Quadcept V10.1　Release Notes

6/9/2020

01

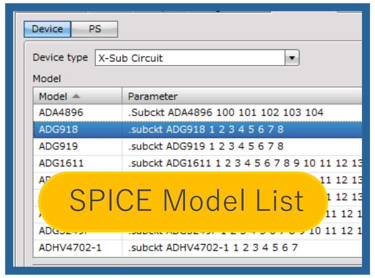
Quadcept Now Linked with LTspice

01

Acquire and Link SPICE Models

A SPICE model can be obtained from a variety of sources such as downloading from a manufacturer's website, creating by hand and purchasing from modeling companies.

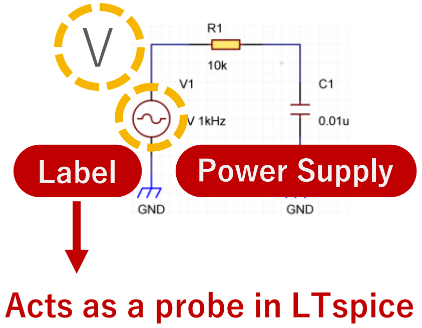
Once you have the required model, adding the model to LTspice allows you to link it to your component in Quadcept.



02

Place a Net Label and Power Supply Symbol to a Wire

In order to run a SPICE simulation in LTspice, it is necessary to place a net label and a power supply symbol on the wire you want to simulate during circuit design in Quadcept. The label will act as a probe in LTspice.



Export SPICE Netlist

Circuit Designer

Quadcept

Innovative EDA Solutions

Link/Analyze

LTspice

ANALOG DEVICES

AHEAD OF WHAT'S POSSIBLE™

Settings in Quadcept

- Link a SPICE model to each component
- Place a label on each wire
- Place a power supply symbol on each wire
- Set simulation commands

Refine Circuits/ Change Settings/ Reanalyze

Operations in LTspice

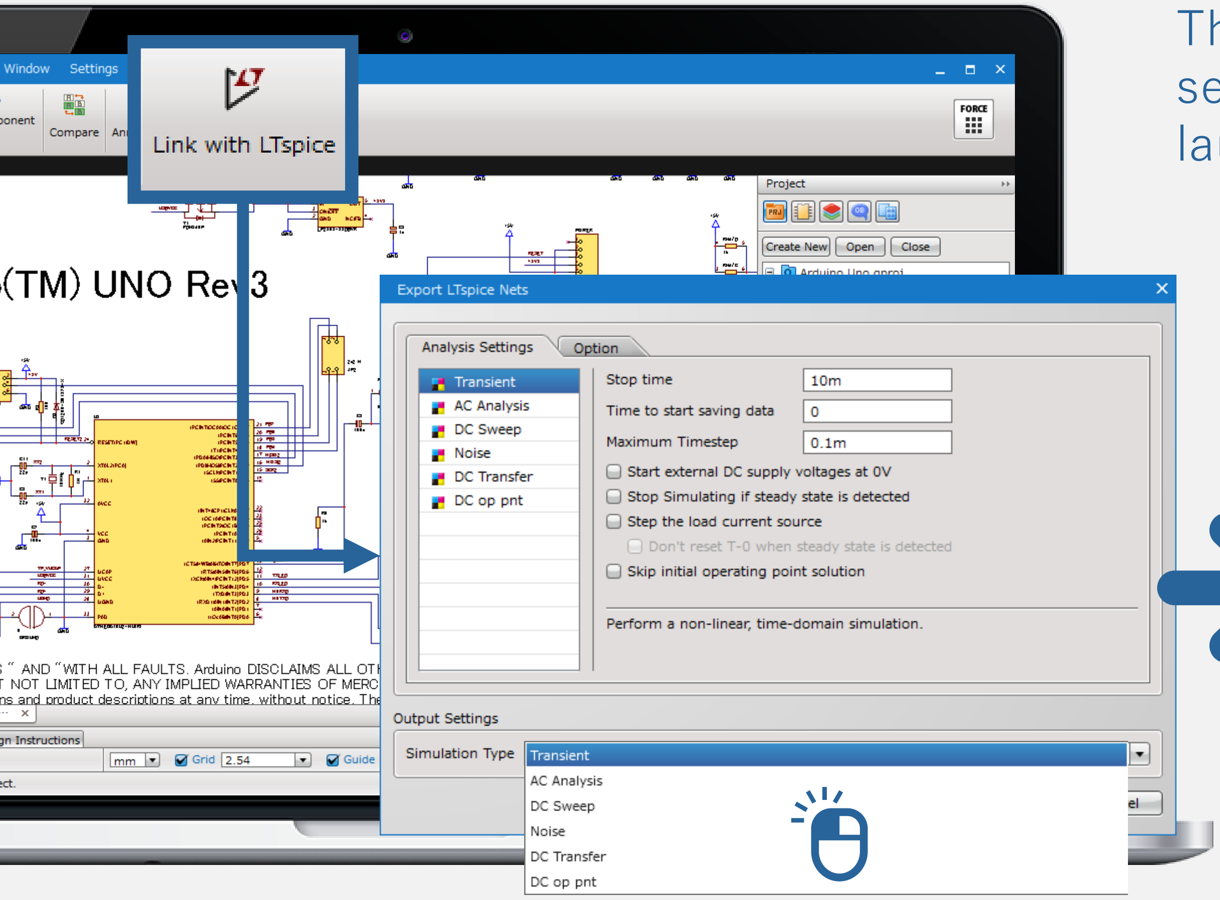
- Specify net names
- Run a simulation
- Check simulation results

