

LTspice Tutorial Part 2- Components and Basic Interface



Prerequisites

- Please make sure you have completed the following:
 - LTspice tutorial part 1 (download and installation)



Note: This guide is intended for Windows platforms. If you are using Mac OS, please refer to Prerequisites in LTspice Tutorial Part 1- Download and Installation for a video link on running LTspice or on how to access Windows based EWS machines remotely.

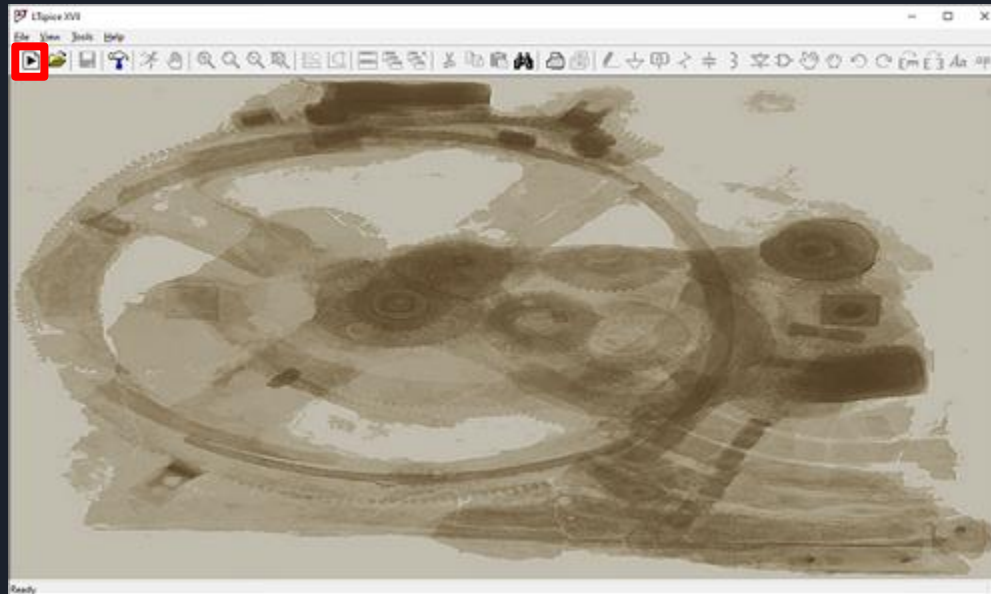


Tutorial 2 Objectives

1. Learn the basic layout of LTspice (version XVII)
2. Be able to create circuit schematics, adjust component values, and run basic simulations

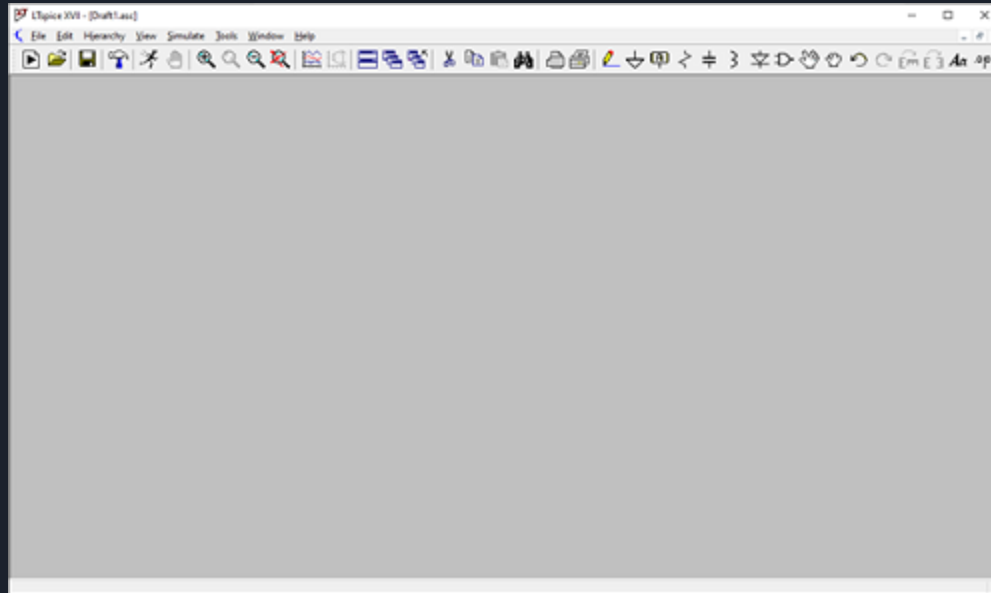
Main Interface

- Double click on the executable for LTspice XVII. The following window should appear. To create a new schematic to add components to, click on the  symbol. Alternatively, you may see  as the new schematic symbol.



