

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/



About this tutorial
Chapter 1 Getting Started
Screen Layout
Customizing Menu
Customizing Ribbon9
Customizing Toolbar11
Using the Mouse
Scroll the design area13
Zoom in/out the image, Fit to screen13
Filter Function
Doing Filter
Setting a Filter Group15
Stroke Function
Customizing Stroke Menu17
Chapter 2 Circuit Designer
Schematic Design Flow19
Concept of Component
Shape of Schematic Component
Creating Symbol
Pin23
Rotation and Mirroring24
Registering Component
Attribute
Input by setting attribute in the Attribute Category26
Creating Schematic
Step 1. Creating a New Schematic
■ Step 2. Placing Component



Step 3-a. Wiring	31
How to draw wire/line	32
This section describes various ways to draw [Line] to create shape or [Wire]	32
■Step 3-b. Auto Wiring	35
Step 3-c. Bus/Parallel Wiring	
Step 3-d. Placing Label	
Step 3-e. Placing Port	
■ Step 4. Verifying Schematics (ERC/DRC)	
Step 5. Export BOM	41
■Step 6. Export Netlist	42
About NET CHANGER	43
Step 7. Print Schematic	44
Chapter 3 PCB Designer	45
PCB Design Flow	
Creating Shape of PCB Component (Footprint)	
Creating Shape of PCB Component (IPC Footprint)	
Registering Component	
Creating PCB	50
Step 1. Annotation	
1-a. When designed the schematic on Quadcept	
1-b. When designed the schematic with other CAD	
Step 2. Layer Settings	
■ Step 3. Design Rules	55
■ Step 4. Drawing Board Outline	58
4-a. Drawing a board outline directly	58
4-a. Drawing a board outline unectry	
4-c. Reading IDF data for a board outline	
■ Step 5. Moving Component	
5-a. Move a component by mouse drag	
5-b. Move component in Move mode	



5-c. Move component by entering the coordinates	63
5-d. Move by the center of the pad as the reference point	64
5-e. Move by selecting the component in the Object window	65
5-f. Place (move) component from schematic	66
■Step 7. Routing	67
Convenient Functions for Routing	72
■Step 8. Types of Plane/Creating Plane	75
■Step 9. Board Slit	79
■Step 10. Verifying PCB (DRC/MRC)	80
■Step 11. Export Manufacturing Data	82
Chapter 4 Handling of Data	86
Annotation	87
Sharing Data	90
Import/ Export Data (Quadcept File)	91
On a Final Note	93



Chapter 1 Getting Started





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 $Settings \rightarrow Select Customize Menu$
- 2. Click [+] tab that appears on the ribbon menu
- 3. The [Create New Tab] dialog appears. Enter the tab name and click $\hbox{\tt [OK]}$.

Quadcept File	Create New Tab X Enter Tab Name. User O K Cancel
Create New Open	*To delete a ribbon tab: 【 Settings 】→ Select【 Customize Menu 】 Click [x] that appears next to the tab



Add a menu

- 1. Menu bar $[Settings] \rightarrow Select [Customize Menu]$
- 2. Select the icon you want to add from the [Customize Menu] dialog. Drag and drop into 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].
- 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.

Customize Meru Customize Meru	Create New Toolbar Enter Toolbar Name User O K Cancel *When a new toolbar is created, you'll see
Erabes Name Default User Use Default Use Default New [Deese]	the toolbar name in the [Toolbar] tab and the new toolbar appears around the bottom left corner.
Cutomize Meru Meru Tooler Enabled Name Cutomize Loriout Uder Default New Coest	* To delete a toolbar: Menu 【Settings】→ Select【Customize Menu】 Select the toolbar you want to delete in the <toolbar> tab and click【Delete】.</toolbar>



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.







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.



* The	re are three types of shortcuts for fit to scr	een;	
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.

ilter List				All ON All C
Name		Object	×	Draw
All Items		Pin		Line
Component		Pin point		Rectangle
Net		Power Supply	1	Polygon
Text		Port		Circle
Draw		Port Address	1	Arc
	1	Symbol	1	Image
	V	Component		Text
		Wire		Reference
		Junction		Attribute
		Bus		
		Label		
		Note		
		Arrow		
		Free Line		
		Single Point Grounding		

