## LT Spice Setup

## Last updated 3/6/19

## **LT Spice Setup**

- LT Spice is a circuit schematic capture and simulation tool from Linear Technologies / Analog Devices
  - Download the installation file <u>http://ltspice.analog.com/software/LTspiceXVII.exe</u>
  - Run and Accept the license agreement
  - Leave defaults and Install Now