This LTspice tutorial will show the basic functions needed to solve problem 1.6 in Problem Set 1 on a Windows machine. A half-wave rectifier will be shown as an example of how to use LTspice, and the full version name of the one used is LTspice XVII.

1. After installing and opening the file, the default screen will appear.

2. Going to File>New Schematic or pressing Ctrl+N will open a new schematic to create a circuit.
3. To create an AC voltage source, first go to Edit > Component or press F2 to open the component window to choose a component to place. Clicking “voltage” will allow you to place multiple voltage sources, pressing escape will cancel placing additional ones. One source is needed is needed for the half-wave rectifier.

4. The voltage source will need to be edited to be an AC source. Right click on the source to open a window to edit the source, and click on “Advanced”.

![Image of voltage source](image.png)
5. The “Advanced” window that opens up will allow various types of sources to be chosen. On the left under the “Functions” column, choose the second option (“SINE”) to create an AC source. More options will appear under the “Functions” column that allow various properties to be chosen. Most options are typical AC source properties, with “Ncycles” being how many cycles the AC source will be on for. A large number will be needed to run the source for 1 second (60 for 60 Hz). The following options will be used for the example.

![Image of the Independent Voltage Source - V1 window with selected options and values.

- DC offset [V]: 0
- Amplitude [V]: 5
- Freq [Hz]: 60
- Tdelay [s]: 0
- Theta [1/s]: 0
- Phi [deg]: 0
- Ncycles: 60

Make this information visible on schematic: ✔️

DC Value:

Make this information visible on schematic: ✔️

Small signal AC analysis (AC):

AC Amplitude: [Blank]
AC Phase: [Blank]

Make this information visible on schematic: ✔️

Parasitic Properties:

Series Resistance [Ω]: [Blank]
Parallel Capacitance [F]: [Blank]

Make this information visible on schematic: ✔️
6. The source is now an AC source at 60Hz with a peak voltage of 5V that will run for 60 cycles (1 second). Notice how it lists “SINE(0 5 60 0 0 0 60)” on the voltage source to display its properties. To place a resistor and diode, open the components window and choose “res” and “diode” respectively (use Ctrl+R to rotate the diode).

7. To change the value of the resistor, right click on “R” to open a window that will allow you to edit the value. A value of 5 Ω will be used.
8. Setting the diode properties requires the usage of a SPICE Directive. It can be found in Edit>SPICE Directive or by pressing “S” on the keyboard. A custom diode with certain properties can be defined here, but only the forward voltage will be changed and the rest left as defaults for this example. The line “.model Diode D(Vfwd=1)” will create a diode type with the default properties in addition to a voltage drop of 1V when active. The model will be named “Diode” but can be named anything as long as the name is placed after “.model”.

9. After pressing “OK”, the directive will need to be placed as a line of text to be active. The diode value can be changed to “Diode” so it will have the properties defined in the directive.
10. Pressing “G” on the keyboard will allow a ground node to be placed. Pressing “F3” on the keyboard will allow the components to be wired by connecting the nodes. To make turns while using the wire tool, clicking will fix that length of wire in place and allow you to place wire in a different direction.
11. The simulation settings window can be found in Simulate> Edit Simulation Cmd. The first three time settings in the “Transient” tab are measured in seconds and allow for a small window to be simulated, such as after 10 seconds if steady-state is analyzed. Since this is a resistive circuit, no delay is necessary but the start time will be set after 0 seconds as an example. Note that the window of sampling should contain an integer number of cycles for measuring RMS values later. The “Maximum Timestep” is the amount of time between sampling data points, so a lower number gives a higher resolution output. The following settings will be used for this example. After pressing “OK”, the line of text will have to be placed on the schematic.
12. Pressing the “Run” button (the running person in the toolbar) will run the simulation. A new window will show once the simulation is done, and this window is where the waveforms will be shown later.
13. To view the waveform of a component, clicking on the wire between components will display the voltage waveform in the above window while clicking on the component itself displays the current through it. Note that multiple waveforms can be displayed, and they can be deleted by right clicking on the name of the trace to open the settings window. The voltage and current of R1 are displayed below.
Due to the purely resistive nature of the circuit, the waveforms overlap. Notice that the peak value of the resistor is 4V compared to the 5V source due to the diode setting of 1V. The time axis also starts at 0 seconds rather than 0.25 seconds. While it is not apparent, the simulation did sample starting at 0.25 seconds; and the figure below shows the steady-state result of a capacitor placed in parallel.
14. Using Ctrl+Left Click on the name of the waveform will open a window showing the average and RMS values.

15. To display the power trace of the voltage source, Alt+Left Click on the voltage source will display the result of its voltage and current output multiplied together. This can be treated as any other trace, so Ctrl+Left Click will display the average and RMS values.
16. Left clicking and dragging on the simulation window will allow you to zoom in on the selected area. Note that the x and y axis scale as well with the window.