

## Instructions to Install LTSpice

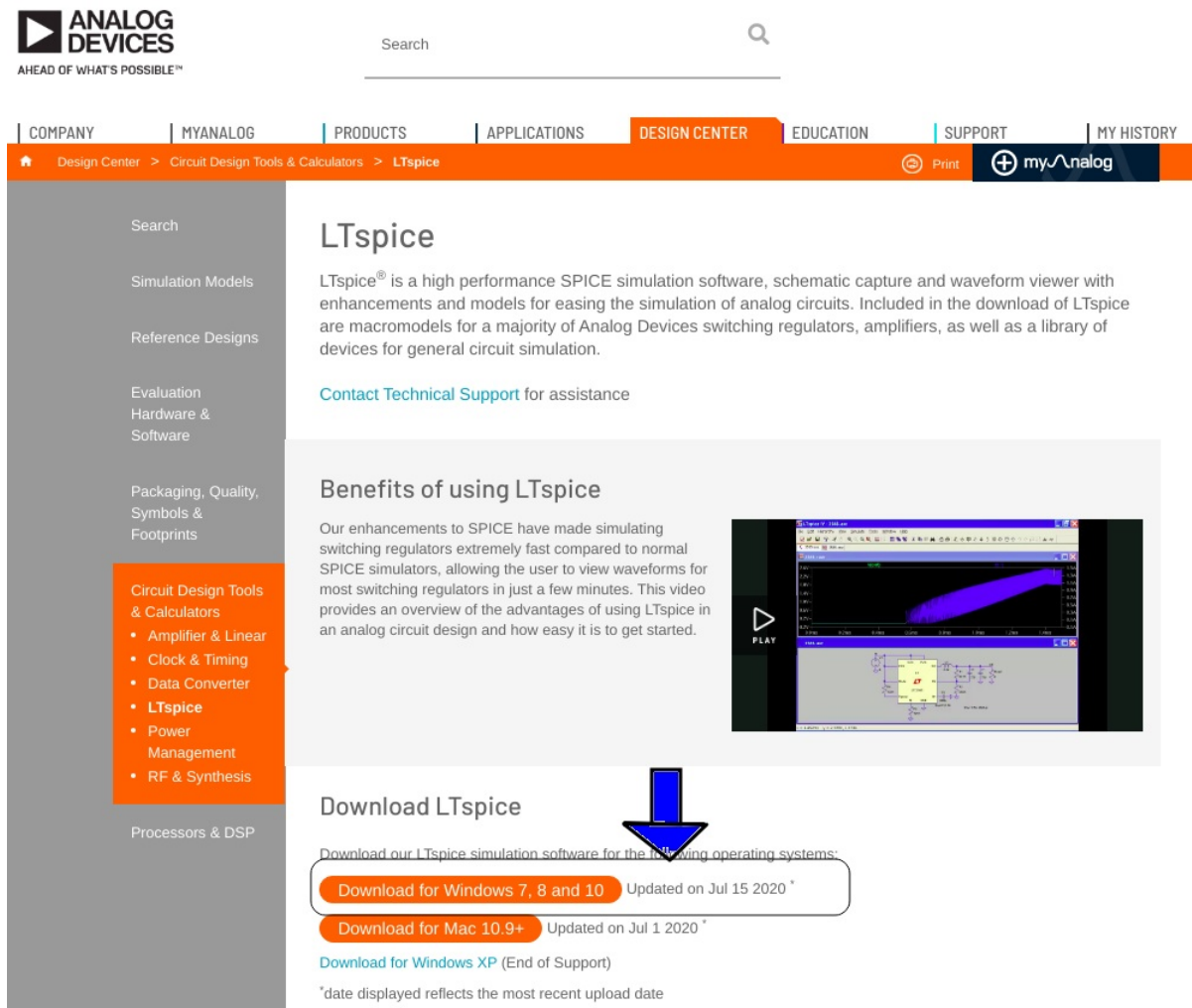
LTspice software was developed by Linear Technology, Inc. It has been bought by Analog Devices, Inc., who now maintains the software. It is freely available for downloading and for use.

- **Download LTspice :**

This software can be freely downloaded from Analog Devices website which is given below.

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

Or you can search in google with keyword 'ltspice', first link will direct you to same above link. Download the LTspice executable file under the 'Download LTspice' section. LTspice is supported for windows and Mac operating systems. Click on highlighted area "Download for Windows 7, 8 and 10" as shown in the following picture and save .exe file for installation in windows machine.



- **Install LTspice :**

Double click on LTspiceXVII.exe and installation window will pop up. Accept the license agreement, modify installation directory if needed then click on "Install Now" button. After successful installation, LTspice will start automatically within few seconds.

- **How to use LTspice :**

Analog Devices has many resources, including spice files, application notes, and videos on their web site, with a link to many of them at below link:

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

Another very good LTspice tutorial can be found at the following link which is maintained by Prof. R. Jacob Baker.

<http://cmosedu.com/cmos1/ltspice/ltspice.htm>