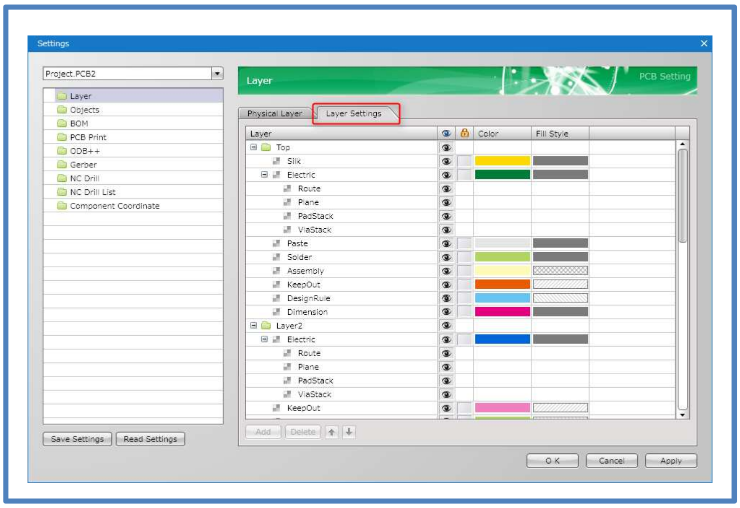
Set the physical information for the impedance calculation. The default setting is four layers.



Layer Setting Screen

Object layers are available.

You can also add a customized layer, set colors as well as set show/hide.



Description of Layers

■Silk

Silk object layer for non-electrical objects such as reference, line, character etc.

■Electric

Copper layer for route, plane, pad land, via land etc. Electrical objects are automatically placed on this [Electric layer].

■Route

Route object layer in the Electric layer.

■Copper

Plane object layer in the Electric layer.

■PadStack

Pad land object layer in the Electric layer.

■ViaStack

Via land object layer in the Electric layer.

■Paste

Metal mask and paste mask layer. When you enter the metal mask data for the SMD pad, the object is created automatically on this layer. Objects such as line can also be placed on this layer.

■Solder

Solder resist and solder mask layer

When you enter the solder resist data for the pad or via, the object is created automatically on this layer. Objects such as line can also be placed on this layer.

■Assembly

Component outline layer

When you use the IPC footprint, the outline is created automatically on this layer. When you create a footprint, its outline can also be generated.

■KeepOut

Keep Out Area layer

When you set the Keep Out Area on other layers, it is created on the KeepOut layer in the individual layer. When you apply the Keep Out Area to all layers, it will be created on the KeepOut layer in the Other.

■DesignRule

Design Rule Area layer

When you set the Design Rule Area on other layers, it is created on the DesignRule layer in the individual layer. When you apply the Design Rule Area to all layers, it will be created on the DesignRule layer in the Other.

■Dimension

Dimension layer

When you use [Create Component Dimension Automatically], the dimension is created automatically on the Dimension layer as default.When you use [Create Board Outline Dimension Automatically], the dimension is created automatically on the Dimension layer in the Other as default.

■Board

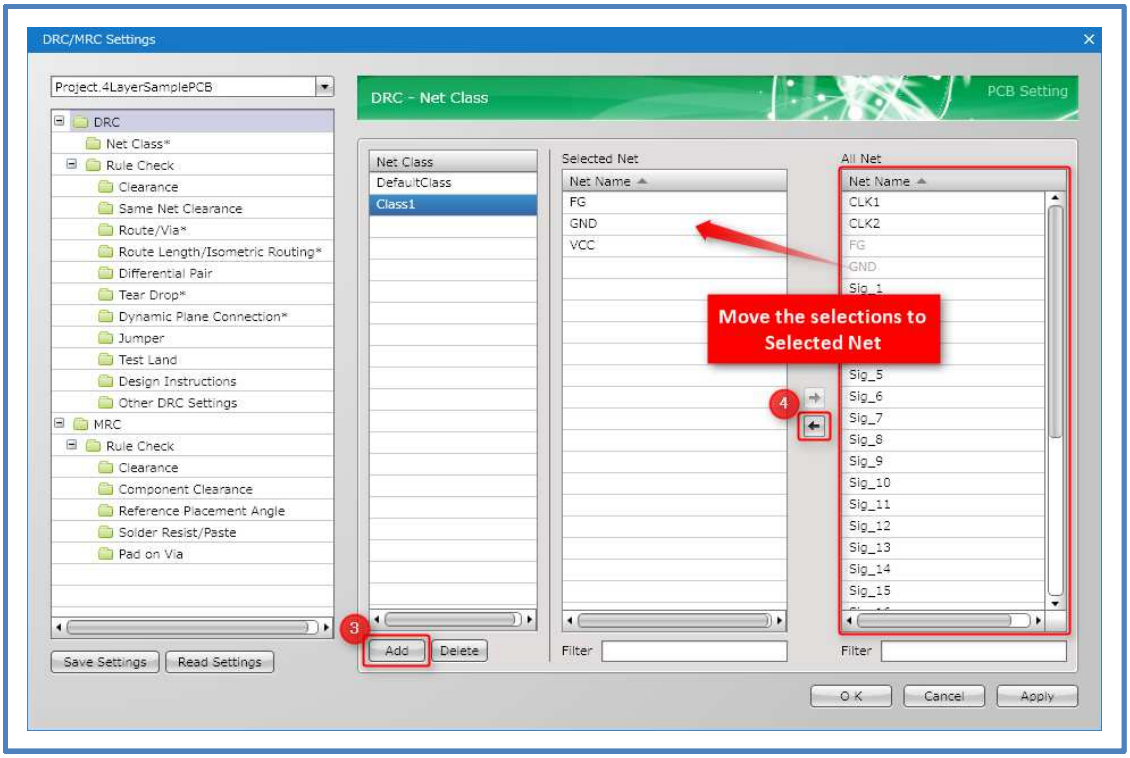
Board Outline layer. Draw closed graphics with line or arc.

■**Step 3. Design Rules**

You can set the clearance, route width or other design rules to the net class (a specific group of nets you selected).

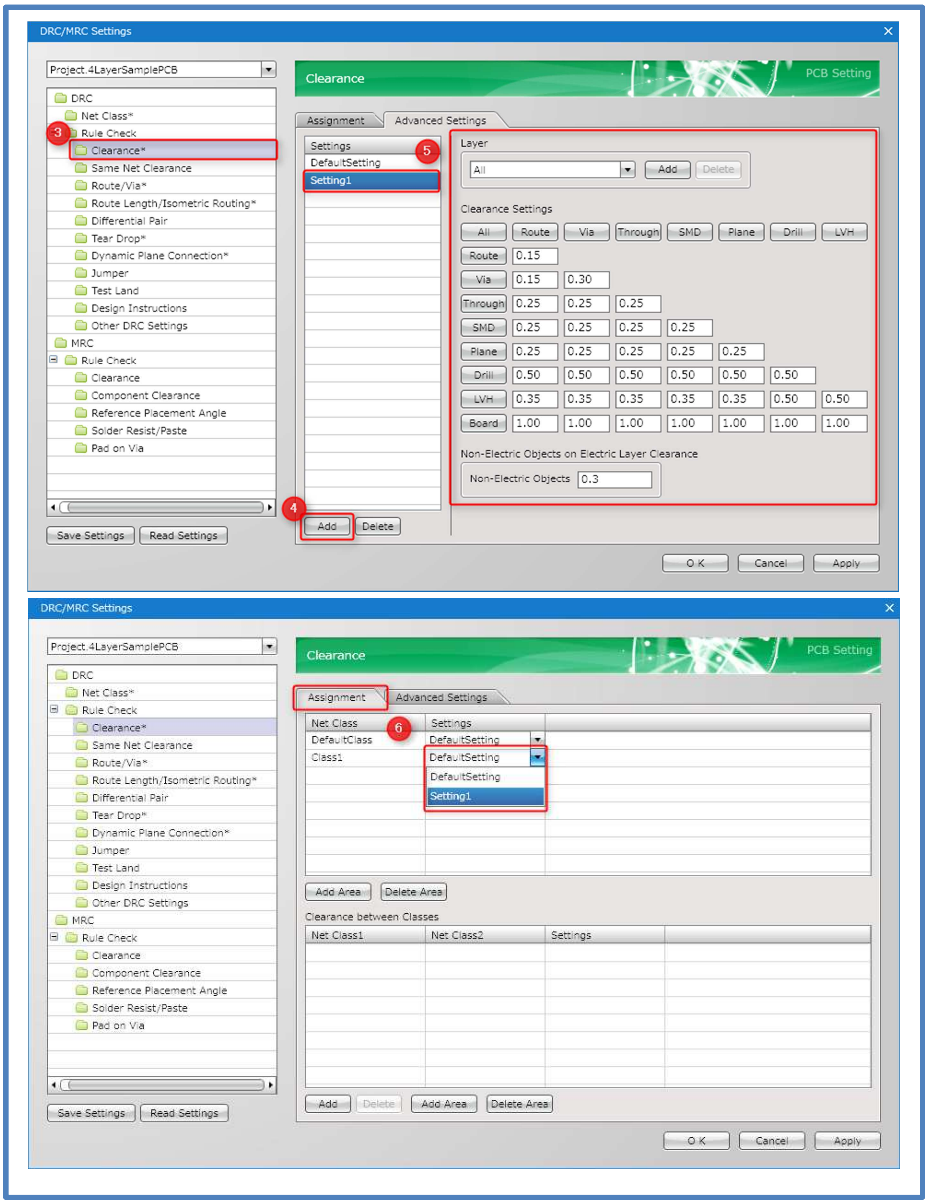
Net Class Settings

1. Open the PCB sheet you want to set
2. Ribbon < Completion > → Select【DRC/MRC Settings】→【Net Class】
3. Click【Add】at the bottom of [Net Class] to add a "Class"
   *At first, all nets are included to "Default Class".
4. Choose nets you want to add to the "Class" from "All Net"
   Click【←】to select [multiple selections available]
5. When you finish selecting, click【Apply】→【OK】



Clearance Settings

1. Open the PCB sheet you want to set
2. Ribbon <Complete> → Select【DRC/MRC】→【DRC/MRC Settings】
3. Select【Clearance】from the list on the left
4. Select <Advance Settings> tab, and click【Add】at the bottom of 【Settings】to add a clearance. "Setting1" is created.
   *At first, all belong to the "DefaultSetting".
5. Select "Setting1" from the list, and set [Layer] and enter the clearance value in the [Clearance Settings] matrix.
6. Select <Assignment> tab → Assign the clearance setting "Setting1" by selecting from the [Setting]
   The clearance "Setting1" is set.



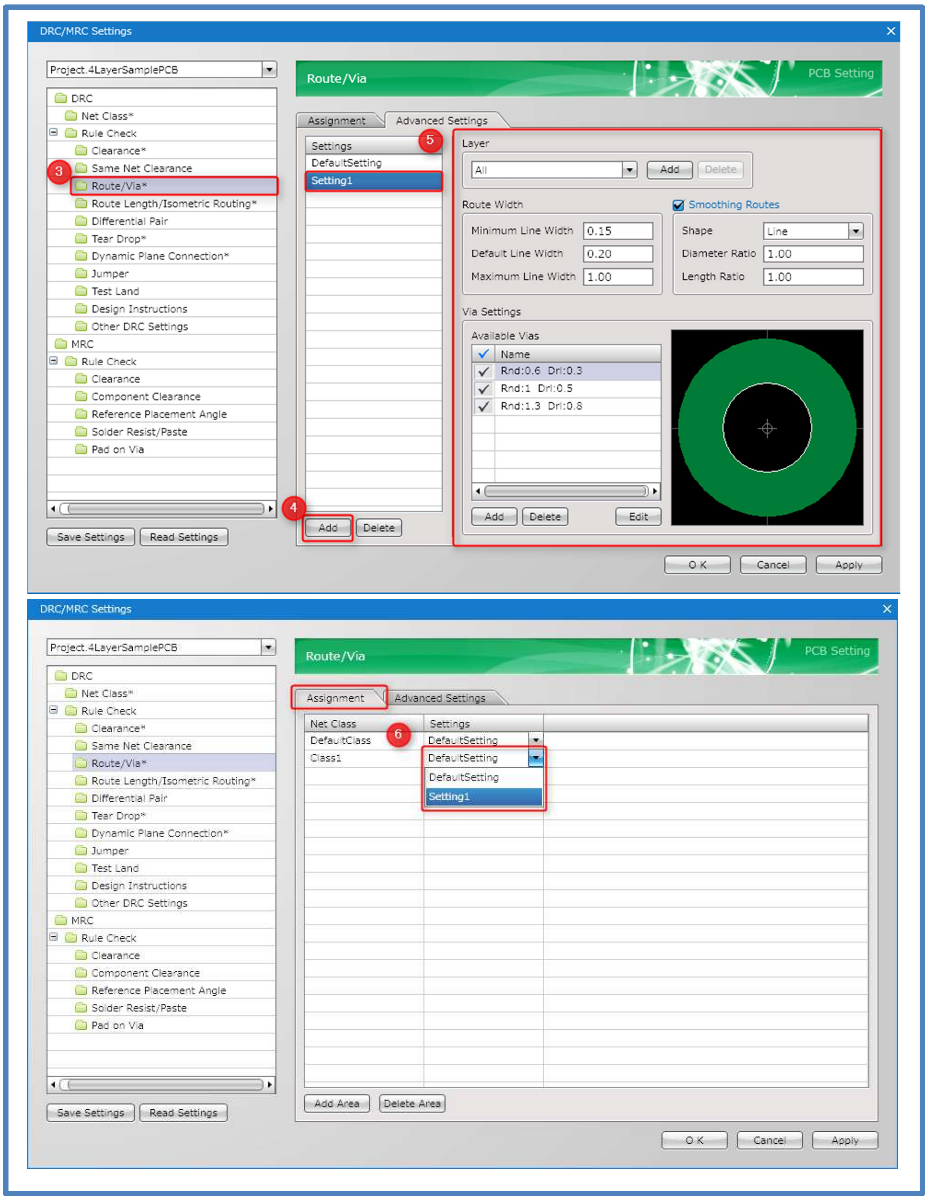
Route Settings

1. Open the PCB sheet you want to set
2. Ribbon <Complete> → Select【DRC/MRC】→【DRC/MRC Settings】
3. Select【Route/Via】from the list on the left
4. Select <Advance Settings> tab, and click【Add】at the bottom of【Settings】to add a route/via.

   "Setting1" is created.

   *At first, all belong to "DefaultSetting".
5. Select the "Setting1" from the list, set [Layer], [Route Width] and [Available Vias]
6. Select <Assignment> tab → Assign the net class "Setting1" by selecting from the [Setting]

   The Via which uses "Setting1" is set.



