We released Quadcept 9.3.15 that includes part link with the world's leading online electronic part companies, Mouser Electronics. In this document, we have highlighted some of newly added and enhanced features.

| New/Improved | Category | Page | |
|---|---|---|---|
| **Common** | | | |
| New | Link | P1 | Quadcept signed a partnership alliance with Mouser Electronics. You can now search for their parts, import their part info to your libraries and export a BOM file in their format during design. |
| New | Setting | P11 | Added indicators on the screen which show license acquisition status. |
| **Circuit Designer** | | | |
| New | SCH | P2 | Added a feature that allows you to insert a power symbol while wiring. |
| **PCB Designer** | | | |
| New | Link | - | Added a feature to export a PCB data in ASCII format. (Quadcept Design File) |
| New | Drawing | P3 | Enhanced a differential pair routing feature. |
| New | Drawing | - | Added a feature that permits you to directly change route length. |
| New | Drawing | P4 | Enhanced a copy feature, and it now allows you to designate a reference point for pasting copied objects. |
| New | Drawing | - | Fixed an issue where a mouse cursor did not move according to a GUIDE setting when designating a reference point in Move and Route Move mode. |
| New | Drawing | - | Added a feature to copy a pad shape when creating a pad stack. |
| New | Drawing | P5 | Added a Subtract feature which allows you to cut selected objects. |
| New | Drawing | P6 | Added a feature that allows you to edit footprints placed on a PCB/panel sheet. |
| New | Drawing | P7 | Multiple objects can be now converted into filled objects in a lump. |
| New | Drawing | - | Filtering can be now enabled or disabled by clicking an item name in the Filter dialog. |
| New | Setting | P8,P9 | Expanded GRID settings.      `Specification Change` |
| New | Setting | P10 | Increased verifiable objects by a MRC item, Text Placement Angle. |
| New | Setting | - | The display location of a drill list is now adjustable by entering offset values. |
| **Improvements** | | | |
| Improved | Common | P11 | Improved the performance when searching for objects. |
| Improved | Common | - | Enhanced the performance when opening and removing project files. |
| Improved | Common | - | Enhanced the Array Placement function for pins and pads, and it now allows you to specify an incremental value. |
| Improved | Common | - | Changed to display customized layers in a Preview window. |

| New/Improved | Category | Page | |
|---|---|---|---|
| **Improvements** | | | |
| Improved | Common | - | A confirmation message is now shown when dragging and dropping a file to a different directory. |
| Improved | Common | - | Fixed an issue where sorting a list was not working properly in a Component Attributes List dialog. |
| Improved | Common | - | Fixed an issue where a license would not be able to be acquired once updating to "Windows 10 April 2018 Update" during standalone authentication. |
| Improved | SCH | P12 | Improved the processing of determining wire width when starting to draw a wire. |
| Improved | SCH | - | Extended the limit number of divisions of rows and columns when creating a drawing frame. (Max 702) |
| Improved | SCH | P12 | Changed the lower left of the maximum area to be a reference point when copying multiple objects. |
| Improved | SCH | - | Fixed an issue where unconnected pin settings were deleted when updating components. |
| Improved | SCH | - | Fixed an issue which caused an error when moving objects with Relative Move after specifying copy number. |
| Improved | SCH | - | Fixed an issue which caused an error when replacing a certain port. |
| Improved | SCH | - | Fixed an issue where the application became sluggish when many ports were arranged on a device block sheet. |
| Improved | SCH | - | Fixed an issue where a connection point placed on a device block sheet was not processed properly by annotation. |
| Improved | PCB | P13 | Changed the drawing order, and a current layer and layer type are now displayed at the forefront. |
| Improved | PCB | - | Object Snap is now available when placing a connection point. |
| Improved | PCB | - | Changed a Convert to Plane function, and it now converts selected objects into static planes. |
| Improved | PCB | - | An overwrite confirmation message is now shown when exporting Gerber files. |
| Improved | PCB | - | Fixed an issue where a Gerber aperture may not be able to be read with some tools. |
| Improved | PCB | - | Improved the display of drill objects. |
| Improved | PCB | - | Fixed an issue where highlights of objects were canceled when performing Cross Probe. |
| Improved | PCB | - | Fixed an issue where objects may be illegally placed in the Other layer with Move between Layers. |
| Improved | PCB | - | Fixed an issue where the clearance error between a board outline and a square / rectangular pad could not be detected properly. |
| Improved | PCB | - | Fixed an issue where a "Check Clearance of Displayed Objects only" function was not working properly. |
| Improved | PCB | - | Fixed an issue where a changed PCB sheet name was not reflected in a panel sheet. |
| Improved | PCB | - | Fixed an issue where the contents of "MountType" and "ElectricalpartType" were not output into a BOM file. |

# Quadcept Is Now Linked with Mouser Electronics.

## Access Their Part Information in Real Time Through Quadcept.



| Search Parts | Their parts are now searchable whenever you want during design. |
| --- | --- |
| Import Information | Part attributes such as manufacturer can be imported to your libraries. |
| Purchase Parts | You can check their part prices and export a BOM file in their format. |

### Description

■**Quadcept signed a partnership alliance with Mouser Electronics.**

- Quadcept is now linked with the world's leading part companies, Digi-Key, Chip1Stop, RS Components and Mouser Electronics.

