



THE CLOUD EDA
Quadcept supports IoT

[Release notes Ver9.2.0](#)

We are offering the new version 9.2.0 that provides new features, improvements and enhancements for a better development environment and service. We have also fixed problems found by users' feedback and our validation. The following is a part of fixed problems and our support team will inform the users of the fixed issue who pointed out the problem individually by e-mail.

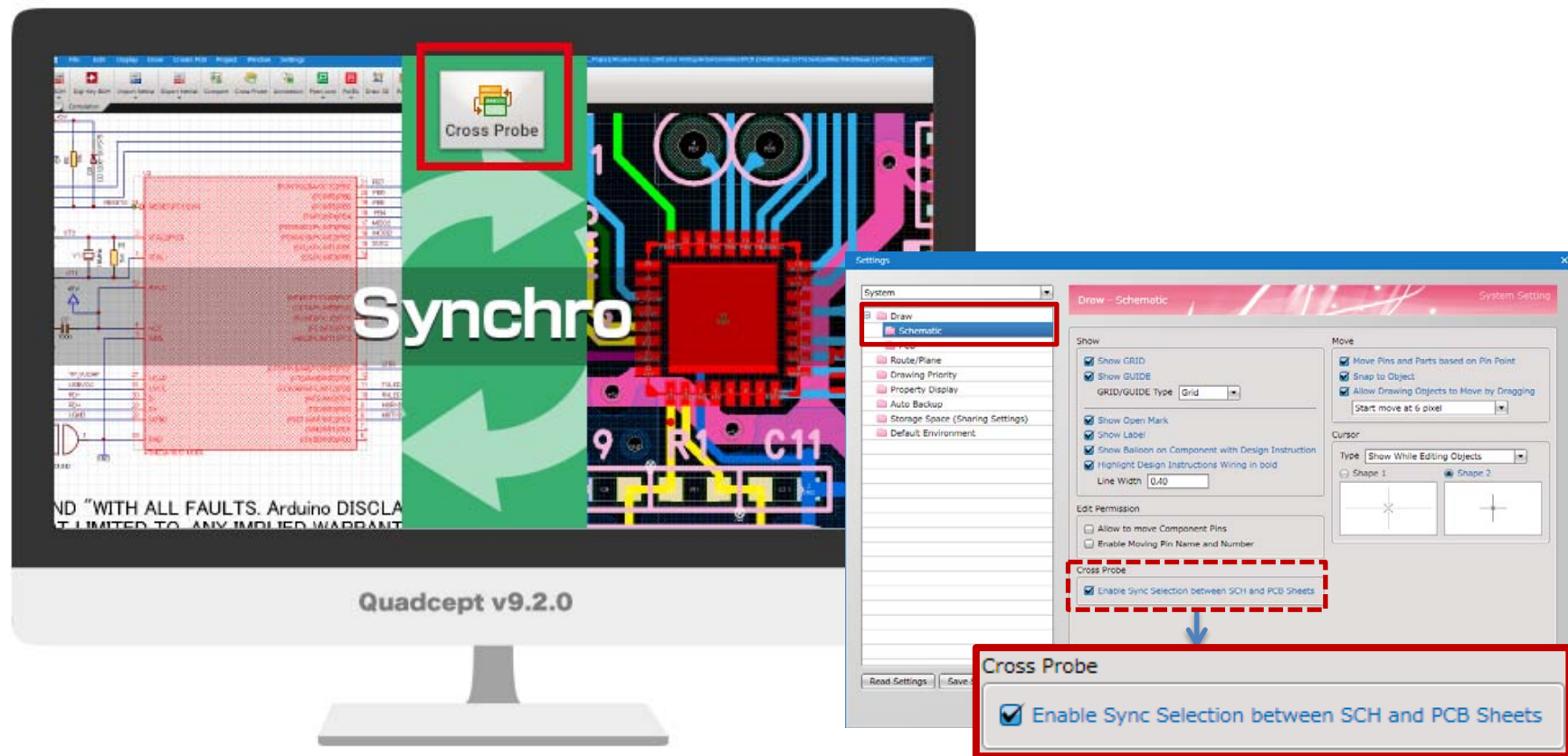
New/Improved	Category	Page	Description
Common			
New	Linkage	P1	Enhanced the cross probe feature for more efficient schematic and PCB designing.
New	Verification	P2	ERC/DRC/MRC windows are now easier to use. (It is now possible to move to an error description in an error list from an error mark on a design sheet.)
New	Setting	P3	The setting dialogs for menu customization have been combined into one dialog and it is now easier to use.
New	Edit	—	Objects can be now searched and moved even when filters are off.
Quadcept Circuit Designer			
New	Setting	P4	Pins can be now selected regardless of the settings of "Move Pin" and "Move Pin Attribute" in Property Window.
New	ERC	—	An ERC error, "Duplicated references" is now detected even when mixed-case references exist. (Ex: R1/r1)
Quadcept PCB Designer			
New	Setting	P5	The history of route width input when routing now remains and you can select the width from the history.
New	Setting	P6	Enhanced the Online DRC feature and you can now check route width (Minimum / Maximum Line Width) while designing.
New	Edit	P7	Enhanced the measure feature.