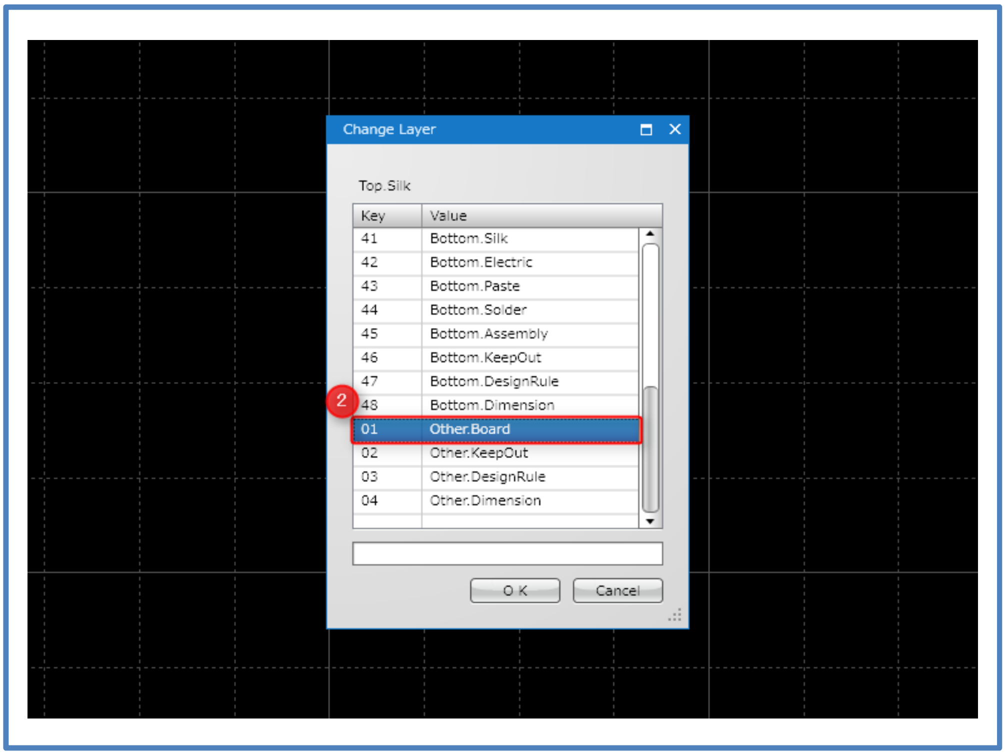
■Step 4. Drawing Board Outline

Quadcept recognize the line drawn on the [Board (type)] of [Other (layer)] as board outline.

Besides drawing a line, board outlines in DXF file or IDF file can be imported.

4-a. Drawing a board outline directly

1. Open the PCB sheet where you want to draw a board outline.
2. Press [L] key and set the current sheet as【Other.Board】 (or【Right click】→【Edit Layer】→set it to [Other.Board])
3. Ribbon <Draw> → Use【Line】or【Rectangle】or【Circle】to draw the line



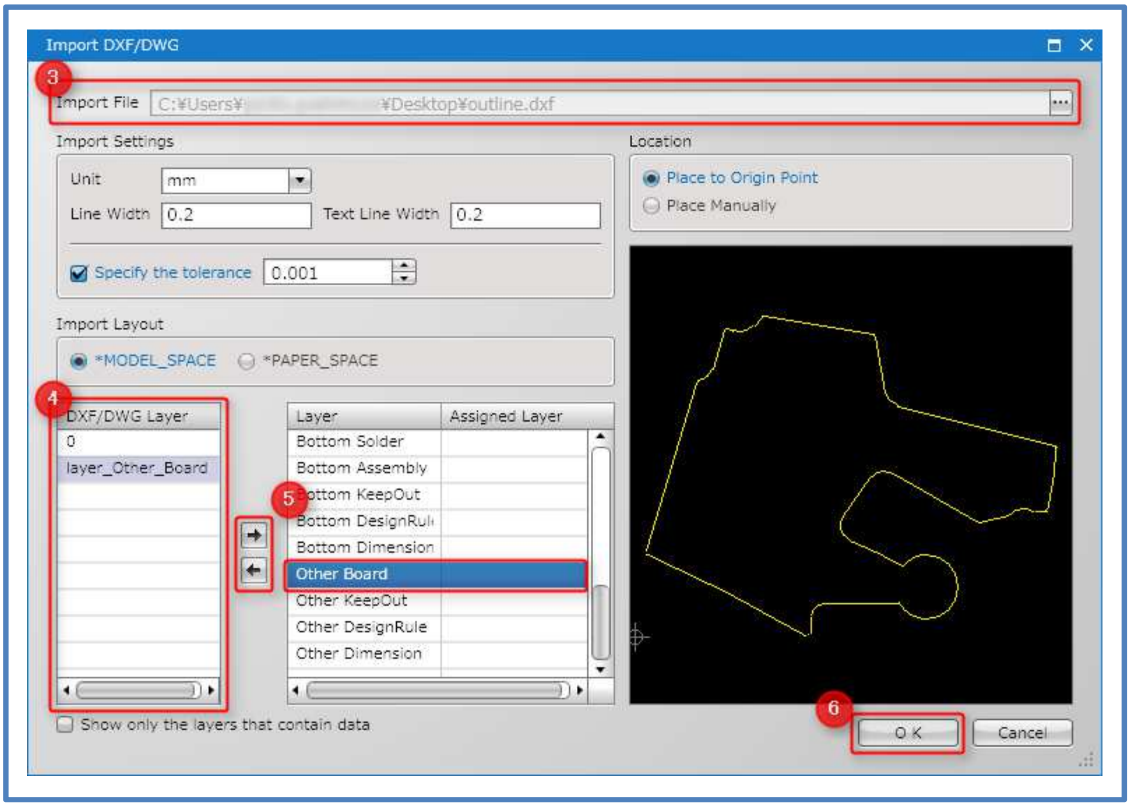
4-b. Reading DXF/DWG data for a board outline

1. Open the PCB sheet where you want to draw the board outline.
2. Read DXF/DWG data

   Ribbon <File> → Select【Import】→【Import DXF/DWG】
3. The <DXF/DW> dialog appears

   Click【…】at the end line of [Input File] → Select the file to import
4. The layer names in the DXF/DWG file will be displayed. Select the importing layer
5. Specify where the layer to be imported on Quadcept:

   Specify【Other： Board】in the Layer list and click【→】
6. Click【OK】



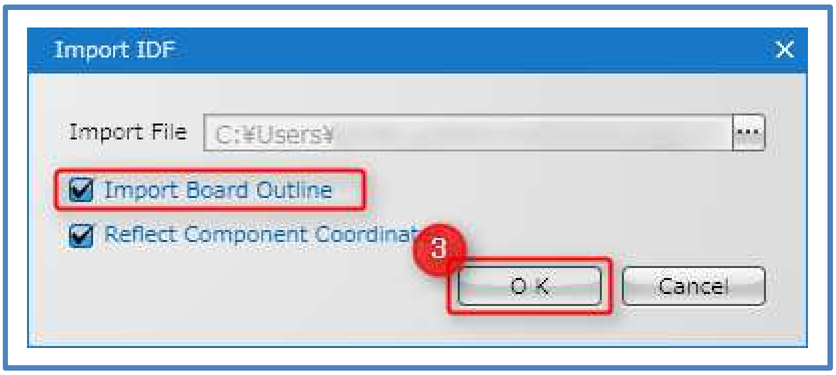
4-c. Reading IDF data for a board outline

1. Open the PCB sheet where you want to draw a board outline.
2. Read IDF data

   Ribbon <File> → Select【Import】→【Import IDF】
3. The <Import IDF> dialog appears

   Click【...】at the end line of [Input File] to open → Select the IDF file

   Check the【Enter Board Outline】box →click【OK】

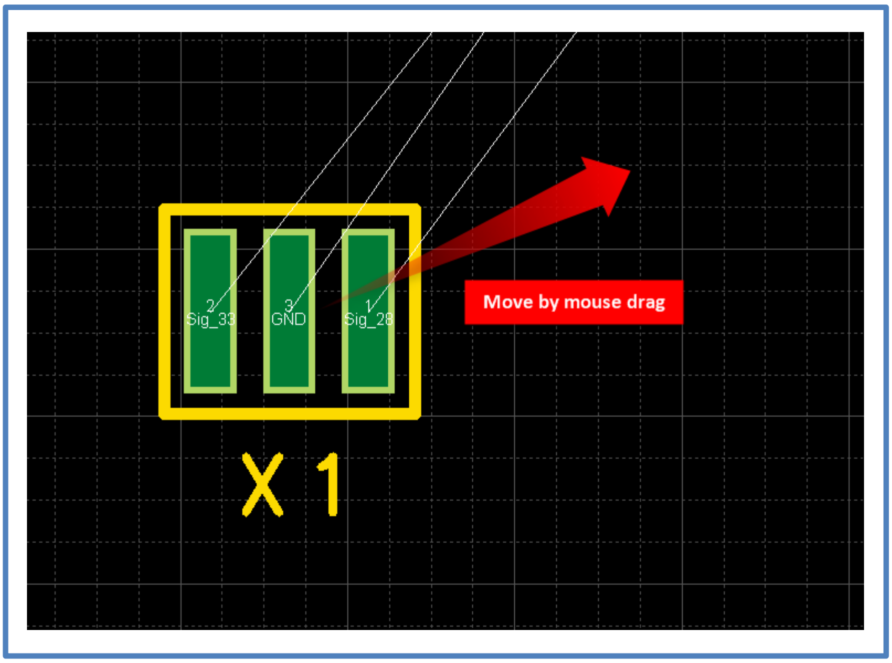


■Step 5. Moving Component

There are several ways to move and place components (footprints).

5-a. Move a component by mouse drag

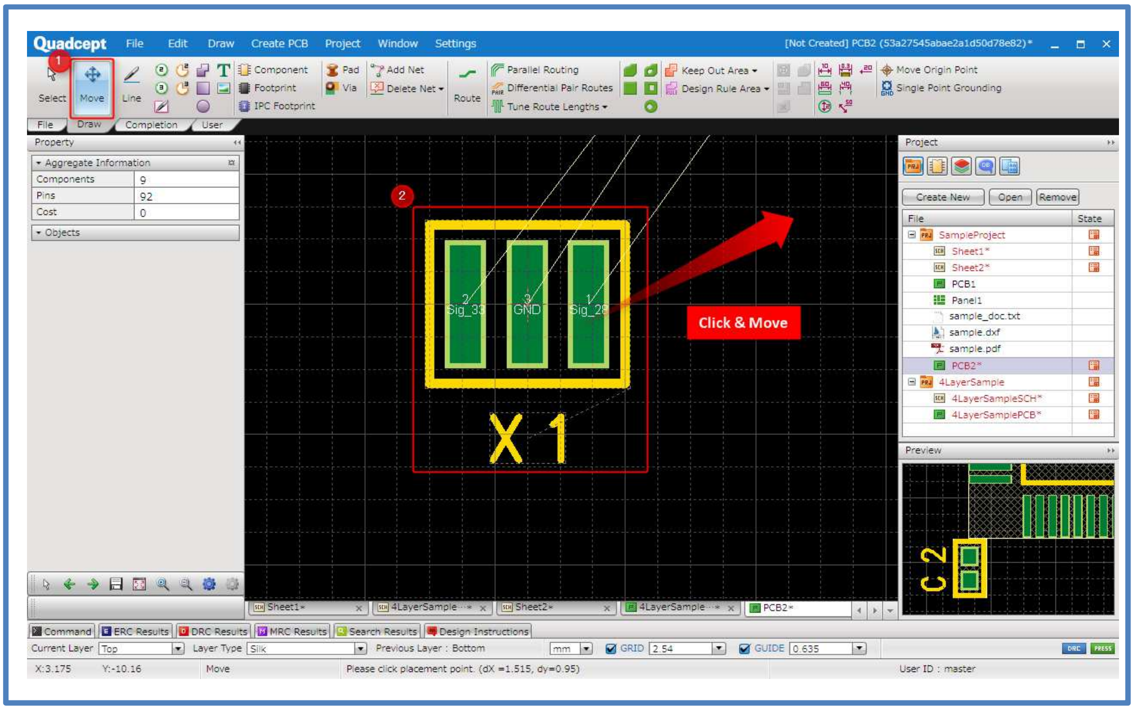
1. Open the PCB sheet → 【Settings】 → 【Environment Settings】 → Select 【PCB】
   Check whether【Allow Moving Objects by Dragging】box in the [Move] section is checked.
2. Drag the component to move



5-b. Move component in Move mode

*The origin point of component is the reference point when moving components.

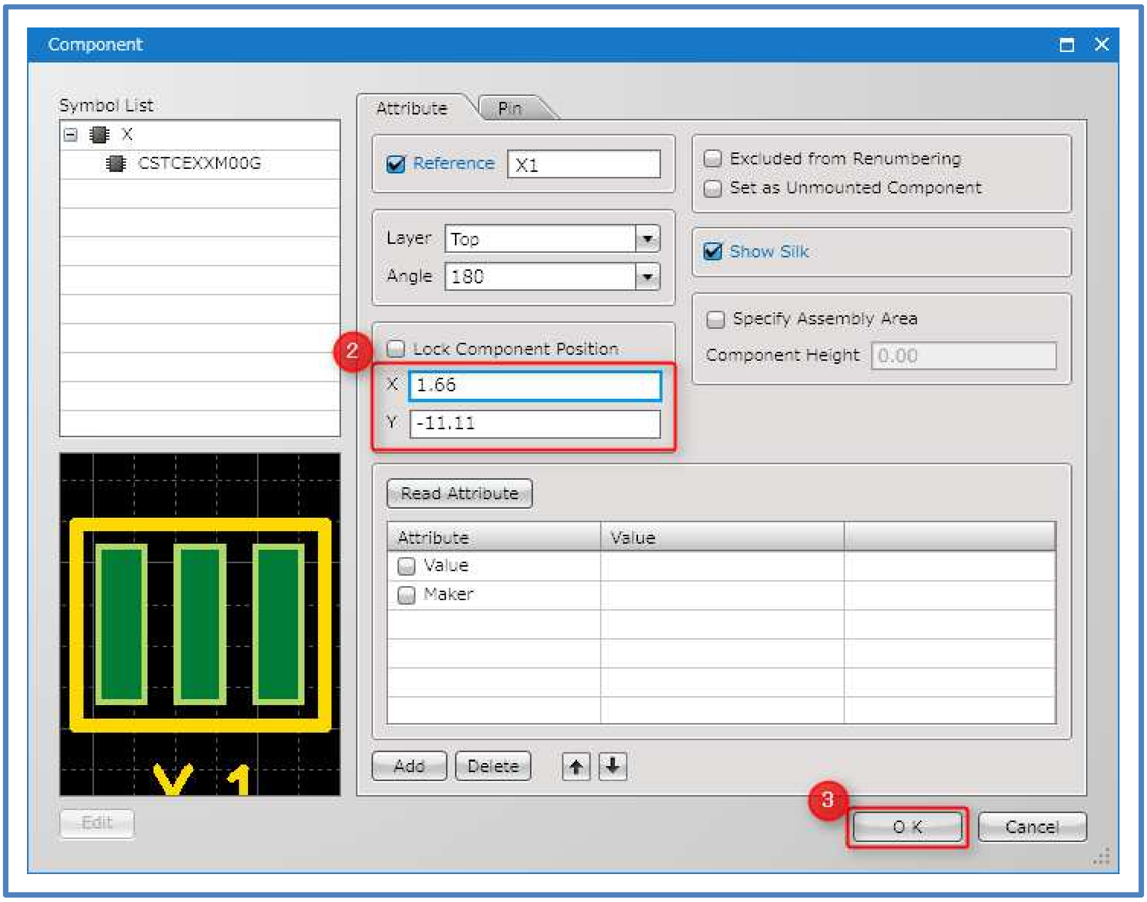
1. Open the PCB sheet → Ribbon【Draw】→ Select【Move】
2. By click the component, it will be attached to the cursor
3. Click where you want to place the component



5-c. Move component by entering the coordinates

*The origin point of component is the reference point when moving components.

