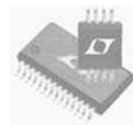


LTSpice IV

Intermediate Lab Class Volume 1

Presented by:
Mats Hellberg
Linear Technology FAE

Copyright © 2010 Linear Technology. All rights reserved



Topics

1. **Importing Third-Party SPICE Models**
2. **Selecting a MOSFET for a DC/DC Converter**
3. **Managing and Customizing Model Libraries**
4. **Piece-wise Linear Voltage Sources**
5. **Appendix**

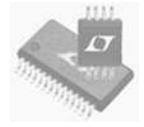


Summary of Hotlinks Used in this Presentation

- C** Class exercise
- S** Solution to exercise
- ⚡** Circuits to explorer at your leisure



Importing Third-Party SPICE Models



Importing Third-Party Spice Models

To import a third party spice model:

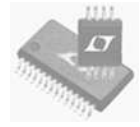
- 1.) Download the spice model file from the manufacturer's website
- 2.) Make sure that the spice model file is located in the same directory as the LTspice simulation file
- 3.) Open up the spice model file and note the device name
- 4.) Add the following spice directive to the LTspice simulation file (Edit pull-down menu ---> SPICE Directive):
`.include spice_model_file_name.abc`
- 5.) Modify the device name in the LTspice schematic to match the device name contained in the spice model file (Right-Click on the device name, and modify the text accordingly)



Importing Third-Party Spice Models

The following items are **CRITICAL!**

- 1.) The file name in the .include statement must match the spice model file name identically! The file name syntax is can be anything, just make sure that all of the characters match.
- 2.) The device name in the spice model file must match the device name in the LTspice schematic identically! The model name syntax can be anything, just make sure that all of the characters match.



Importing Third-Party Spice Models

Spice Model Example #1:

```
1N5244B.mod - Notepad
File Edit Format View Help
* 1N5244B Zener Diode
*
.MODEL 1N5244B1 D
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
+ NR = 3.907
+ BV = 14.00
+ IBV = 0.001
```

File name = 1N5244B.mod

Model name = 1N5244B1

Summary: The file and model names are irrelevant. Just make sure that the device name in the schematic and .include file_name match those of the spice model file.

Spice Model Example #2:

```
Joe.txt - Notepad
File Edit Format View Help
* 1N5244B Zener Diode
*
.MODEL Everest D
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
+ NR = 3.907
+ BV = 14.00
+ IBV = 0.001
```

File name = Joe.txt

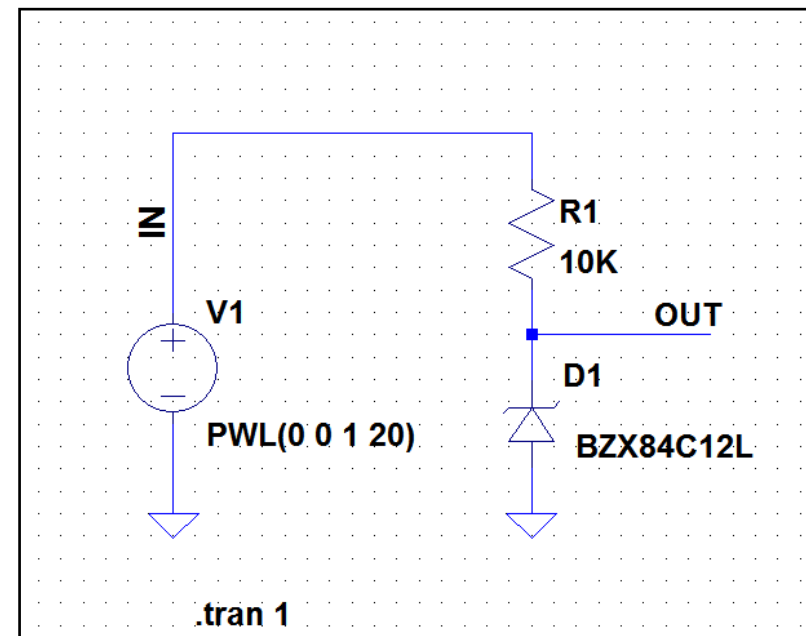
Model name = Everest

Importing Third-Party Spice Models

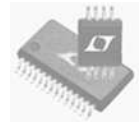


Hands-on Exercise:

