About LTspice, Download and Install

- SPICE : Simulation Program with Integrated Circuit Emphasis
- LTspice is a high performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits
- For SPICE, any circuit is described as an interconnection of various active, passive elements. This interconnection of elements is also called **Net-List**
- DC, transient, AC, pole-zero, noise - analysis can be performed using LTspice
- Result plots can be viewed and saved
- Download LTspice from following path and install: *https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html*
- LTspice documentation can be also downloaded from the same site

Getting Started

- Circuit schematic
- Models used to describe circuit elements is included
- Type of analysis to be done on the circuit to be mentioned in SPICE directive
- Results can be plotted with the help of probes and expressions
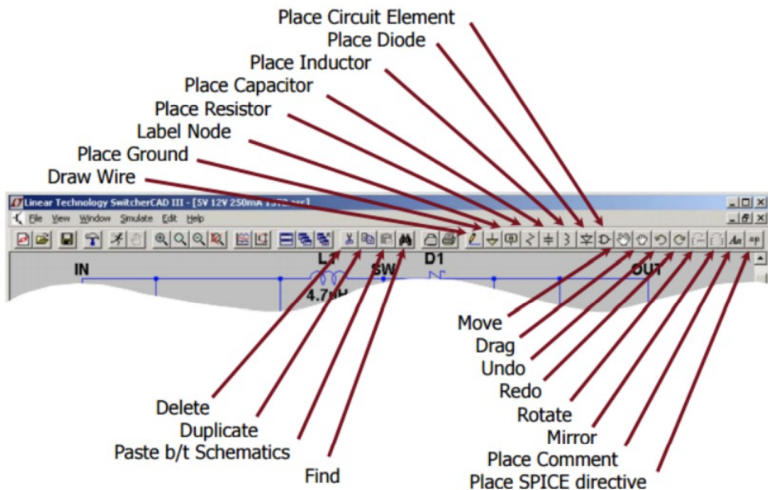
Start with a New Schematic

New Schematic

Summary of Schematic Editor Toolbar

Units

Use the following labels to specify units for circuit element attributes

- **K** = k = kilo = 10^3
- **MEG** = meg = 10^6
- **G** = g = giga = 10^9
- **T** = t = terra = 10^{12}

- **m** = M = milli = 10^{-3}
- **u** = U = micro = 10^{-6}
- **n** = N = nano = 10^{-9}
- **p** = P = pico = 10^{-12}
- **f** = F = femto = 10^{-15}

Simulation Setup

- Left click on simulation menu
- Left click on Edit Simulation Cmd
- Left click on requires simulation type tab
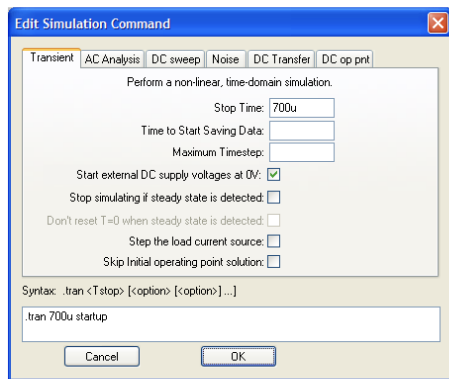- Enter necessary parameters
- Click OK

Figure: Edit simulation command window

Probing Circuit

- Left click on any wire to plot the voltage on the waveform viewer

Voltage probe cursor

- Left click on the body of the component to plot the current on the waveform viewer

Current probe cursor

- For voltage difference across 2 nodes, left click and hold on a node and drag the mouse to the other node
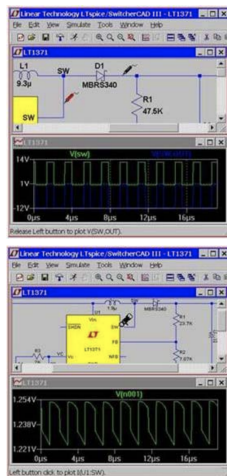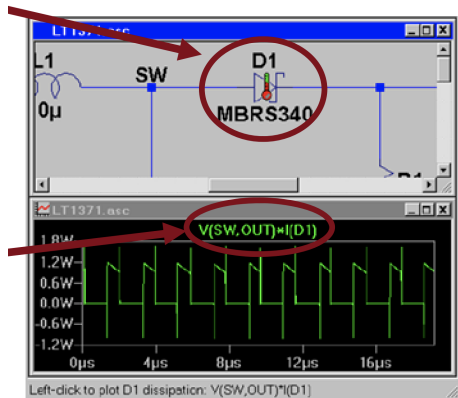