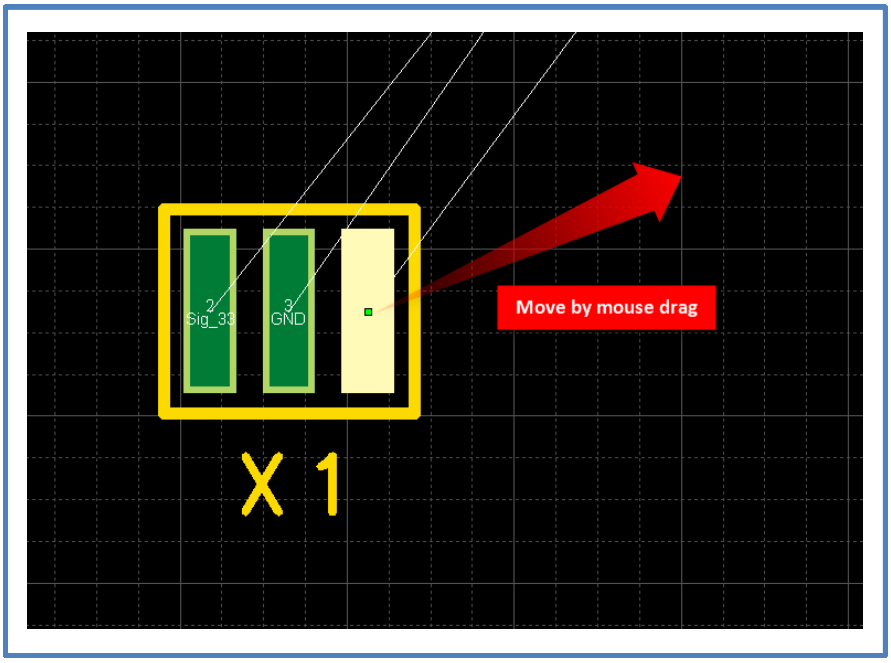
1. Double click the component you want to move → The [Component] dialog will open.
2. Enter the coordinate values in the X and Y fields around the middle of the [Attribute] tab
3. Click【OK】



5-d. Move by the center of the pad as the reference point.

*The center of pad is the reference point when moving components.

1. Select the pad of the component
2. Move the mouse cursor on the pad handle and drag

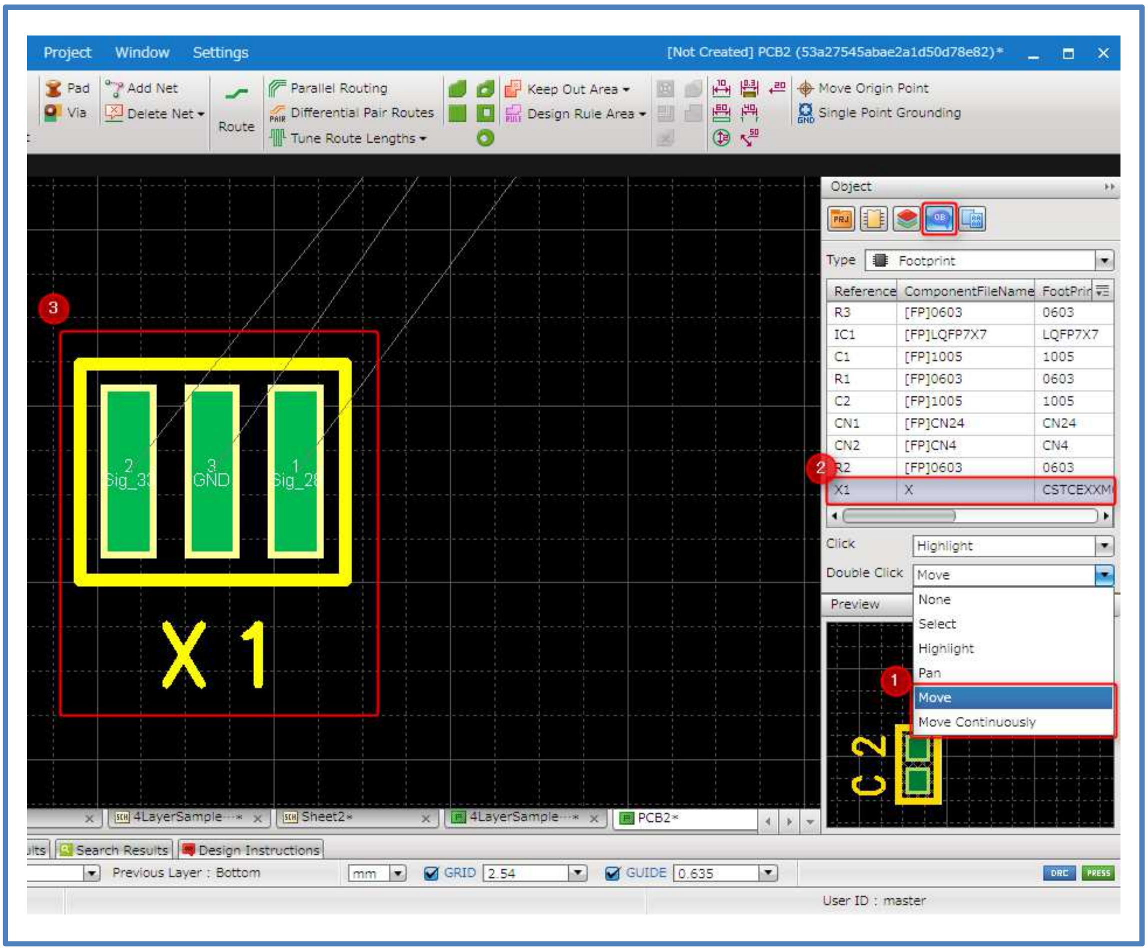


5-e. Move by selecting the component in the Object window

*The origin point of component is the reference point when moving components

1. Open the PCB sheet and select the <Object> window. Assign a【Move】or【Move Continuously】task to [Double Click]
2. Select the target component in the <Object> window and [Double click]
3. When you navigate the mouse to the document area, the component will be attached to be moved
4. Click where you want to place it

   When【Move Continuously】is selected, the next component in the <Object> window becomes movable automatically.

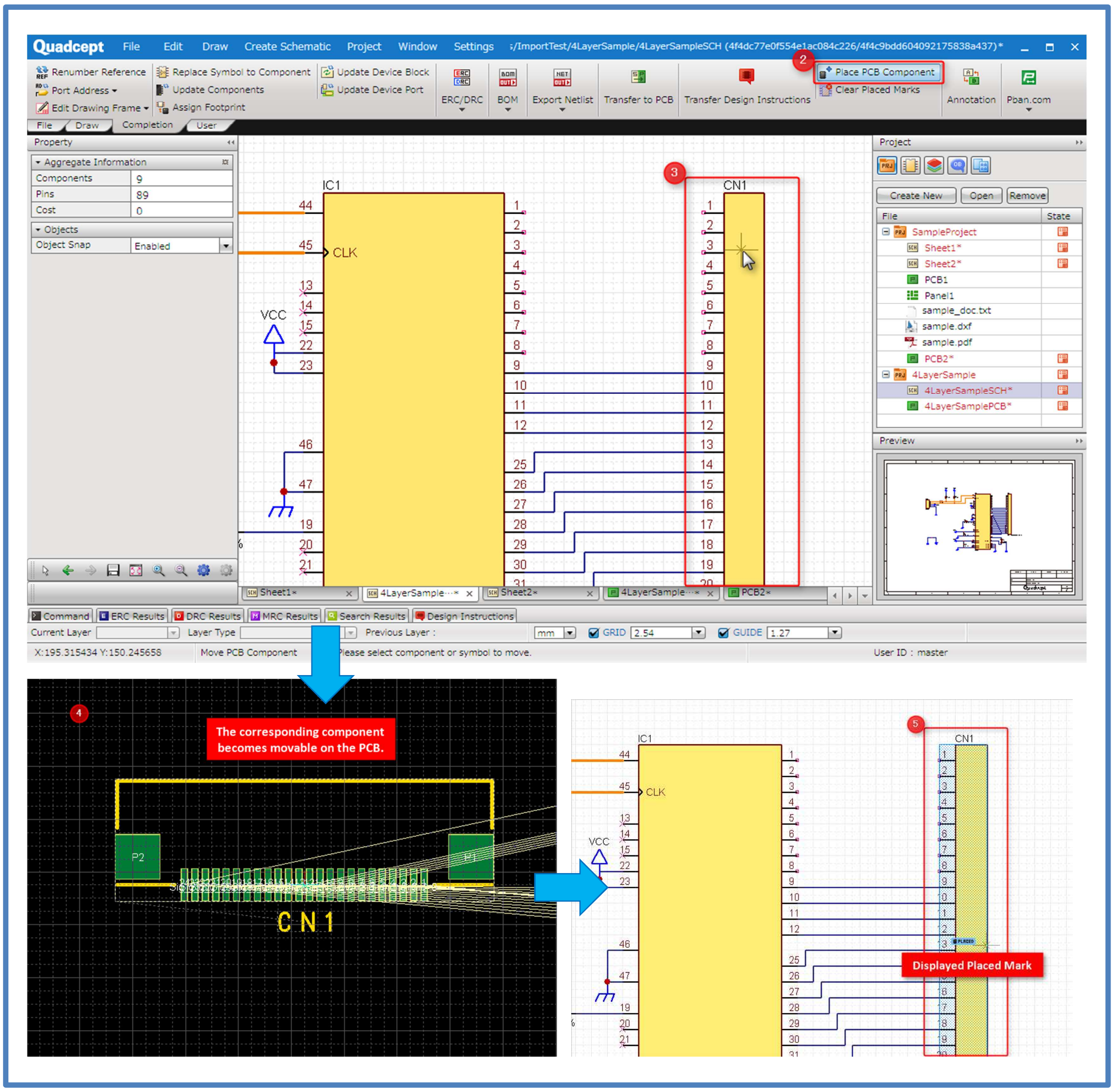


5-f. Place (move) component from schematic

*The origin point of component is the reference point when moving components.

If the component is within the same project, you can place (move) the component while checking with the schematic.

1. Open the schematic sheet
2. Ribbon <Completion> → Select【Place PCB Component】
3. Click the component, which you want to place in the PCB sheet, on the schematic
4. Automatically switches to the PCB sheet. Place the PCB component as it is now movable
5. After placing the component, it automatically returns to the schematic and the [Placed Mark] is displayed on the component.

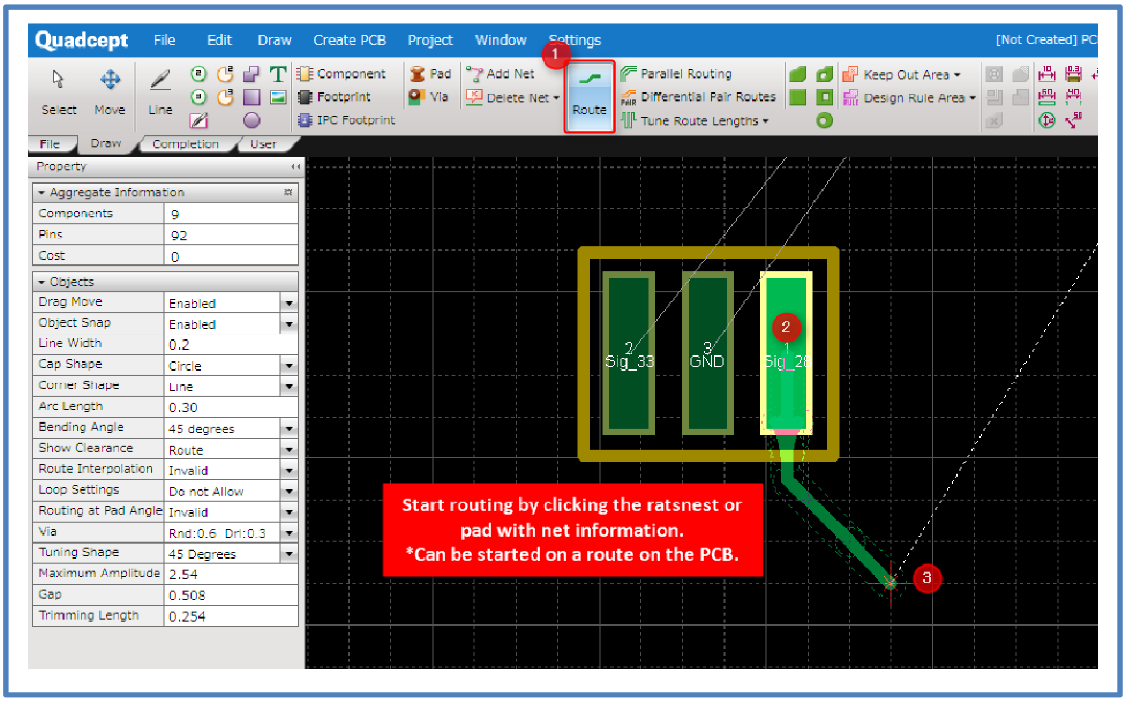


■**Step 7. Routing**

Drawing routes based on the netlist information.

Routing Task

1. Open the PCB sheet → Ribbon <Draw> → Select【Route】
2. Start routing by clicking the ratsnest, pad with net information or the route in the PCB.
3. Draw the path by setting points with a click along the route. To finish routing, click or double-click on the pad.

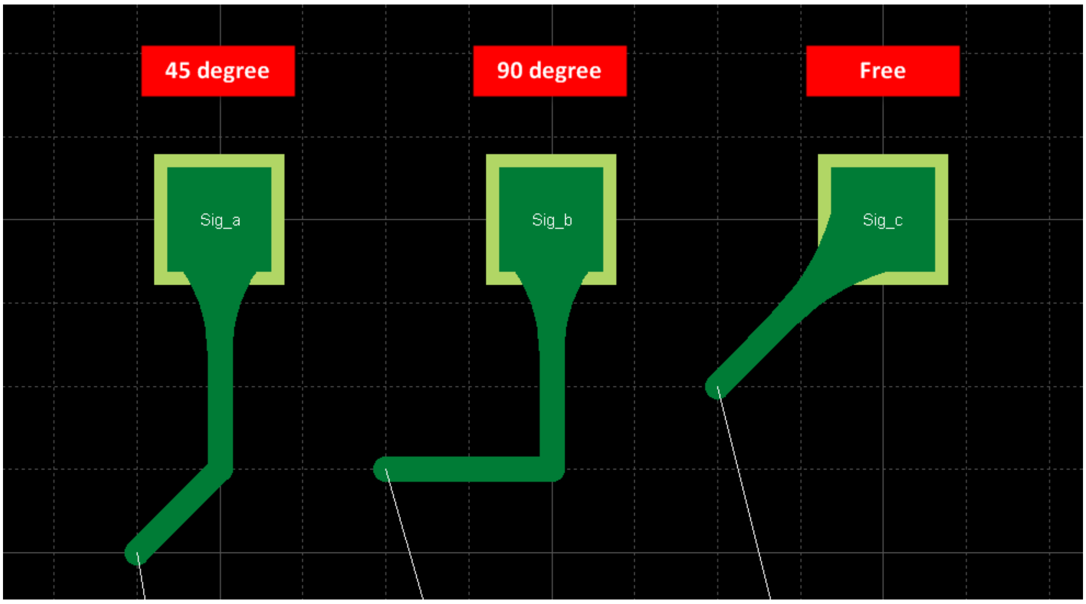


Bending Angle of Routing

Bending Angle：

Press【S】key at the corner of routing, and change the angle to "45 degree","90 degree" or "Free"

*Other methods:【Right click】→【Change the Bending Angle】or【Bending Angle】in the [Property] window

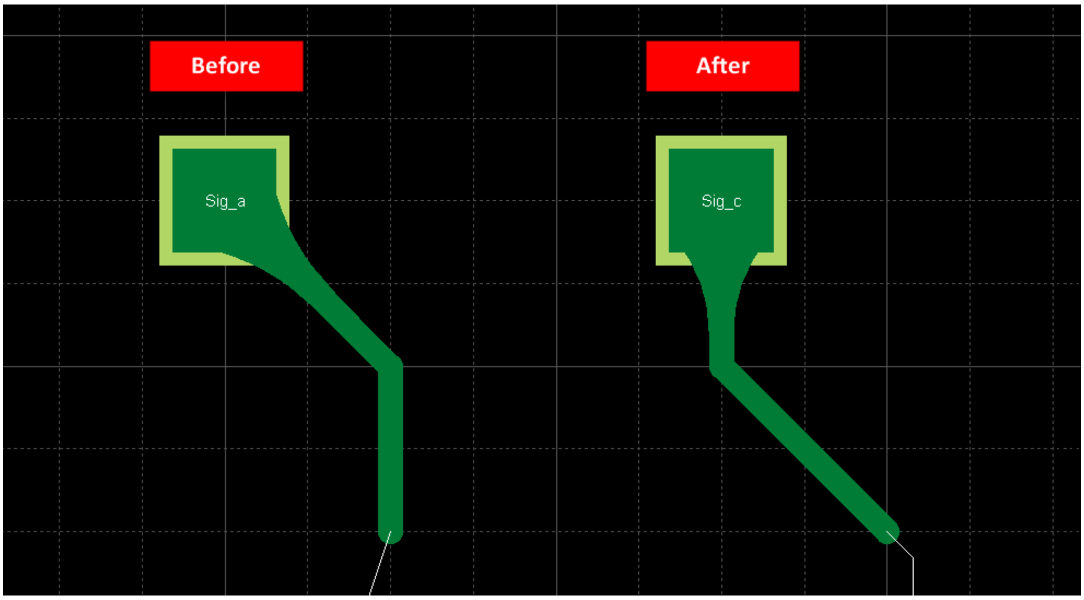


Switch Angle of Routing

Switch Angle of Routing：

Press【X】key at the corner of routing, and change the routing angle.

