

LTspice Tutorial Part 1- Download and Installation



Prerequisites

- A Windows or Mac OS based platform
 - All subsequent LTspice tutorials will be based on Windows.
 - For Mac users, please see <https://www.youtube.com/watch?v=MsiYqaGPQw&feature=youtu.be> and similar guides. Alternatively, the University's Windows based EWS machines can be accessed remotely via Citrix <https://it.engineering.illinois.edu/ews/lab-information/remote-connections/connecting-citrix>. The EWS machines should already have LTspice installed.

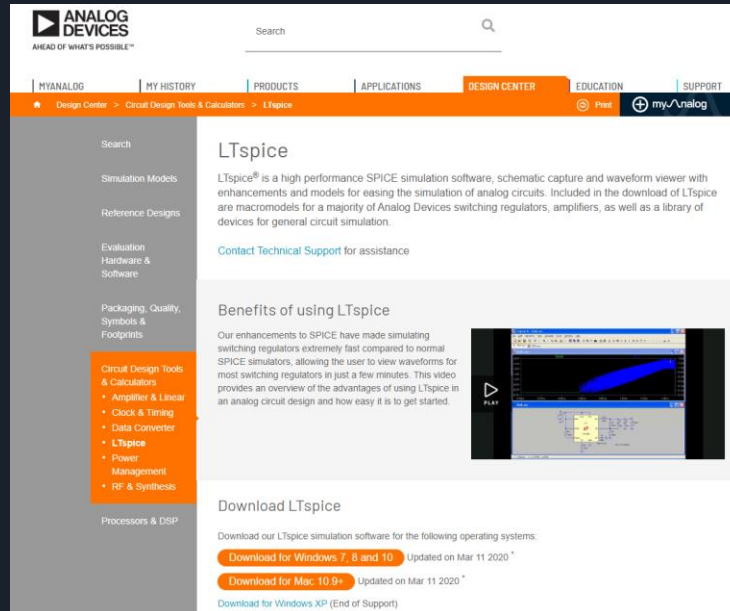


Tutorial 1 Objectives

1. Install LTspice XVII on your computer
2. Learn where the main and library directories are for LTspice

Download

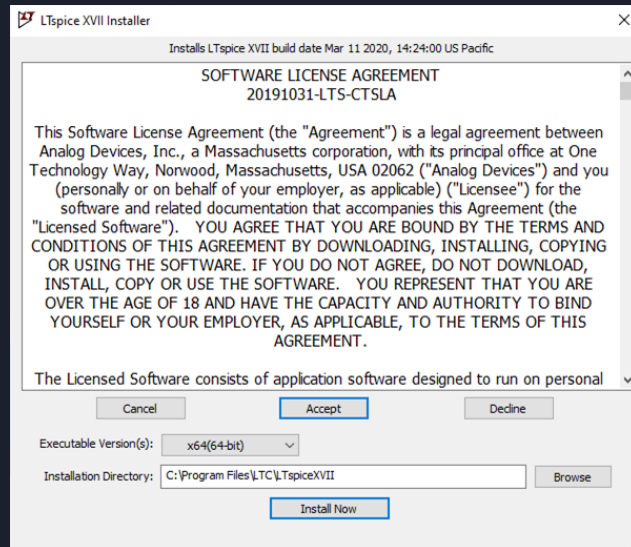
- The current version of LTspice (as of March 20, 2020), is XVII.
- Download LTspice at <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
- Select appropriate platform (Windows or Mac).



The screenshot displays the Analog Devices website's LTspice download page. The page features the Analog Devices logo and tagline 'AHEAD OF WHAT'S POSSIBLE™' at the top left. A search bar is located at the top center. The navigation menu includes 'MY ANALOG', 'MY HISTORY', 'PRODUCTS', 'APPLICATIONS', 'DESIGN CENTER', 'EDUCATION', and 'SUPPORT'. The 'DESIGN CENTER' tab is active, and the breadcrumb trail shows 'Design Center > Circuit Design Tools & Calculators > LTspice'. The main content area is titled 'LTspice' and includes a description of the software as a high-performance SPICE simulation tool. A 'Contact Technical Support' link is provided. Below this, a section titled 'Benefits of using LTspice' features a video player showing a simulation waveform. The 'Download LTspice' section lists two download options: 'Download for Windows 7, 8 and 10' (updated Mar 11 2020) and 'Download for Mac 10.9+' (updated Mar 11 2020). A link for 'Download for Windows XP (End of Support)' is also present.

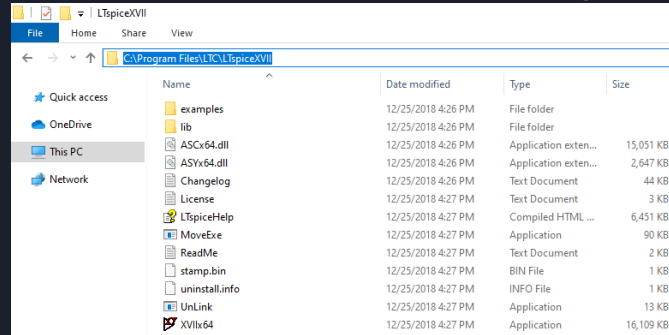
Install

- Double click on the downloaded installer.
- After reading through the license agreement, click “Accept”.
- Choose an appropriate directory and click “Install Now”.
- After installation is complete, the software should automatically open. If not, find the LTspice XVII executable and open it.



Main Directories

- The main directory is the directory you specify during the installation.



- LTspice also contains a “library” of custom components, often stored in a different directory. For Windows 10 users, this is often in your (OneDrive) Documents folder. We will learn about custom components in Tutorial 2.

