

Quadcept V10.4.0 Release Notes

Release Date 2021/01/14

© Quadcept Inc. All Rights Reserved.

□ 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.



Copyright © Quadcept, Inc. All Rights Reserved.



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	Eccape	.		
	Selection Method	•	÷.	Toggle Selection of Object (Next)	н
	Move/Rotate/Mirror	,	**	Toggle Selection of Object (Prev)	Shift+H
	Filter	F	2	Select Same Attribute	
í٦	Toggle Object Snap	0	4	Select Same Components	
. :	Change Component Scale]•× Pin	Select Same Pins	
=	Align	•	Ľ.	Select Same Nets	Shift+N
0	Update Components		3	Spread Selecting	Shift+C
12 REF	Update Reference				
iii)	Extract Selected Symbol/Compone	ent			
٥	Replace Component				
3	Replace Pin				
de	Replace Power Supply			Command	S
	Port Address				
æ	Cross Probe			 Select Same Attr 	ribute
Ð	Сору	Ctrl+C		• Select Same Con	nonent
r	Cut	Ctrl+X		Select Same Pins	•
ĉ	Paste	Ctrl+V			•
×	Delete	Delete		 Select Same Net 	S
24	Select All	Ctrl+A			

* 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 be deleted.







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		1	\sim				
Display Digit	4)			7
Zero Suppres	sion Trailing 🔹		<u> </u>				
Header						ιc	•
Footer						0	
Layer						• •	
Layer	Тор					- -	
Layer Type	Dimension •						
			∕∧_	•••	•••		\mathbf{N}

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	uting N	Mode Line Width	
Ve	rsion AC1032 (201	18) 🔹		Outpu	t Tear Drop	
Un	it mm	•		Draw I	Polygon/Plane Wi	ith…
_	Output in Block Mo			Outpu	t Text as Line	
			_		on the edge of lir	
					ngle on the edge	or line
	out Layer	Color			out Objects	
)utp ✓	out Layer Type Top Silk	Color				
~	Туре	Color			out Objects Type	
✓ √	Type Top Silk	Color			out Objects Type Line	
✓ ✓ ✓	Type Top Silk Top Electric	Color			ut Objects Type Line Circle	
✓ ✓ ✓	Type Top Silk Top Electric Top Paste	Color			Type Line Circle Arc	
✓ ✓ ✓ ✓	Type Top Silk Top Electric Top Paste Top Solder	Color			ut Objects Type Line Circle Arc Rectangle	
\checkmark \checkmark \checkmark \checkmark \checkmark	Type Top Silk Top Electric Top Paste Top Solder Top Assembly	Color			ut Objects Type Line Circle Arc Rectangle Text	



Enhanced CR5000-BD Converter

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

CR-5000 Layer (Doc.]FRAME-A (Doc.]SIZE-1 (Doc.]SIZE-2 (Doc.]Symbol-A-1 (Doc.]Symbol-B-1	Layer Top:Silk Top:Electric Top:Paste Top:Solder	Assigned Layer conductive-1	
UDC: JFRAME-A [Doc.]SIZE-1 [Doc.]SIZE-2 [Doc.]Symbol-A-1 [Doc.]Symbol-B-1	Top:Electric Top:Paste	conductive-1	
[Doc.]SIZE-2 [Doc.]Symbol-A-1 [Doc.]Symbol-B-1	Top:Paste	conductive-1	
[Doc.]Symbol-A-1 [Doc.]Symbol-B-1			
[Doc.]Symbol-B-1	Top:Solder		
	Top: Assembly		
board_figure	Top:KeepOut		
LENTER-A	Top: Designikule		
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	
CompAres-B	Laver3-KeenOut		
Show only layers that contain data	Save Settings Read	Settings	ayer Setting
oort Settings			