E	nable Filter		All ON All OF
~	Object	V	Draw
\checkmark	Pin	\checkmark	Line
\checkmark	Pin point	\checkmark	Rectangle
~	Power Supply	\checkmark	Polygon
\checkmark	Port	\checkmark	Circle
\checkmark	Port Address	\checkmark	Arc
\checkmark	Symbol	\checkmark	Image
\checkmark	Component	\checkmark	Text
\checkmark	Wire	\checkmark	Reference
\checkmark	Junction	\checkmark	Attribute
\checkmark	Bus		
\checkmark	Label		
\checkmark	Note		
\checkmark	Arrow		
\checkmark	Free Line		
\checkmark	Single Point Grounding		
- 1	Filter List		
	Add current filter to the lis	it.	Ctrl+A
	Edit Filter List		Ctrl+E
	All Items		1
	Component		2
	Net		3
	Text		4
	Draw		5



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] \rightarrow 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

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)

	ing inte	Induon	can be set	up into the Quadcept of	component.
mponer	nt Nam	e (74L	508)		
Referer				Symbol	Link
		ce text of ent can b	e set up.	Gate A	ternate
Value II	nforma	tion		Gate B	ternate
Maker	0	N Semico	nductor	Gate C	ternate
Cost	\$	2			
Stock	2	,000		Gate D 🗐 🔶	
Discontir	nued M	ay 2012			ternate
Compone an be se		oute info	rmation	Component shape or created as a symbol,	
Pin Info	rmatio	n		Footprint	Link
PinNumber	PinName	Electrical	Swap		
1	1A	Imput	SWAP/A/1		14
2	2B	Imput	SWAP/A/2		19
3	1Y	Output	SWAP/A/3		U1
4	2A	Imput	SWAP/B/1		18
ŧ	÷	ŧ			
	VCC	Power			



a	Reference U1		Set as Mech	om Renumbering nanical Components (excluded from net oounted Component	list)
	Read Attribute				
	Attribute	Value			
	Value				
	Package		Attribute		
	Large Category		Attribute		
	Small Category			_	
	🔒 Maker				
Add Delete	Part Number				
	G Shape				
01	Cost				
300 10R1 400 10R2 1 c ⁵ の 500 10R3					
50 10R3					
11(1) 20R1					
12(12) 20R2 2 0 ¹¹² (10)					
			\		
Value	Add Delete 1	• •			



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> \rightarrow Select [Pin] \rightarrow [Sample]

Creating Symbol

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

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

Ribbon <Draw> \rightarrow 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 $\langle Draw \rangle \rightarrow Select \ Pin \ \rightarrow Sample$

- Open a sheet to create the new pin
 Ribbon <File> → 【Create New】 → 【Other】 →Select 【Pin】
- 2. Create the pin shape
 - Ribbon $\langle Draw \rangle \rightarrow Select$ [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> \rightarrow 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.

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.







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.

- 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

- Register the symbol (shape of schematic component)
 Double click and open [Select Component] dialog and select the symbol
- Register the footprint (shape of PCB component)
 Double click and open [Select Component] dialog and select the symbol
- 6. Save

Ribbon <File> \rightarrow [Save] or [Save as] Select a directory and save it with a name

Component1	Reference U	Set a	uded from Renumbering as Mechanical Components (excluded from netli as Unmounted Component
	Read Attribute		
	Attribute	Value	
	Value Cost		
Add Delete			



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>
- 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 **]**
- 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> \rightarrow Click [Create New] \rightarrow [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.

Project) ()
🔤 💷 📚 🔍 🛅	
Create New Open Rem	ove
File	State
🖃 💀 SampleProject	
Sheet1	
Sheet2	
PCB1	
Panel1	
sample_doc.txt	
sample.pdf	
Project	
Project	
Project Schematic sheet	
Project Schematic sheet PCB sheet	

*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】
- 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> \rightarrow 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> \rightarrow 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] \rightarrow Select [Change the Bending Angle] or press [S] key *You can check the bending angle in the [Property] window.



How to switch angle





Return to a previous state

[Right click] \rightarrow Select [Pushback] or press [Back Space] key





Edit line width





Cancel drawing





■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> \rightarrow 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> \rightarrow Select [Bus] to create bus routing
- 2. Ribbon <Draw> \rightarrow 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> \rightarrow 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> \rightarrow 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

				5 results	Advance
🖯 🗮 Library		÷	Name	Updated Date	History
🗄 🧰 Clotary		*	E BIDIR	4/24/2012 11:43 /	1 changed
🗄 🧰 UserD.		*	P BIDIRCircle	N 77	
	13775	*	🗠 Left	4/24/2012 11:43 /	
		1	C LeftCircle	4/24/2012 11:43 4	6 changed
		1	🖻 Right	4/24/2012 11:43 4	3 changed
🗅 Includa Su	hdirertoriae		<u> </u>	A[1:8]	
] Include Su			<u> </u>	A[1:8]	
Port Name	[A[1:8]		<u> </u>		
Port Name			<u> </u>		Auto Adjust
	[A[1:8]		<u> </u>		Auto Adjust

