

# Quadcept V10.4.0 Release Notes

**Release Date 2021/01/14** 

© Quadcept Inc. All Rights Reserved.

### 1 SCH PCB

### 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.





### 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 local-db < < share Filter Reference Object 🔳 Footprint . • \* show the components you need. This is New Columns Show unassigned components only Q useful when you want to assign Reference A Pin Match Name Value Footprint footprints only to specific components. 6x1F-H8.5 6x1F-H8.5 AD 0 6x1F-H8.5 The Name(ComponentFileName) and Value **T** 🖃 🗍 default.go 📋 C1 0 03(R) 100n 100n columns have been added. Reference . C2 0 03(R) 100n 100n 🗏 🧰 Sampl Pin Match 0 100n 🗋 C3 03(R) 1u 01\_DrawingFrame 6x1F-H8.5 2/6/2017 2:58-0 🗋 C4 100n 03(R) Footprint 📄 02 Pin 8x1F-H8.5 2/6/2017 2:58-📋 C5 0 03(R) 100n 100n Name 03\_PowerSupply 03(R) 2/6/2017 2:59-0 C6 100n 100n 03(R) 04\_Port 04(R) 2/6/2017 2:59 C7 0 100n 100n 03(R) Value 05\_Symbol 10x1F-H8.5 2/6/2017 2:59 C8 🗊 0 100n 1u 03(R) 2/6/2017 2:59-📋 C9 0 100n 06 Component 0603[1608] 03(R) 22p Import/Export Function C11 0 100n 22p 03(R) Assign/Remove Function 0 M7 💼 d 🗎 D1 M7 0 = M7 You can now assign footprints to -4 ( This buttons allow you to add/remove a )• multiple components at once by using Import Export Assign Remove selected footprint to/from a component. CSV format files. 15\_ne 14 Spice Component \* The following 4 columns are required in the CSV 15 Spice Power files to be imported. The key column is "Reference". C1📄 UserData - Reference : Reference designator 100n - ComponentFileName : Component name - FootprintFileName : Active footprint Include Subdirectories - AssignedFootprintFileName1 : Alternate footprint. You can add this column up to 9. ОК Cancel



### Enhanced Same Object Selection

3

📮 SCH



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



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

	Cancel	Ferana			
	Selection Method		+	Toggle Selection of Object (Next)	н
	Move/ Rotate/ Mirror	,	_ <b>1</b> *	Toggle Selection of Object (Prev)	Shift+H
	Filter	F		Select Same Attribute	
ត	Toggle Object Snap	0	4	Select Same Components	
	Change Component Scale		]ex Pin	Select Same Pins	
	Align	•	r <u>L</u>	Select Same Nets	Shift+N
•	Update Components		3	Spread Selecting	Shift+C
REF	Update Reference				
(à	Extract Selected Symbol/Componen	t			
	Replace Component				
1	Replace Pin				
<b>P</b>	Replace Power Supply			Commands	5
	Port Address				
8	Cross Probe			<ul> <li>Select Same Attr</li> </ul>	ibute
6	Сору	Ctrl+C		Select Same Con	nonents
*	Cut	Ctrl+X		- Select Same Ding	iponenes
â	Paste	Ctrl+V		<ul> <li>Select Same Pins</li> </ul>	
×	Delete	Delete		<ul> <li>Select Same Net</li> </ul>	s
**	Select All	Ctrl+A			
	Show Attribute	А			

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

Online Manual

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

### 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.



PCB



### **Enhanced Routing Capabilities**

PCB

 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







6

PCB

### Trimming Now Available for More Objects

for more objects.

Shape of Corner

🖂 C cut

Orthogonal Angle

Rounding 2.54

2.54



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





Copyright © Quadcept, Inc. All Rights Reserved.

PCB

### Added Ability to Suppress Trailing Zeros in Dimensions

Quadcept

- > 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

2 Point Distance Dimension	□ ×
Display Text Dimension Show	
Display Digit 4	·( )
Zero Suppression None	
Header	
Footer	·····································
Layer	· · · · · · · · · · · · · · · · · · ·
Layer Top 🔹	
Layer Type Dimension 💌	· · · · · · · · · · · · · · · · · · ·
Settings	O K Cancel

#### Zero Suppression : Trailing

Show					P	-						
Display Digit	4		- P			)				Г		7
Zero Suppres	sion Trailir	g	•		<u>~</u>	1						í
Header				•							10	
Footer											õ	Ì.
Layer				•								·
Layer	Тор		-								Ţ	• •
Layer Type	Dimension		-	•						•		÷
				1	2							$\mathbb{N}$
					Ð	•	•	• •	+			
				•								

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





Copyright © Quadcept, Inc. All Rights Reserved.

PCB



## 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.

Component C	oordinates
Format	
Unit	mm
Coordinates	Pad Center
Offset X	0.00
Offset Y	0.00
Mirror	No Mirror
File Type	CSV
	Text
	CSV



### New Option Added to DXF Exporter

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

Fo	rmat DXF ASCII	•	Rou	iting I	Mode Line Width	-		
Ve	rsion AC1032 (201	18) 🔹		Dutpu	t Tear Drop			
Un	it mm		Draw Polygon/Plane With…					
_			<b>S</b>	Dutpu	t Text as Line			
						- 6 11		
utr	out Laver			Outr	ngle on the edge	or line		
utp	out Layer Type	Color		Outp	out Objects			
outp ✓	out Layer Type Top Silk	Color			out Objects Type Line			
outp ✓ ✓	out Layer Type Top Silk Top Electric	Color			ngle on the edge out Objects Type Line Circle			
	ut Layer Type Top Silk Top Electric Top Paste	Color			out Objects Type Line Circle Arc			
	Type Top Silk Top Electric Top Paste Top Solder	Color			ngle on the edge of out Objects Type Line Circle Arc Rectangle			
	tut Layer Type Top Silk Top Electric Top Paste Top Solder Top Assembly	Color			ngle on the edge i but Objects Type Line Circle Arc Rectangle Text			
	Type Top Silk Top Electric Top Paste Top Solder Top Assembly Top KeepOut	Color		Outp V V V V	ngle on the edge i but Objects Type Line Circle Arc Rectangle Text Dimension			
	Type Top Silk Top Electric Top Paste Top Solder Top Assembly Top KeepOut Top DesignRule	Color			ut Objects Type Line Circle Arc Rectangle Text Dimension Filled Polygon			

### Online Manual

#### Enhanced CR5000-BD Converter

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

signment Settings				
CR-5000 Layer		Layer	Assigned Layer	
[Doc.]FRAME-A	1	Top:Silk		
[Doc.]SIZE-1		Top:Electric	conductive-1	
[Doc.]SIZE-2		Top:Paste		
[Doc.]Symbol-A-1		Top:Solder		
[Doc.]Symbol-B-1		Top:Assembly		
board_figure		Top:KeepOut		
CENTER-A	+	Top:DesignRule		
CENTER-B	+	Top:Dimension		
CMP-BODY-A		Default Inner:Electric		
CMP-BODY-B		Layer2:Electric	conductive-2	
CMP-NOTE-A		Layer2:KeepOut		
CMP-NOTE-B		Layer2:DesignRule		
CompArea-A		Layer3:Electric	conductive-3	
CompArea-B		Laver3-KeenOut		
•		•	)	)
Show only layers that contain data		Save Settings Read	Settings	Layer Settin
port Settings				
port Settings		seight 0		
port Settings 7 Delete Ibanovski seto of ibanovski 7 Comunit English av Comunity seto		eight O		
oort Settings X Dalate the assembly area of the com I Convert Footprint as Component		eight 0		