- 1.) Open up the simulation file titled
“ZenerImportExample.asc”.
- 3.) Open up the SPICE
model file titled
“1N5244B.mod” and note the
device model name.
- 4.) Modify the simulation file
so that it uses the 1N5244B
third-party SPICE model
based on the instructions
provided on the previous
slides.
- 5.) Run the simulation and
probe the IN and OUT nodes.



Importing Third-Party Spice Models

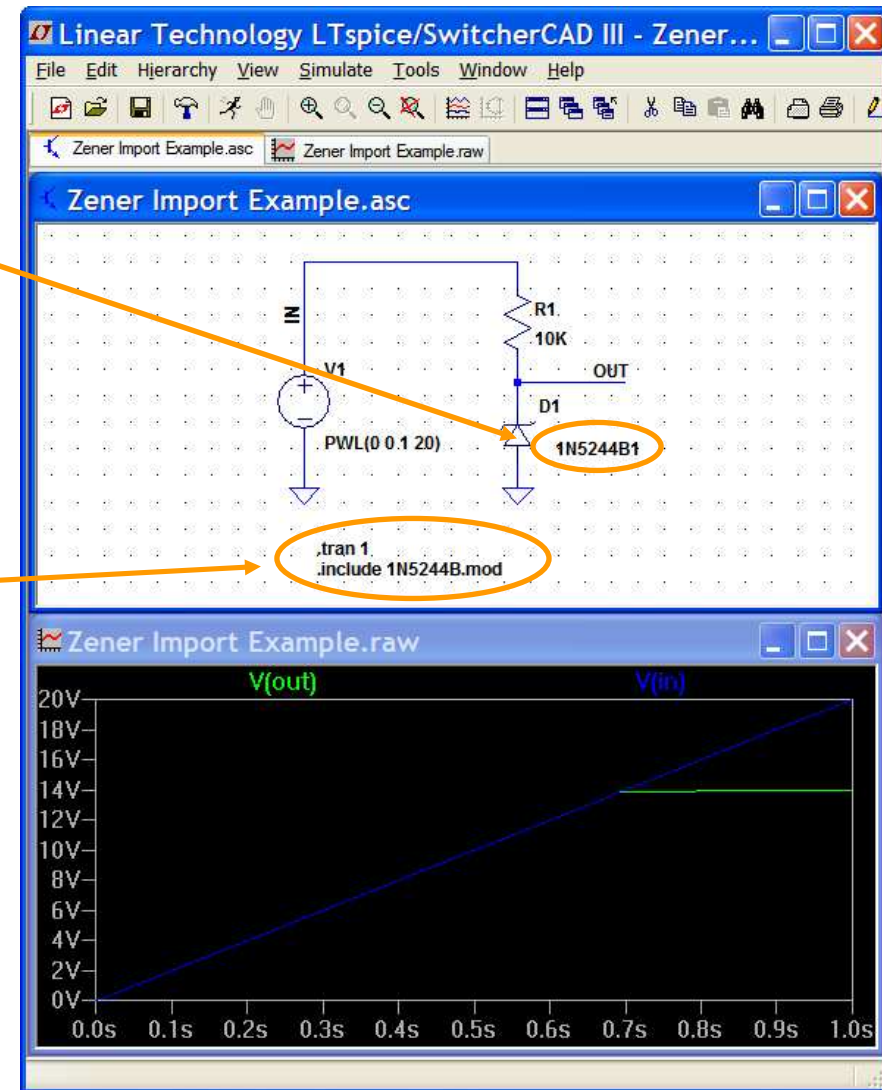


Solution:

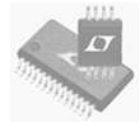
1.) Zener name changed to 1N5244B1 to match model name in the SPICE model file. Right-Click on the diode name text to change.