Main Interface

- After creating a new schematic, some of the main icons will be colored. The schematic can be saved as a .asc file. Begin by creating an empty draft.



Main Interface

- Below are some useful icons on the interface. Once clicked, press **right click** or **Esc** to exit. Use the mouse wheel or the microscope/zoom icons to zoom in or out.

Icon

Description



Run simulation. Prior to running, simulation parameters must be specified through **Simulate > Edit Simulation Cmd.**



Create new schematic (.asc file).



Establish wires or nodal connections. Continuously left click to add straight lines and establish “wire” connections. Right click to end.



Add ground point.



Label node or point in circuit. Useful in place of wire connections in complex circuits, and for plotting.

Main Interface

- Below are some useful icons on the interface. Once clicked, press **right click** or **Esc** to exit. Use the mouse wheel or the microscope/zoom icons to zoom in or out.

Icon

Description



Add resistor.



Add capacitor.



Add inductor.



Add diode.



Add custom component, e.g. voltage source, transistors, ICs, etc.



Delete component. Left click on component to remove, or left click and drag to delete whole portions.

Components – Basic

- Adding components
 - Components such as resistors and capacitors can be added by clicking on the appropriate icon (for shortcuts, see the list after clicking on the **Edit** tab).
 - Once clicked, hover to the area you wish to place the component.
 - Left click to place the component.
- Component properties
 - **Right click** on the component to modify its properties.
 - “k”, “m”, “u”, “n”, etc. can be used as multiplication factors e.g. “10k” on resistor = 10 k Ω , “10u” on capacitor = 10 μ F

Suffix (case insensitive)	Power
T	10^{12}
G	10^9
Meg	10^6
k	10^3
m	10^{-3}
u	10^{-6}
n	10^{-9}
p	10^{-12}
f	10^{-15}

Resistor - R1

Manufacturer: -----

Part Number: -----

Select Resistor

OK

Cancel







Resistor Properties

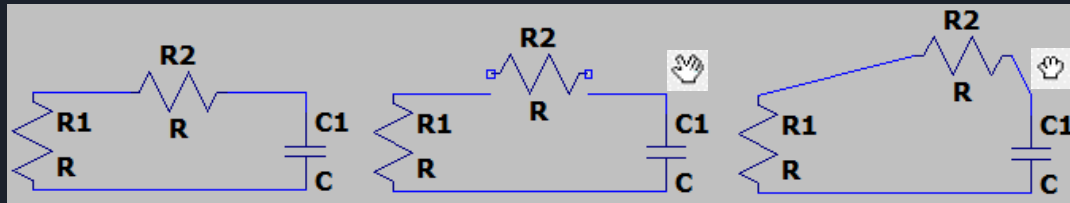
Resistance[Ω]: R

Tolerance[%]:


Power Rating[W]:

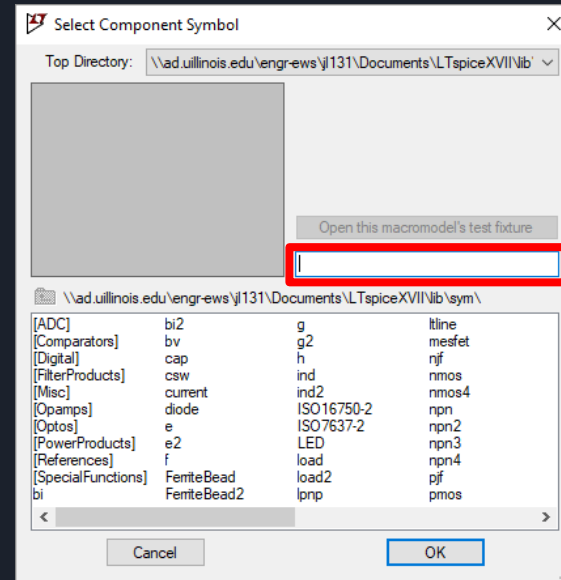
Components – Basic

- Moving components
 - To move a component, click on  then left click on a component to move it. Click and drag to select an entire area. Alternatively use  by clicking and dragging.  Can be used to move components out of nodes, and  drags the node or “wires” along with the components.
 - To rotate or mirror a component, ensure it is selected by using  (initially when placing a component it is automatically selected). Press **CTRL+R** to rotate the component if selected, **CTRL+E** to mirror the component.
- Deleting components
 - Press **Delete** or click on , and left click on component to delete it, or click and drag to delete an area.