*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> \rightarrow Select [ERC/DRC] \rightarrow [ERC/DRC Settings]
- 2. Set each item for the rule checks
- 3. Select [Apply] and click [OK]
- 4. Ribbon < Completion > \rightarrow Select [ERC/DRC] \rightarrow [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

Project.Schematic				Net and Bus Rule Circuit Settin
Color	ER	c - Inconsis	tent	Net and Bus Rule
Objects	_			
	2	Error Type		Check Contents
Memo	1	C Error		Net connection that violates bus rule.
ERC	~	C Error		Two-byte characters in net name.
Inconsistent Component/Reference Rule	V	C Error		Same net power supplies but different symbol.
Inconsistent Net and Bus Rule	V	C Error		Same pins connected to different nets.
Undefined/Unconnected Objects	1			
Pin Connection Matrix				
DRC				
Clearance				
Distance				
Design Instructions				
Design Instructions	-	1 1 1	12	
	1			
		AO	_	
		Al		
		A2		
		В		
	-			
•(() •				A[0:2]/B
		1 1 1		
Save Settings Read 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 > \rightarrow 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 > \rightarrow Select **[** Export Netlist **]**
- 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/

NET CHANGER 🤤 Conversion Q Comparison FAQ Manual Releasenote Inquiry 🗾 Login	
Net list Conversion & Comparison	
You can compare schematics CAD nellists into other CAD formats! Wherever the Internet is available, whoever can use the latest version any lime, anywhere!	
 ▶ conversion ▶ comparison 	



■Step 7. Print Schematic

Print the schematic.

- 1. Ribbon <File> \rightarrow 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.

Rrinter Bullzip PDF Printer	Properties
Number of Copies 1	
Scaling 0.99	
Offset X: 0.00 + Centering Y: 0.00 + Centering	
Print Direction Horizontal	
Show Memo Memo Layer	
Comment 💿	
Todo S Knowledge S	-
Knowledge 🔮	aun an an ann ann ann ann ann ann ann an
Print Target	
Name 🗸	
4LayerSampleSCH 🗸	
Print All Sheets in the Project	



Chapter 3 PCB Designer





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



6. DRC/MRC

 $\label{eq:perform} \mbox{Perform DRC} \ (\mbox{Design Rule Check}) \ \mbox{and MRC} \ (\mbox{Manufacturing Rule Check})$

