

Quadcept V10.4.0 Release Notes

Release Date 2021/01/14

Added Ability to Import SamacSys-Format Library Files

- ▶ Quadcept has signed a partnership with SamacSys, supplier of high-quality schematic symbols, PCB footprints and 3D models. This allows users to download design-ready, verified models from SamacSys and focus on their design without having trouble in component generation.

1 How to Use

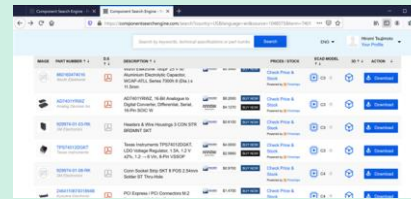
Access the SamacSys website.
[\(https://componentsearchengine.com/\)](https://componentsearchengine.com/)
 Search for a component by entering the desired search terms.



* A SamacSys account is required to use this service.

2

Once you locate a part from the resulting list, click the Download button.



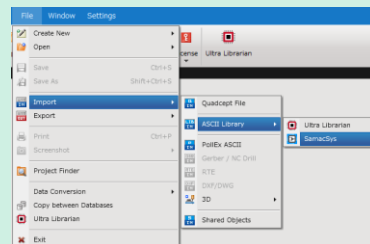
3

Click the Download ECAD Model button to download.



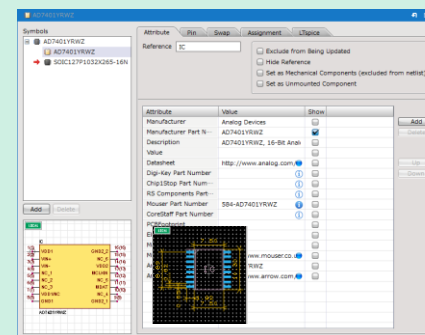
4

Open Quadcept and select the File >> Import >> ASCII Library >> SamacSys menu items. Once the file selection dialog appears, browse to select the xml file included with your download.



5

You are now ready to use the component.



Over 15 million
components now
available for download

* Please be sure to verify models to ensure they suit your design requirements before using.

Enhanced Footprint Assign

- ▶ The Assign Footprint dialog now provides the ability to import/export CSV-format files.
- ▶ This allows you to assign footprints to multiple components at once.

Filter Function

This allows you to filter the list to only show the components you need. This is useful when you want to assign footprints only to specific components.