- You can now search for their parts, import their part info to your libraries and buy their parts whenever you need during design.

- A BOM file for Mouser can be now exported from Quadcept.

# A Power Symbol Can Be Now Inserted While Wiring.
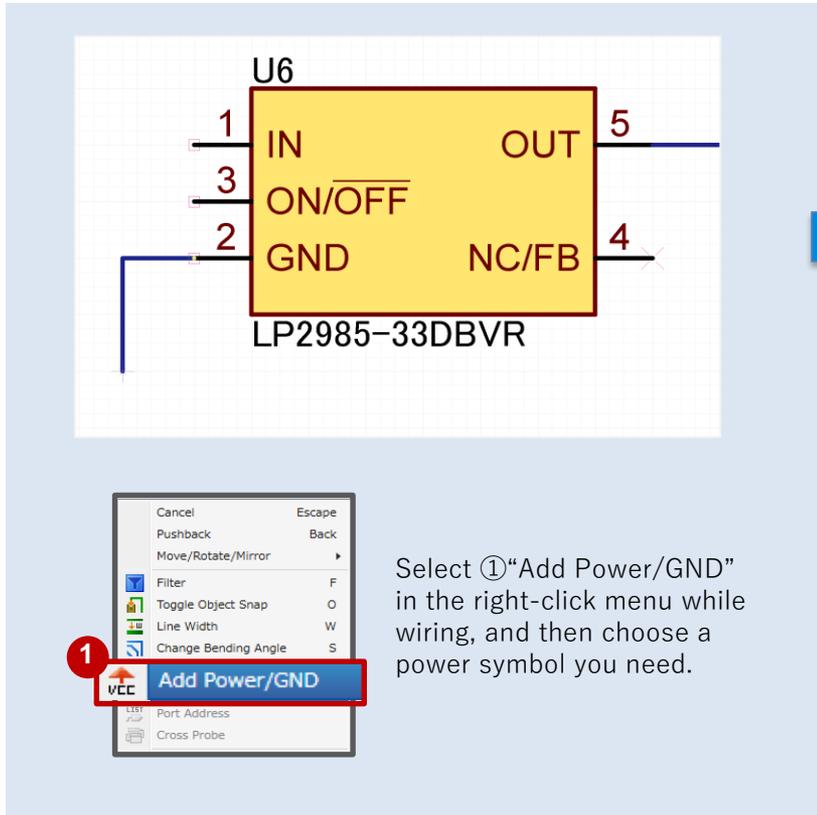


Select ①"Add Power/GND" in the right-click menu while wiring, and then choose a power symbol you need.
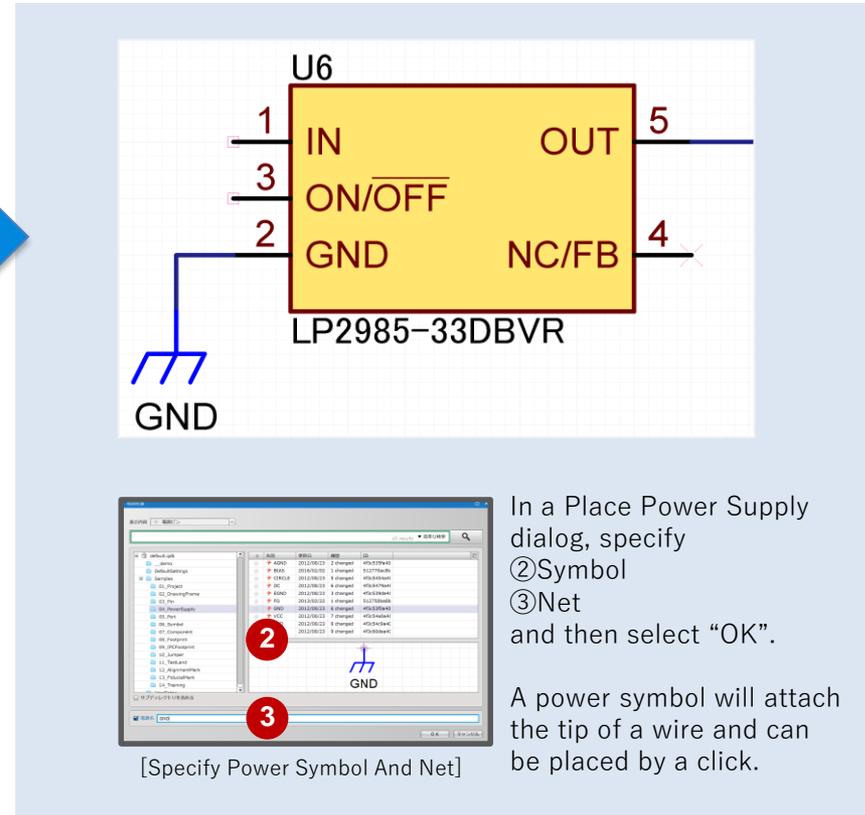
In a Place Power Supply dialog, specify
②Symbol
③Net
and then select "OK".