Set up simulation commands in Quadcept, and run a simulation in LTspice

See Also: Tips for Using Quadcept and LTspice

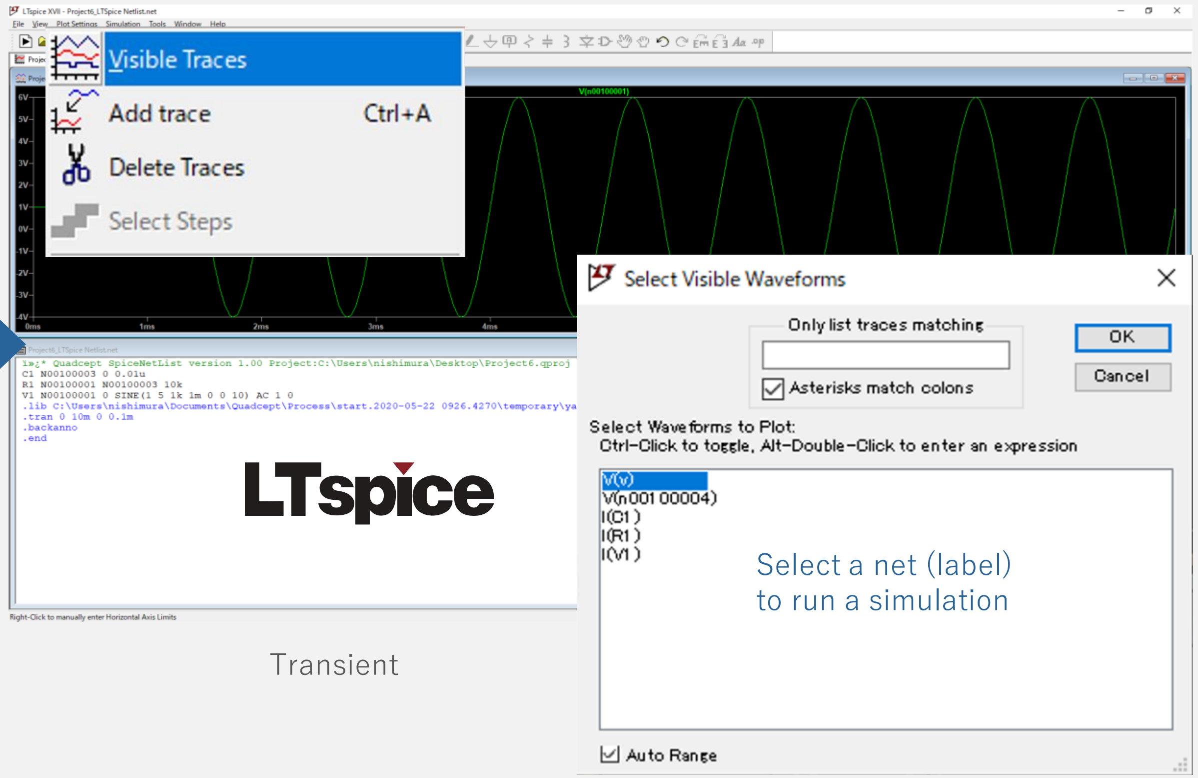
BLOG

Run Simulations and View Waveforms with LTspice



Select SPICE Simulation Type

The simulation commands for 6 simulation types can be set up in Quadcept. Once set up, LTspice will automatically launch and you can run the simulation immediately.



Transient

See Also: Tips for Using Quadcept and LTspice

New Cloud Services Released: Component Management System(CCM) / BOM Management System(CBM)

Design with Real-time Information