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 \langle File \rangle \rightarrow \langle Create New \rangle \rightarrow Select \langle Footprint \rangle
2. Place a pad
     Ribbon \langle Draw \rangle \rightarrow Select [Pad] to place
3. Input silk
     Press [L] key to set the current layer as [Silk] (or [Right click] \rightarrow Select [Change Layer] and
     set as [Silk]
     Ribbon <Draw> \rightarrow Use [Line] or [Rectangle] or [Circle]
4. Adjust the Reference position
     Move the reference [U] by mouse drag
5. Set the origin point
     Ribbon \langle Draw \rangle \rightarrow Select [Move Origin Point] and click on the desire point or while
     selecting the object [Right click] \rightarrow [Move Origin Point to Center]
6. Input dimensions
       [Create PCB] \rightarrow Select [Create Component Dimension Automatically]
7. Save
```

Ribbon <File> \rightarrow 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.

- 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> \rightarrow 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).

- Open a new sheet to create a component Ribbon <File> → 【Create New】 → Select 【Component】
 Set Reference
 Input attributes
 Register Symbol (shape of schematic component) Double click and open the [Select Component] dialog and select a symbol
- Register Footprint (shape of PCB component)
 Double click and open [Select Component] dialog and select a symbol

6. Save

Ribbon <File> \rightarrow Select [Save] or [Save as]

Select a directory and save it with a name.

5 II NEW	Reference U	G Set	luded from Renumbering as Mechanical Components (excluded from netil as Unmounted Component
	Read Attribute		
	Attribute	Value	
	Value		
	😡 cost		
Add Delete			
	_		
		Li.	



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 > \rightarrow 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 $\langle File \rangle \rightarrow Select \ Create New \ \rightarrow \ PCB Project \$
- 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.
- 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 $[Settings] \rightarrow [Settings] \rightarrow [Layer]$
- 2. Set each layer in the [Physical Layer] and [Layer Settings] and Click [Apply] \rightarrow [OK]

Physical Layer Setting Screen

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

Project.PCB2	Laver						7	35	P	
👝 Layer					-		1		V	
Dojects	Physical Layer Lay	er Settings								
BOM										
DCB Print	Layer	Thickness				Materia				
COB++	Тор	0.043	Conductor	Route	10	Copper				
Gerber		0.2	Insulator		_	FR4	٠		4.6	0.018
Dr. NC Drill	Layer2	0.035	Conductor	Plane	•			59		
🗀 NC Drill List		1.1	Insulator		_	FR4			4.6	0.018
Component Coordinate	Layer3	0.035	Conductor	Plane	٠			59		2.2.2
		0.2	Insulator		_	FR4			4.6	0.018
	Bottom	0.043	Conducto:	Route		Copper		59		***
									9	
	Add Delete								Mater	ial Settings
	Board Thickness 1.6	Platic	g Thickness	0.025						

Layer Setting Screen

Object layers are available.

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

Project.PCB2	Layer		100	
🗀 Layer				_
Objects	Physical Layer Layer Settings			
BOM				
DCB Print	Layer	👁 🔒 Calor	Fill Style	-
CDB++	🗃 🧰 Тор	3		m
📄 Gerber	UT SIK	3		
NC Drill	🗃 🖉 Electric	a		
NC Drill List	all Route	Ð		
Component Coordinate	il Plane	9		
	all PadStack	9		
	J ViaStack	9		
	all Paste	3		
	ill Solder	9		
	all Assembly	a	5000000000	
	ill KeepOut	3		
	J DesignRule	1		
	2 Dimension	•		
	🗄 🧰 Layer2	9		
	🖯 🖉 Electric	•		
	all Route	3		
	ill Plane	Ð		
	ill PadStack	9		
	ill ViaStack	3		
	I KeepOut	3		9
Save Settings Read Settings	Add Delete			



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 > \rightarrow Select (DRC/MRC Settings) \rightarrow (Net Class)
- Click 【Add】 at the bottom of [Net Class] to add a "Class"
 *At first, all nets are included to "Default Class".
- Choose nets you want to add to the "Class" from "All Net" Click 【←】 to select [multiple selections available]
- 5. When you finish selecting, click $[Apply] \rightarrow [OK]$

Project.4LayerSamplePCB	DRC - Net Class		PCB Setting
Net Class*	[
🖻 🧰 Rule Check	Net Class	Selected Net	All Net
🗀 Clearance	DefaultClass	Net Name 🔺	Net Name 🔺
Same Net Clearance	Class1	FG	
🗀 Route/Via*		GND 🔶	CLK2
🙆 Route Length/Isometric Routing*		VCC	FG
🛄 Differential Pair			GND
🛅 Tear Drop*			Sig_1
Dynamic Plane Connection*		1 I I I I I I I I I I I I I I I I I I I	Move the selections to
🗀 Jumper			Selected Net
📋 Test Land			
Design Instructions			Sig_5
Other DRC Settings			4 - Sig_6
🗏 🤖 MRC			Sig_7
🗏 🦲 Rule Check			Sig_8
Clearance			Sig_9
Component Clearance			Sig_10
Reference Placement Angle			Sig_11
Solder Resist/Paste			Sig_12
Pad on Via			Sig_13
			Sig_14