*Other method:【Right click】→【Switch Angle】



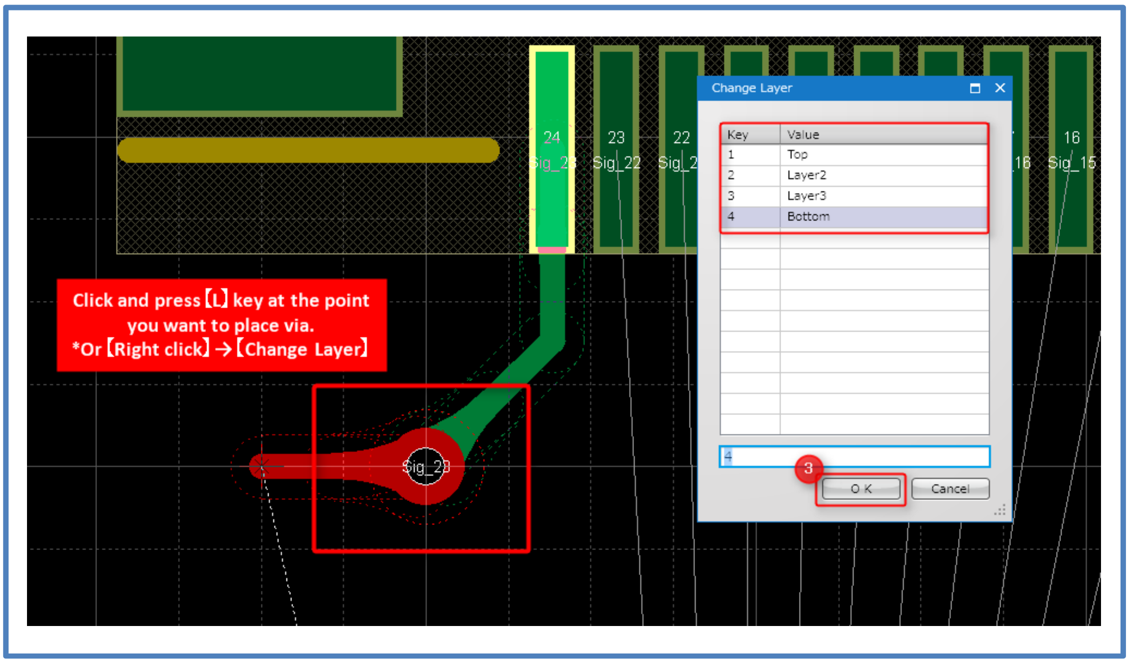
Placing Via / Change the Current Layer

When you change layers while routing, a via is automatically created.

1. Click where you want to place the via while routing
2. Press【L】key to change layers（or with a right click →【Change Layer】）
3. Specify the layer you want to switch to the current layer, and click【OK】

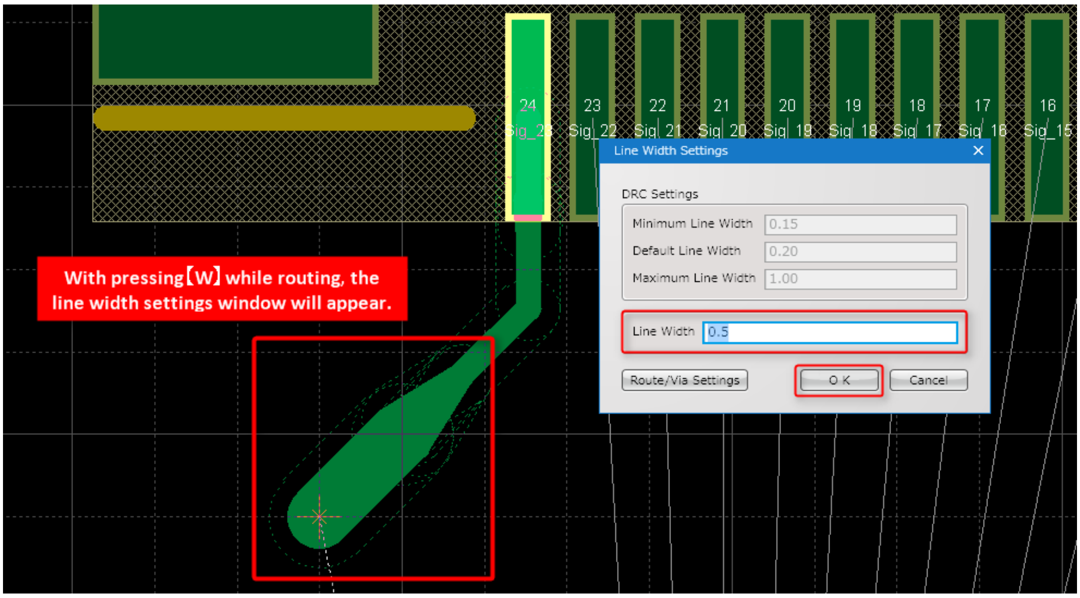
*You can also place a via with pressing【Tab】key (Move to Previous Layer) at the step 2.

*You can place via stacks and fix creating routes with pressing【V】key at the step 2.



Editing Route Width

1. Click on the route you want to edit the width.
2. Press【W】key and edit the width (or with a right click →【Edit Line Width】)
3. Enter the line width and click【OK】



Convenient Functions for Routing

Here are some tips for routing.

Online DRC

You can check the DRC settings in real time while designing the PCB.

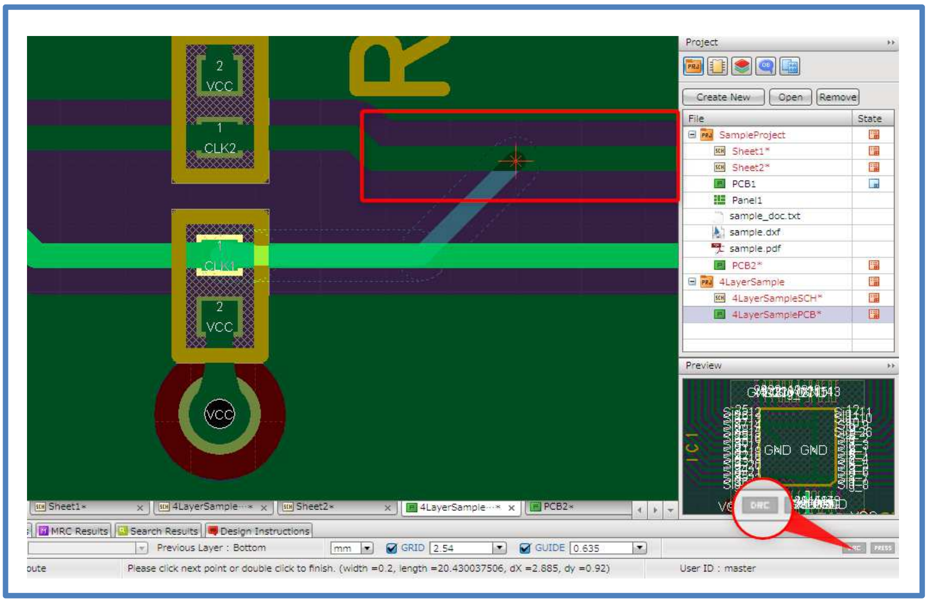
The reduction of work time and quality improvement is expected as you can check the DRC settings while designing; the task previously done after the design is completed.

Toggle ON/OFF

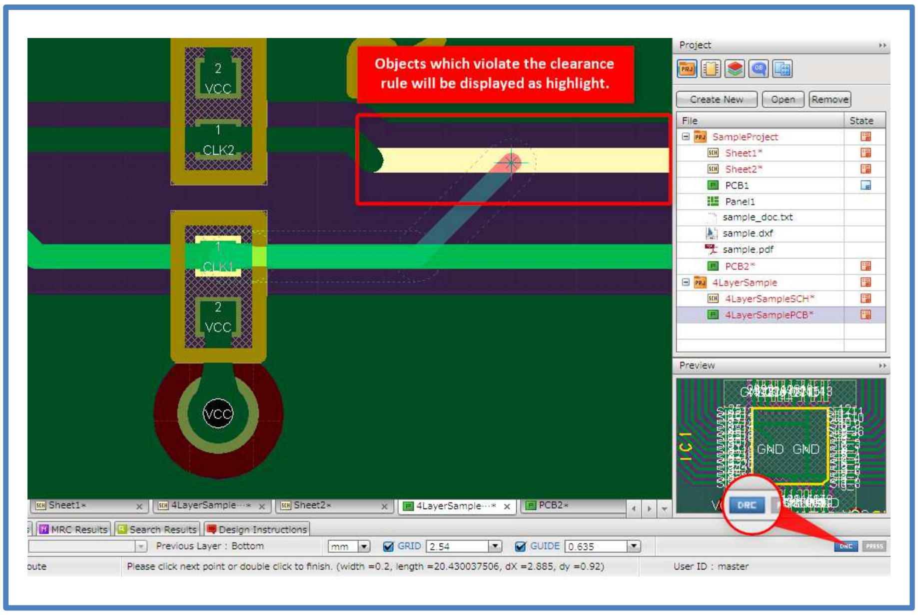
・Toggle enable/disable with the【DRC】button at the bottom right corner.

・Select【Settings】→【Settings】→【System】→【Route】

Toggle check/uncheck [Enable Online DRC]



*Objects which violate the clearance rule will be displayed as highlight.

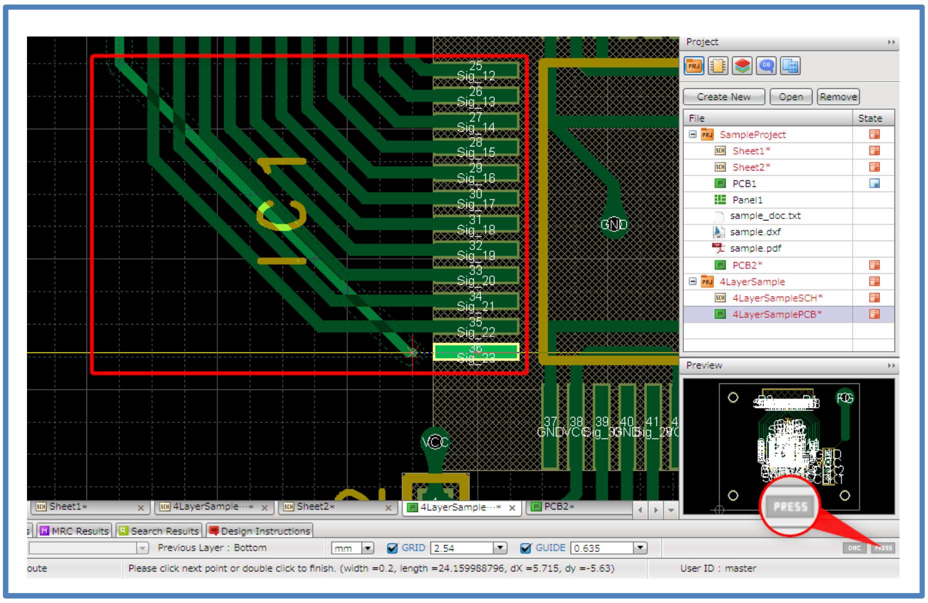


Stop at First Obstacle

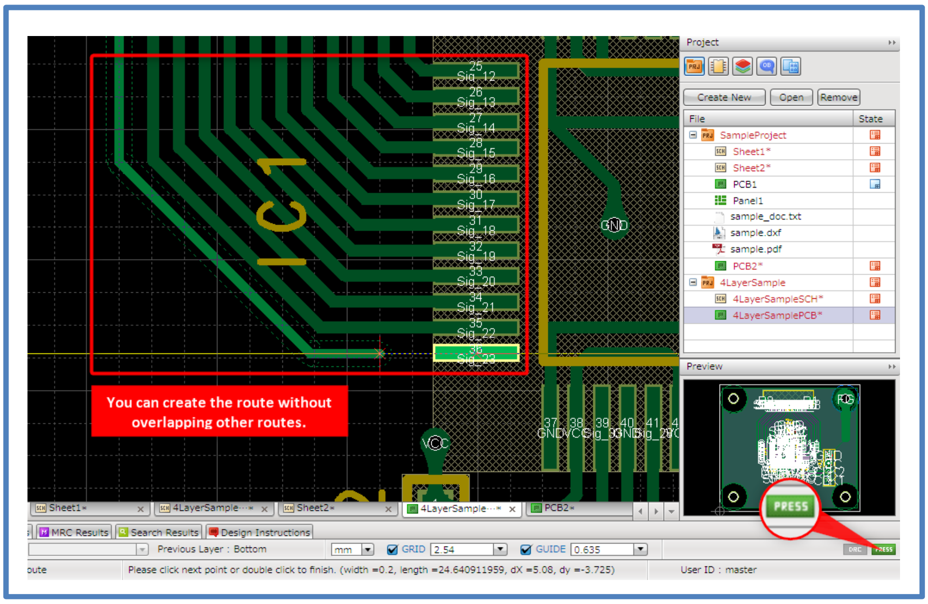
It allows you to stop at first obstacle while routing. So you can create the route with the shortest path along other routes or planes taking into consideration the DRC clearance settings and avoiding obstacles automatically.

Toggle ON/OFF

- Toggle enable/disable with the【PRESS】button at the bottom right corner.
- Select【Settings】→【Settings】→【System】→【Route】



*You can create the route with the shortest path along other routes and without overlapping them taking into consideration the DRC clearance settings.



