

# Setting up LTSpice XVII

date: 06/06/2022 author: Iz

## Downloading the software

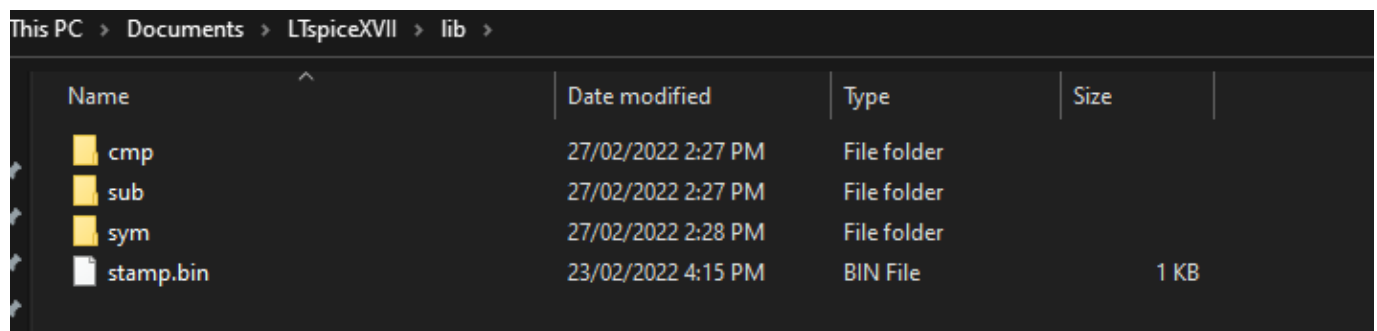
Visit the following link to download the software [LTspice Simulator | Analog Devices](#)

## Additional SPICE Models for certain IC's

Download the additional libraries from [this website](#)

[Additional library for LTspice, file is lib.zip ~18M](#)

Copy the contents of the downloaded file into the `lib` folder of your LTSpice in your documents. For example:



Name	Date modified	Type	Size
cmp	27/02/2022 2:27 PM	File folder	
sub	27/02/2022 2:27 PM	File folder	
sym	27/02/2022 2:28 PM	File folder	
stamp.bin	23/02/2022 4:15 PM	BIN File	1 KB

Replace the files in the directory. You should now be able to use 74HC IC models and other SPICE models.