			Sig_15
• ()) • (3	•	
•()•]			
Save Settings Read Settings	Add Delete	Filter	Filter



Clearance Settings

- 1. Open the PCB sheet you want to set
- 2. Ribbon <Complete> \rightarrow Select [DRC/MRC] \rightarrow [DRC/MRC Settings]
- 3. Select 【Clearance】 from the list on the left
- 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.
- Select <Assignment> tab → Assign the clearance setting "Setting1" by selecting from the [Setting]

The clearance "Setting1" is set.

Project.4LayerSamplePCB	Clearance	РСВ 5	
DRC			_
Net Class*	Assignment	Advanced Settings	
3 Rule Check			
Clearance*	Settings	5 Layer	
Same Net Clearance	DefaultSetting	All Ald Delete	
Route/Via*	Setting1		
📄 Route Length/Isometric Routir	ng*	Clearance Settings	
🚞 Differential Pair			
🚞 Tear Drop*		All Route Via Through SMD Plane Drill LV	VH
Dynamic Plane Connection*		Route 0.15	
Dumper 📄		Via 0.15 0.30	
i Test Land			
Design Instructions		Through 0.25 0.25 0.25	
🗀 Other DRC Settings		SMD 0.25 0.25 0.25 0.25	
MRC		Plane 0.25 0.25 0.25 0.25 0.25	
🗏 🧰 Rule Check			
🚞 Clearance		Drill 0.50 0.50 0.50 0.50 0.50 0.50	
🗀 Component Clearance		LVH 0.35 0.35 0.35 0.35 0.35 0.50 0.50	50
🗀 Reference Placement Angle			
🚞 Solder Resist/Paste		Board 1.00 1.00 1.00 1.00 1.00 1.00 1.00	0
🚞 Pad on Via		Non-Electric Objects on Electric Layer Clearance	
		Non-Electric Objects 0.3	
•	→		
Save Settings Read Settings		O K Cancel As	ppiy
		O K Cancel As	ppiy
RC/MRC Settings		O K Cancel As	ppiy
RC/MRC Settings			
	Clearance		
RC/MRC Settings Project.4LayerSamplePCB	Clearance	O K Cancel As	
RC/MRC Settings Project.4LayerSamplePCB	Clearance	PCB Sc	
RC/MRC Settings Project, 4LayerSamplePCB DRC Net Class*	Clearance		
RC/MRC Settings Project 4LayerSamplePCB DRC Net Class* B Rule Check	Clearance	PCB Sc	
RC/MRC Settings Project 4LayerSamplePC5 DRC Ref Cless* Rule Check CleanAce*	Assignment	Advanced Settings	
RC/MRC Settings Project 4LayerSamplePCB DRC Net Class* Clearance* Clearance* Same Net Clearance	Assignment	Advanced Settings	
RC/MRC Settings Project 4LayerSamplePCB DRC Net Class* Rule Check Clearance* Same Net Clearance Rule Clearance Rule/Ne*	Assignment Net Class DefaultClass Class1	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePC5 DRC Net Class* Class* Clasance* Clasance* Same Net Clearance Route Length/Isometric Routh	Assignment Net Class DefaultClass Class1	Advanced Settings Office PCB Settings Opfaul Setting Perfaul Setting Defaul Setting Perfaul Setting	
RC/MRC Settings Project, 4LayerSamplePCB DRC RC RC Learance* Clearance* Route Check Route //la* Route Length/Isometric Routif Differential Pair	Assignment Net Class DefaultClass Class1	Advanced Settings	
RC/MRC Settings Project ALayerSamplePCB © DRC © Net Class* © Rule Check © Clearance* © Same Net Clearance © Route Length/Dametric Routif © Differential Pair © Tear Drop*	Assignment Net Class DefaultClass Class1	Advanced Settings Office PCB Settings Opfaul Setting Perfaul Setting Defaul Setting Perfaul Setting	
RC/MRC Settings Project. 4 LayerSamplePCB BC RC Net Class* Clasance* Clasance* Clasance* Route Length/Isometric Routh Differential Pair Tear Drop* Chymatic Plane Connection*	Assignment Net Class DefaultClass Class1	Advanced Settings Office PCB Settings Opfaul Setting Perfaul Setting Defaul Setting Perfaul Setting	
RC/MRC Settings Project, 4LayerSamplePCB DRC RC RC Learance* Clearance* Route_Vla* Route_Leath/Isometric Routif Differential Pair Tear Drop* Dynamic Plane Connection* Dynamic	Assignment Net Class DefaultClass Class1	Advanced Settings Office PCB Settings Opfaul Setting Perfaul Setting Defaul Setting Perfaul Setting	
RC/MRC Settings Project. 4 LayerSamplePC5 DRC Rev Clease Rule Check Clearace* Same Net Clearance Route/Via* Route Length/Isometric Routi Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land	Assignment Net Class DefaultClass Class1	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePCB RC Net Class* RC	Assignment Net Class DefaultClass Class1	Advanced Settings	
RC/MRC Settings Project. 4LayerSamplePCB Project. 4LayerSamplePCB Project. 4LayerSamplePCB Rule Check Clearance* Same Net Clearance Route, Vila* Route Length/Isometric Routhi Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land Design Instructions Other DRC Settings	Assignment Net Class DefaultClass Class1	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePCB DRC Net Class* Clasance* Same Net Cleance* Route/Via* Route Length/Isometric Routir Differential Pair Test Drop* Dynamic Plane Connecton* Jumper Test Land Design Instructions Other DRC Settings MRC	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePCB RC	Assignment Net Class DefaultClass Class1	Advanced Settings	