2.) .include SPICE directive added to link to the SPICE model file. Use the Edit pulldown menu ---> Spice Directive to add this SPICE directive to your simulation.

3.) Result after clicking on the Running Person symbol on the toolbar and probing the IN and OUT nodes.

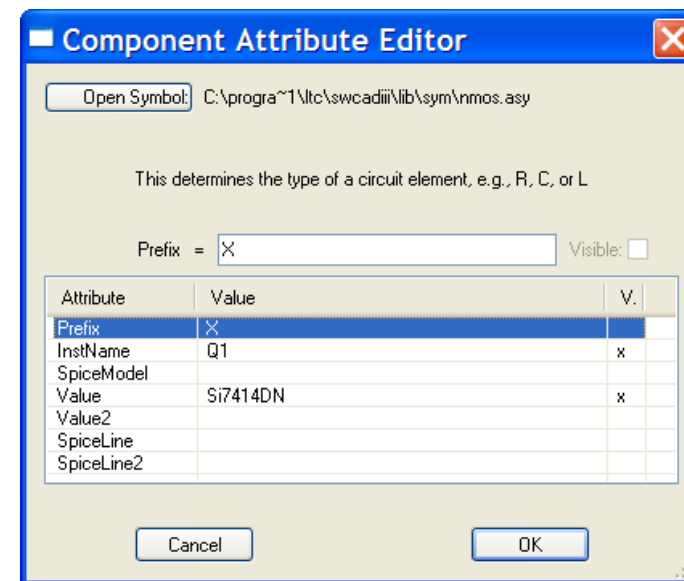


Importing Third-Party Spice Models



Types of SPICE Models (open up the SPICE model file to determine)

- ❖ **.MODEL** definition (as covered in the previous Zener example)
 1. Change the device name in the simulation schematic to match the device name in the SPICE model file
 2. Add the SPICE directive to the schematic `.include spice_model_file.abc`
- ❖ **.SUBCKT** definition
 1. Same as above
 2. Same as above
 3. Must Ctrl-Right-Click on the device and change the Prefix to “X”.



Importing Third-Party Spice Models



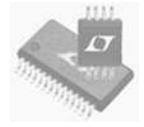
Exercise:

- ❖ Open up the simulation file titled “LTC1871FETImport.asc” and follow the instructions in the simulation file.





Selecting aMOSFET for a DC/DC Converter (LTC1871-7 Example)



Verifying an Appropriate MOSFET for a DC/DC Converter

Exercise:

- ❖ Open up the simulation file titled “LTC1871Boost.asc” and follow the instructions in the simulation file.





MOSFET Parasitics and Power Dissipation (LTC3728 Example)

MOSFET Parasitics and Power Dissipation



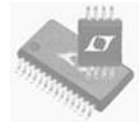
Exercise:

- ❖ Open up the simulation file titled “LTC3728ShootThrough.asc” and follow the instructions in the simulation file.





Managing and Customizing Model Libraries



Managing and Customizing Model Libraries

LTspice Standard Library Files (these can be opened and edited using LTspice)

❖ RCL databases (easy to edit and expand)

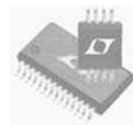
- ❖ standard.res
- ❖ standard.cap
- ❖ standard.ind
- ❖ standard.bead

Custom entries into the library files will not be removed by a Sync Release

❖ Intrinsic Devices (more complicated to edit and expand)

- ❖ standard.dio
- ❖ standard.bjt
- ❖ standard.mos
- ❖ standard.jft

File path for standard library files:
C:\Program Files\LTC\SwCADIII\lib\cmp



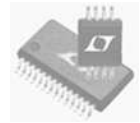
Managing and Customizing Model Libraries

- ❖ Using LTspice, open up the RCL (resistor, capacitor, inductor) libraries and explore. File path to the library files: C:\Program Files\LTC\SwCADIII\lib\cmp
- ❖ Using LTspice, open up intrinsic devices and explore. Mention VDMOS tool (download from LTspice User's Group)
- ❖ Modify "ZenerLibraryExample.asc" to use a library file. Follow the instructions in the simulation file.
- ❖ Run "NPNandLibrary.asc". Notice that the simulation calls out the same library file as the ZenerLibraryExample.asc. A library file can contain models for multiple devices.
- ❖ If you add devices to the library files, your devices will *not* be removed when running a Sync Release.