A power symbol will attach the tip of a wire and can be placed by a click.

[Specify Power Symbol And Net]

## Description

■ 【Circuit Designer】　Added an Add Power/GND command that allows you to insert a power symbol while wiring.

Step1: While wiring, right-click your mouse and select Add Power/GND.

Step2: A Place Power Supply dialog will appear. Select a power symbol you want to place and click OK.

Step3: The selected symbol will be attached to the tip of the wire you are currently editing.

Step4: Click where you want to place the power symbol.

# The Differential Pair Routing Has Been Greatly Enhanced.

**Via objects can be now added from the right-click menu while routing signals.**

Horizontal
Vertical
Diagonally Upward
Diagonally Downward

**Via direction can be switched from the Via Pattern menus.**

| Cancel | Escape |
| Pushback | Back |
| Change Layer | L |
| Move to Previous Layer | Tab |
| Move/Rotate/Mirror | ▶ |
| Filter | F |
| Toggle Object Snap | O |
| Net Class Settings | Ctrl+N |
| Line Width | W |
| Toggle Corner Shape | Shift+S |
| Change Bending Angle | S |
| Switch Angle | X |
| Add Via | |
| Via Pattern | ▶ |
| Differential Via Spacing | |
| Switch Reference Point | |
| Differential Pair Spacing | C |
| Trimming | |
| Subtract | |
| Cross Probe | |

**Existing functions have also been enhanced and optimized for much easier differential pair routing.**

Description

■ 【PCB Designer】　Enhanced the differential pair routing.

- The existing functions have been enhanced and optimized for easier and smoother differential pair routing.

- Detailed settings such as switching via direction and via spacing can be made now while routing pair signals.

# A Reference Point Can Be Now Designated When Copying Objects.

**Enhanced** **Reference Point Designation**
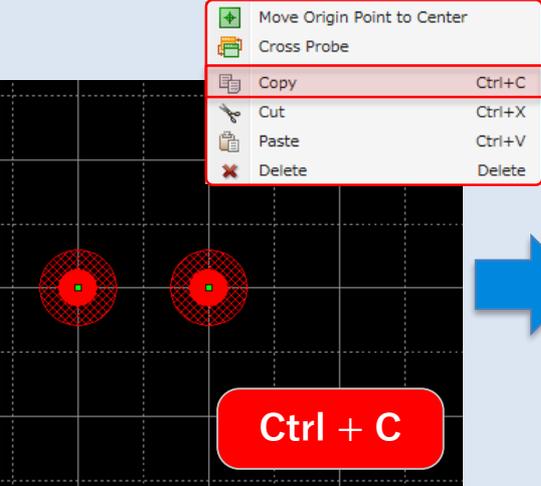


**[Step 1]**
**Select objects and perform a copy command.**
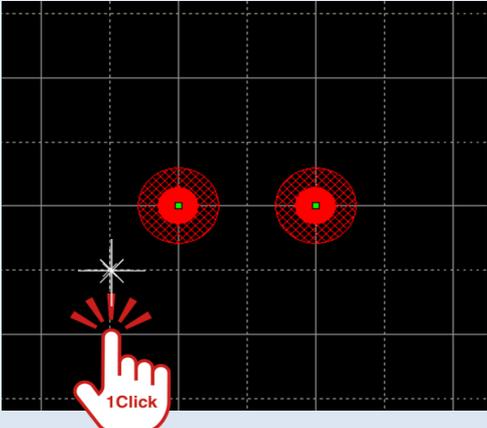
**[Step 2]**
**Designate a reference point by a click.**
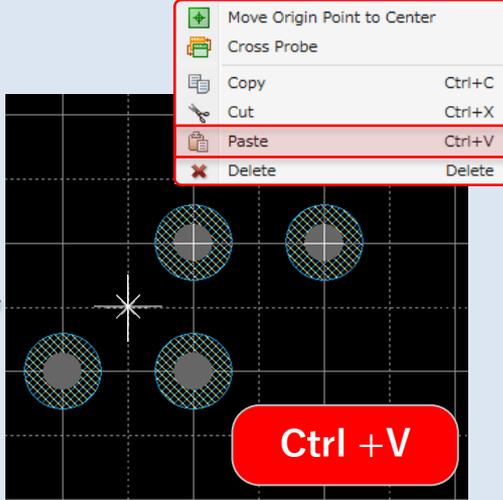
**[Step 3]**
**After performing a paste command, you can paste the copied objects based on the specified reference point.**