New	Edit	—	An origin point can be now moved by inputting absolute coordinates.
Improved Items			
Improved	Common	—	Improved the speed of processing when starting up the application and editing design by up to 50% in in-house comparison with previous versions.
Improved	Common	—	Changed the timing to check 100 pin limitation when licenses are not obtained.
Improved	Common	—	Fixed an issue where Quadcept failed to start under certain circumstances.
Improved	Circuit	—	Improved not to hide a "X" mark that means an "Unconnected Pin" when connecting a wire to the pin.
Improved	Circuit	—	Fixed an issue that caused an error when moving a column in the BOM dialog. (Only on Windows 10)
Improved	Circuit	—	Fixed an issue with a label position when moving a wire.
Improved	Circuit	—	Fixed an issue where the font size changed when importing Altium data.
Improved	Circuit	—	Fixed an issue with the selection of pin attributes.
Improved	PCB	—	Improved not to generate illegal objects such as objects of 0mm long or radius 0mm.
Improved	PCB	—	Fixed an issue where "Align Routes" did not work.
Improved	PCB	—	Fixed an issue with cutout objects.
Improved	PCB	—	Fixed an issue with Property Window.
Improved	PCB	—	Fixed an issue where a DRC error, "Floating Via" was erroneously detected when a route "Cap Shape" is set as "Square".
Improved	PCB	—	Fixed an issue where hidden rats could be selected.
Improved	PCB	—	Fixed an issue with References generated by Create Test Land Automatically.
Improved	PCB	—	Fixed an issue where the column sorting feature in a list did not work correctly.