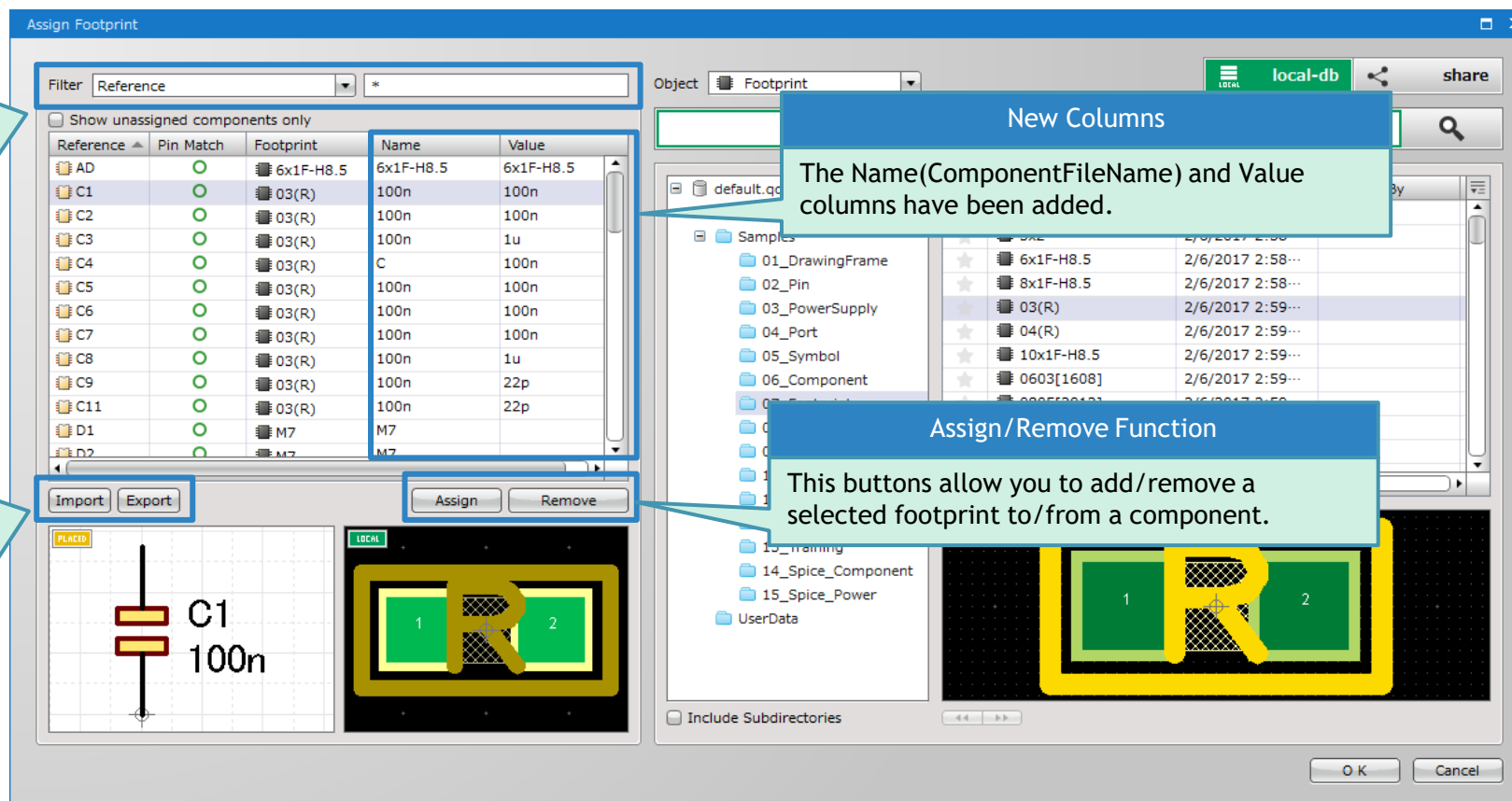
Reference
Pin Match
Footprint
Name
Value

Import/Export Function

You can now assign footprints to multiple components at once by using CSV format files.

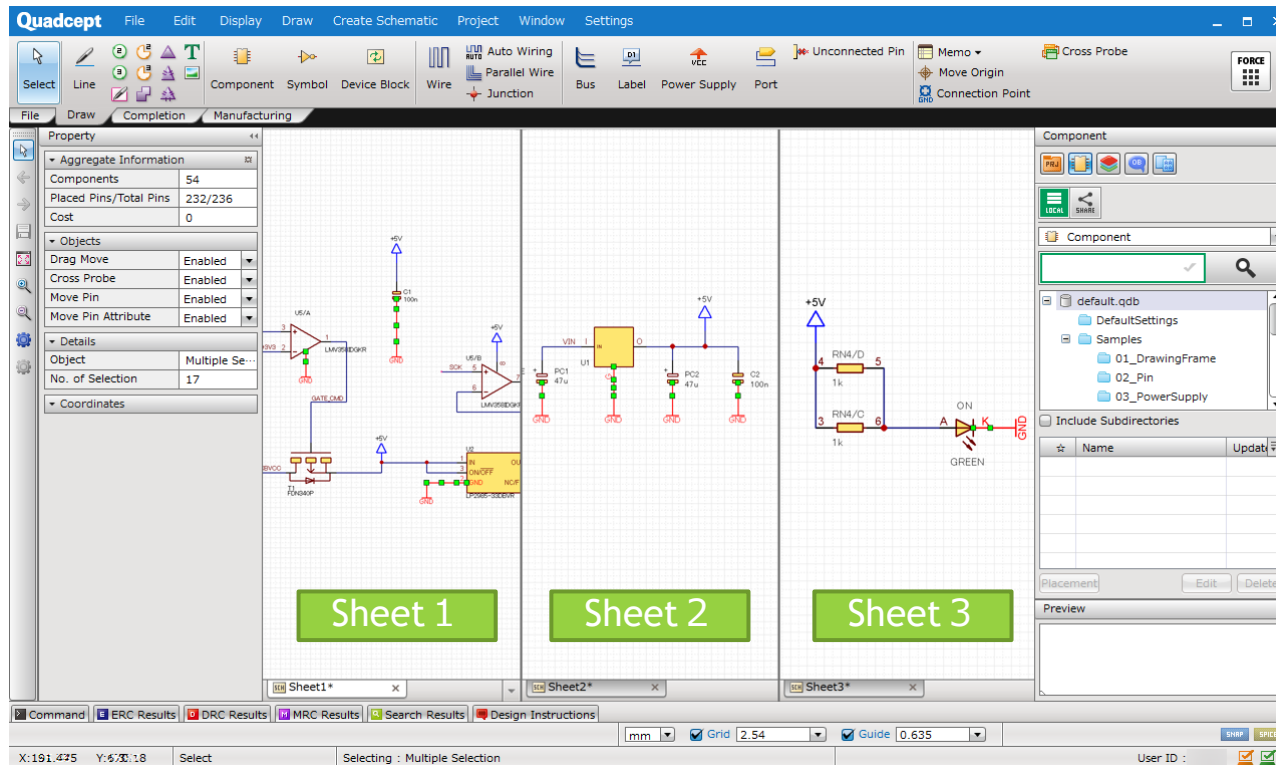
* The following 4 columns are required in the CSV files to be imported. The key column is "Reference".