RC/MRC Settings Project, 4LayerSamplePCB Project, 4LayerSamplePCB Project, 4LayerSamplePCB PRC Rule Check Clearance* Rule Check Rule Check Rule Check Rule Check Prop* Project All Pair Prop* Pr	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePCB DRC Net Class* Clasance* Clasance* Same Net Clearance Route Length/Izometric Routir Differential Pair Tear Dop* Dynamic Plane Connection* Jumper Test Land Design Instructions Other DRC Settings Rule Check Clearance	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePCB Project 4 LayerSamplePCB RC Net Class* Clasance* Clasance* Same Net Clearance Route Length/Izometric Routin Ufferential Pair Tear Orop* Ufferential Pair Tear Comp* Ufferential Pair Teat Land Design Instructions Other DRC Settings MRC RC RC Clearance Clearance Clearance Reference Pacement Angle	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project. 4 LayerSamplePC5 DRC Net Class* Rule Check Clearance* Same Net Clearance Rule/Via* Route Length/Isometric Routy Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land Design Instructions Other DRC Settings MRC Clearance Component Clearance Camponent Clearance Camponent Clearance Camponent Clearance Reference Plane	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project 4 LayerSamplePCB Project 4 LayerSamplePCB RC Net Class* Clasance* Clasance* Same Net Clearance Route Length/Izometric Routin Ufferential Pair Tear Orop* Ufferential Pair Tear Comp* Ufferential Pair Teat Land Design Instructions Other DRC Settings MRC RC RC Clearance Clearance Clearance Reference Pacement Angle	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project. 4 LayerSamplePC5 DRC Net Class* Rule Check Clearance* Same Net Clearance Rule/Via* Route Length/Isometric Routy Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land Design Instructions Other DRC Settings MRC Clearance Component Clearance Camponent Clearance Camponent Clearance Camponent Clearance Reference Plane	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project. 4 LayerSamplePC5 DRC Net Class* Rule Check Clearance* Same Net Clearance Rule/Via* Route Length/Isometric Routy Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land Design Instructions Other DRC Settings MRC Clearance Component Clearance Camponent Clearance Camponent Clearance Camponent Clearance Reference Plane	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project. 4 LayerSamplePC5 DRC Net Class* Rule Check Clearance* Same Net Clearance Rule/Via* Route Length/Isometric Routy Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land Design Instructions Other DRC Settings MRC Clearance Component Clearance Camponent Clearance Camponent Clearance Camponent Clearance Reference Plane	Asignment Net Class DefaultClass Class1 Add Area Clearance between	Advanced Settings	
RC/MRC Settings Project. 4 LayerSamplePCB Project. 4 LayerSamplePCB RC RCM RC Clearance* Clearance* Route./vla* Route.eught/Isometric Routin Differential Pair Tear Drop* Dynamic Plane Connection* Jumper Test Land Delign Instructions Other DRC Settings MRC RC RC Clearance Clearance Clearance Clearance Reference Pacement Angle Solder Resist/Paste Ped on Via	Asignment Net Class DefaultClass Class1 Add Area Clearance betwe Net Class1	Advanced Settings	
RC/MRC Settings Project. 4 LayerSamplePCB Project. 4 LayerSamplePCB RC	Asignment Net Class DefaultClass Class1 Add Area Clearance betwe Net Class1	Advanced Settings	



Route Settings

- 1. Open the PCB sheet you want to set
- 2. Ribbon <Complete> \rightarrow Select [DRC/MRC] \rightarrow [DRC/MRC Settings]
- 3. Select [Route/Via] from the list on the left
- 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]
- 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.
- Press [L] key and set the current sheet as 【Other.Board】 (or 【Right click】→ 【Edit Layer】
 →set it to [Other.Board])
- 3. Ribbon <Draw> \rightarrow Use [Line] or [Rectangle] or [Circle] to draw the line

	Change La Top.Silk	yer		
	Key 41 42 43 44 45 46 47 2 48 01 02 03 04	Value Value Bottom.Silk Bottom.Electric Bottom.Paste Bottom.Assembly Bottom.KeepOut Bottom.DesignRule Bottom.Dimension Other.ReepOut Other.DesignRule Other.DesignRule Other.Dimension		
		O K Car	icel	



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

- 1. Open the PCB sheet where you want to draw the board outline.
- 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] \rightarrow 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 (\rightarrow)

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

 $\mathsf{Ribbon} < \mathsf{File} > \rightarrow \mathsf{Select} \texttt{[Import]} \rightarrow \texttt{[Import IDF]}$