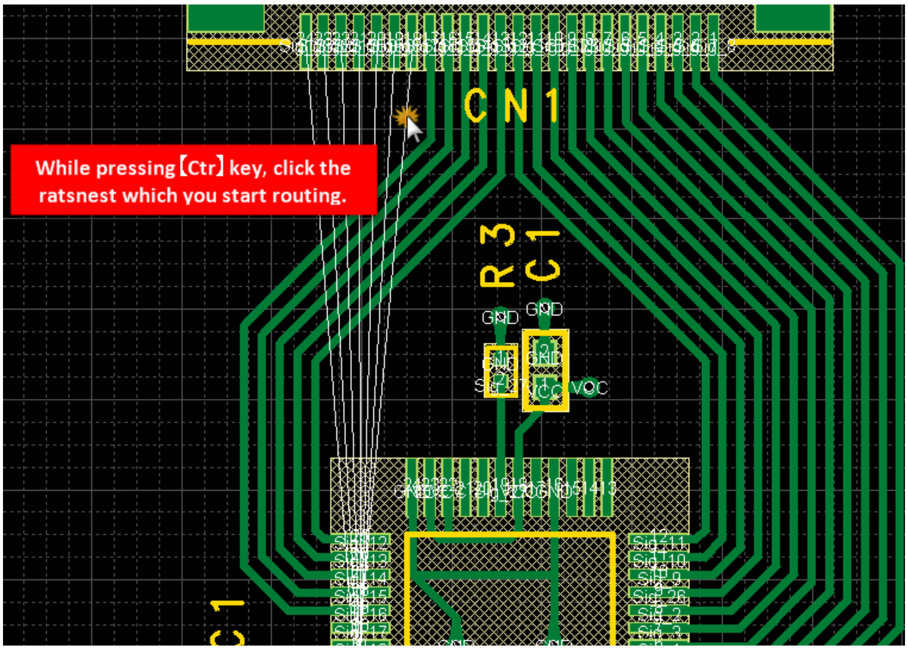
Semi-Auto Routing

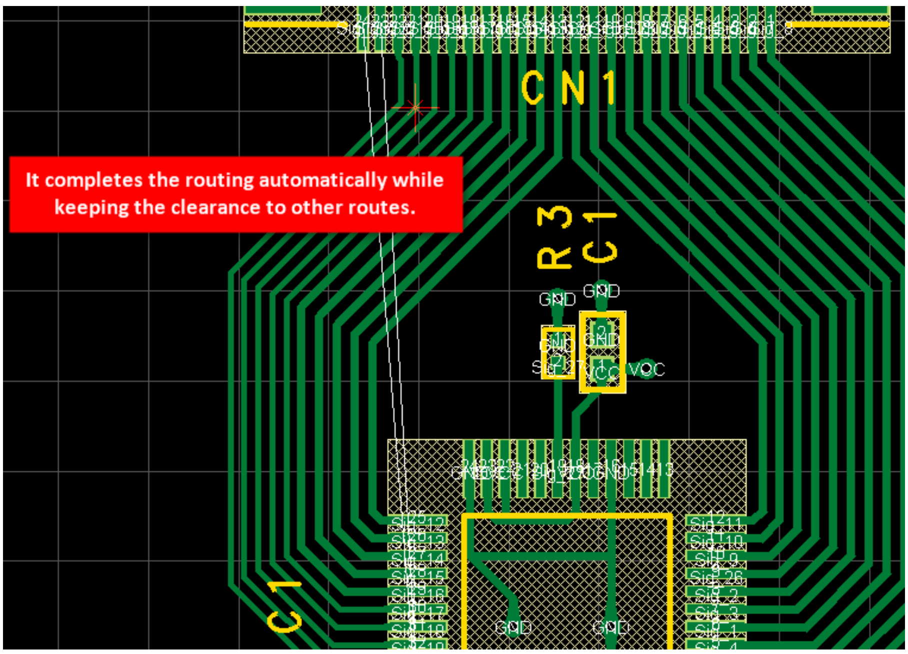
It calculates the shortest path automatically and completes routing with one click.

Together with the Stop at First Obstacle function, it can be utilized more conveniently.

1. Open the PCB sheet, select【Create PCB】→【Route】
2. While pressing and holding down【Ctrl】key, click the rats
3. Completes routing with the shortest path

*You can also activate the function by a click with pressing and holding down【Ctrl】key while routing.





■Step 8. Types of Plane/Creating Plane

Plane, Quadcept has three types of plane.

Dynamic Plane

Specify a net name to the created plane. Dynamic plane automatically connects to objects with the same net name according to the rule you set in the [Dynamic Plane Connection]. It also automatically creates clearance for objects other than the specified net name and will not connect to them.

Static Plane

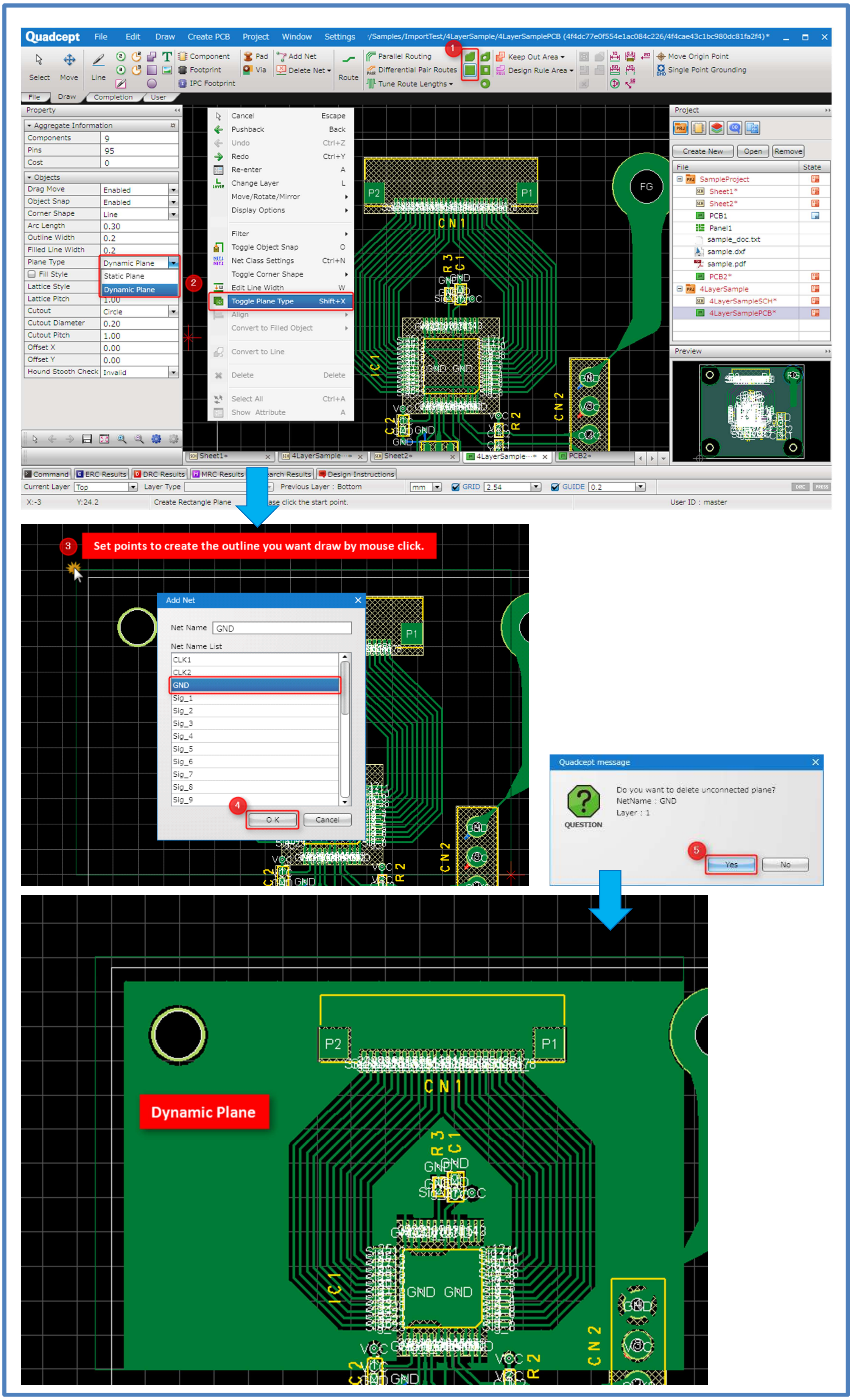
It fills the area you draw. It is suitable for reinforcing the route.

1. Ribbon <Draw> → Select【Polygon Plane】 or【Rectangle Plane】
2. Check the [Plane Type] in the property window, and select【Dynamic Plane】 or【Static Plane】

   *【Right click】→【Switch Plane Type】or you can also change it with the keyboard shortcut【Shift + X】
3. Set points to create the outline you want draw by mouse click.

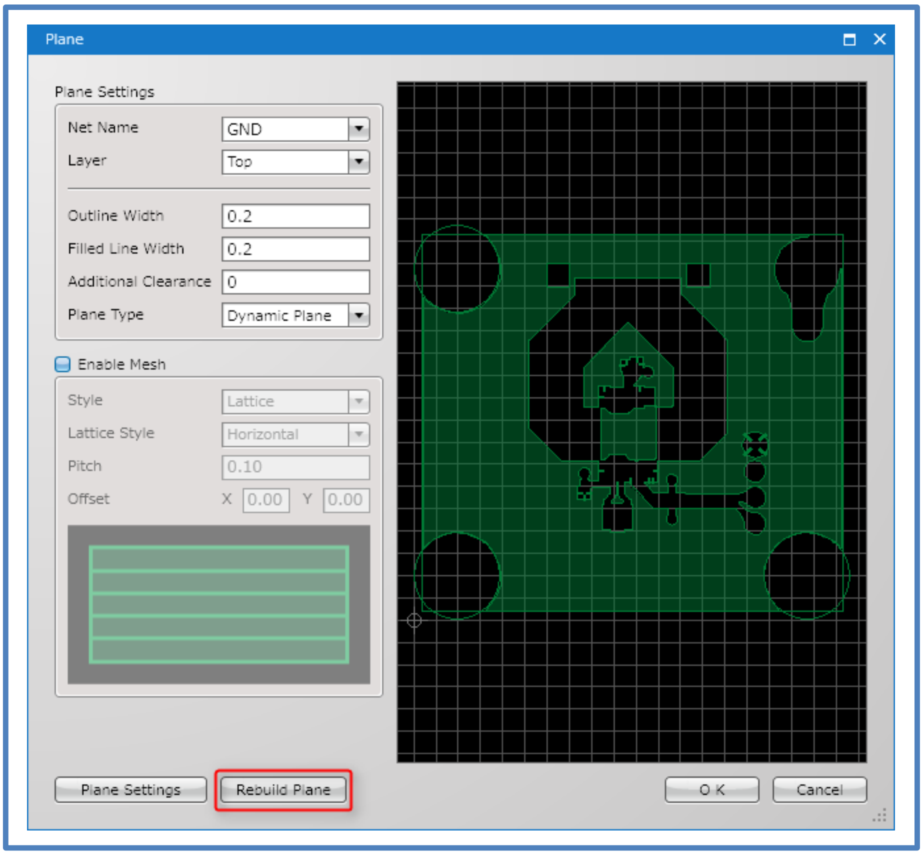
   When it is the dynamic plane
4. Net names within the drawn area will be displayed in the [Add Net] dialog.

   Specify the net names you want to connect to and click【OK】
5. [Do you want to delete unconnected plane?] message appears, select【Yes】



Toggle Plane Type Shift+X
Plane Type
Static Plane
Dynamic Plane
Set points to create the outline you want draw by mouse click.
Add Net
Net Name GND
Net Name List
CLK1
CLK2
GND
Sig_1
Sig_2
Sig_3
Sig_4
Sig_5
Sig_6
Sig_7
Sig_8
Sig_9
O K
Cancel
Quadcept message
Do you want to delete unconnected plane?
NetName : GND
Layer : 1
QUESTION
Yes
No
Dynamic Plane

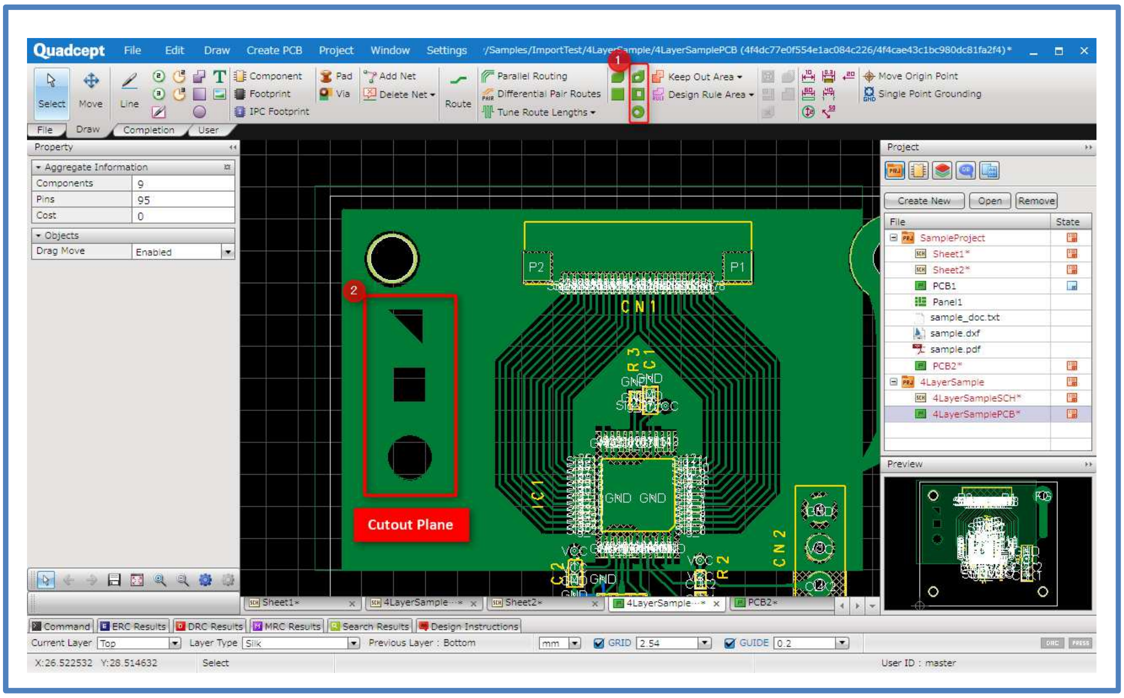
*When you edit the routing or component placement after you draw the dynamic plane, you can update the plane by;【Double click】the plane → 【Rebuild Plane】 or with the keyboard shortcut 【 I 】.



Cutout Plane

Cutout plane is used to create much larger clearance or for the area you don't want to create plane, etc. By placing it in the plane, that area will be cut out.

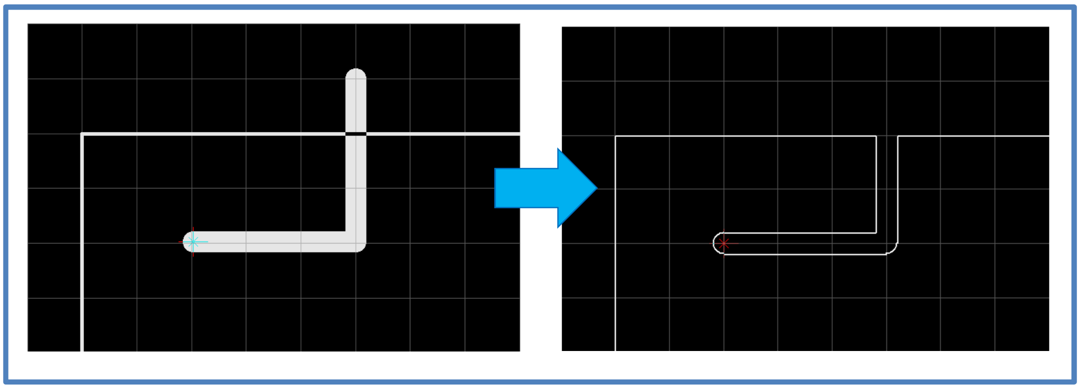
1. Ribbon <Draw> → Select【Polygon Cutout】,【Rectangle Cutout】or【Circle Cutout】
2. Set points by mouse click to draw the outline of the cutout plane on the sheet.