- Reference : Reference designator
- ComponentFileName : Component name
- FootprintFileName : Active footprint
- AssignedFootprintFileName1 : Alternate footprint. You can add this column up to 9.



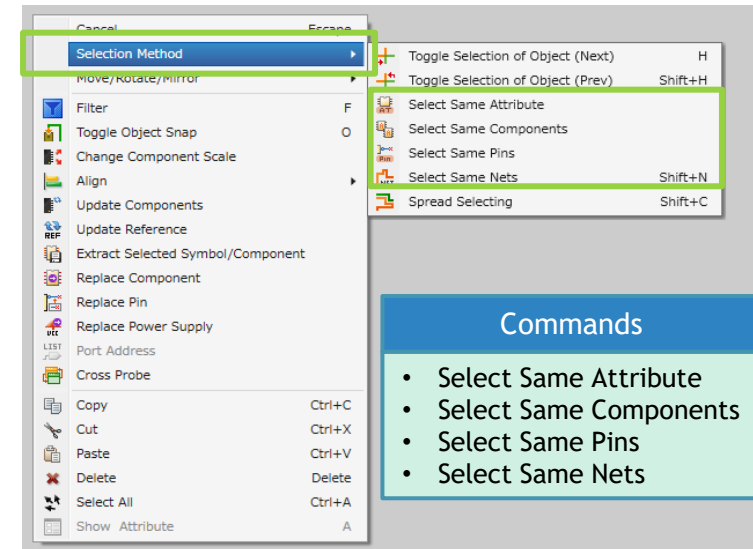
Enhanced Same Object Selection

- ▶ Same Object Selection has been enhanced with the ability to select the same objects across all the schematic sheets in a project.



You can now select/highlight the same objects across all the schematics in a project.