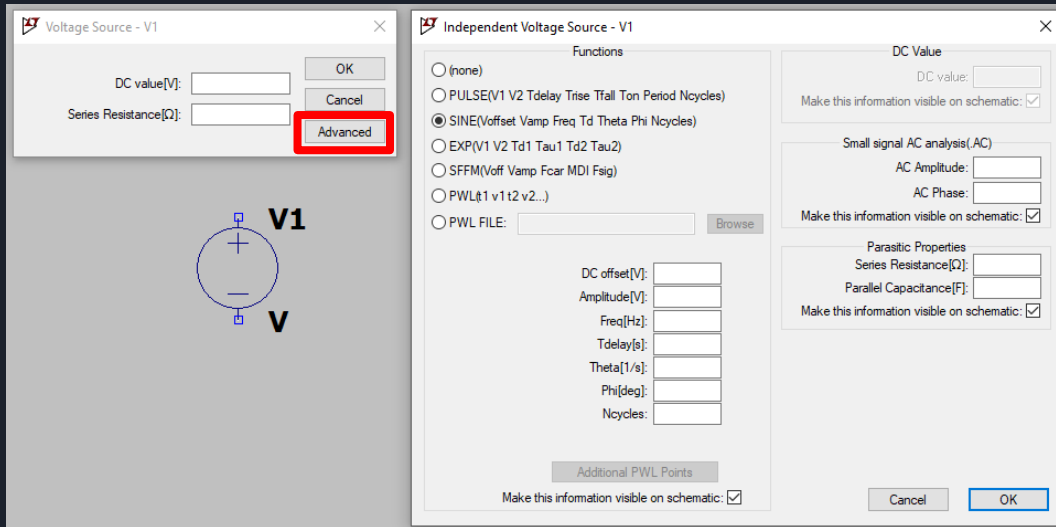
Components – Custom

- Can be selected using  symbol (shortcut: F2).
- LTspice comes with a library of common components
 - Different operational amplifiers, IC chips, etc. common in industry are modeled based on datasheet specifications. The models for the CD40106 and HEF40106B inverting schmitt triggers in ECE 110 do not come with the default library, however for this course we will employ a generic schmitt trigger component and modify its voltage thresholds.
- Current and voltage sources can be selected
 - On the **Select Component Symbol** interface, scroll to the right and click on **voltage** to select the component for the voltage source. Alternatively, enter “voltage” on the box indicated on the right.





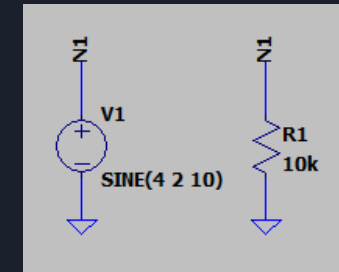
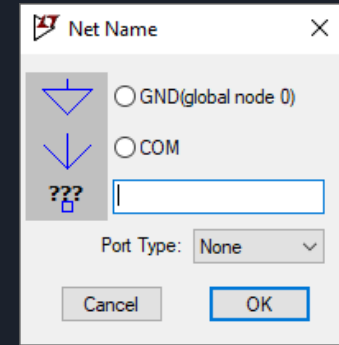
Components – Custom

- Custom components can be modified
 - **Right click** on, e.g., voltage source once placed on the schematic.
 - Click on **Advanced**. This allows the user to specify additional properties of the source, e.g. shape of waveform.
 - Example: SINE, DC offset[V]: 4, Amplitude[V]: 2, Freq[Hz]: 10
This creates a sinusoid beginning at $t = 0$ and ranging from 2 V to 6 V at a frequency of 10 Hz.



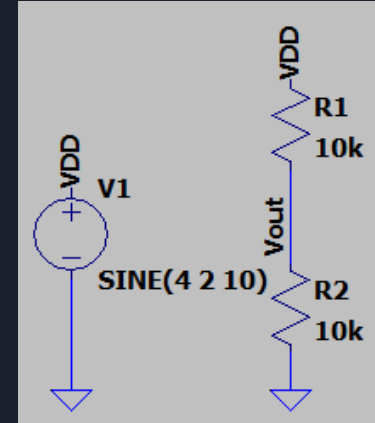
Net Labels

- Net labels are very useful when there are many nodal connections in your circuit. Can be selected using the  symbol.
- Once clicked, the **Net Name** window pops up. Any name can be chosen for a nodal point.
- Similar to the existing ground symbol , net labels establish nodal connections for all lines connected to the symbol with that specified name.
- Also useful when running simulation and creating plots; net labels appear in plots.
- Example: Two “N1” net labels are placed on the diagram to the right. Although no “wires” are drawn connecting the voltage source and resistor, they are connected via the “N1” net label.





