LTspice Tutorial Part 1- Download and Installation

Prerequisites

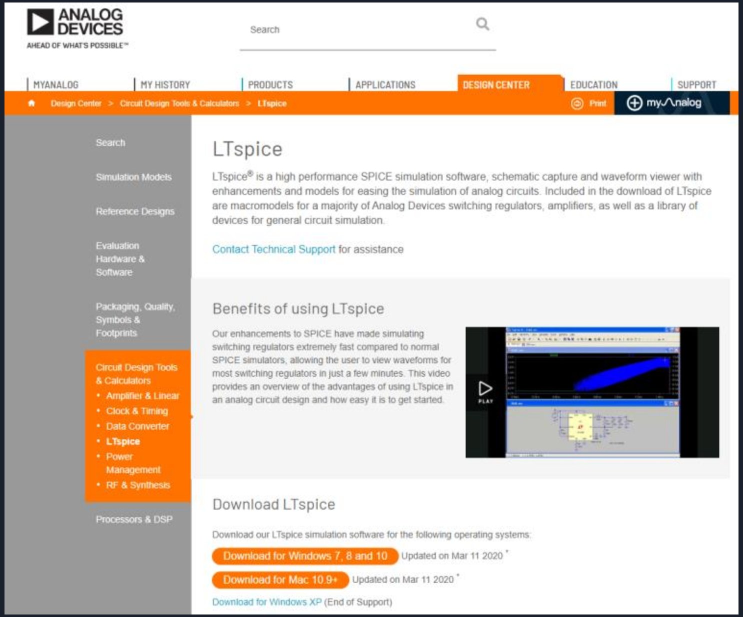
- A Windows or Mac OS based platform
    - All subsequent LTspice tutorials will be based on Windows.
    - For Mac users, please see https://www.youtube.com/watch?v=MsilYqaGPQw&feature=youtu.be and similar guides. Alternatively, the University's Windows based EWS machines can be accessed remotely via 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

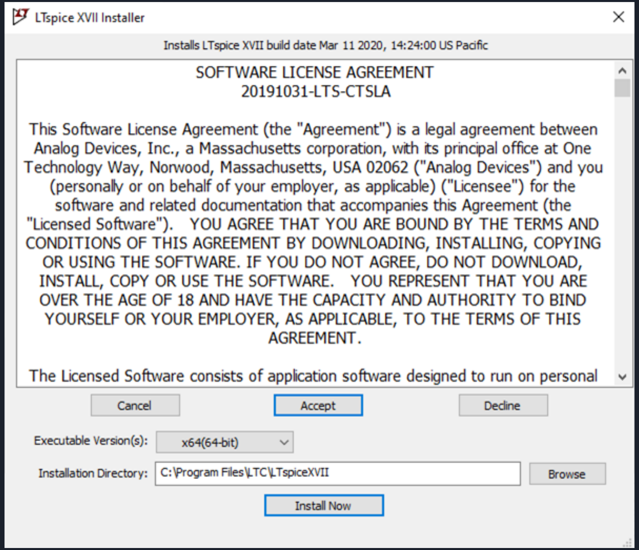
Download

- 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).



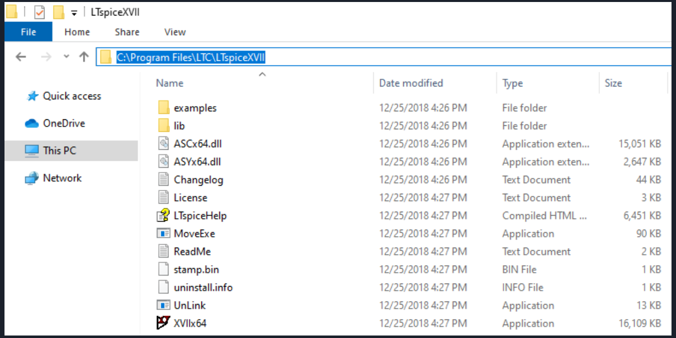
Install

- 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.