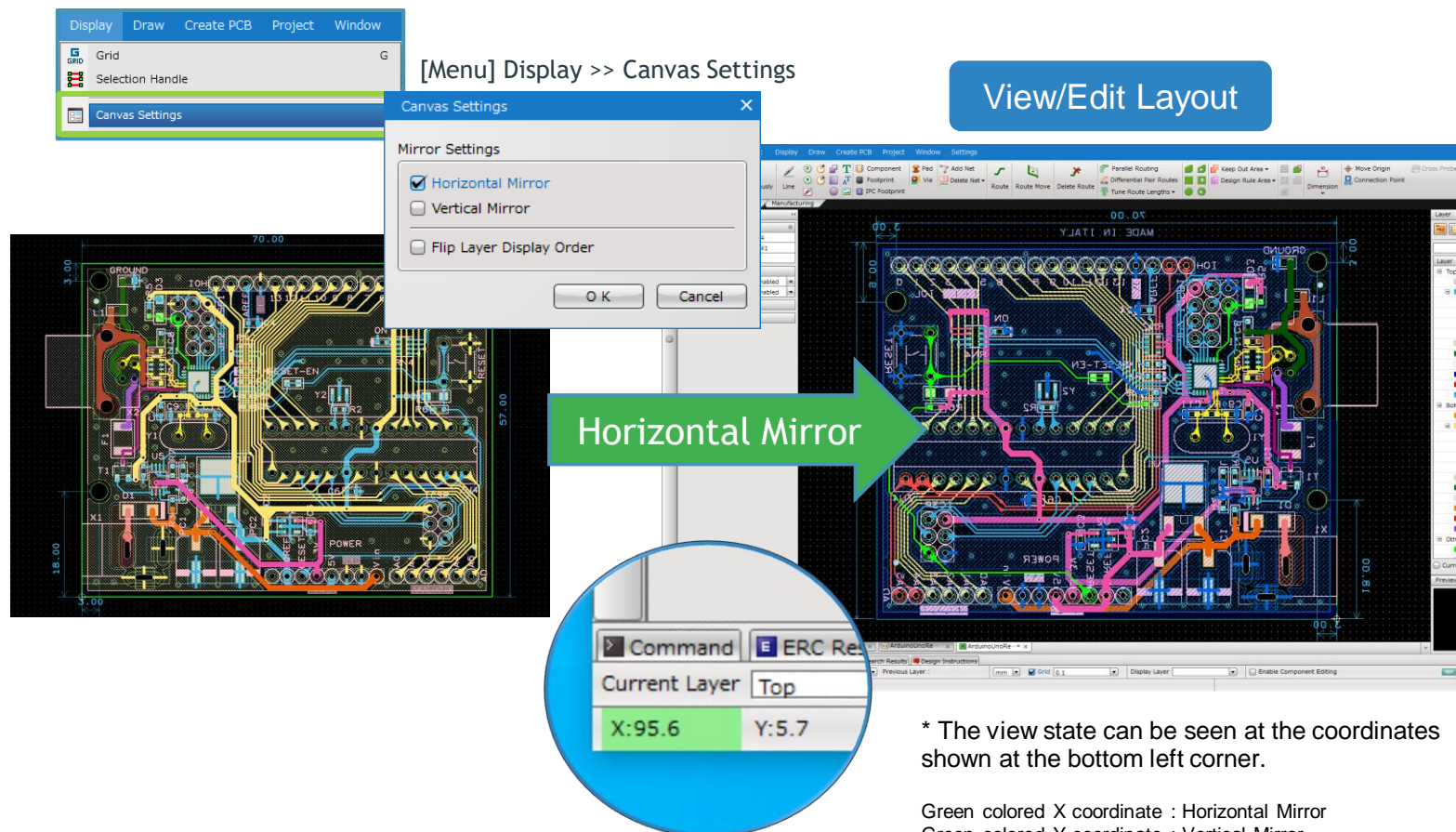
Select a target object and right-click to choose any of the same object selection commands.



* The values you have changed in the Property window will apply only to the objects in the active schematic sheet.

Added Ability to Flip Board During PCB Design

- ▶ In response to many requests, the ability to flip the entire board is now available for a PCB document.
- ▶ This allows you to view/edit your layout from the bottom side and to generate manufacturing outputs such as Gerber files with the viewed-from-bottom coordinate information.



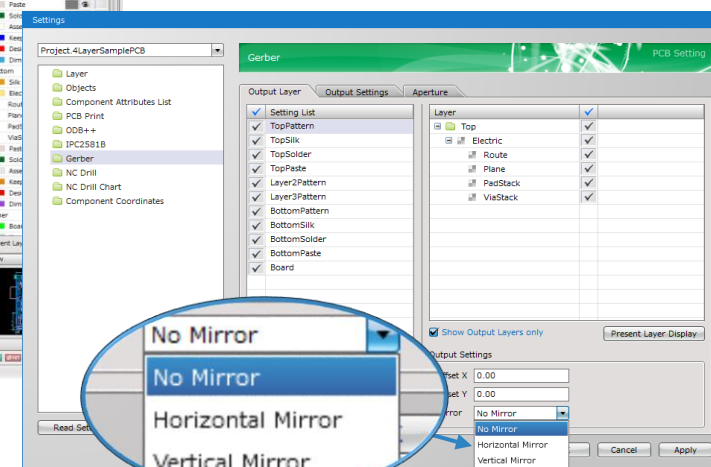
* The view state can be seen at the coordinates shown at the bottom left corner.

