Installation of LTspice on Windows and Mac

Dr. Abdulrahman Kalbat Electrical Engineering Department, United Arab Emirates University

Last Update: May 22, 2019

Table of Contents

- 1. Instructions for Installing LTspice on Windows
- 2. Instructions for Installing LTspice on Mac OSX
 - <u>Step 1: Install WineBottler</u>
 - <u>Step 2: Install LTspice</u>
- 3. Installing University of Evansville LTspice Library (for additional components)
 - Instructions for Windows
 - Instructions for MAC OSX

Instructions for Installing LTspice on Windows

1. Go to the link below:

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html



2. In the "Download LTspice" section, click on "Download for Windows 7, 8 and 10" button to start the downloading process.



3. Once the file is downloaded, please double click the file and follow the instructions to install the software.



e take no lator whether

Browse

- 🗆 🗙

4. Done.

software exponsibility fo

LTspice XVII has b

have OK rates in the Upgrades, ns, or repairs to this program will be strictly at the discretion of LTC. If Cancel Cancel Additional Control Contr

Install Now

Instructions for Installing LTspice on Mac OSX

The instructions below are for installing the Windows version of LTspice on MAC. We use this approach since the Mac version of LTspice is not user friendly and some features are even missing.

Step 1: Install WineBottler

This is a software used to run some windows softwares on Mac without having a windows license, which is required when using a virtual machine. It will be used here to run LTspice on Mac OSX.

1. Go to the link below:

http://winebottler.kronenberg.org/



 Click on the second button which includes the stable version of WineBottler "WineBottler 1.8.6 Stable" → Click on Download → Skip the advertisement → If your download does not start in 5 seconds, click on "WineBottlerCombo_1.8.6.dmg".





2	ave As: WineBottlerCombo_1.8.6	
< > ** = ··· **	Downloads	Q Search
	Format: Disk Image	
Hide extension New Folder		Cancel Save

Double click the downloaded file "WineBottler Combo" → A new small window will pop up → Highlight the icons "Wine" and "WineBottler" → drag both of them to the "Applications" folder to install them.



4. Go to the "Applications" folder and check if both apps are working properly by double clicking the icon of each one of them.

5. Below is a window of a properly working "WineBottler".



- 6. In case of "Wine", if it works properly, then it will show up in the status menu at the top of the Mac OSX window as shown below (the icon is highlighted in red on the left).
 - 💡 🤝 🖬 10% 🖓 🛄 Thu 12:39 PM 🔍 🔕 ≔
- If any error is produced when double clicking on any app, then right click on the app icon → hold the "option" key on the keyboard → click on "open". Below is an example of an error produced by "Wine".



8. If both apps are working properly, then proceed to the next step.

Step 2: Install LTspice

This is a simple yet powerful free spice simulation software (Multisim and orCad are also good but Multisim's free version is time-limited and OrCad is very heavy)

1. Go to the link below:

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html



2. In the "Download LTspice" section, click on "Download for Windows 7, 8 and 10" button to start the downloading process.



3. Open the application "WineBottler" that was installed in <u>Step 1</u>.



4. Once the application window show up, go to "Advanced" and a yellow window should show up.



 Click on "Select File" → select the .EXE file for LTspice downloaded in point 2 of Step 2 (file name is LTspiceXVII.exe)



6. Click "Install" at the lower right corner of the yellow window.



7. Select the location of where you want the installed application to be (later, you will double click on the icon created in this location to run the windows application on Mac). You could call this file "LTspice WineBottler" and the location could be "Applications" as shown below.