*Altium is registered trademark or trademark of Altium Limited.

*Ver 9.2.0 provides still more improvements other than those above. For more information, please refer to the following release note.

URL : <https://blog.quadcept.com/en/quadcept-9-2-0-released/>

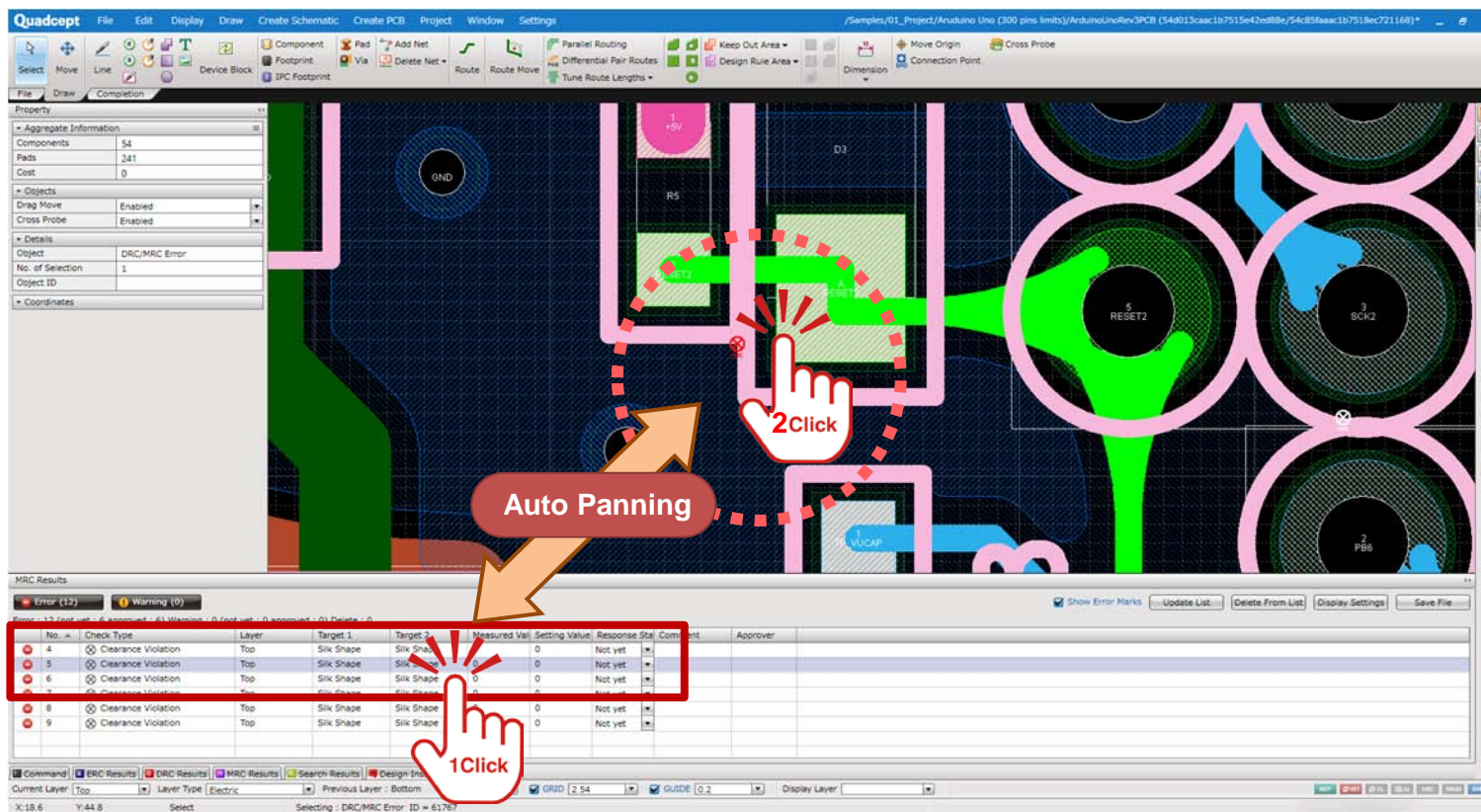
Schematic and PCB Now Synchronize by Cross Probe.



Added Feature

- Seamless Link between Schematic and PCB
 - Pull-down menu at the top of the screen : Select [Environment Settings] → "Schematic" and check the checkbox in the "Cross Probe".
 - By selecting a "component" / "net" and clicking a "Cross Probe" icon in the ribbon menu or choosing "Cross Probe" in the right-click menu, the other "component" / "net" in a schematic or PCB sheet will be selected and displayed in the screen.
- *When a sheet containing a "component" / "net" selected by "Cross Probe" is not open, the sheet will be automatically open and displayed.