Green colored X coordinate : Horizontal Mirror
 Green colored Y coordinate : Vertical Mirror

Generate Outputs

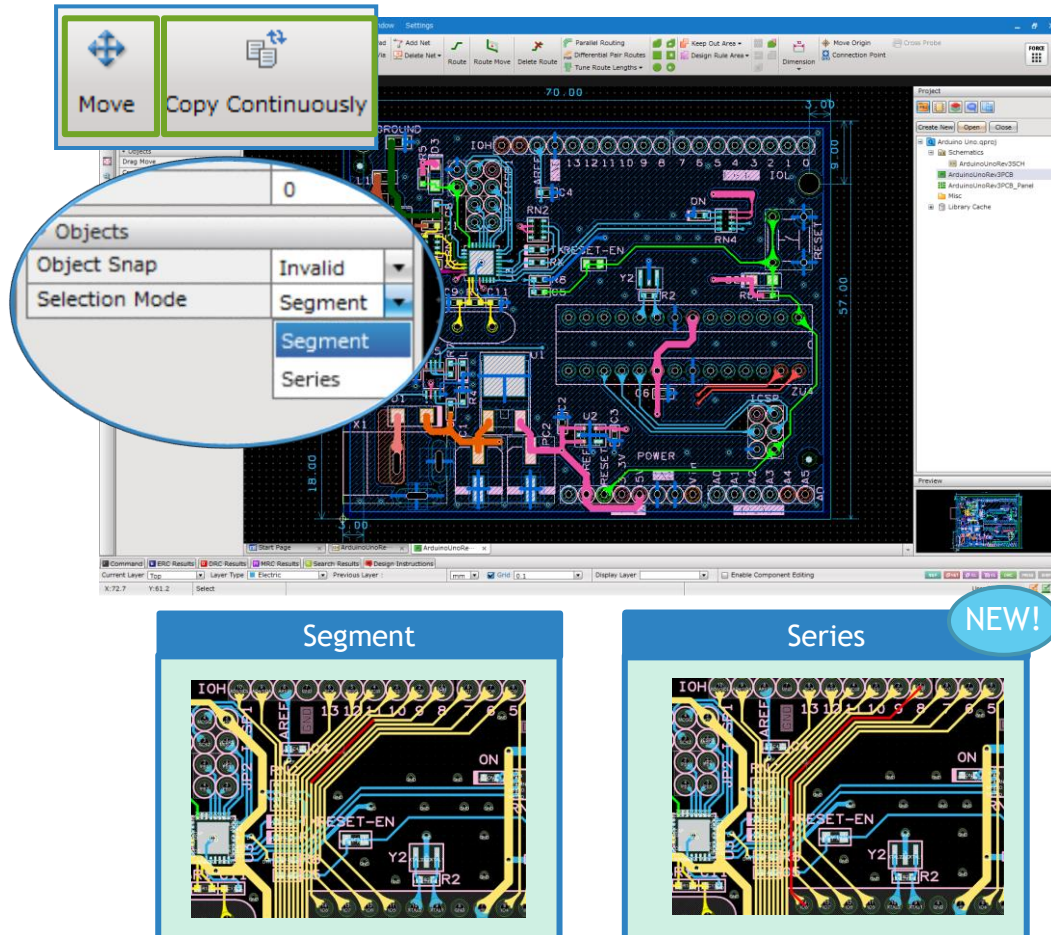
The formats below can be generated with the viewed-from-bottom coordinate information.

- Print
 - Gerber
 - NC Drill
 - NC Drill Chart
 - Pick-and-Place File
- * The view state can be set for each layer.

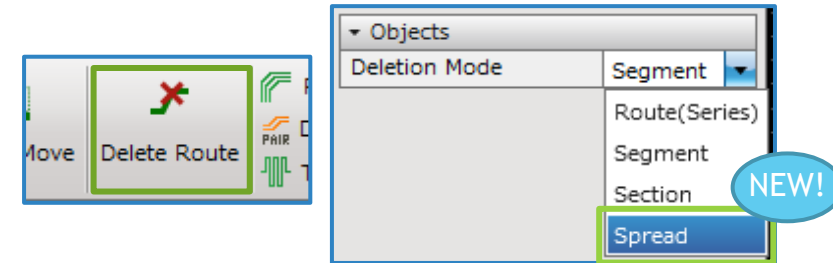


Enhanced Routing Capabilities

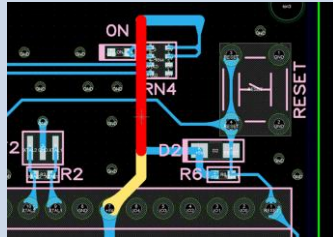
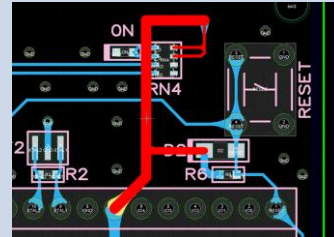
- The ability has been added to specify how routing objects are selected while in Move / Copy Continuously mode.