8. Click "save".



9. A new window with the title "LTspice XVII Installer" will show up.



10. Click on "Accept" \rightarrow then click on "Install Now" \rightarrow once the installation is done, click on

۷	٢	ſ	١	k	٢	,	,
		Ľ	,	P			

	C LTspice XVII Installer	
LTspice XVII Installer	Installs LTspice XVII build date Mar 15 2019, 12:10:06 US Pacific	
Installs LTspice XVII build date Mar 15 2019, 12:10:06 US Pacific		
LTspice XVII License Agreement and Disclarmer Copyright © 2016-2018 Linear Technology Corporation Al rights reserved.	LTspice XVII License Agreement and Disdaimer Copyright © 2016-2018 Linear Technology Corporation All rights reserved.	<u>-</u>
LTspice XVII is Linear Technology Corporation's analog circuit simulation software.	LTspice XVII is Linear Technology Corporation's analog circuit simulation software.	
This software is copyrighted. You are granted a non-exclusive, non-transferable, non-suble-meakle, novaly-free right to evaluate ICT conducts and also to perform general circuit smalation. Linear Technology Corporation owns the software. You may not modify, adapt, translate, neverse engineer, decomple, or disassemble the software executable(s) or models of ILT ponducts provided. We take no responsibility for the accuracy of third party models used in the simulator whether provided by LTC or the user.	This software is copyrighted. You are granted a non-exclusive, non-transferable, non-sublenceable, roydly-there infit to exvaluate LTC products and also to perform general arout simulation. Linear Technology Corporation owns the software. You may not modify, adapt, translate, reverse engineer, decomptle, or dessemble there software executable(s) or modes of LTC products provided. We take no responsibility for the accuracy of third party models used in the simulator whether provided by LTC or the user.	-
While we have made every effort to ensure that LTspice XVII operates in the manner desorbed, we do not guarantee operation to be error free. Upgrades, modifications, or repairs to this program will be strictly at the discretion of	While we have made every effort to ensure that LTspice XVII operates in the manner described, we do not guarantee operation to be error free. Upgrades, modifications, or repairs to this program will be strictly at the discretion of	🗉 🔵 🔘 LTspiceXVII
Cancel Accept Dedine	Cancel Accept Decline	
Executable Version(s): x86(32-bit)	Executable Version(s): x86(32-bit) *	LTspice XVII has been successfully installed
Installation Directory; CCIPhogram Files/LTCI, Topice/VII Drowse Drotal from Drotal from	Installation Directory: C-IProgram Files(LTC)(LTSpiceWIT Browse Installation Directory: C-IProgram Files(LTC)(LTSpiceWIT Brows	OK

11. A new window with the title "Select Startfile" will show up → from the dropdown menu, select "Program Files/LTC/LtspiceXVII/XVIIx86.exe" → click "OK" → then click "OK" again.

	Select Startfile This file will be run when you double-click the created application. Program Files/Internet Explorer/lexplore.exe 🗘	WineFile Program Files/LTC/LTspiceXVII/XVIIx86.exe Program Files/LTC/LTspiceXVII/UnLink.exe Program Files/LTC/LTspiceXVII/MoveExe.exe Program Files/LTC/LTspiceXVII/MoveExe.exe Program Files/Windows Media Player/wmplayer.exe Program Files/Windows NT/Accessories/wordpad.exe Y Program Files/Internet Explorer/iexplore.exe
X	OK Select Startfile This file will be run when you double-click the created application. Program Files/LTC/LTspiceXVII/XVIIx86.exe C OK	OK Prefix created sucessfully. OK

12. Close all windows.

13. Go to the location of the file "LTspice WineBottler" selected in point 7 of step 2 → double click the file.



14. This will run the windows application. The first time the application is run, it will take some time until everything is configured. Then, LTspice software window should show up, as shown below.





15. Done.

Installing University of Evansville LTspice Library (for additional components)

The UE archive supplements the components and libraries supplied with LTSpice. The components and libraries in the archive are intended for students of circuits and electronics.

Instructions for Windows

1. First, go to the following path: C:\Users\UserName\Documents\LTspiceXVII\lib\ where

UserName should be the name of the user on the PC. Keep this folder open to use it next in Step 4 and Step 5.