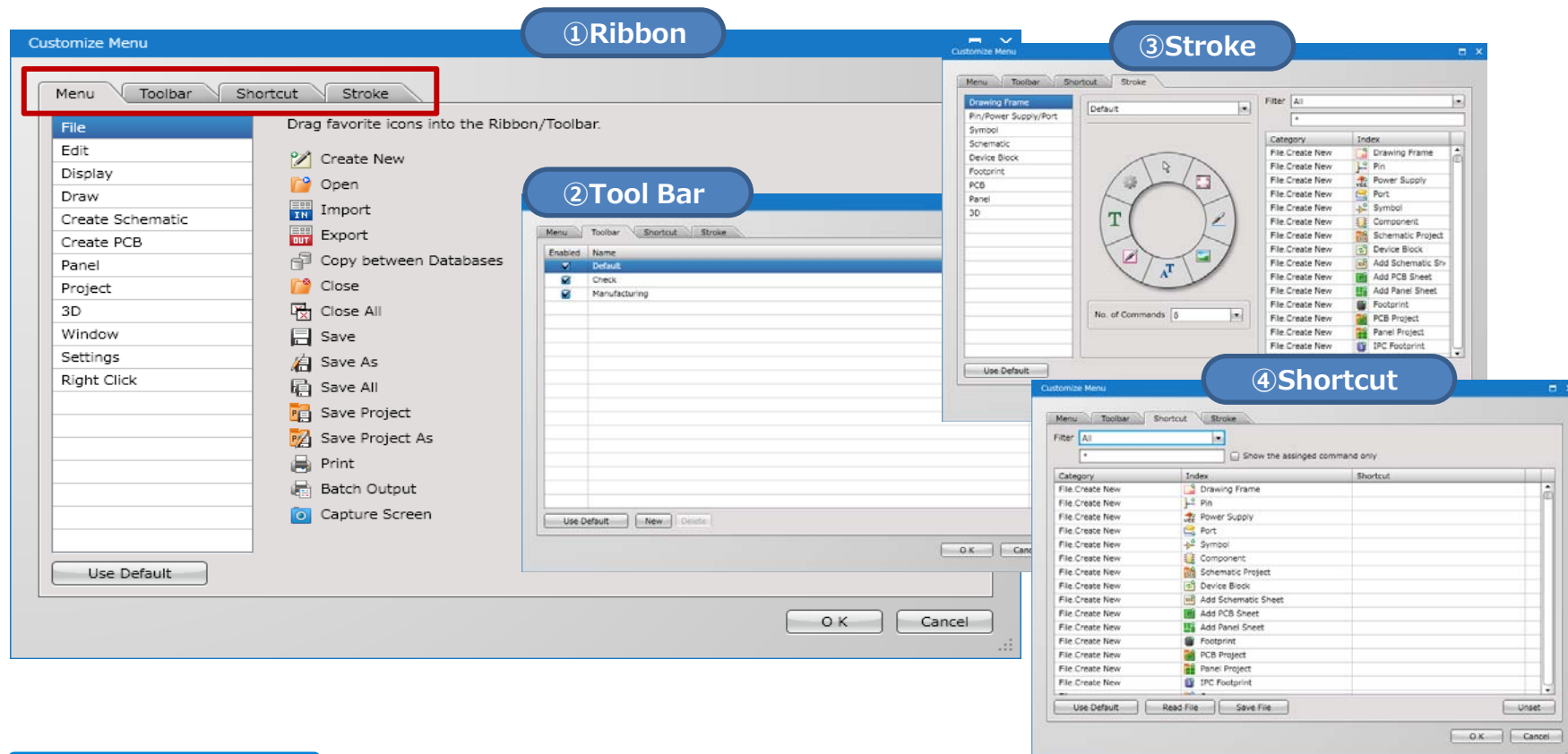
Now Possible to Move to Error Description in Error List from Error Mark.



Added Feature

- Link between Error Mark and List
 - By double-clicking error marks displayed as verification results by DRC / MRC, an error list will be displayed and you can confirm its description. Also, the screen will automatically move and show the error location by clicking an item in the list.

4 Settings for Menu Customization Now Combined into One Dialog.



Added Feature

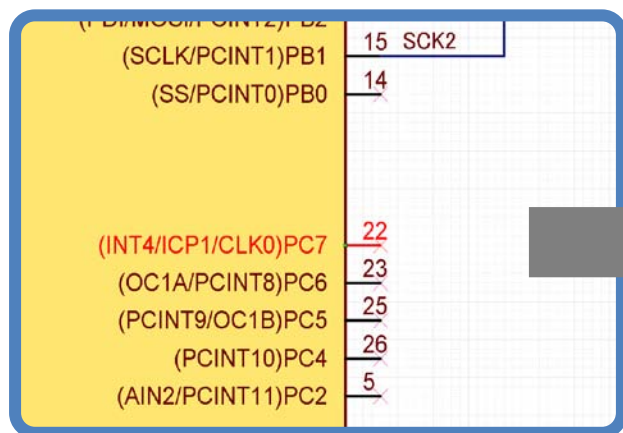
■ Customize Menu Integration

Stroke and shortcut settings in the "Environment Settings" have been integrated into the "Customize Menu" dialog ("Settings" → "Customize Menu"), and it is now easier to use.