Figure: Edit simulation command window

Instantaneous and average power dissipation

- Instantaneous power dissipation
  - Hold down ALT key and left click on the component
  - Pointer ill change to a thermometer

- Average power dissipation
  - Hold down Ctrl key and left click on the trace label of the power dissipation waveform
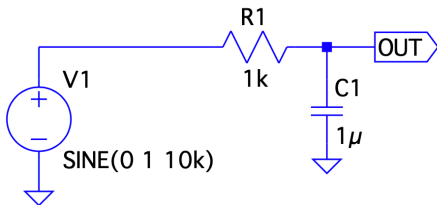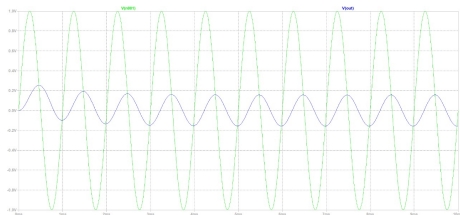
RC circuit

Figure: Schematic
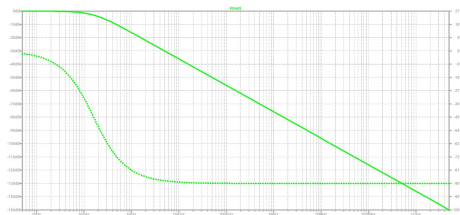

Figure: Transient Simulation Results
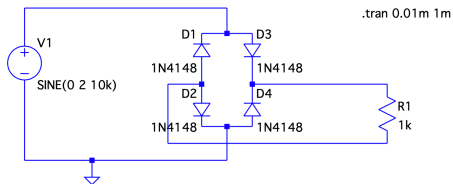

Figure: AC Simulation Results

Diode Rectifier

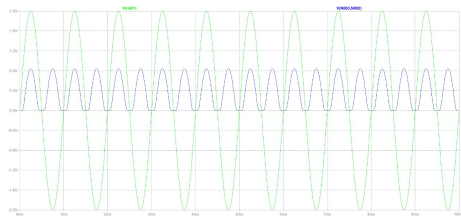Figure: Schematic

.tran 0.01m 1m

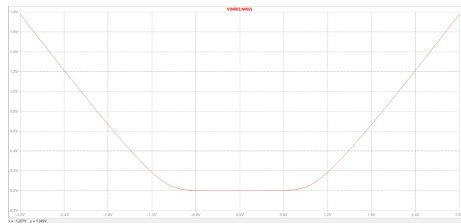

Figure: Transient analysis results

Figure: DC analysis results

OPamp inverting amplifier
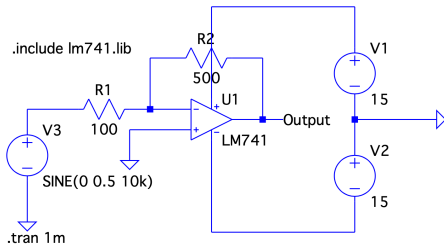
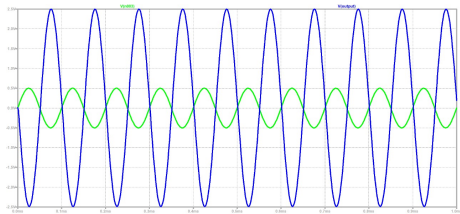Include library file lm741.lib when using this circuit.

Figure: Schematic

Figure: Transient analysis results

LTspice shortcuts on OS X

- G   GROUND
- S   ADD SPICE DIRECTIVE
- CMD+N   NEW SCHEMATIC
- CMD+O   OPEN
- CMD+S   SAVE
- CMD+Z   UNDO
- CMD+SHIFT+Z   REDO
- F2   COMPONENT
- F3   WIRE
- F4   NET NAME
- F5   DELETE
- F6   DUPLICATE
- F7   MOVE
- F8   DRAG
- SPACE BAR   ZOOM TO FIT