**Description**

■ **【PCB Designer】 Enhanced the copy function.**

The copy feature has been enhanced, and it now allows you to designate a reference point for pasting copied objects.

It is useful when copying multiple objects or when you want to place based on specific points such as the tip of a route.

Note: A reference point can be set only when drag-selecting an object or selecting multiple objects.

(If only one object is selected by a click, it will be copied without reference point designation.)

# Cutting Object Is Now Much Easier with "Subtract"  .

**Cut Silk**



**Cut Plane**



**Cut Silk**

A1 ➡ A1
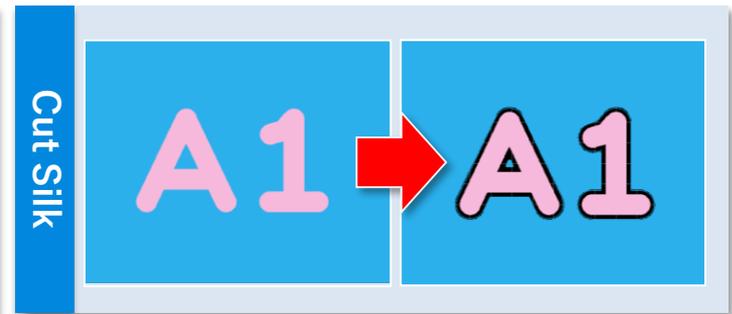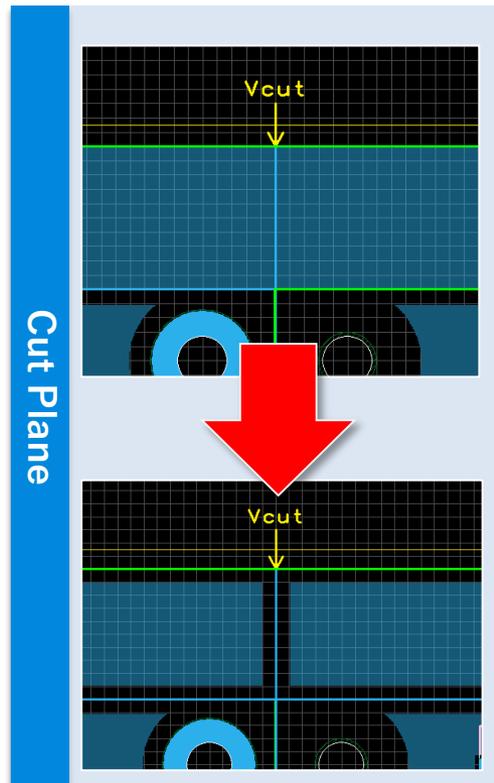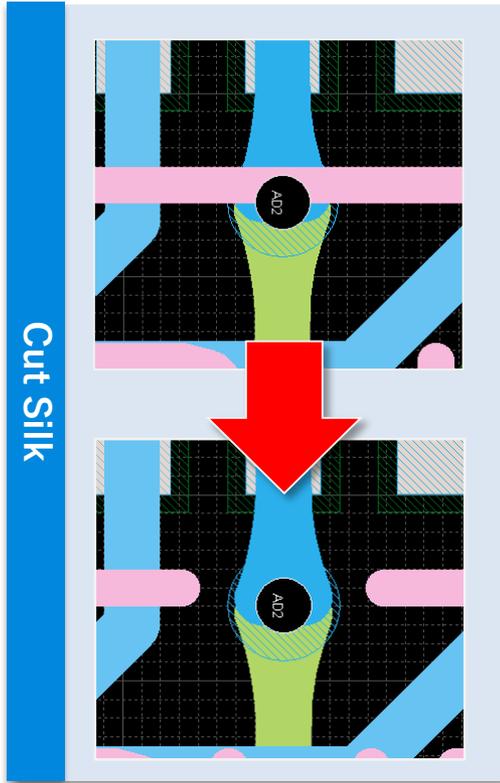
| Property | ◀◀ |
|---|---|
| ▾ Aggregate Information | ⚙ |
| Components | 54 |
| Pads | 241 |
| Cost | 0 |
| ▾ Objects | |
| Target Layer of Pad | Electric [ ▾ |
| Clearance | 0.15 |

\* Clearance is settable in the Property window.