①Ribbon Menu ②Tool Bar Menu ③Stroke Menu ④Shortcut Key

Now Possible to Select Pins and Check Attributes Even when Setting “Move Pin” as “Invalid”.

Pin attributes can be selected and checked
regardless of “Enabled” / “Invalid”.



Property	
Placed Pins/Total Pins	233/237
Cost	0
▼ Objects	
Drag Move	Enabled
Cross Probe	Enabled
Move Pin	Enabled
Move Pin Attribute	Enabled
▼ Details	
Object	Pin
No. of Selection	1
ID	4f9629dca405361e9c9e7540
Name	Line
Show	Show
<input checked="" type="checkbox"/> Pin No	22
<input checked="" type="checkbox"/> Pin Name	(INT4/ICP1/CLK0)PC7
▼ Coordinates	
X	186.69
Y	115.57

Property	
Placed Pins/Total Pins	233/237
Cost	0
▼ Objects	
Drag Move	Enabled
Cross Probe	Enabled
Move Pin	Invalid
Move Pin Attribute	Invalid
▼ Details	
Object	Pin
No. of Selection	1
ID	4f9629dca405361e9c9e7540
Name	Line
Show	Show
<input checked="" type="checkbox"/> Pin No	22
<input checked="" type="checkbox"/> Pin Name	(INT4/ICP1/CLK0)PC7
▼ Coordinates	
X	186.69
Y	115.57

【Attributes Displayed when Selecting Pins】

Added Feature

■ Pin Attribute Display

When set as “Enabled”, pin attributes had not been selected and checked. From ver.9.2.0., you can select and confirm pin attributes regardless of the settings of “Move Pin” and “Move Pin Attribute” in Property Window.

The History of Line Width Input when Routing Now Remains and You Can Specify the Width from the History.

The screenshot shows the Quadcept PCB design software interface. On the left, a 'Property' window is open, displaying a list of objects and their properties. The 'Line Width [HOLD]' property is highlighted, and a red box around it contains the text 'Up to 5 remain'. Below the property list, a text box explains: 'You can now confirm the history of line width input when routing. The number of history is up to 5, and the history will be overwritten from the sixth and the followings. It is possible to specify a line width from the list.'

The main workspace shows a PCB layout with several components and traces. A vertical pink rectangle labeled 'CN2' is on the left. Traces are colored blue, pink, yellow, and green. Dimensions are shown next to the traces: 1.2, 1.3, 1.5, 0.2, and 0.1. A component labeled 'R1' is on the right. A green vertical line is also visible.