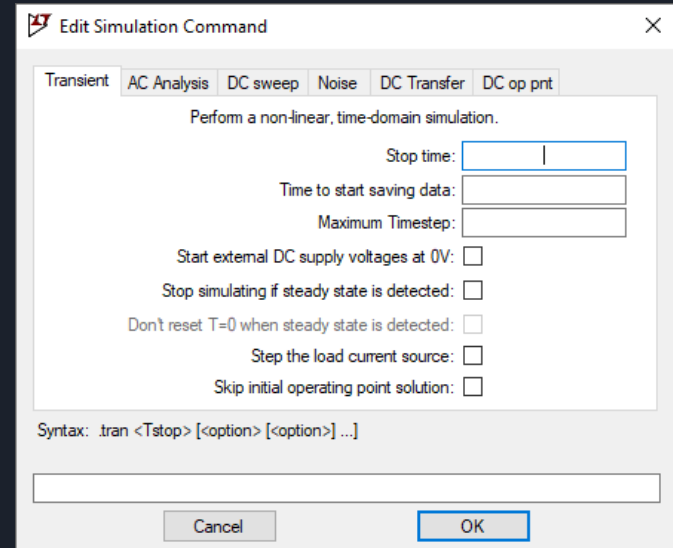
Creating Your First Circuit

- Exercise 1:
 1. Create a voltage source with the following parameters:
 - Type: Sinusoidal
 - Amplitude: 2 V
 - Offset: 4 V
 - Frequency: 10 Hz
 2. Create two resistors, later to be connected in series next to the voltage source, with the following resistance values:
 - R1 = 10 k Ω
 - R2 = 10 k Ω
 3. Connect R1 and R2 in series, and ground the other side of either R1 or R2. Ground the negative terminal of the voltage source.
 4. Add net labels "VDD" to the positive side of the voltage source and the remaining open-ended side of the resistive network.
 5. Add the net label "Vout" between R1 and R2.



Simulation

- **FIRST STEP:** Once we have a basic circuit, click on **Simulate > Edit Simulation Cmd.**
 - The options in the **Transient** tab should suffice for simulating circuit behavior for the ECE 110 course.
 - **AC Analysis**, **DC sweep**, etc. are more advanced features that you may look up on.
 - Under **Stop time**, set an appropriate time based on frequencies in your circuit. The stop time is by default in units of seconds.
 - Click Ok and place the “.tran ...” label anywhere on your schematic.
- **SECOND STEP:** Click on  to run the simulation.
 - Empty graph should appear.
 - Click on nodal point or wire to probe it.
- Once component values are updated, or extra components are added, the simulation needs to be run again by clicking on , provided that the first step is complete.