3. The <Import IDF> dialog appears

Click [...] at the end line of [Input File] to open \rightarrow Select the IDF file

Check the **[**Enter Board Outline **]** box \rightarrow click **[**OK **]**

sers¥		123
utline		
	A	
		Cancel
		hent Coordinat



■Step 5. Moving Component

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

5-a. Move a component by mouse drag

- 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 \rightarrow Ribbon $[Draw] \rightarrow$ 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 \rightarrow 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】

Symbol List	Attribute Pin			
CSTCEXXM00G			Excluded from R	
	Layer Top Angle 180		Show Silk]
	2 🖸 Lock Componen		Specify Assemble Component Height	
	X 1.66 Y -11.11			
	Read Attribute			
	Attribute	Value	1	
	Value			
	Maker		= =	
	Add Delete	•		



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> \rightarrow 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
- 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 \rightarrow Ribbon < Draw> \rightarrow 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: $[Right click] \rightarrow [Change the Bending Angle] or [Bending Angle] in the [Property] window$





Switch Angle of Routing







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 \rightarrow **[**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 \rightarrow [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] \rightarrow [Settings] \rightarrow [System] \rightarrow [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] \rightarrow [Settings] \rightarrow [System] \rightarrow [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] \rightarrow [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

- 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]







*When you edit the routing or component placement after you draw the dynamic plane, you can update the plane by; [Double click] the plane \rightarrow [Rebuild Plane] or with the keyboard shortcut [|].





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> \rightarrow 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.





■Step 10. Verifying PCB (DRC/MRC)

Verify the printed circuit board before the output.

- 1. Ribbon< Completion > \rightarrow [DRC/MRC] \rightarrow Select [DRC/MRC Settings]
- 2. Select 【Rule Check】 and set what you want to verify
- 3. Select 【Apply】 and click 【OK】
- 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

Project.4LayerSamplePCB	DI	RC - Rule Cl	neck		PCB Setting
🗏 🧰 DRC	2	_			
Net Class		Error Type		Туре	Check Contents
😑 🚞 Rule Check*	\checkmark	😂 Error	•	Unconnected Net	Unconnected routes.
Clearance	V	Error	•	Clearance	Clearance rule.
Same Net Clearance	V	Error	•	Via Rules	Via rule.
🛅 Route/Via	V	Error	-	Route Width	Route width rule.
🚞 Route Length/Isometric Routing	\checkmark	Error	•	Route Length/Isometric Routing	Route length and isometric routing rule.
🚞 Differential Pair	\checkmark	Error	•	Differential Pair	Gap/route length/tolerance of differential pair.
🛅 Tear Drop	\checkmark	Error	•	Arc Route	If the edge shape of routes is arc.
Dynamic Plane Connection	\checkmark	C Error	•	Dangling Route	The existence of dangling routes.
🚞 Jumper	\checkmark	Error	•	Route Angle	Unexpected route angle.
🚞 Test Land	\checkmark	Error	•	Tear drop (Through)	The existence of tear drop of Through.
Design Instructions	V	Error	•	Tear drop (SMD)	The existence of tear drop of SMD.
Other DRC Settings	\checkmark	C Error	-	Tear drop (Via)	The existence of tear drop of via.
B 💼 MRC	V	Error	•	Dynamic Plane is divided	If a dynamic plane is divided.
🗏 🧰 Rule Check	\checkmark	Error	-	Polygon/Twisted Plane	The existence of twisted polygon/twisted plane.
Clearance	V	Error	•	Unconnected Plane	The existence of unconnected plane.
Component Clearance	\checkmark	C Error	•	Keep Out Area	Keep Out violation.
🚞 Reference Placement Angle	\checkmark	Error	-	Thermal	The minimum number of thermal connection.
🚞 Solder Resist/Paste	V	Error	•	Test Land	The existence of test land.
🚞 Pad on Via	V	Error	•	Design Instructions	Unconfirmed design instructions.
	V	Error	•	Edit Component	Edited component.
	\checkmark	Error	•	Annotation	The differences between PCB and Schematic.
Save Settings Read Settings		Check Clearar	nce o	f Displayed Objects only Ma	aximum Number of Errors 100



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

roject.4LayerSamplePCB	Gerber		PCB Se	
🗀 Layer				-
Dbjects	Output Layer Output	t Settings Aperture		
BOM	2 V Batch List	Laver	0	
PCB Print ODB++	✓ TopPattern	🗄 📄 Top	3	_
Gerber	TopSilk	B B Electric	0	
Gerber	✓ TopSolder	Route	3	
	✓ TopPaste	Plane	3	
Component Coordinate	✓ Layer2Pattern	PadStack	3	