■**Step 9. Board Slit**

You can create a slit easily sense to enter the line.

1. 【Create PCB】→ Select【Create Board Slit】
2. Press【W】key, set the line width and click【OK】

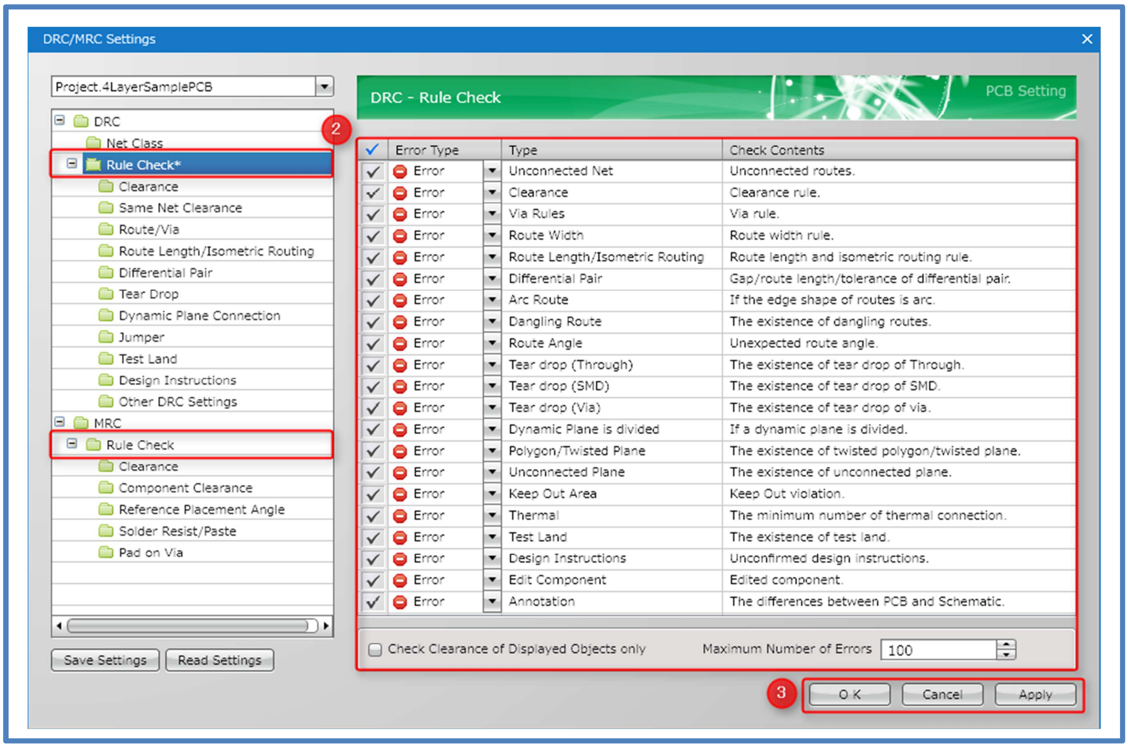


■Step 10. Verifying PCB (DRC/MRC)

Verify the printed circuit board before the output.

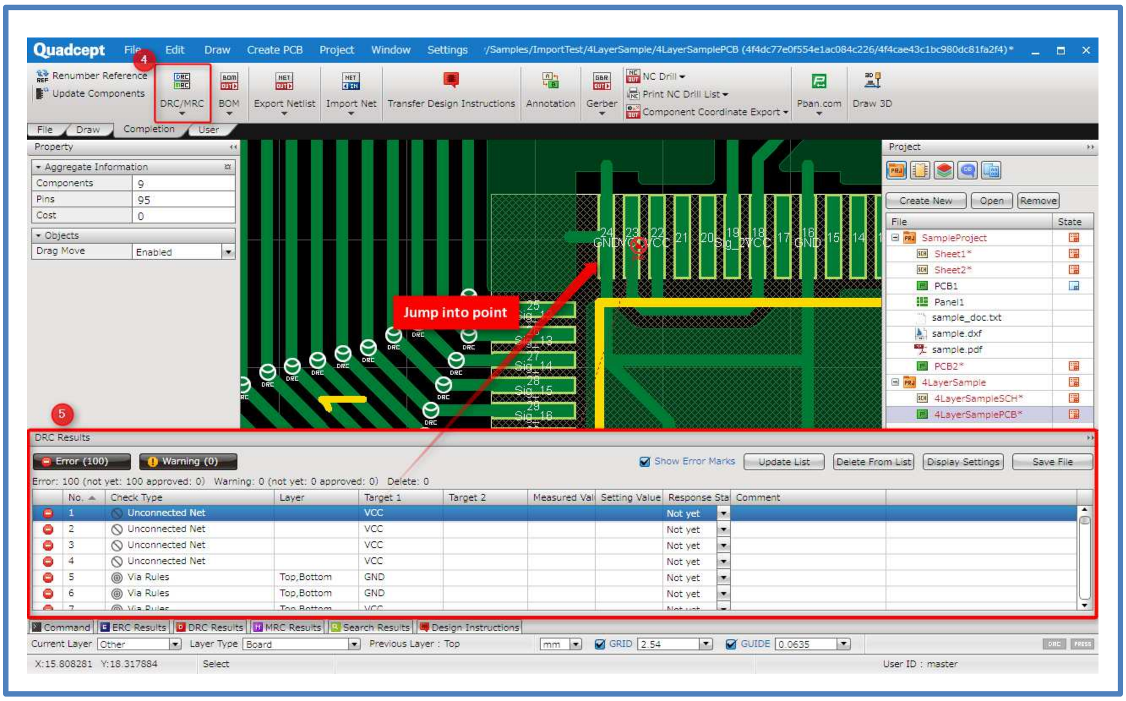
1. Ribbon< Completion > → 【DRC/MRC】 → Select 【DRC/MRC Settings】
2. Select 【Rule Check】 and set what you want to verify
3. Select 【Apply】 and click 【OK】
4. Ribbon < Completion > → Select 【DRC/MRC】
   Check the items for verification and click 【Run DRC/MRC】
5. Check the results of [DRC] and [MRC]
   When there are errors, correct them and run the checks until there are no errors.

ERC/DRC settings



DRC/DRC Results

DRC/MRC correct error points after verifying.（double click to jump into that point）



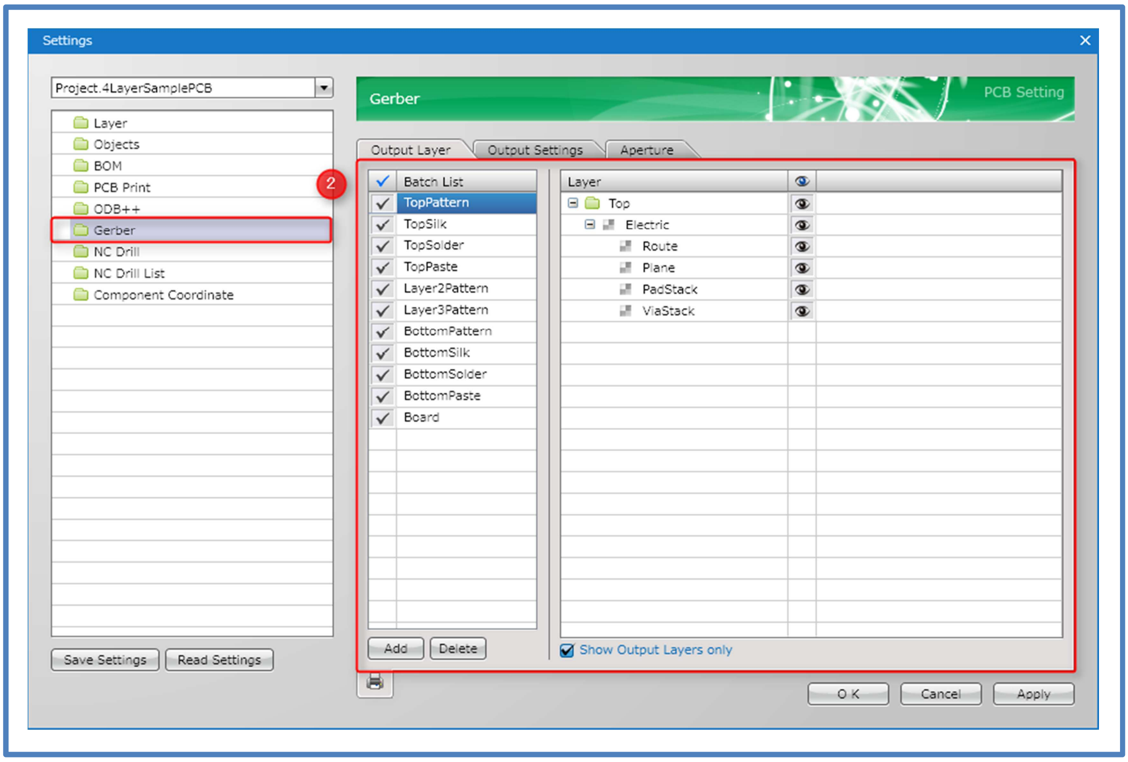
■Step 11. Export Manufacturing Data

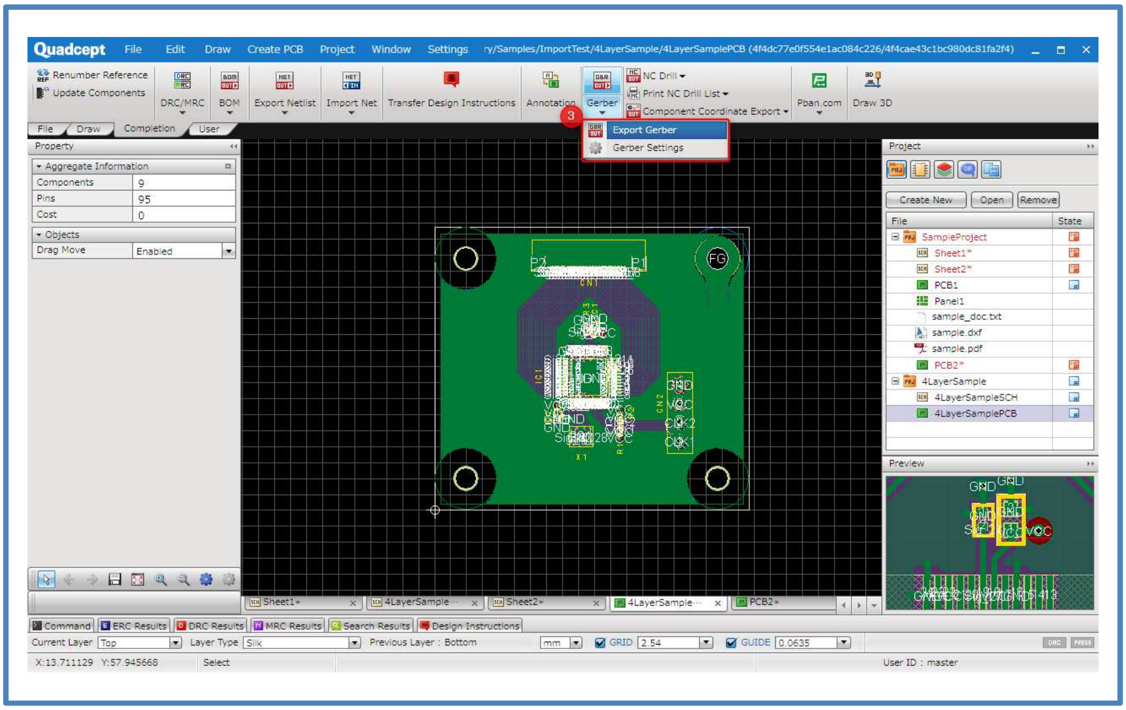
When completed the PCB design, output the data for the manufacturer.

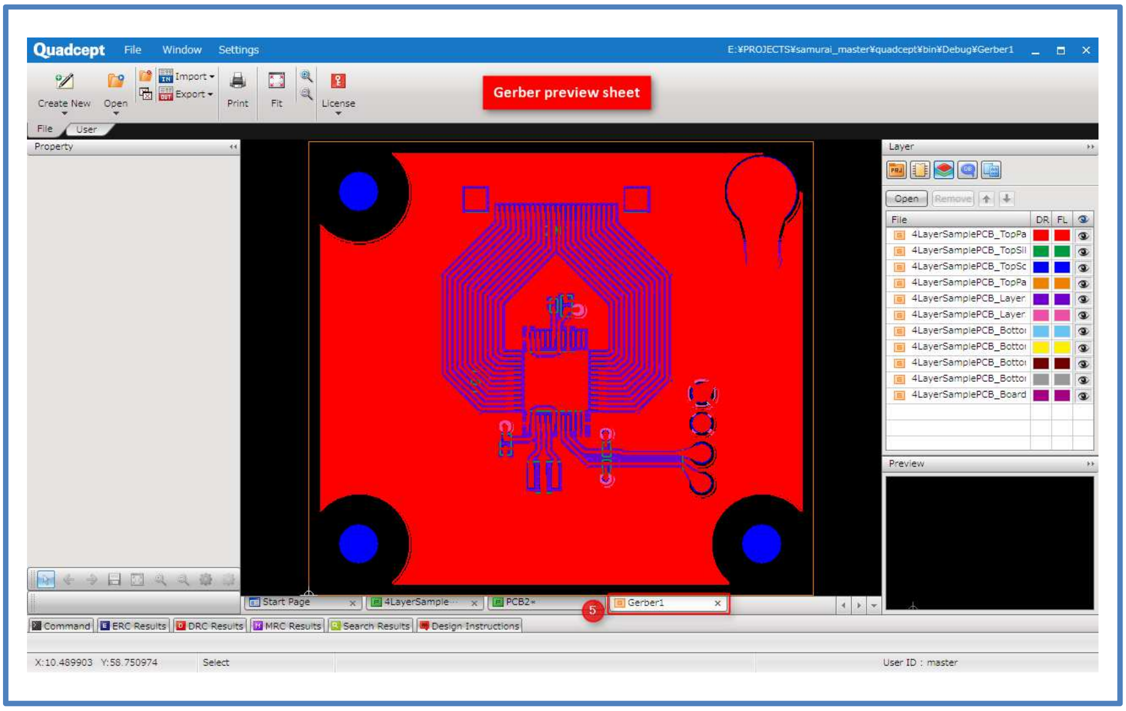
This section describes the method to export gerber files.