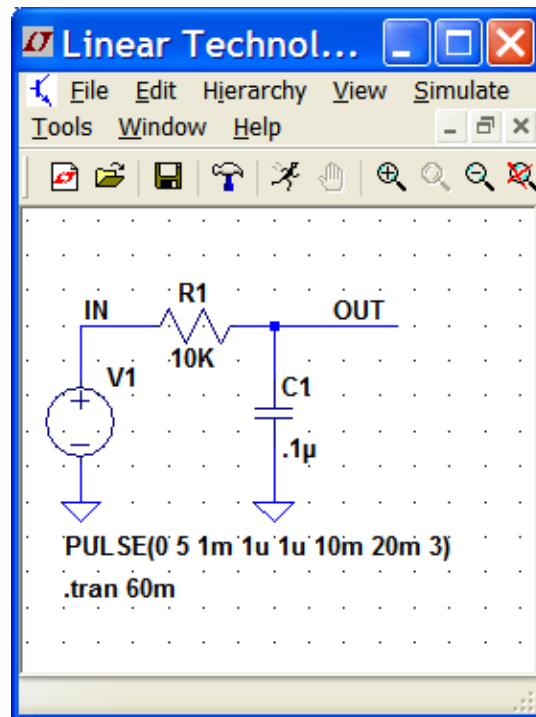


Piece-Wise Linear (PWL) Voltage Sources



Creating a PWL Voltage Source

- ❖ Open up the simulation file titled “RCFilterTimeDomain.asc”
- ❖ Run the simulation and probe the IN and OUT nodes
- ❖ Right-Click on the voltage source and select the PWL function
- ❖ Configure the PWL source to manually recreate the pulse waveform as shown in the voltage source window on the right
- ❖ Rerun the simulation. Notice a single pulse is now present.



Independent Voltage Source - V1

Functions

- ☐ (none)
- ☐ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- ☐ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
- ☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)
- ☐ SFFM(Voff Vamp Fcar MDI Fsig)
- ☒ PWL(t1 v1 t2 v2...)
- ☐ PWL FILE: Browse

time1[s]: 0
value1[V]: 0
time2[s]: 1u
value2[V]: 5
time3[s]: 10m
value3[V]: 5
time4[s]: 10.001m
value4[V]: 0

Additional PWL Points

Make this information visible on schematic: ☒

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis (AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

