

Businesses
Information

Semiconductor business
Technical

Semiconductors
business's
HOME Products/Services

Macnica, Inc.
Technical event Handling Support
Informati seminar Manufacturer

New article

foundation

← LTspice

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10



日本語

Business inquiry

Article search

Please enter a keyword

Search



Click here to narrow down by specifying

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

When using LTspice in practice, does the simulation time become abnormally long?

In such cases, you might think, "It would be easier if we could automatically run simulations of different circuits in the middle of the night."

This time, I will introduce a method to automatically run multiple circuit simulations using the command prompt as a method to eliminate such troubles.

If you are just starting LTspice, we recommend that you look at the "basics" from the list below.

[Let's use LTspice series list is here](#)

Cookie Settings

Reject All Cookies

Accept All Cookies

menu
Businesses Semiconductor business Technical
Information

Semiconductor, Inc.
Technical event Handling Support
business 's Informati seminar Manufacturer
HOME Products/Services

New article

foundation

← LTspice

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

日本語

Business inquiry

Article search



Click here to narrow down by specifying

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

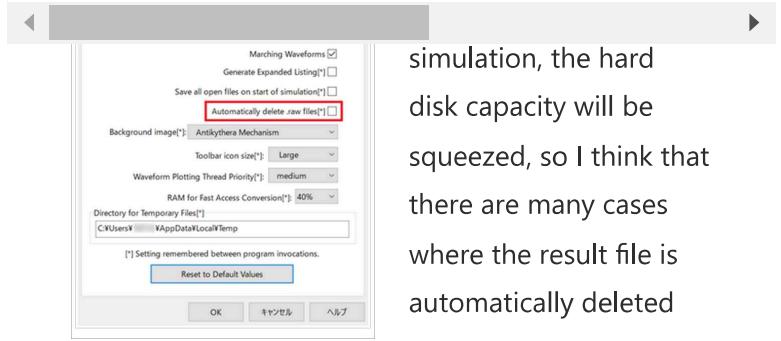


Figure 1: Automatically delete .raw files[*] in Control Panel

simulation, the hard disk capacity will be squeezed, so I think that there are many cases where the result file is automatically deleted after the simulation is finished. If you run a simulation using the command prompt with that setting, all the simulation results will disappear.

Therefore, when executing the simulation using the method explained here, uncheck "Automatically delete .raw files[*]" in the control panel in order to keep the simulation results.

Businesses
Information

Semiconductor business
menu

Technical

Semiconductors
business's
HOME Products/Services

Macnica, Inc.
Technical event Handling Support
Informati seminar Manufacturer

New article

foundation

← LTspice

Figure 2. Change to the directory with the LTspice executable

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

Business inquiry

日本語

Article search

2. Netlist generation

Generate a netlist from a circuit file (.asc). A netlist is generated by executing XVIIx64.exe -netlist "file directory" on the command prompt (Figure 3). file directory specifies the folder that contains the circuit files to run.

This time, I will use the JIG file of the LDO product called LT1117 prepared by LTspice.

On the command prompt, specify C:\Users\username\Documents\LTspiceXVII\examples\jigs\1117.asc and execute. Then, a netlist will be generated in the same folder containing the schematic file (Fig. 4).

For how to use the files prepared in the JIG folder, please refer to the article "[Easy! How to draw a circuit diagram in 5 steps](#)".



Click here to narrow down by specifying

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)



Businesses
Information

Semiconductor business

menu
TechnicalSemiconductor, Inc.
business's Technical event Handling Support
HOME Products/Services Informati seminar Manufacturer

New article

foundation

← LTspice



Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

日本語

Business inquiry

Article search



Click here to narrow down by specifying

3. Run the simulation

Now try running the simulation using the netlist (.net).

By executing XVIIx64.exe -b

C:\Users\username\Documents\LTspiceXVII\examples\jigs\1117.net on the command prompt, the simulation result (waveform file) 1117.raw is generated.

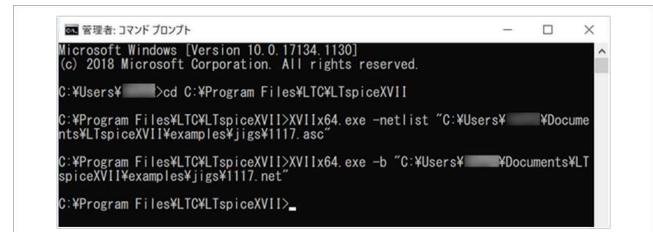
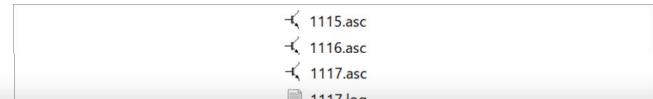


Figure 5: Running the simulation

Waveform data called 1117.raw is output as follows.



We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

Businesses
Information

Semiconductor business
Technical

Semiconductor, Inc.
business's Technical event Handling Support
HOME Products/Services Informati seminar Manufacturer

New article

foundation

← LTspice

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

日本語

Business inquiry

Article search



Click here to narrow down by specifying

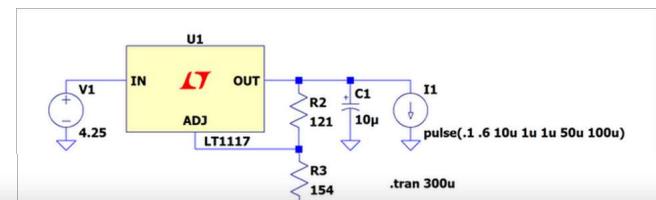


Figure 7: How to display simulation results

In this window, select the node whose waveform you want to see. Figure 10 is the circuit diagram of the simulation we ran this time. I want to check the output waveform, so I check the number of the OUT node.. When I checked with the JIG circuit file, V(n002) corresponds. Be careful because you can't know the number of the node of the OUT pin in advance.