- The ability has been added to perform spread selection when deleting routing objects using the Delete Route command.

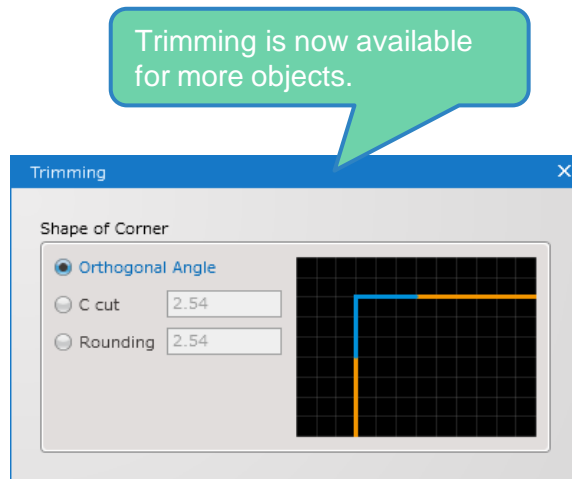


What are the differences between “Route(Series)” and “Spread”?

Route(Series)	Spread
Between junctions	Between pads
This option allows you to delete a series of routing objects, i.e., all contiguous routes until a via, pad or junction is encountered.	This option allows you to delete all routes between pads, including any vias. * The objects on hidden or locked layers will not be deleted.
	

Trimming Now Available for More Objects

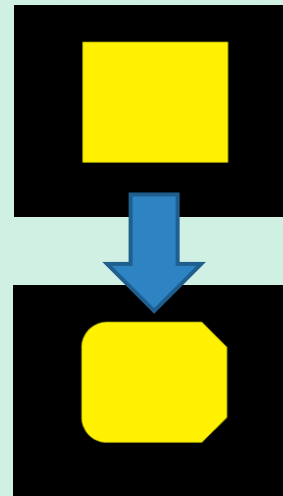
- ▶ The Trimming function has been enhanced and is now available for filled objects and square routes.



Objects Available for Trimming

NEW! Filled Objects

- Plane (Dynamic • Static)
- Design Rule Area
- Keep Out Area
- Cutout
- Assembly Area
- Filled Polygon

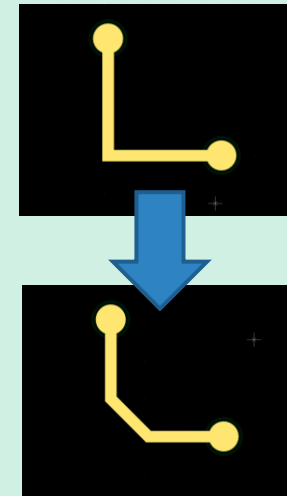


Line • Route

- Line
- Arc

NEW!

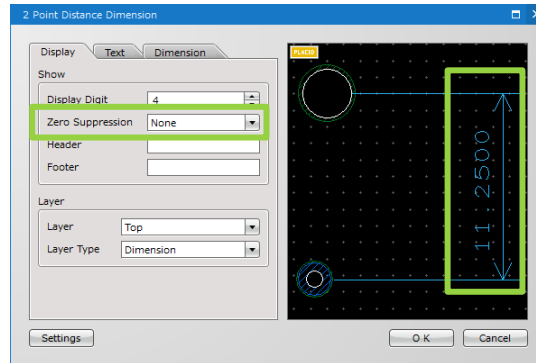
- Square Route(i.e., route with its Cap Shape set as Square)
* Rounding is not supported.