**Description**

■ 【PCB Designer】 **Added a Subtract function which allows you to cut selected objects.**

　- With a Subtract function, you can subtract the second selected object from the first selected object.

　- The clearance between selected objects is adjustable in the Property window.

# Footprints Are Now Editable on a PCB/Panel Design Sheet

# with an Enable Component Editing Feature.



A silk contained in a footprint can be deleted.

By checking "Enable Component Editing", a library on a PCB will be disassembled, and you will be able to edit its shape.

Enable Component Editing

---
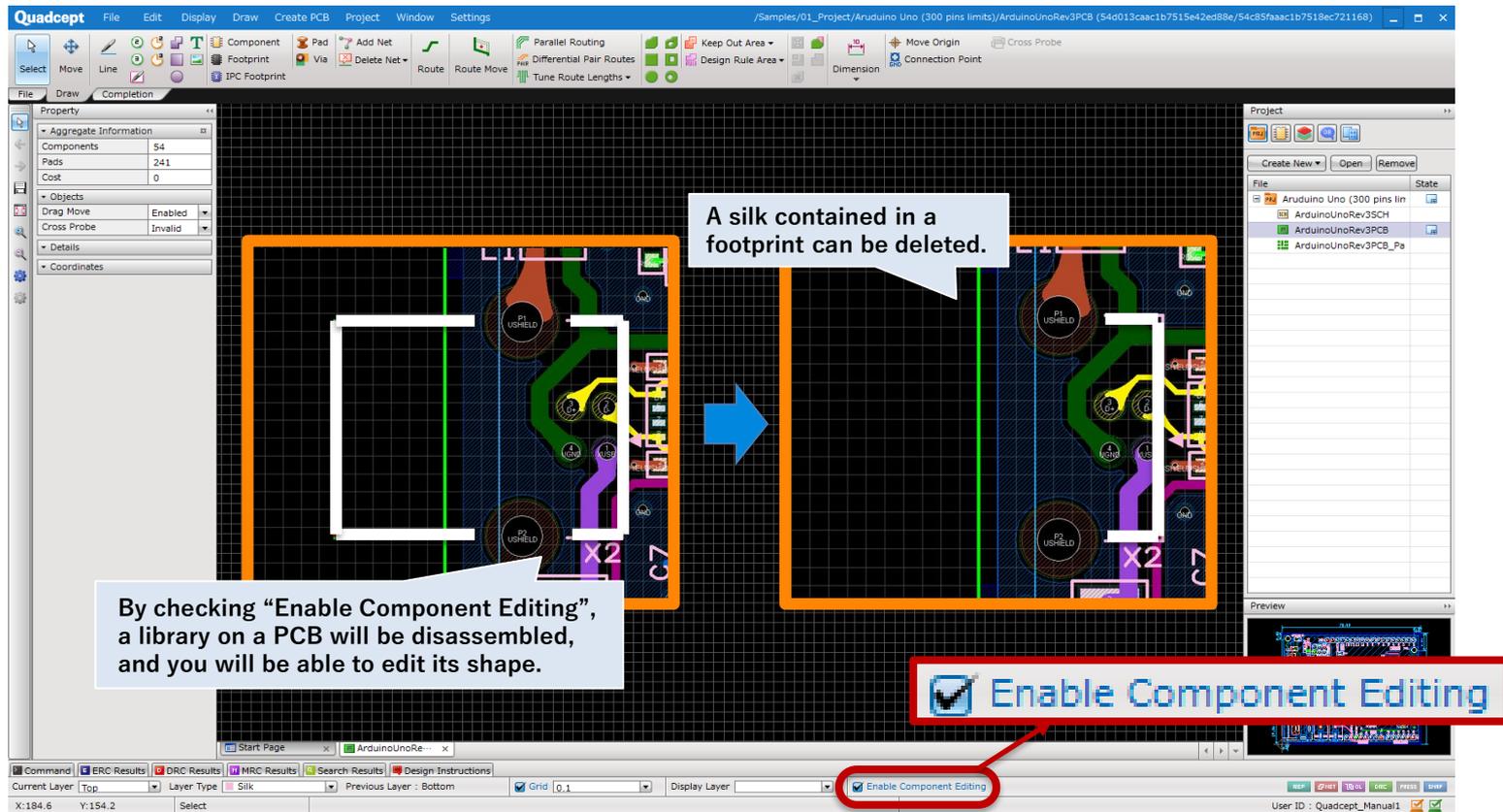
## Description

■ 【PCB Designer】 Added an "Enable Component Editing" function which allows footprint editing on a design sheet.

- By enabling component editing, you can edit footprints while designing a PCB/panel sheet.

