Quadcept V10.2　Release Notes

Release Date：08/20/2020

1

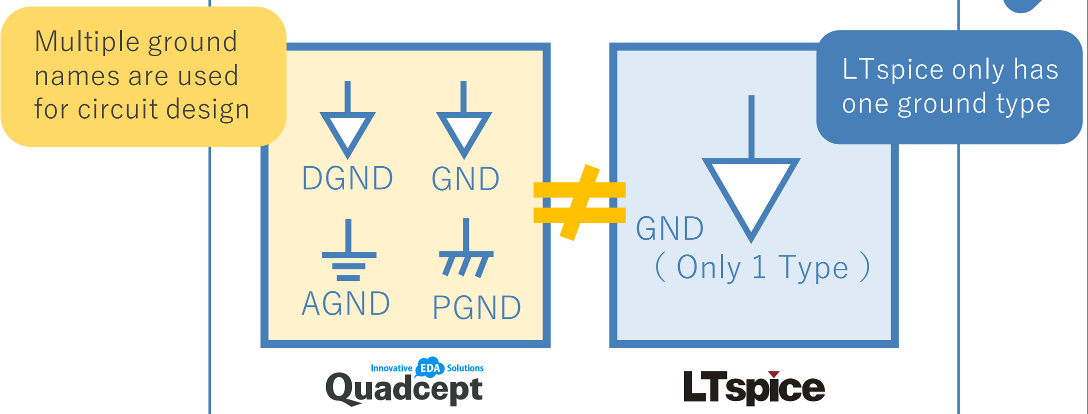
Multiple Nets Now Assignable to a Ground for LTspice Simulations

Added the ability to export user-selected nets as a ground when linking with LTspice.

LTspice has only one type of GND...

Request

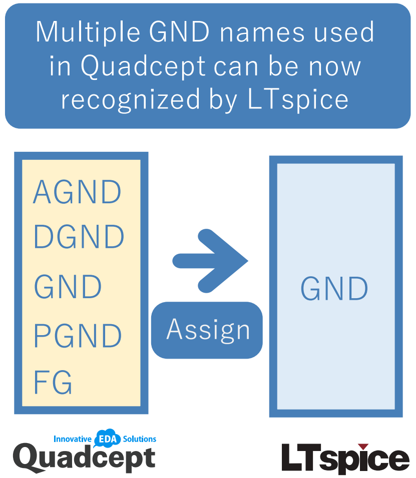
In order to run simulations in LTspice, I need to use a single ground name for circuit design, as LTspice only has one ground type. I want to use multiple ground names even when doing simulations in LTspice.

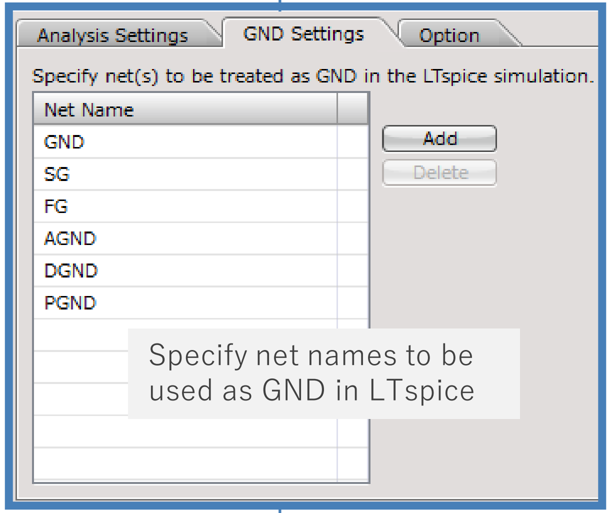


User-selected nets now assignable to a ground

Solution

Quadcept V10.2.0 now supports the ability to export user-selected nets as a ground when linking with LTspice.





[GND Settings Tab]

2-1

Enhanced User Interface and Usability : Direct Connection to LTspice

Each SPICE value and parameter can be now displayed on a schematic and easily edited during design.

Want to run simulations more smoothly...

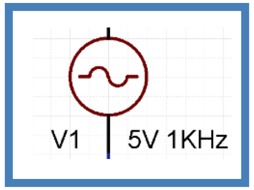
Request.

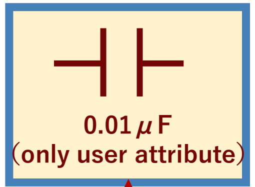
【SPICE Value/Power Component】
SPICE Value is not currently displayed on a schematic, so I need to open its edit dialog every time I want to edit its value. Also, it is hard to notice the difference between the value of a user-defined attribute and that of a SPICE attribute, which can lead to mistakes.

Target	Setting items	Show/hide	edit	Property window editing	Dialog editing
Part	spice Value	×	×	×	O
Power Part	Power settings	×	×	×	O

【SPICE Value＝LTspice Value】

The SPICE Value is not shown on a schematic



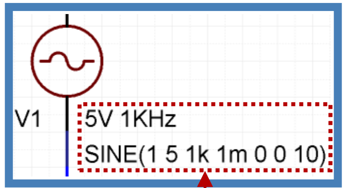


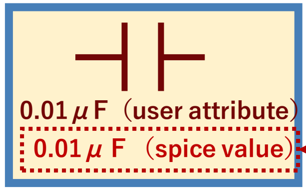
Each SPICE parameter now editable easily

Solution.