2. Then, download the zip file that contains the library files from the link below:

http://csserver.evansville.edu/~richardson/courseware/resources/LTSpice/ue_ltspice_archive.zip

- Unzip the downloaded file "ue_ltspice_archive". Go to the unzipped folder "ue_ltspice_archive" → then go to the subfolder "lib"
- Copy the <u>files</u> from the path \ue_ltspice_archive\lib\sub\ to the path
 C:\Users\UserName\Documents\LTspiceXVII\lib\sub\
- Copy the <u>folders</u> from the path \ue_ltspice_archive\lib\sym\ to the path
 C:\Users\UserName\Documents\LTspiceXVII\lib\sym\
- 6. To test whether the new components were correctly added to LTspice or not, please run LTspice and then create "New Schematic". Go to "Edit" and then "Component". You should see a window like the one below. The component folders that start with numbers are the newly added components. For example, "03_coupled_inudctors" is a newly added component library.

	S	elect Componer	nt Symbol	
Top Directory:	C:\Use	ers \abdulrahmankalbai	t\Documents\LTspiceXVII\	lib∖syr ∨
)pen this macromodel's test	fixture
C:\Users\ab	odulrahma	ankalbat\Documents\	LTspiceXVII\lib\sym\	
C:\Users\ab [00_components] [00_Test_Circuits [01_sources] [02_dependent_s [02_dependent_s [10_diodes] [20_bt]_transiton [21_mosfet_trans [22_fet_transiston [22_fet_transiston [40_opamps] [70_laplace_sour <] sources] uctors] s] sistors] rs]	[80_miscellaneous] [85_analog] [90_digital]	LTspiceXVII\ib\sym\ [PowerProducts] [References] [SpecialFunctions] [Switches] bi bi2 bv cap csw current diode	e e2 f Fentte g g2 h ind ind2 ISO1!

7. Done.

Instructions for MAC OSX

 First, go to the location of the file "LTspice WineBottler" selected in point 7 of step 2 → Right click on the file "LTspice WineBottler" → Select "Show Package Contents" → Follow the following path /Contents/Resources/wineprefix/drive_c/Program

Files/LTC/LTspiceXVII/lib/. Keep this folder open to use it next in Step 4 and Step 5.

2. Then, download the zip file that contains the library files from the link below:

http://csserver.evansville.edu/~richardson/courseware/resources/LTSpice/ue_ltspice_archive.zip

- Unzip the downloaded file "ue_ltspice_archive". Go to the unzipped folder "ue_ltspice_archive" → then go to the subfolder "lib"
- Copy the <u>files</u> from the path /ue_ltspice_archive/lib/sub/ to the path /Contents/Resources/wineprefix/drive_c/Program Files/LTC/LTspiceXVII/lib/sub/
- 5. Copy the <u>folders</u> from the path /ue_ltspice_archive/lib/0_UofEvansville/sym/ to the path /Contents/Resources/wineprefix/drive_c/Program Files/LTC/LTspiceXVII/lib/sym/
- 6. To test whether the new components were correctly added to LTspice or not, please run LTspice and then create "New Schematic". Go to "Edit" and then "Component". You should see a window like the one below. The component folders that start with numbers are the newly added components. For example, "03_coupled_inudctors" is a newly added component library.

9 🔴 S	elect Component	Symbol	
Top Directory: C:\user	s\abdulrahmankalbat\	My Documents\LTspiceXVII	lib\s_▼
	One	en this macromodel's test fix	ture
			cour o
C:\users\abdulrahma	nkalbat\My Document:	s\LTspiceXVII\lib\sym\	
[00_components]	[80_miscellaneous]	[PowerProducts]	e
[00_Test_Circuits] [01_sources]	[85_analog] [90_digital]	[References] [SpecialFunctions]	e2 f
[02 dependent sources]		[Switches]	Fe
[03_coupled_inductors]	[Comparators]	bi	Fe
[10_diodes]	[DAC]	bi2	g
[20_bjt_transistors]	[Digital]	bv	g2
[21_mosfet_transistors] [22_jfet_transistors]	[FilterProducts] [Misc]	cap csw	h ind
[40 opamps]	[Opamps]	current	inc
[70_laplace_sources]	[Optos]	diode	IS
•			Þ
Cancel	1	ОК	

7. Done