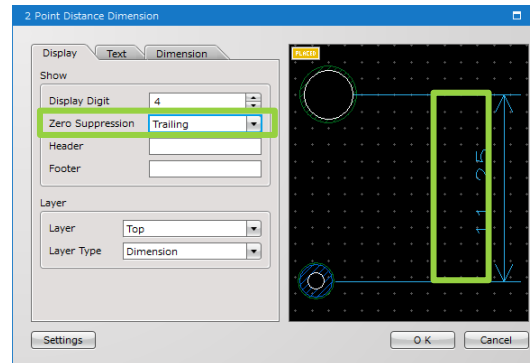
Added Ability to Suppress Trailing Zeros in Dimensions

- ▶ The ability has been added to suppress trailing zeros in the dimension values.
- ▶ Removing unnecessary zeros makes your layout easier to read.

Zero Suppression : None



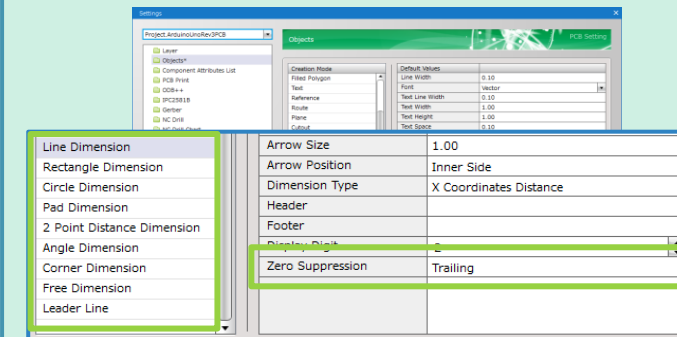
Zero Suppression : Trailing



Example : Trailing

Actual Scale : 0.635
 Decimals 4 : 0.635
 Decimals 3 : 0.635
 Decimals 2 : 0.64
 Decimals 1 : 0.6
 Decimals 0 : 1

* The default Zero Suppression setting can be configured in the Object section of the PCB settings (Settings >> Settings).



Added Direct Interface to DEMITASNX®

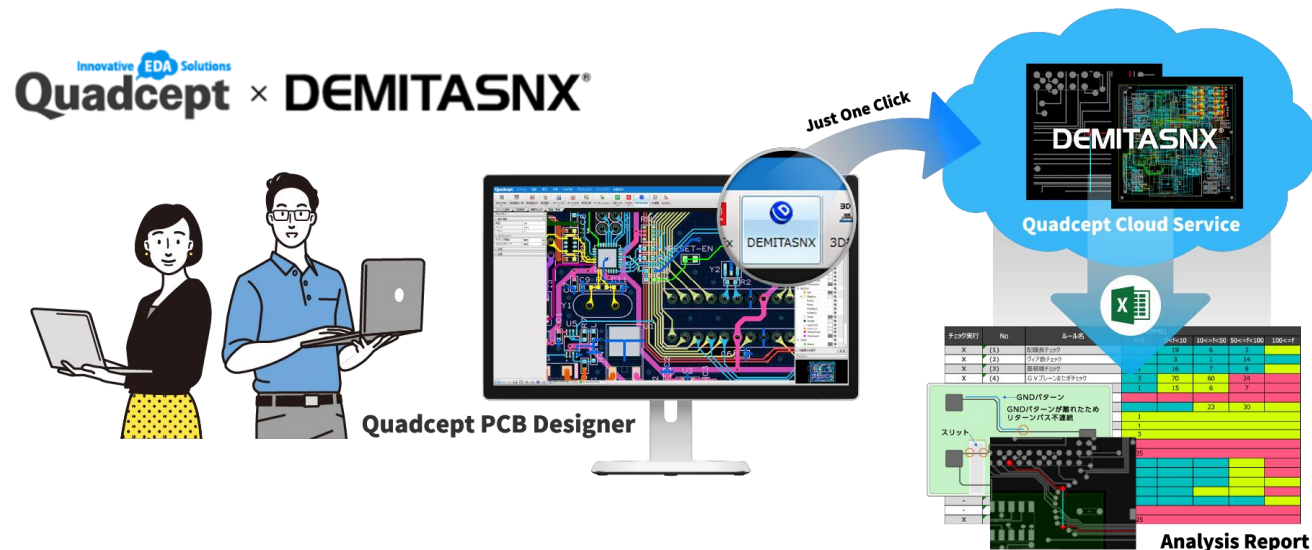
- ▶ The ability has been added to run an EMI analysis by directly connecting to NEC DEMITASNX® which has an established reputation as an EMI suppression support tool.
- ▶ Through direct connection to DEMITASNX®, you can perform the following four EMI check items: Traces Near Plane Edge, Grounding Vias Along Plane Outline, Return Current Path Discontinuity and Decoupling Capacitor.
- ▶ The analysis reports of these check items are downloadable in Microsoft Excel format.