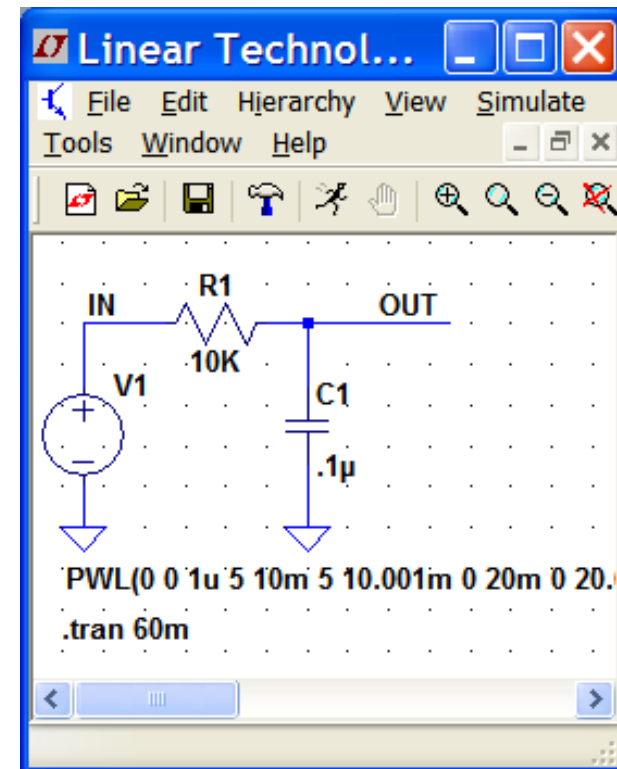
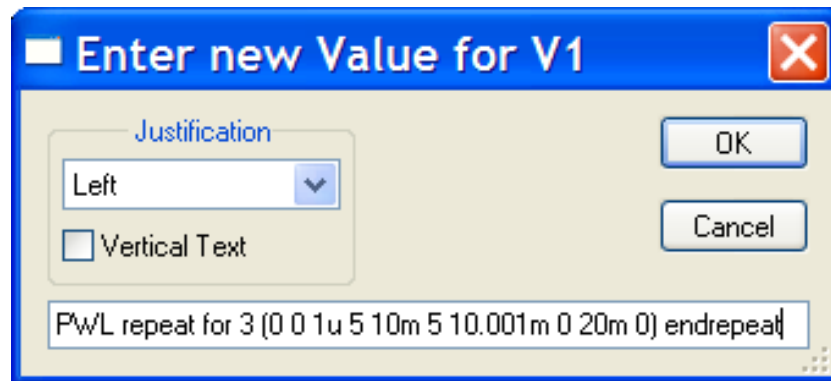
Make this information visible on schematic: ☒

Cancel OK

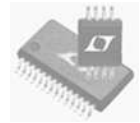
Repeating PWL Source – RC Circuit Revisited



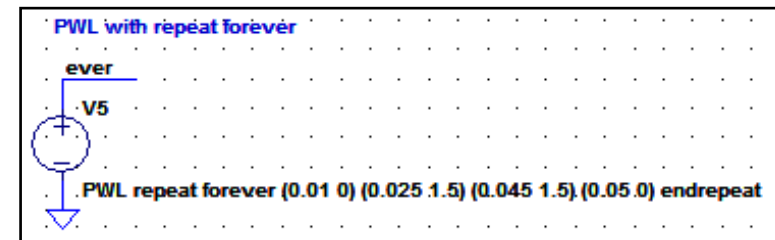
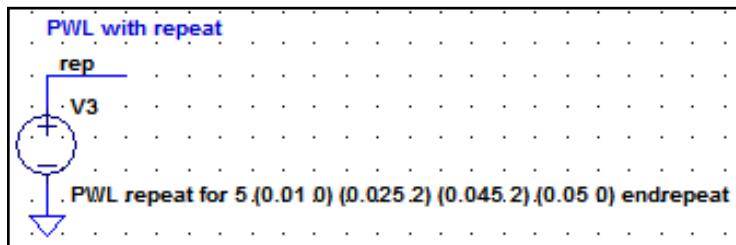
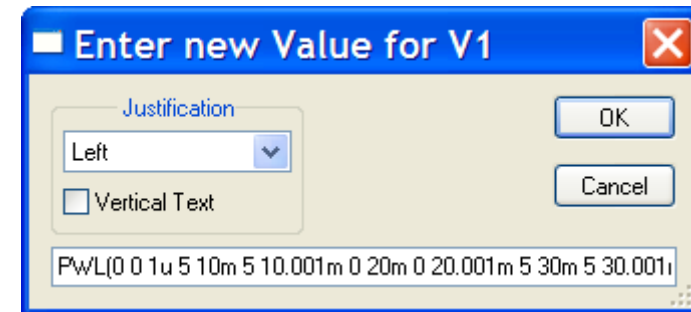
- ❖ Open up the simulation file titled “RCFilterTimeDomainPWL.asc”
- ❖ Run the simulation and probe the IN and OUT nodes
- ❖ Right-Click on the PWL text string and use the repeat command to create three cycles of the input square wave.