- It is useful when you want to move the pin 1 mark or when you want to remove unnecessary silk.

- Various editing tools including Trimming, Split Shape and Subtract(New function) are available for footprint edition.

# DXF Data Now Easier to Convert to Keep Out, Cutout and More.

**After selecting multiple objects, you can convert to various objects from the right-click menus.**

As DXF data is composed of line objects, it is troublesome to convert to filled objects such as keep out areas and so on.

In the new version, multiple areas can be converted into filled objects in a lump.

| | |
|---|---|
| Cancel | Escape |
| Change Layer | L |
| Move to Previous Layer | Tab |
| Selection Method | ▶ |
| Move/Rotate/Mirror | ▶ |
| Filter | F |
| Toggle Object Snap | O |
| Net Class Settings | Ctrl+N |
| Trimming | |
| Subtract | |
| Align | ▶ |
| Align Routes | |
| Split Shape | |
| Convert to Filled Object | ▶ |
| Convert to Line | |
| Auto Create Restricted Area | |
| Assign Net | |
| Move Origin Point to Center | |

- Convert to Filled Polygon
- Convert to Plane
- Convert to Cutout
- Convert to Keep Out Area
- Convert to Design Rule Area

[Filled Objects]　　　[Cutouts]

---

### Description

■ 【PCB Designer】　Enhanced the Convert to Filled Object.

The Convert to Filled Object function has been enhanced, and it now allows you to convert multiple areas into filled objects in a lump.

Step1: Select multiple objects.

Step2: Select [Convert to Filled Object] and an object you want to convert to in the right-click menu.

# GRID and GUIDE Now Integrated for Much Easier Operation.

| Enhanced | **Enhanced GRID Feature** |
|----------|---------------------------|

**Easier to see by GRID change**

When setting the pitch to [2.54 mm] and the highlight to [2], the dotted line is displayed at intervals of 2.54 mm and solid line is shown at 5.08 mm intervals (2.54 x 2).
Solid lines will be highlighted.

2.54   2.54

Y:5.08

X:5.08

[GRID Display]

Dot            Line

### Grid dialog
Show Grid

Pitch
X: 2.54000 : Highlight 2
Y: 2.54000 : 2

Offset
X: 0.00000  Y: 0.00000

Display Type  ○ Dot  ● Line

Grid List        O K

| Description |
|-------------|

■ 【PCB Designer】 **Enhanced the GRID Feature. / Integrated GRID and GUIDE for Much Easier Operation.**

- The setting screen can be opened by selecting [Display] → [Grid] [GRID] on the menu bar or

  selecting [Grid Settings] on the bottom of the screen.  ☑ Grid <0.1/0.1>

- In a Grid dialog, "Pitch (interval of GRID)" and "Highlight (interval of highlighting)" are settable.
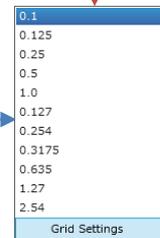
⚠ **!** GRID and GUIDE have been integrated, and the display GRID and the operation GRID are now the same.

0.1
0.125
0.25
0.5
1.0
0.127
0.254
0.3175
0.635
1.27
2.54
Grid Settings

# Easier to Design a PCB with GRID Offset Settings.

## Enhanced　GRID Offset Now Available
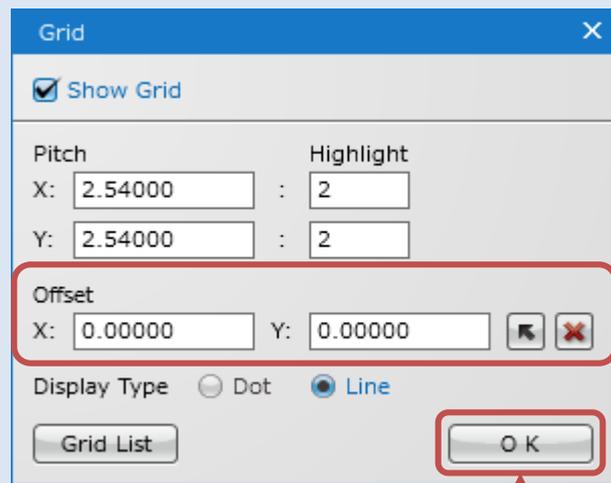


If pads or objects are off the grid, routing signals becomes difficult.

In the new version, you can move the grid line to any location for easier operation.

In the Grid dialog,
① Click on 🔖 .
② Click on any location you want to move the grid.
③ Select the OK button.

The grid line will be moved to the specified location.

\*　❌ : Resets offset settings.

### Description

■ 【PCB Designer】 Added GRID Offset Settings.
- GRID offset settings are now available.
- It is easier to arrange objects and routing signals by moving the grid position to an object which is off the grid.