Export Gerber Data

1. Ribbon <Completion> → Select【Gerber】
2. Select【Gerber Settings】, set and check the output content
3. Ribbon < Completion > → Select【Gerber】→【Export Gerber】
4. Select where to save, and click【OK】
5. Open the gerber preview sheet




Quadcept
File
Edit
Draw
Create PCB
Project
Window
Settings
Renumber Reference
Update Components
DRC/MRC
BOM
Export Netlist
Import Net
Transfer Design Instructions
Annotation
Gerber
NC Drill
Print NC Drill List
Component Coordinate Export
Pban.com
Draw 3D
3
Export Gerber
Gerber Settings
Property
Aggregate Information
Components
Pins
Cost
Objects
Drag Move
Enabled
Project
Create New
Open
Remove
File
State
SampleProject
Sheet1*
Sheet2*
PCB1
Panel1
sample_doc.txt
sample.dxf
sample.pdf
PCB2*
4LayerSample
4LayerSampleSCH
4LayerSamplePCB
Preview
Command
ERC Results
DRC Results
MRC Results
Search Results
Design Instructions
Current Layer
Top
Layer Type
Silk
Previous Layer : Bottom
GRID
2.54
GUIDE
0.0635
X:13.711129 Y:57.945668
Select
User ID : master

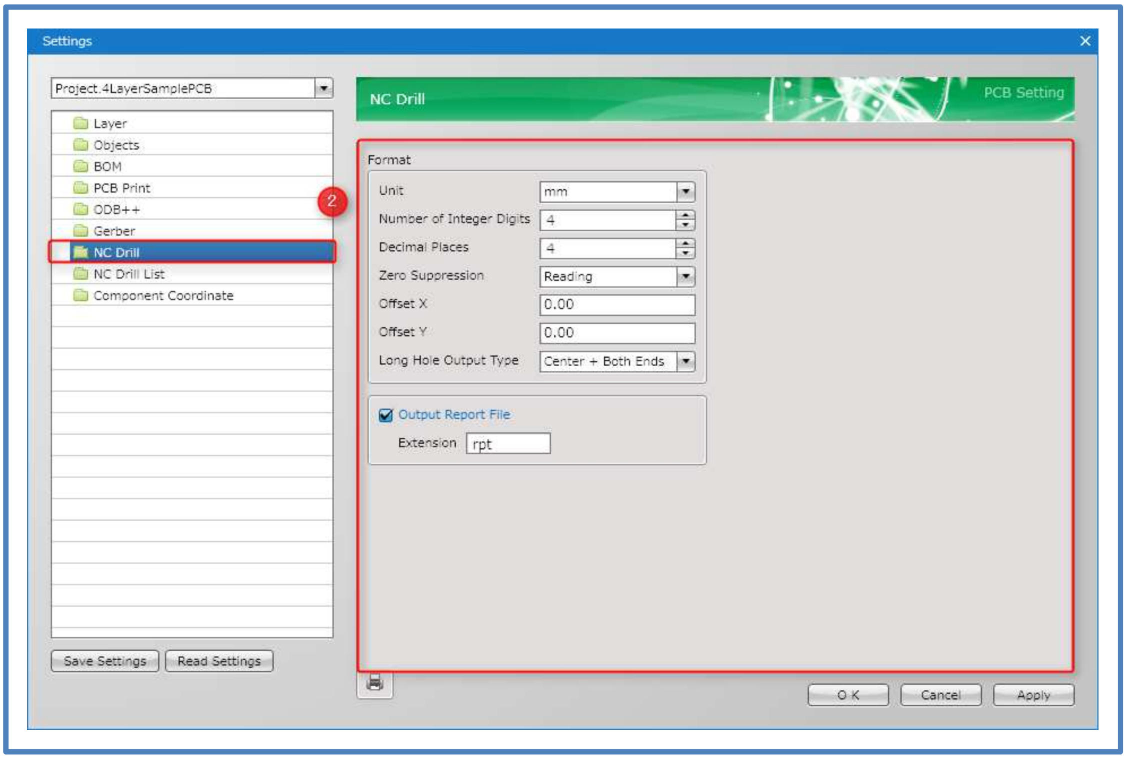

Quadcept
File
Window
Settings
E:¥PROJECTS¥samurai_master¥quadcept¥bin¥Debug¥Gerber1
Create New
Open
Import
Export
Print
Fit
License
Gerber preview sheet
Property
Layer
Open
Remove
File
DR
FL
4LayerSamplePCB_TopPa
4LayerSamplePCB_TopSil
4LayerSamplePCB_TopSc
4LayerSamplePCB_TopPa
4LayerSamplePCB_Layer
4LayerSamplePCB_Layer
4LayerSamplePCB_Bottoı
4LayerSamplePCB_Bottoı
4LayerSamplePCB_Bottoı
4LayerSamplePCB_Bottoı
4LayerSamplePCB_Board
Preview
Start Page
4LayerSample
PCB2*
5
Gerber1
Command
ERC Results
DRC Results
MRC Results
Search Results
Design Instructions
X:10.489903 Y:58.750974
Select
User ID : master

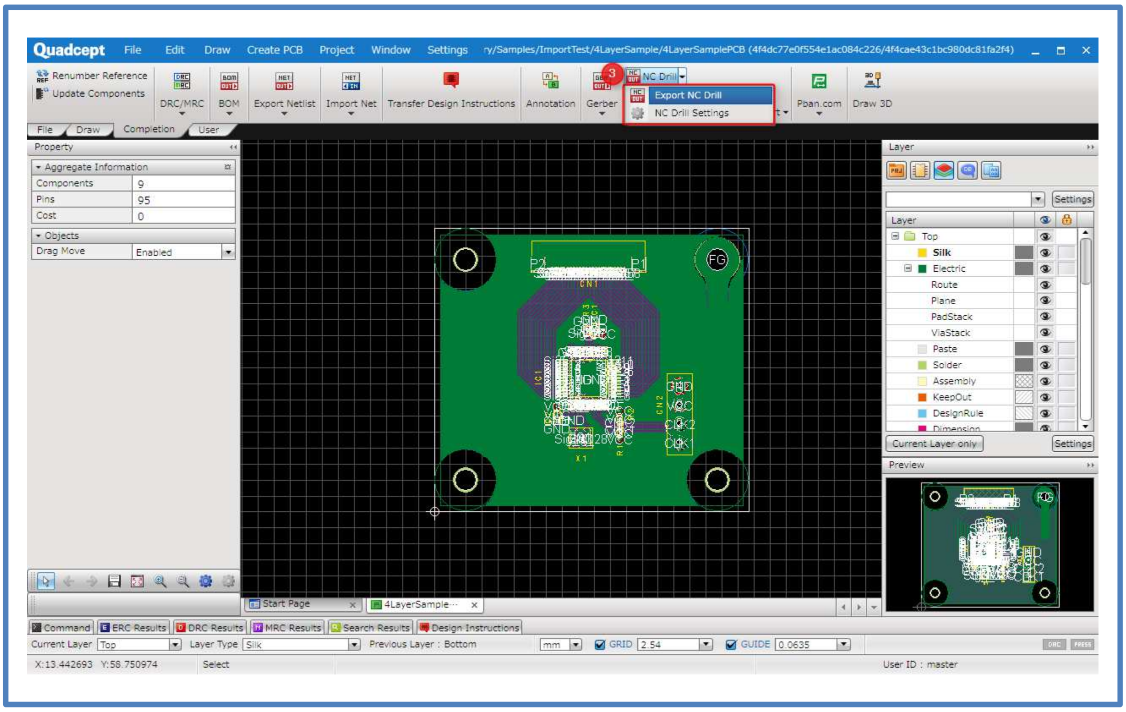
Export Drill Data

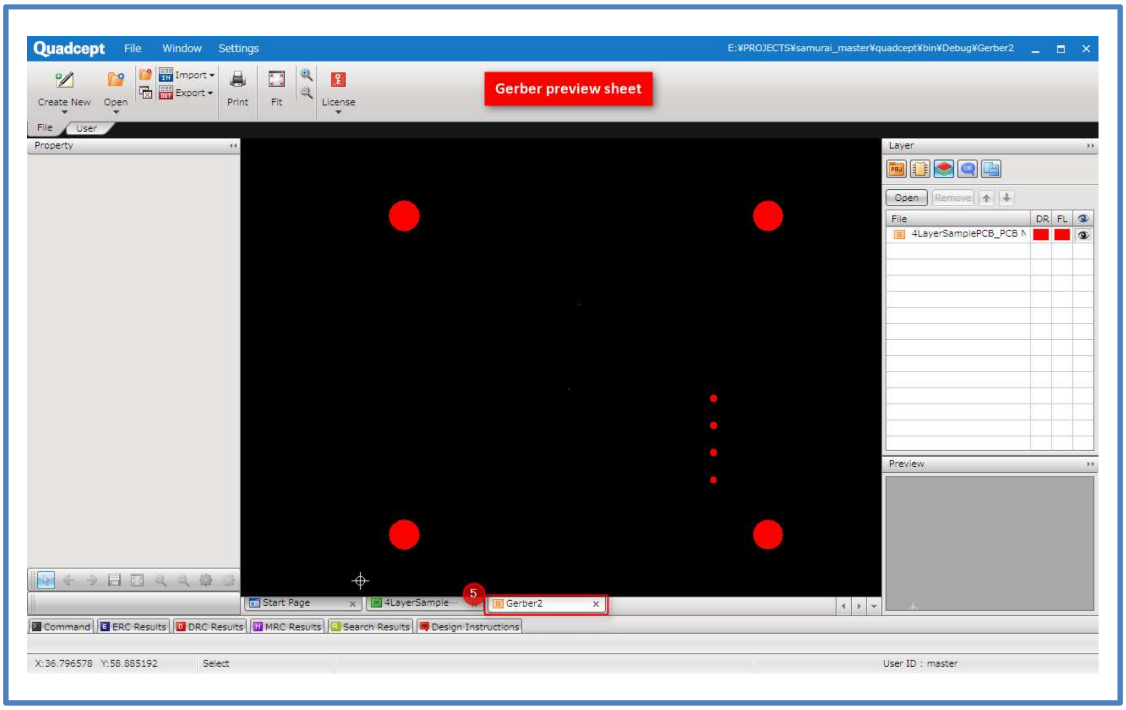
1. Ribbon < Completion > → Select【NC Drill】
2. Select【NC Drill Settings】 and set and confirm the export content
3. Ribbon < Completion > → Select【NC Drill】 →【Export NC Drill】
4. Select where to save, name the file and click【OK】
5. Open the gerber preview sheet

* Perform [Print NC Drill List] or [Component Coordinate Export] as needed.

*You can also output all at once;【File】 →【Batch output】




Quadcept
File
Edit
Draw
Create PCB
Project
Window
Settings
Export NC Drill
NC Drill Settings
Gerber
Draw 3D


Quadcept
File
Window
Settings
Gerber preview sheet
Create New
Open
Import
Export
Print
Fit
License
Gerber2

Chapter 4　Handling of Data

Annotation

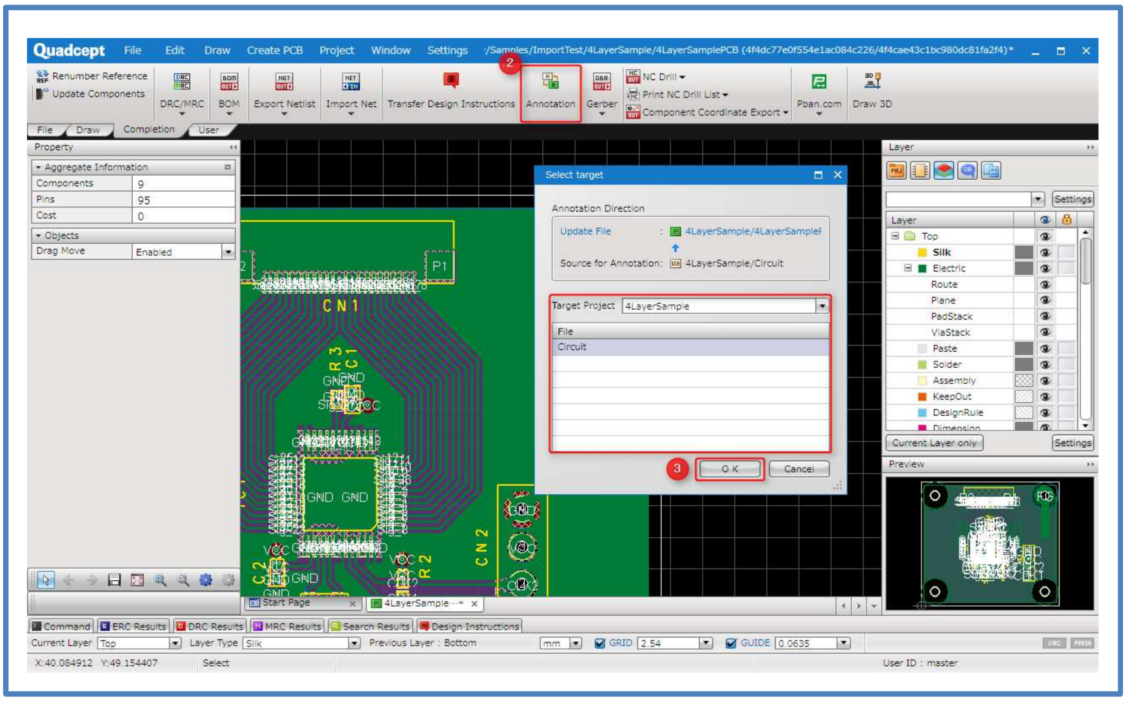
On Quadcept you can extract the differences, occurred through design changes, between schematic/PCB.

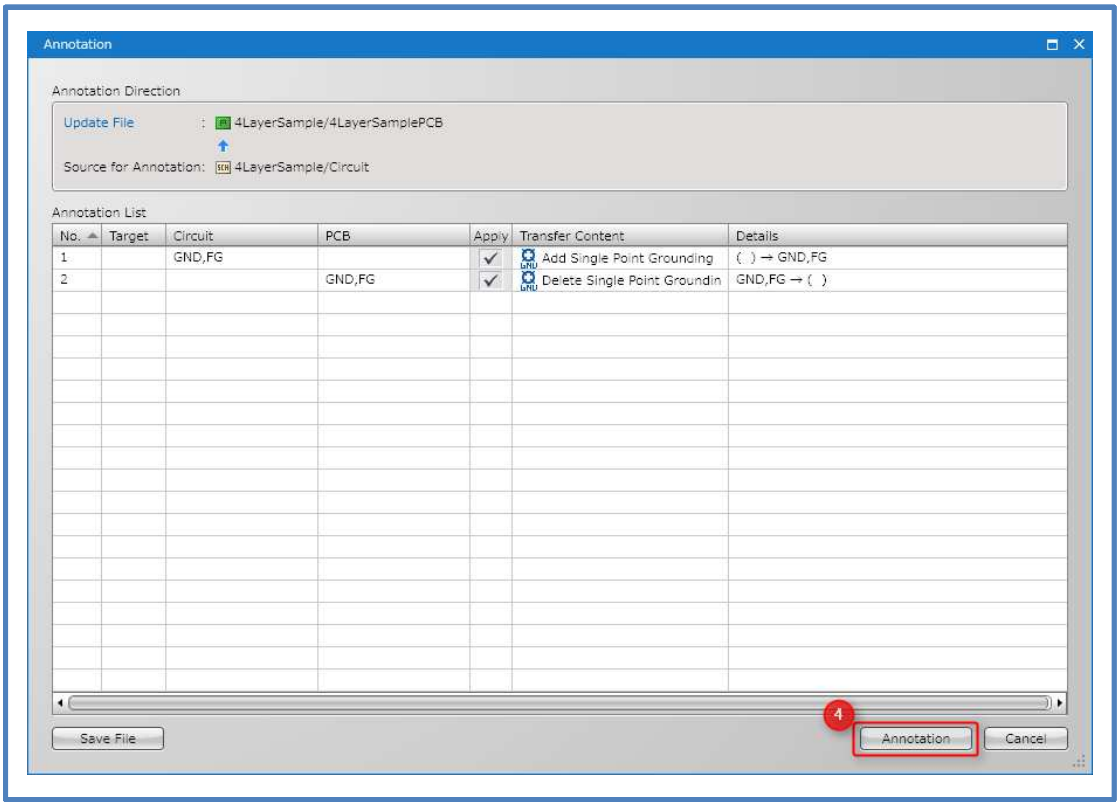
This section describes forward annotation and back annotation.