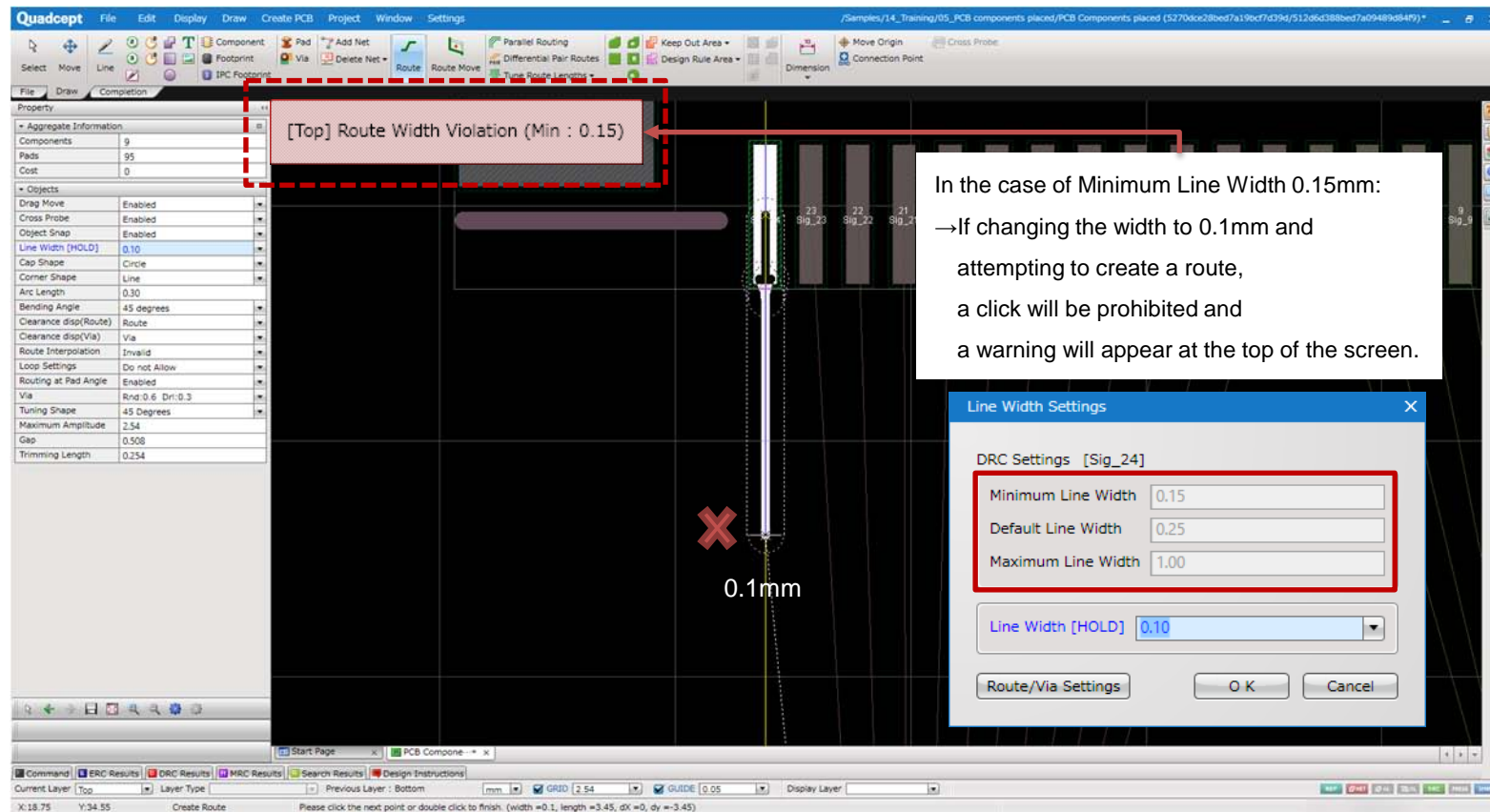
*The line width out of the range set in DRC route settings can be input when routing from ver.9.2.0.

Added Feature

- You Can Now Specify the Line Width from a History.

The history of line width set when routing now remains. It is possible to select a line width from "Line Width [Hold]" in Property Window.

Minimum / Maximum Line Width Violation Now Detected by Online DRC.