Component Coordinate	✓ Layer3Pattern	ViaStack	0	
	✓ BottomPattern			
	✓ BottomSilk			
	✓ BottomSolder			
	✓ BottomPaste			
	V Board			
	1			
		-		
Save Settings Read Settings	Add Delete	Show Output Layers only		
	8			









Export Drill Data

- 1. Ribbon < Completion > \rightarrow Select [NC Drill]
- 2. Select [NC Drill Settings] and set and confirm the export content
- 3. Ribbon < Completion > \rightarrow Select [NC Drill] \rightarrow [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 $\$ \rightarrow $\$ Batch output $\$

Project.4LayerSamplePCB	NC Drill		2005	PCB Setting
🗀 Layer				
i Objects	Format			
BOM	Format			
PCB Print	Unit	mm		
CDB++	Number of Integer Digits	4		
Gerber	Decimal Places	4		
NC Drill List	Zero Suppression			
Component Coordinate	and and a second s	Reading		
Component Coordinate	Offset X	0.00		
	Offset Y	0.00		
	Long Hole Output Type	Center + Both Ends		
	Output Report File	- 14		
	Extension rpt			
Save Settings Read Settings				









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 > \rightarrow 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】





Updat	e File	: 🖪 4Laye	rSample/4LayerSampleF	СВ		
Souro	e for Ann	otation: 💷 4Laye	rSample/Circuit			
	ion List	-				
	Target	Circuit	PCB		Transfer Content	Details
1		GND,FG		~	Add Single Point Grounding	$() \rightarrow \text{GND}, \text{FG}$
2			GND,FG	~	Delete Single Point Groundin	$GND, FG \rightarrow ()$



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 > \rightarrow 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.



Updat	e File	: 🗰 4Laye	rSample/Circuit			
		†				
Souro	e for Ann	otation: 🔳 4Laye	rSample/4LayerSampleF	СВ		
nnotat	ion List					
No. 🔺	Target	Circuit	PCB	Apply	Transfer Content	Details
1			GND,FG	1	Add Single Point Grounding	$() \rightarrow \text{GND}, \text{FG}$
2		GND,FG		~	👰 Delete Single Point Groundin	$GND,FG \rightarrow ()$
		1				



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.

- [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

System 🔹	Storage Space (Sharing Settings)	System Setting
🗏 💼 Draw		- · ·
Schematic		
DCB	Use Local Storage	
D Route	Storage Space (Sharing Settings)	Use Default
Drawing Priority		
Auto Backup	Store in myArea	
Shortcut Key	Not connected to server due to disabled setting for using my	Area.
Storage Space (Sharing Settings)	https://guadcept.com/team/storage	
Default Environment		
🖻 🧰 Stroke		
📄 Drawing Frame		
Pin/Power Supply/Port		
៉ Symbol		
🔲 Schematic		
Device Block		
i Footprint		
DCB		
🦲 Panel		
👜 3D		
Property Display		
Save Settings		3 OK Cancel Apply

*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> \rightarrow [Export] \rightarrow Select [Export Quadcept File]
- 2. Choose objects you want to export to the "Quadcept File" from left side
- 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

Objects are added to the selected directory

* Object of all in Qcom file to click the $\mbox{[All} \rightarrow \mbox{]}$ 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.

port F	file: C:¥Users¥	¥Desktop¥ç	gomi¥default	qcom	Filter All Objects			
		1,059 results	Advan	ced			1,	061 results 🍳 🛛 Advanc
ਸ਼ੇ	Name	Updated Date	History	=	🖯 🛄 Library	#	Name	Updated Date
*	🔋 4LayerSample	6/13/2013 8:39 PM	50 changed		🕀 📄 Samples	*	3216	4/24/2012 11:43 /
0	1608	4/24/2012 11:43 /	3 changed		🖯 🗋 UserDatas	1 12	1\$1588	2/25/2013 11:17 F
4	2012	4/24/2012 11:43 4	1 changed		PARTS_LIB	*	📳 3 端子	4/24/2012 11:43 4
	3216	4/24/2012 11:43 /	1 changed		CAPACITOR	*	0603	2/25/2013 7:34 PM
A.	3225	4/24/2012 11:43 /	1 changed	1	CONNECTOR	*		3 7:37 PM
	() 4000	4/24/2012 11:43 /	3 changed		DC-DC	*	Import o	4 5:59 PM
	- 4000	2/22/2013 1:18 PM	3 changed		🖃 🚞 DIODE	*	1608	3/26/2014 3:17 PM
	4001	2/21/2013 1:28 PM	7 changed	4 1	💼 SBD	*	1608	4/24/2012 11:43 4
		1) ,	ALL	C VARISTOR	•∈) •
					ZENNER.	1	1.000	
					🖃 🧰 IC	-		
					CPU	3		
					DRAM			
			€ 2		DRIVER			1 🔶 2
					EEPROM	-		
		F.44 [33.]			Include Subdirectories			



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.

©2014 Quadcept, Inc. All rights reserved.

This contents —including images, text, and so on —are owned by Quadcept corporation to use the content. Quadcept cannot grant you permission for content that is owned by third parties. You may only copy, modify, distribute, display, license, or sell the content if you are granted explicit permission within the End-User License Agreement.

*It is prohibited to make unauthorized copies, reproductions, data tapes or data files outside the Copyright Act.

*Please contact us for replacement if the pages are out of order or missing.

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