It is convenient to label the nodes before running the simulation so that you can easily identify the node whose waveform you want to check.

(Reference) [Let's use LTspice - clean circuit diagram with "Label Net"](#)



We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

Businesses
Information

Semiconductor business
Technical

menu

Semiconductor business's Technical event Handling Support
Informati seminar Manufacturer
HOME Products/Services

New article

foundation

← LTspice

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

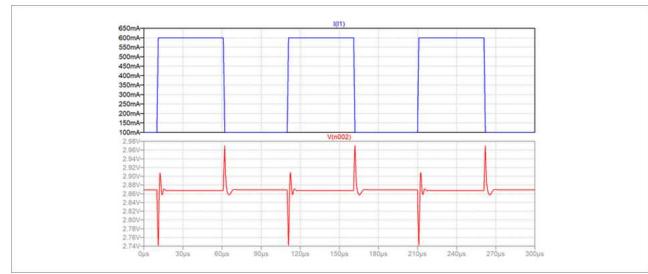


Figure 10: Simulation results

日本語

Batch processing

Business inquiry

Article search

Processing 1 to 5 is the method of manually executing a simulation using commands on the command prompt.

By creating this command processing in text and making it a bat file, it is possible to automatically execute multiple simulations in succession. This is useful when processing multiple time-consuming simulations while you sleep.

As a reference, I made a sample file for batch processing JIG files of LT1117 and LT1118.

Download the sample file below and try it out.



Click here to narrow down by specifying

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

Businesses
Information

Semiconductor business

menu

Technical

Semiconductor, Inc.
business's Technical event Handling Support
HOME Products/Services Informati seminar Manufacturer

New article

foundation

← LTspice



Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

[Download LTspice here](#)

We also hold regular LTspice seminars for beginners. You can learn the basic operation of LTspice, so please participate.

[Click here for LTspice seminar information](#)

日本語

Business inquiry

Article search

Click here for recommended articles/materials

List of articles: [Let's use LTspice Series](#)

LTspice FAQ: [FAQ list](#)

List of technical articles: [technical articles](#)

Manufacturer introduction page: [Analog Devices, Inc.](#)



Click here to narrow down by specifying

Click here for recommended seminars/workshops

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

Businesses
Information

Semiconductor business

menu
HOME Products/Services

Technical

Semiconductor, Inc.
business's Technical event Handling Support
seminar Manufacturer
HOME Products/Services

New article

foundation

← LTspice

◀ Information Top ▶

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

If you want to return to Analog Devices
Manufacturer Information Top, please click below.

[Back to Manufacturer Information
Top](#)

日本語

Previous
of articles

Next article
→

Business inquiry

Article search



Click here to narrow down by specifying

Related information



Features of



Acceleromete

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

Products and Services of Macnica, Inc.

Manufacturing consultation from ideas

Makers/Startup support

Evaluation Board/Development Kit

FPGA IP/Software

Commissioned Development

What is Mpression

Technical Information

New

Foundation

Design

Product Pick Up

The latest information on the semiconductor business

Event/Seminar

Handling Manufacturer

Support

FAQ

Inquiry

MACNICA

M MOUSER ELECTRONICS

[Click here to purchase products](#)

[Click here for details on](#)

registering for the semiconductor business e-mail newsletter

Businesses

Semiconductor

Network

Security

Smart Manufacturing

Smart City/Mobility

AI

DX

Service Robot

About

Macnica, Inc.

About Macnica, Inc.

Company Overview

Base Information

IR Information

CSR Information

Latest Information

Electronic Public

Career

New Graduate Recruitment

Career Recruitment

Employment of People With Disabilities

Entry

Access

[Terms of Use of the Site](#)

[Privacy Policy](#)

[Site Map](#)

[Inquiry](#)

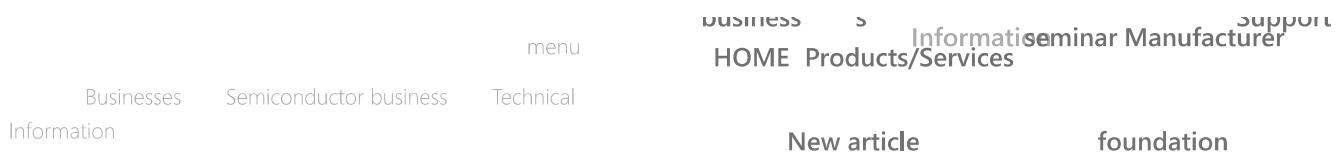
[Support](#)

Measures to prevent the spread of the COVID-19

[Macnica, Inc.'s Response Policy](#)

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)

©Macnica, Inc. All rights reserved.

[← LTspice](#)

Let's use LTspice - let's automatically run multiple simulations with a batch file!

2020.02.10

[日本語](#)[Business inquiry](#)[Article search](#)[Click here to narrow down by specifying](#)

We use cookies to improve your experience on our website, to personalise content and ads, to provide social media features and to analyse our traffic. We share information about your use of our website with our social media, advertising and analytics partners, who may combine it with other information that you have provided to them or that they have collected from your use of their services. Please click [Reject All Cookies] if you reject all cookies. Please click [Accept All Cookies] if you agree with the use of all of our cookies. Please click [Cookie Settings] to customise your cookie settings on our website. You can withdraw your consent at any time via the hover button appears on the bottom left of our website or the Cookie Settings button located in our Cookie Policy page. [Privacy Policy](#)