Forward Annotation

Transfer the content of changes made on the schematic to PCB data

1. Open the file to be updated (PCB data you want to reflect the changes on)
2. Ribbon < Completion > → Select【Annotation】
3. Select the source for annotation (Target project and file), click【OK】
4. Annotation screen will open and the annotation list will be displayed, check the content and click【Annotation】



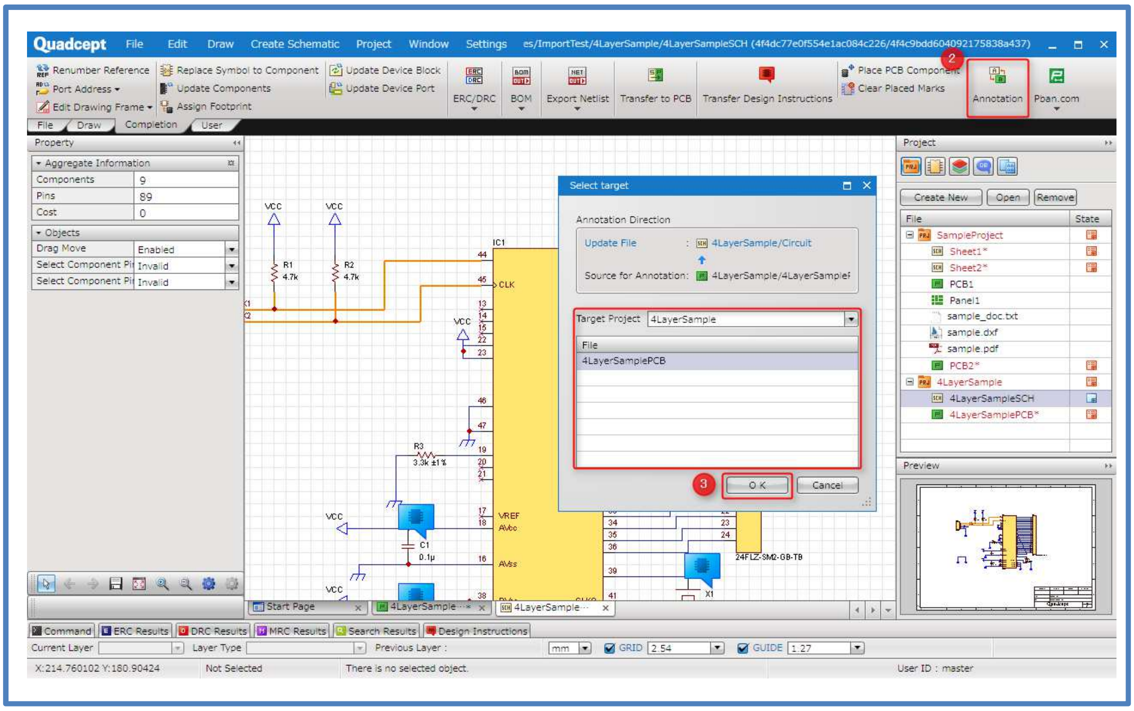

Annotation
Annotation Direction
Update File : 4LayerSample/4LayerSamplePCB
Source for Annotation: 4LayerSample/Circuit
Annotation List
No.
Target
Circuit
PCB
Apply
Transfer Content
Details
1
GND,FG
Add Single Point Grounding
( ) → GND,FG
2
GND,FG
Delete Single Point Groundin
GND,FG → ( )
Save File
4
Annotation
Cancel

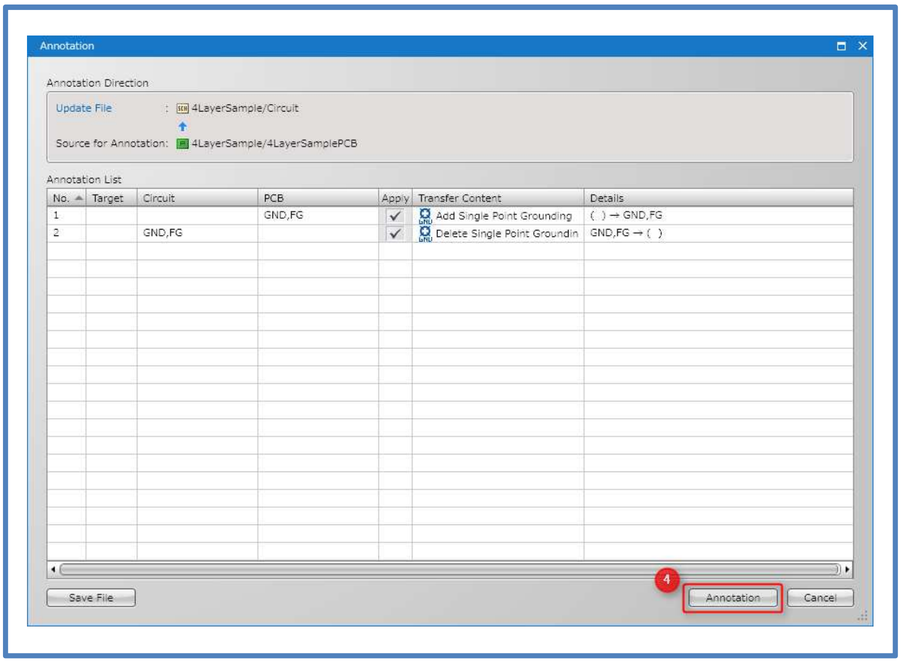
Back Annotation

Transfer the content of changes made on the PCB to schematic.

1. Open the file to be updated (schematic data you want to reflect the changes on)
2. Ribbon < Completion > → Select【Annotation】
3. Select the source for annotation(Target project and file), click【OK】
4. Annotation screen will open and the annotation list will be displayed, check the content and perform【Annotation】

   *Quadcept allows you to perform annotation on new and old data of schematic/PCB.





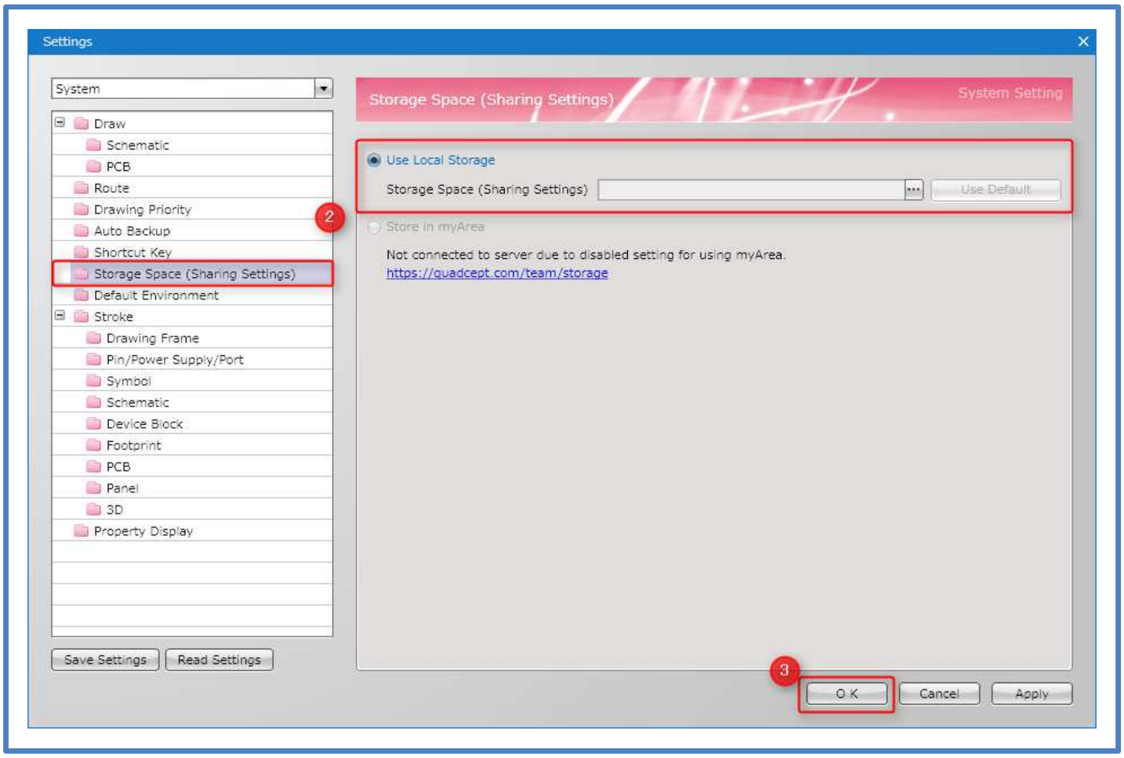
Sharing Data

Qudcept's data are in DB (database) format and the files can be shared.
This section describes the method for sharing Quadcept directories and data with multiple people, within a company or group.

Method for creating database
Create a new database on the server and share the database.

1. 【Settings】→ Select【Environment Settings】→【System】→【Storage Space (Sharing Settings）】
2. Click【...】at the end line of the [Storage Space (Sharing Settings], and select where you want to save it on the server, name the file and click【OK】
3. Now [Storage Space (Sharing Settings)] has been set, so click【OK】
4. Please restart Quadcept as the storage is referenced to after the restart.

*When created a new database space
To transfer Quadcept data to the [Storage Space (Sharing Settings)], you need



*In order for other multiple people to refer to the common database, please follow the above procedure. Set the database, created at the #2 above, in the [Storage Space (Sharing Settings)], and restart Quadcept.

Import/ Export Data (Quadcept File)

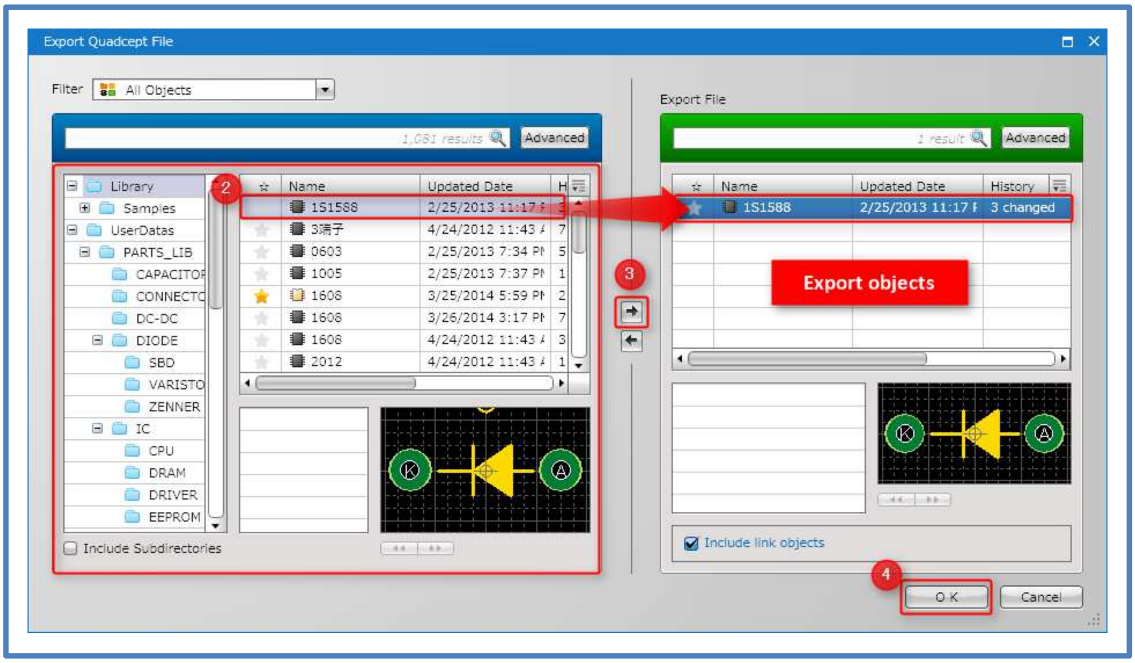
Quadcept manages data such as directories and files in database format and saves them in bulk. So when you want to exchange data with people outside the company or other users, you need to 【Export Quadcept File】/【Import Quadcept File】(its extension is .qcom).

Export Quadcept File

1. Ribbon <File> → 【Export】 → Select 【Export Quadcept File】
2. Choose objects you want to export to the "Quadcept File" from left side
3. Click 【→】 to select export objects
4. Click 【OK】 to export objects you selected.

*When [Include link objects] is checked:

Data that is linked to the object or project will also be export together.



Import Quadcept File

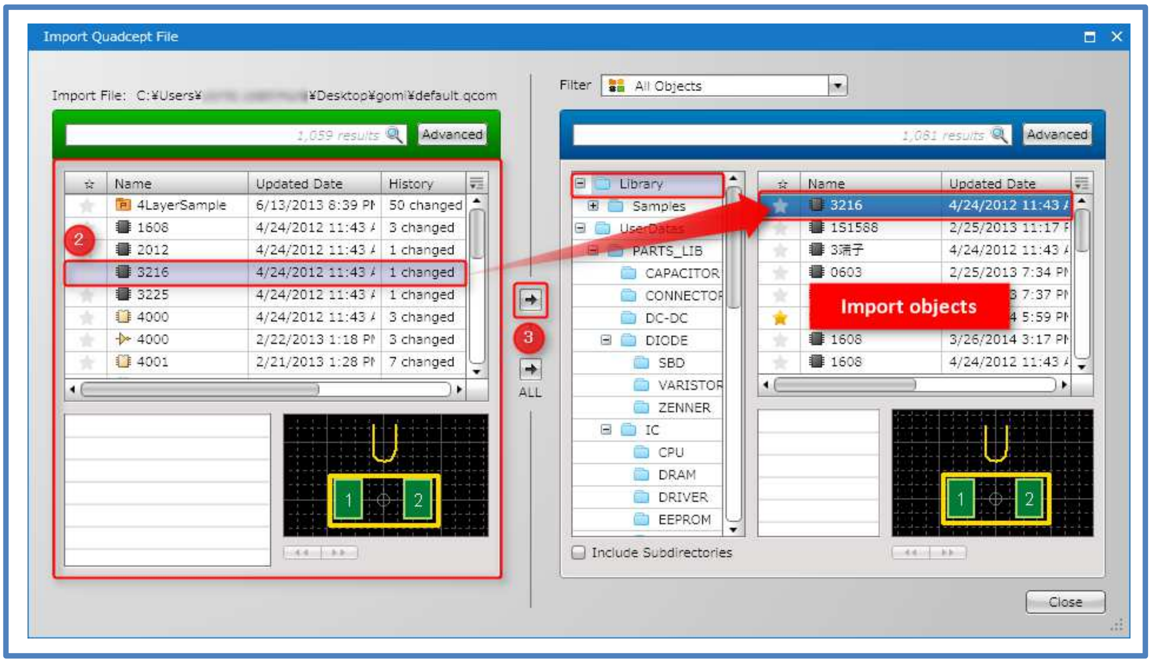
1. Ribbon <File> → 【Import】 → Select【Import Quadcept File】
2. Choose objects you want to import from left side
3. Click【→】to select import objects

   Objects are added to the selected directory

   * Object of all in Qcom file to click the【All →】will be imported.

* If you want to import objects of the same ID, the overwrite confirmation message is displayed. Please choose overwrite or import as another ID.

* It will be imported at the time it was added to the directory list on the right side.



On a Final Note…

We have explained how to create a schematic/PCB and export gerber data based on the workflow.
Having learned these tasks, you will be able to design a basic schematic/PCB on Quadcept.
This tutorial manual provides a basic introduction to Quadcept and covers only basic functions.
In order to use the CAD more efficiently, we not only provide support for the operation of Quadcept, but also try our best to fulfill feature requests from customers. Our aim is to offer the daily evolving CAD system with its users.
If you have any queries or requests, please do not hesitate to contact us.

Quadcept Tutorial
Publication of the 1st edition August 2014
Author Quadcept Inc.