【SPICE Value/Power Component】
Each SPICE parameter can be now shown on a schematic. This allows you to edit its value without opening its edit dialog. Also, each SPICE value is editable in the Property window as well. You can now run your simulations more smoothly without having trouble in setting up SPICE parameters.

Target	Setting items	Show/hide	edit	Property window editing	Dialog editing
Part	spice Value	O	O	O	O
Power Part	Power settings	O	O	O	O





2 -2

Enhanced User Interface and Usability : Direct Connection to LTspice

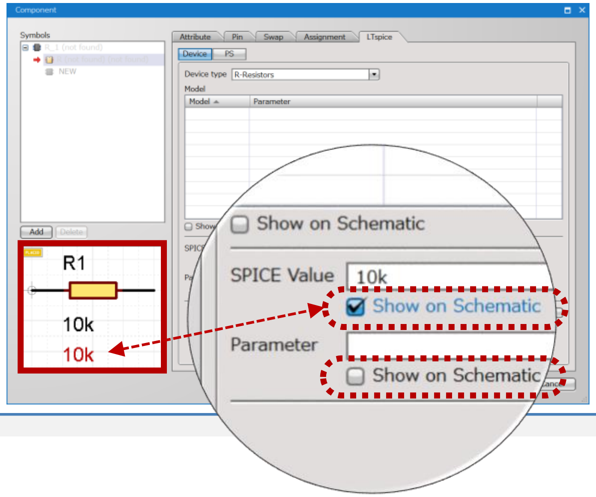
Each SPICE value and parameter can be now displayed on a schematic and easily edited during design.

[SPICE Value]Now Displayable and Editable on a Schematic

In the Component edit dialog, you can toggle the visibility of each SPICE parameter. Showing the SPICE attribute allows you to edit its value on a schematic during design. Each value can be also edited in the Property window located at the left side of the application.

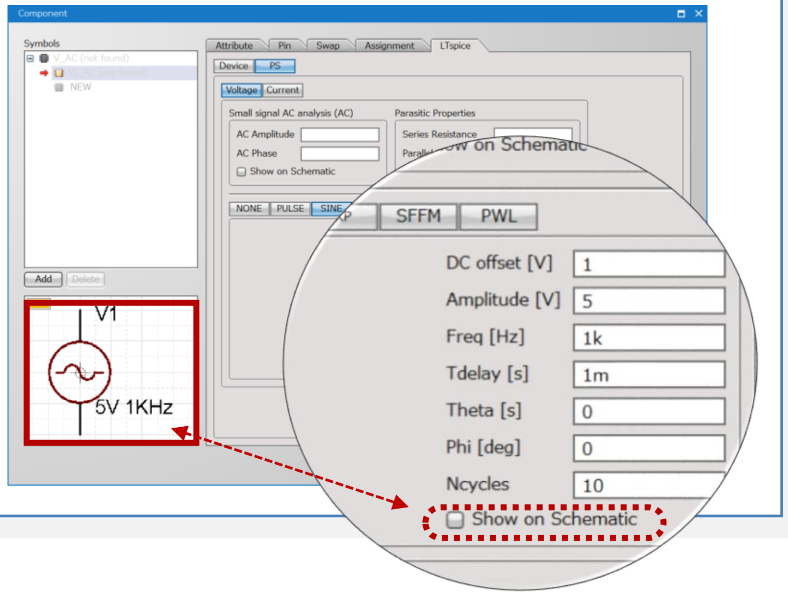
Each SPICE value is now editable on a schematic or in the Property window

Attribute	SPICE Value
Text	10k



[Power Settings] Now Displayable and Editable on a Schematic

Checking the Show on Schematic option of each power setting in the Component edit dialog will allow you to show its value on a schematic sheet.



2 - 3

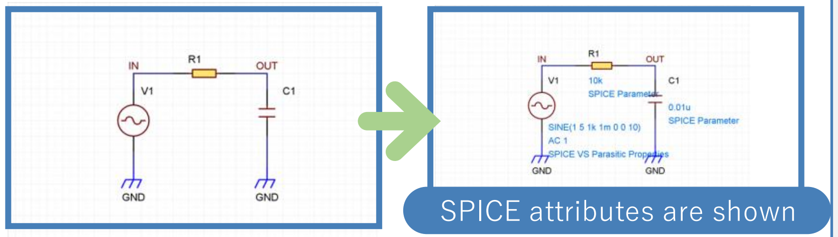
Enhanced User Interface and Usability : Direct Connection to LTspice

Each SPICE value and parameter can be now displayed on a schematic and easily edited during design.