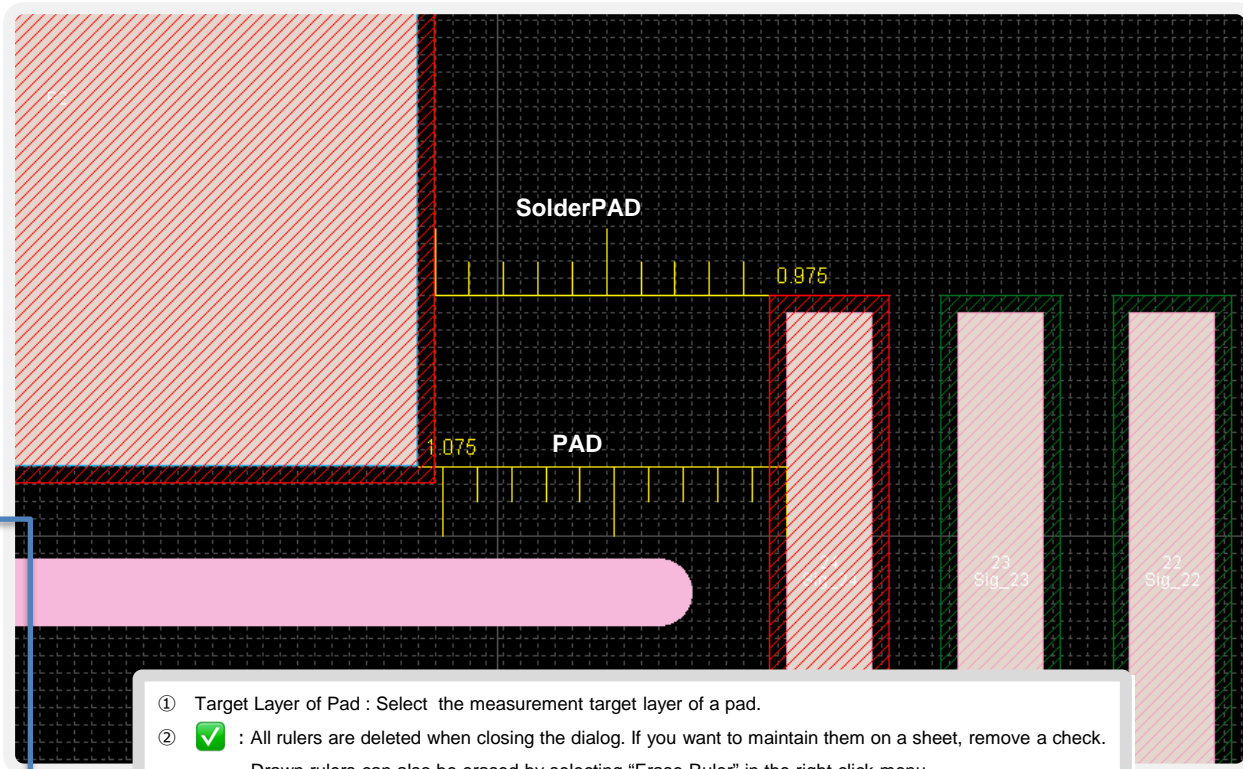
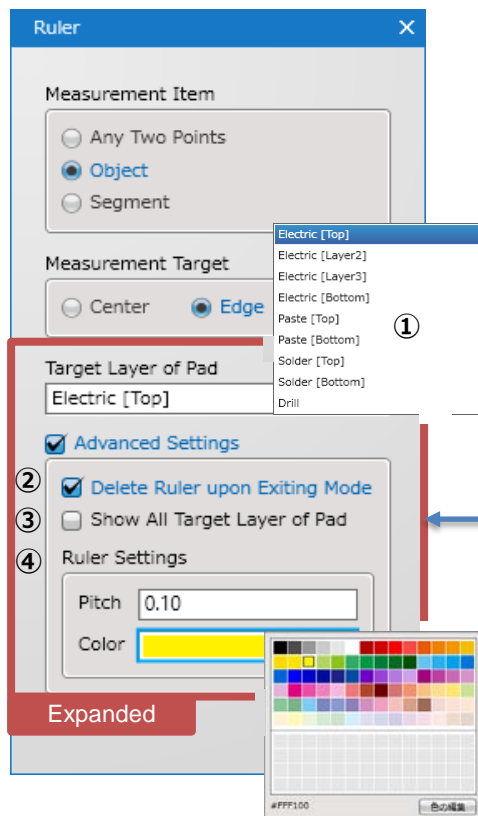




Added Feature

■ Enhanced Online DRC (Route Width Verification)

It is possible to input a line width freely when routing from ver.9.2.0. but if specifying a width out of the range set in Route / Via settings, a route width violation error will be detected by Online DRC and a route will not be created.

From  to  “Ruler” ! Advanced Feature!



- ① Target Layer of Pad : Select the measurement target layer of a pad.
- ②  : All rulers are deleted when closing the dialog. If you want to maintain them on a sheet, remove a check. Drawn rulers can also be erased by selecting “Erase Ruler” in the right-click menu.
- ③  : If checked, show all layers of pad in the drop-down list of Target Layer of Pad.
- ④ Ruler Settings : Set the pitch and color.

Added Feature

■ Advanced “Ruler”

Measurement feature has been enhanced. Advanced settings such as keeping drawn rulers after exiting the mode are now available.



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>