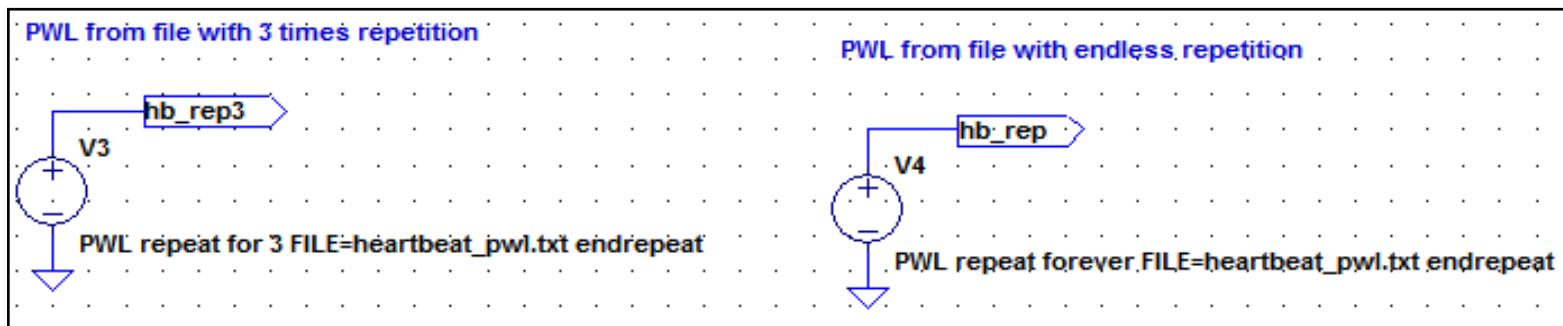
Repeating PWL Source – Additional Info



- ❖ To edit the PWL source attributes, Right-Click on the PWL text string on the schematic
- ❖ The following window will appear -->
- ❖ In the command line, modify the PWL command (examples below)



- ❖ Repeating PWL format using a PWL source file (see next page):





Importing Externally Generated PWL Sources

- ❖ To import a PWL waveform from a file, Right-Click on a voltage source, select “Advanced”, and select “PWL File”
- ❖ The file format must contain pairs of numbers separated by white space (carriage return, spaces, tabs). The first number is time (in seconds) and the second number is voltage.

- ❖ Example 1:

- ❖ 0 0 0.1 1 0.2 0.5 0.5 0 0.7 0.3 1 0



- ❖ Example 2:

- ❖ 0 0

- ❖ 0.1 1

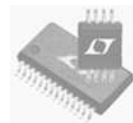
- ❖ 0.2 0.5

- ❖ 0.5 0

- ❖ 0.7 0.3

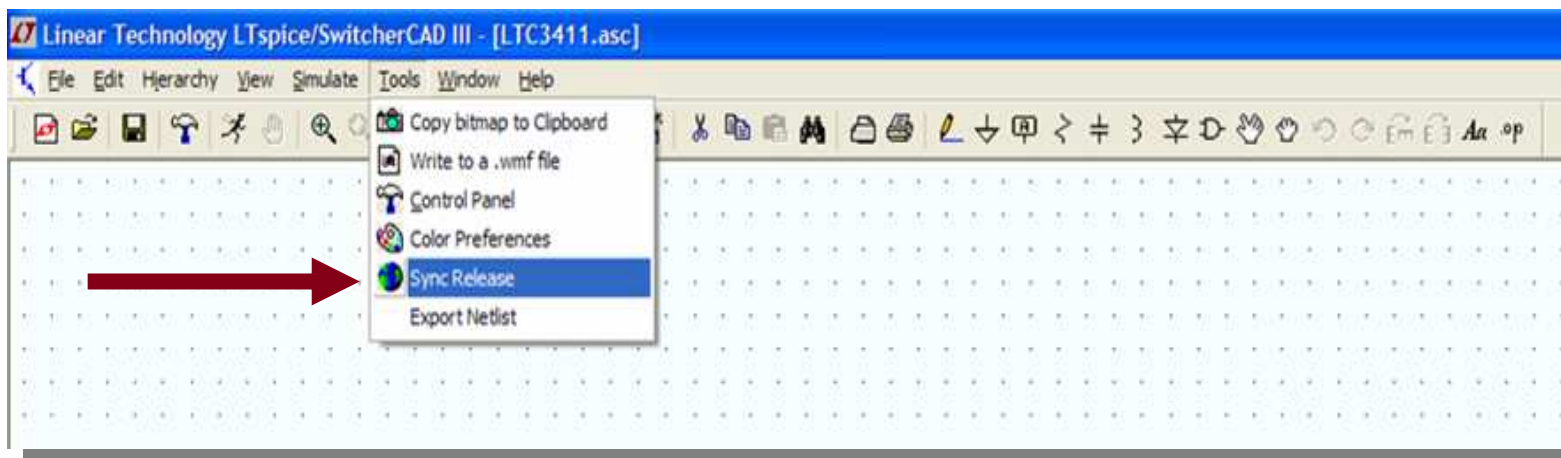
- ❖ 1 0





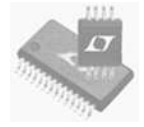
Reminder to Periodically Sync Release

- ❖ Update your release of LTspice to get the latest
 - ❖ Software updates
 - ❖ Models and examples



Vista users:

***You must “Run as administrator” scad.exe
even if you are logged in as an administrator***



Appendix A – Summary of Special Mouse and Keyboard Commands

- ❖ **Schematic-Based Special Commands:**
- ❖ **Alt-Left-Click on a wire**
 - ❖ This will display the waveform for the current flowing in the wire
- ❖ **Alt-Left-Click on a component**
 - ❖ This will display the instantaneous power dissipation in the component
- ❖ **Ctrl-Right-Click on a component**
 - ❖ Allows you to edit embedded component attributes
- ❖ **Waveform-Based Special Commands:**
- ❖ **Ctrl-Left-Click on a waveform title**
 - ❖ Displays the average and RMS values for the waveform

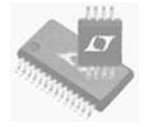


Appendix B – Summary of Additional Features

- ❖ To pause a simulation:
 - ❖ “Simulate” pull down menu ---> Pause
- ❖ To zoom in/out using the schematic editor:
 - ❖ Just use the wheel on your mouse
- ❖ To pan around a schematic
 - ❖ Just Left-Click the mouse and hold, then drag
 - ❖ Tilt wheel to move right and left

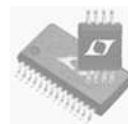


Appendix C – Efficiency Calculations



Steps to Calculate Power Supply Efficiency

- 1. Efficiency will only be calculated in the steady state condition**
- 2. Right-Click the .tran statement on the schematic to bring up the Edit Simulation Command dialog box**
- 3. Check the box “Stop simulating if steady state is detected”**
- 4. Load must be a current source or resistor labeled Rload**
- 5. Run the simulation**
- 6. Upon completion select the View dropdown menu, then Efficiency Report, then Show on Schematic**
- 7. Re-run simulation**
- 8. Efficiency report will be pasted under the schematic**



Edit Simulation Command

Edit Simulation Command

Transient AC Analysis DC sweep Noise DC Transfer DC op pnt

Perform a non-linear, time-domain simulation.

Stop Time: 500u

Time to Start Saving Data:

Maximum Timestep:

Start external DC supply voltages at 0V: ☒

Stop simulating if steady state is detected: ☒

Don't reset T=0 when steady state is detected: ☐

Step the load current source: ☐

Skip Initial operating point solution: ☐

Syntax: .tran <Tstop> [<option> [<option>] ...]

.tran 500u steady startup

Cancel OK



Power Supply Efficiency Caveats

- ❖ LTSpice will not always be able to determine steady state, but this is rare!
- ❖ Probe the OUT node and verify that it has stabilized
 - ❖ If not edit the .tran statement and increase the Stop Time parameter
 - ❖ Re-run simulation
- ❖ Efficiency must be determined partially by hand for multiple output and/or multiple input supplies
- ❖ Right-Clicking any component will report power dissipation

Thank you for attending, and happy simulating!



Homework: Once you return to the office, go back over the training materials within a week!