All SPICE Attributes Displayable at Once

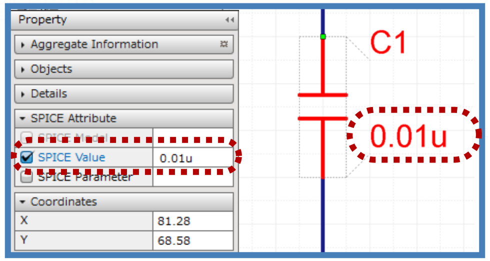
All the SPICE attributes can be shown at once during circuit design, even when you haven't set up any SPICE parameters. This is useful when you want to edit each SPICE parameter to run circuit simulations because you do not have to show each parameter one by one.

Show All SPICE Attributes



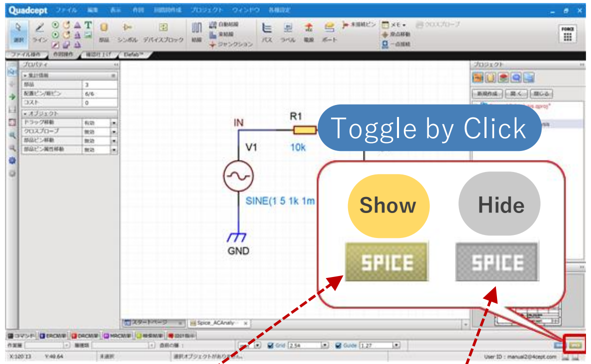
SPICE attributes are shown

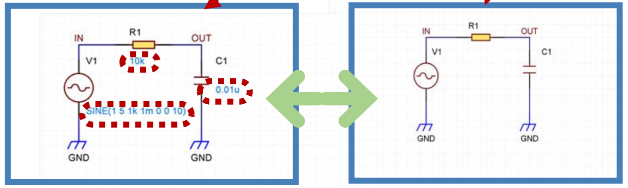
Showing a SPICE attribute enables you to edit its value on a schematic



Toggle Visibility of SPICE Attributes with Ease

You can toggle the visibility of all the SPICE attributes on a whole schematic. This is useful when you want to switch circuit design and SPICE simulation.





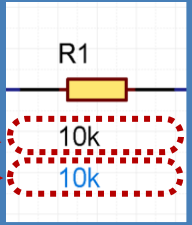
For SPICE Simulation

For Circuit Design

This is also useful when you want to compare the values of a user-defined attribute and a SPICE attribute.

(User-Def)

(SPICE)



3

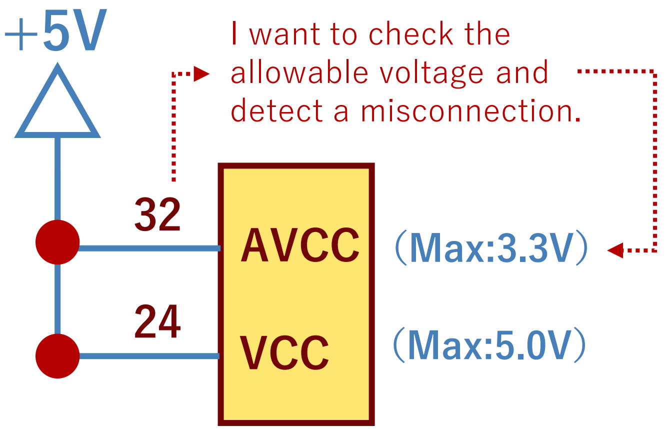
A New Electrical Rule Added for Circuit Verification

Added a new electrical rule to verify that the voltage of power supply connected to a component pin is within the allowable voltage range for the pin.

Want to check the voltage connected to each power supply pin of an IC...

Request.

I want to check the voltage of power supply connected to each power supply pin of an IC component to prevent unintended voltages from being connected.



Unintended voltage now detectable by ERC

Solution.

An error can be now detected when the voltage of power supply connected to a component pin is not within the allowable voltage rage for the pin.

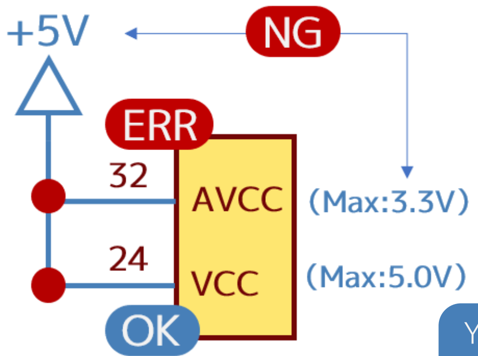
Ex.1 :
Set the allowable voltage for the component pin.

Run ERC

Pin No.	Pin name	ERC Allowable Voltage
32	AVCC	3.3
24	VCC	5

ERC Allowable Voltage
0 to 3.3
0 to 5

Ex.2 :
Set the allowable voltage range in the following format: [Minimum Voltage] to [Maximum Voltage].



You can detect an allowable voltage violation error by specifying the allowable voltage for a component pin

4

The Latest DXF/DWG Version Files Can Be Now Imported/Exported

Importing / Exporting DXF/DWG AC1032(2018) format files is now supported.

Importing / Exporting DXF/DWG AC1032(2018) Format Files Now Supported

Quadcept 10.2 now supports to import/export DXF/DWG AC1032(2018) format files.

This allows you to import/export any DXF/DWG files, including schematics, manufacturing drawings and board outlines, generated in versions up to AC1032(2018).