# A Text Angle Can Be Now Verified by MRC.

**[MRC Error : Text Angle Violation]**

In addition to reference angle check, text angle and unintended mirror can be verified now. For example, if a text is placed on a BOTTOM side without inverting the character, a mirror error will be detected.

**[Text Placement Angle Settings]**

- Set allowed text angles (0° 90° 180° 270°).
- Set a target layer to be checked.



Placed on Bottom Without Mirror

**Mirror Error and Angle Violation Detected**

| No. ▲ | Check Type | Layer | Target 1 | Target 2 | Actual Value | Required Value |
|---|---|---|---|---|---|---|
| 1 | Text Angle Violation | Top | Electric | | 45 | 0,90,180,27 |
| 2 | Text Angle Violation | Bottom | Electric | Mirror Error | | |

| Target Layer | | | |
|---|---|---|---|
| ☑ Top | Silk | | |
| ☑ Bottom | Silk | | |

**Allowed Text Angle**

| | 0 | 90 | 180 | 270 |
|---|---|---|---|---|
| Top | ☑ | ☑ | ☐ | ☐ |
| Bottom | ☑ | ☑ | ☐ | ☐ |

---

**Description**

■ 【PCB Designer】 **Increased verifiable objects by a MRC item, Text Placement Angle.**

- In addition to "Reference", the angles of "Text" objects can be now verified.

- You can now find text angle violation as well as an unintended text mirror.

  The error will be detected if a text in the Top layer is inverted or the one in the Bottom layer is not inverted.

**Enhanced** | **Optimized Object Search**

Added available search words and optimized searchable information for smoother object search.
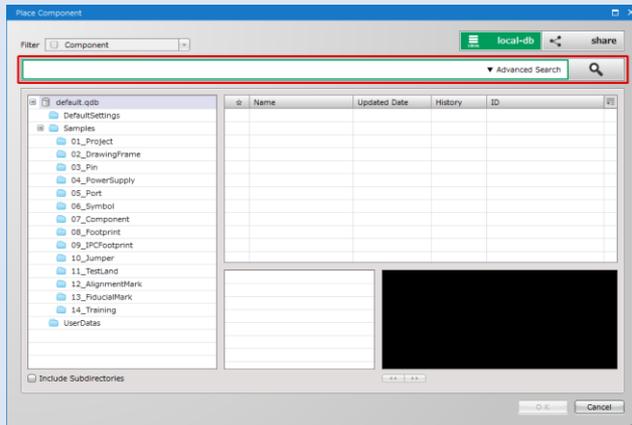
**New** | **License Acquisition Status Indicators**

**No License Authenticated**

**Circuit Designer**

**PCB Designer**

User ID :

**Description**

■ **Optimized Object Search**

Object search has been enhanced and optimized.

You can now search for objects more easily and smoothly.

**Description**

■ **License Acquisition Status Now Referable**

The license icons are now displayed on the lower right of the screen, and you can check your license acquisition status at a glance.

**Improved** **Reference Point When Copying**

In Circuit Designer, the reference point for pasting copied objects has been changed from "the center of the selected objects" to "the lower left of the selected objects".

**[Previous Version]** **[Current Version]**

**Improved** **Default Wire Width**

Improved the processing of determining wire width when starting to draw a wire.

**Wiring from other than a tip => Default Width**

**Wiring from a tip => Width Maintained**

---
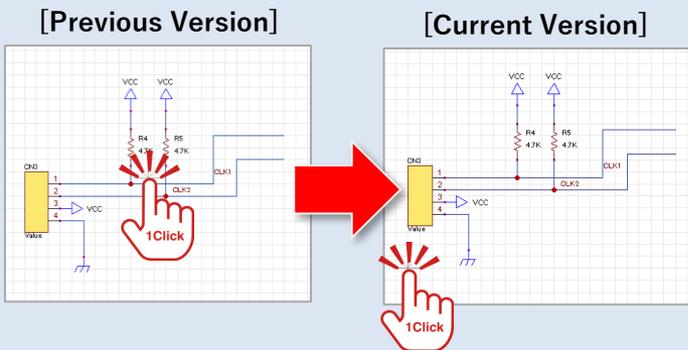
Description

■ 【Circuit Designer】

**Changed a Reference Point When Copying Objects**

The reference point for pasting copied objects has been changed from "the center of the selected objects" to "the lower left of the selected objects".