[License System]

A license enables you to analyze the same project up to 5 times for 30 days.

* For details on how to access this service and notes on use, please visit the website below.

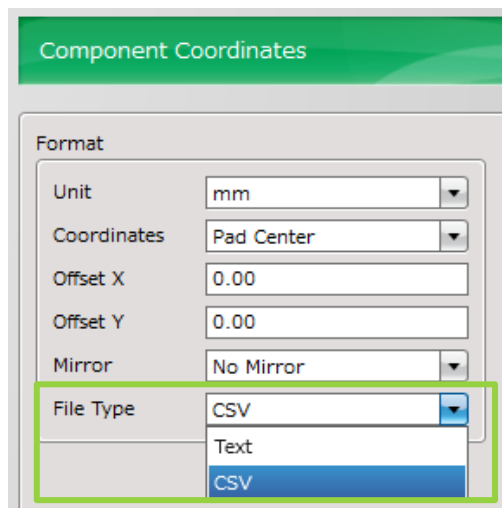
<https://www.quadcept.com/en/service/demitasnx/#notes>



Other Enhancements

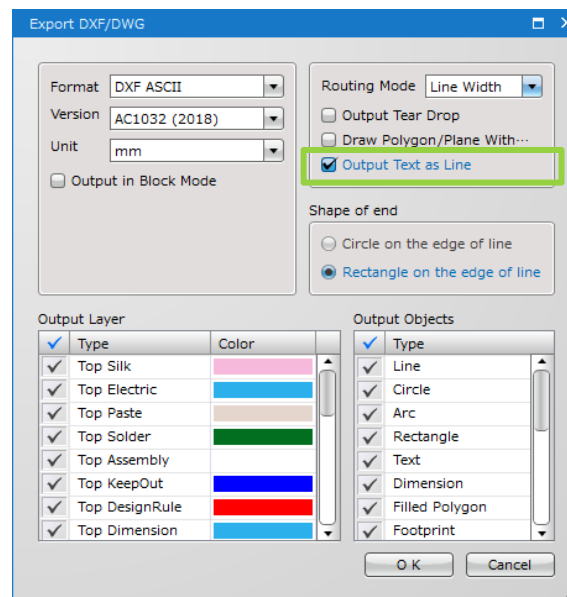
Component Coordinates Now Exportable in CSV Format

- The ability has been added to export component coordinate files in CSV format. The header separating each panel (PCB) has been removed, which makes it easier to edit the file for importing into a manufacturing machine.


[Online Manual](#)

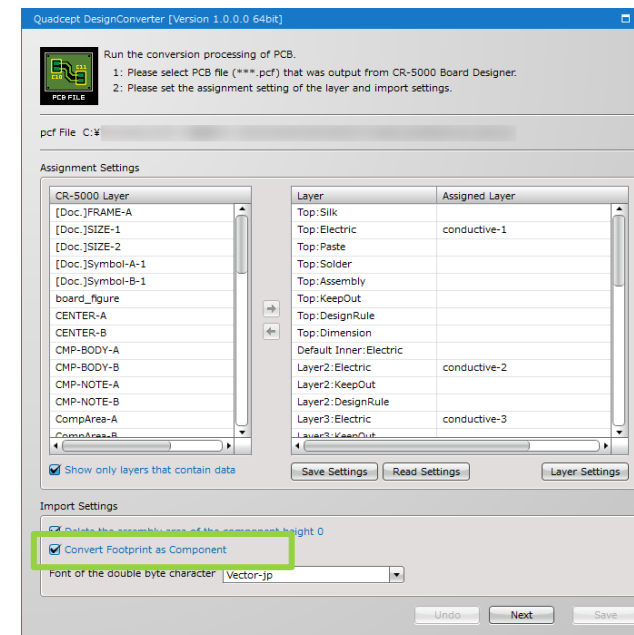

New Option Added to DXF Exporter

- The option to output text objects as line objects has been added to the Export DXF/DWG dialog.


[Online Manual](#)


Enhanced CR5000-BD Converter

- The option to import footprints as components has been added to the CR5000-BD converter.


[Online Manual](#)