---
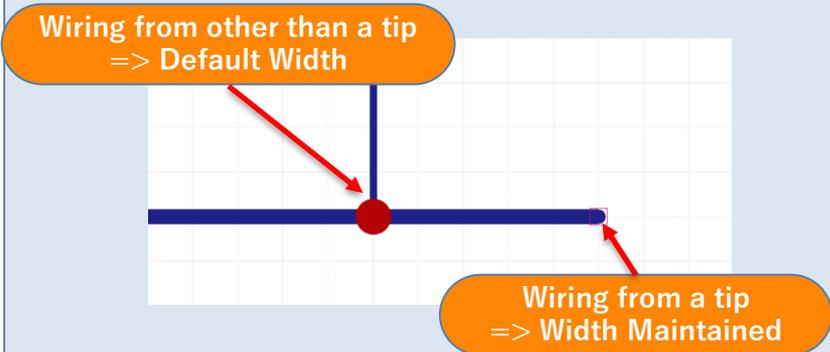
Description

■ 【Circuit Designer】

**Improved the Processing of Determining Wire Width.**

- Wiring from the tip of a wire ⇒ "Width Maintained"

- Wiring from other than the tip of a wire ⇒ "Default Width"

**Improved**　**Drawing Order of Each Layer**

Changed the drawing order, and a current layer and layer type are now displayed at the forefront.



Change Layer

Top：Silk

Top：Electric
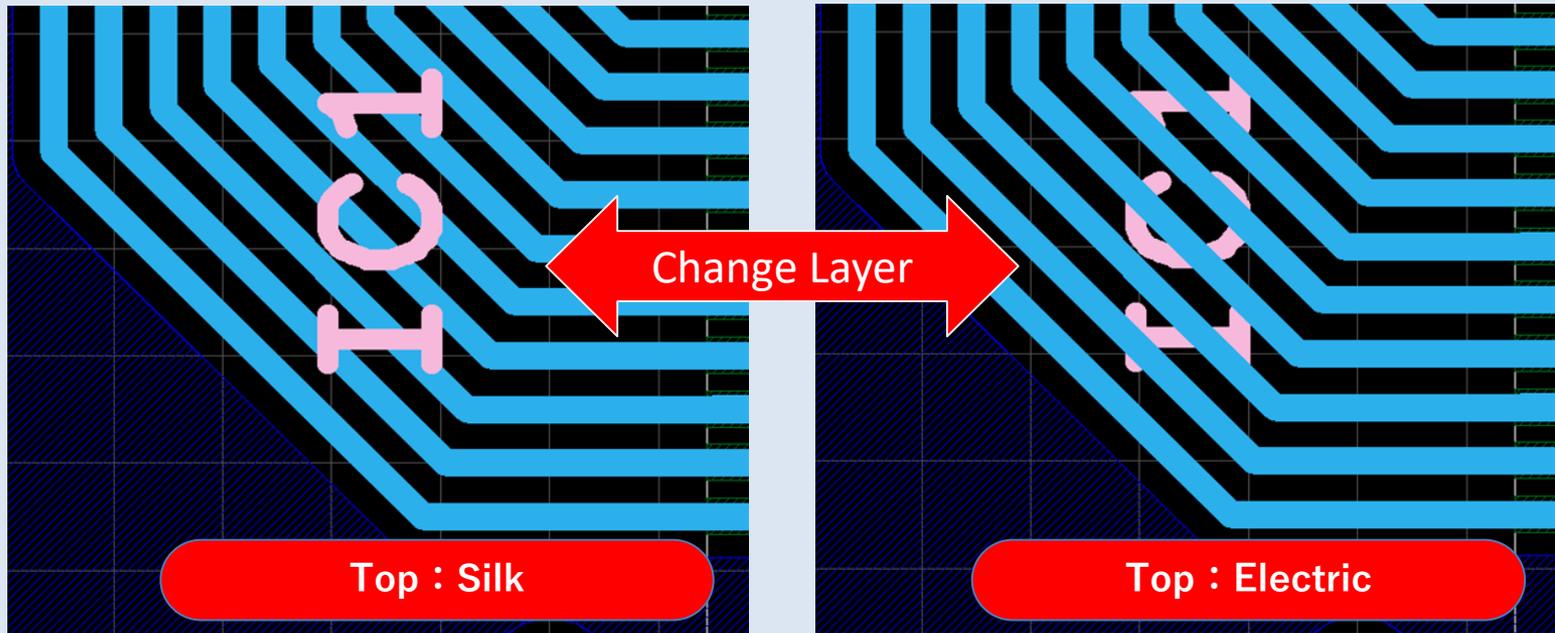
Description

■ 【PCB Designer】　**Changed the Drawing Order of Each Layer.**

　The drawing priority has been changed, and a current layer and layer type are now displayed at the forefront.

# Thank you!

We keep improving and enhancing our Quadcept to offer a better design environment for all users.

If you have any thoughts or suggestions for our tools, please share those with us!

Quadcept Inc.

Support Team

support@4cept.com

http://www.quadcept.com