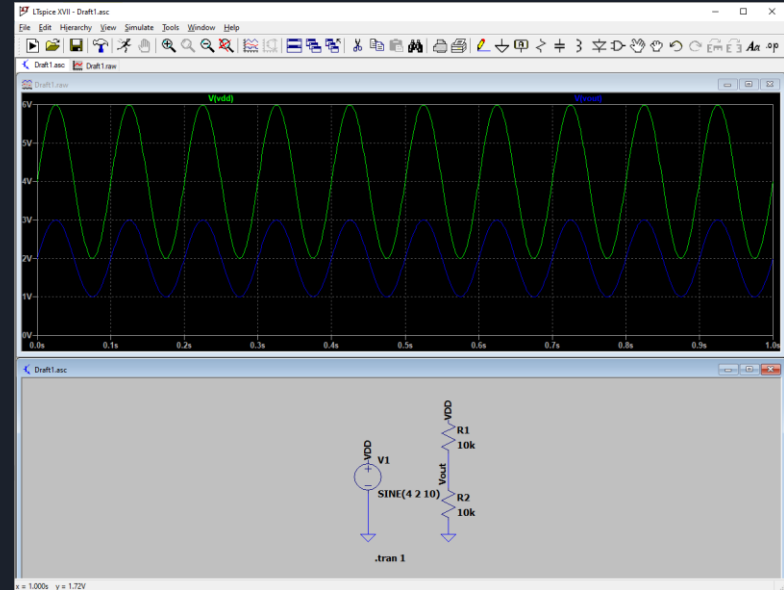
Simulation

- Graphing features/commands
 - Left click a node or wire to graph the voltage at that point.
 - Left click on a component to graph the current flowing through that component.
 - Hold **CTRL** and left click on another location to append the waveform on the graph.
 - **Double left click** on a point to only display that point on the graph.
 - **Right click** on axis values to adjust scaling.
 - **Right click** on the graph to adjust display settings.
 - **Right click** on the name of a waveform above the graph to edit the waveform, e.g. delete the waveform or change the color.

Simulating Your Circuit

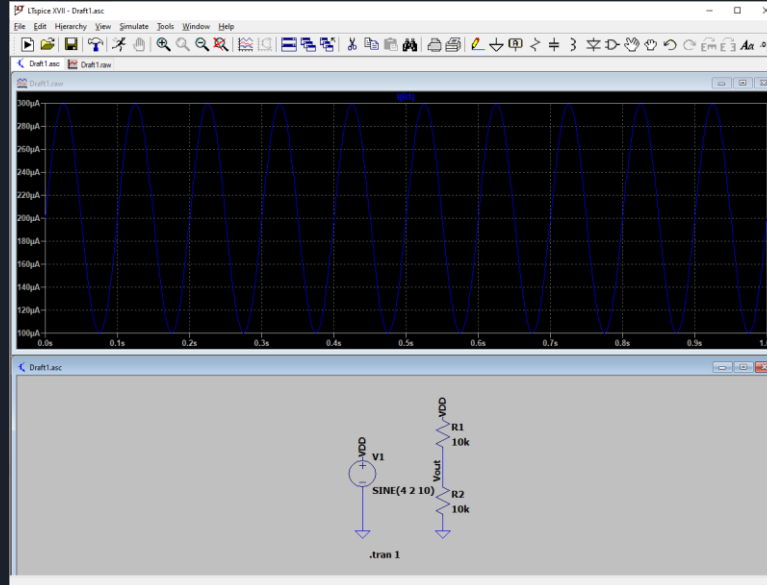
- Exercise 2:

1. Using the circuit from the previous exercise, go to Simulate > Edit Simulation Cmd and set the transient stop time to “1” (for 1 second). Click Ok and place the “.tran ...” label on the schematic.
2. Run the simulation by clicking on .
3. Left click on the node containing the “VDD” net label on your schematic.
A sinusoidal voltage waveform should appear on the graph.
4. Hold CTRL and left click on the node containing the “Vout” net label. Another sinusoidal waveform should be appended to the graph.
5. Right click on the vertical axis and set the following parameters to scale the axis:
Top: 6V
Tick: 1V
Bottom: 0V
6. Right click on the graph, click on View, and tick the Grid box.



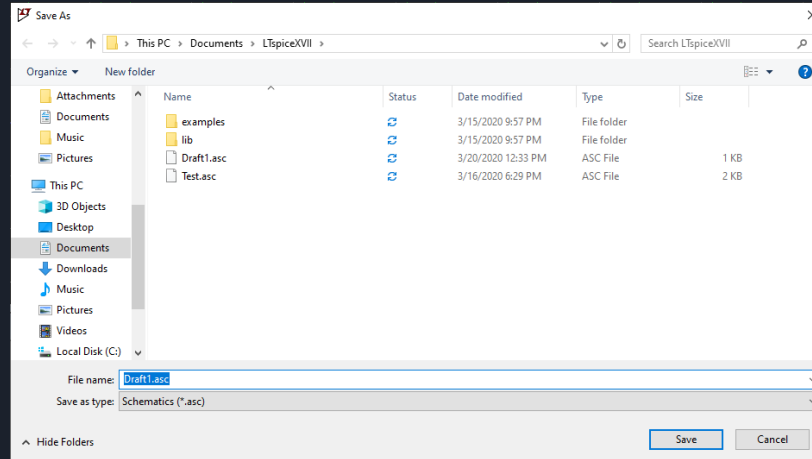
Simulating Your Circuit

- Exercise 2:
 7. Double click on resistor R1 or R2. A single sinusoidal current waveform should appear on the graph. Pay attention to the direction of the arrow denoting the current as you hover over a component.



Saving File

- Click on **File** > **Save** or **Save As** to begin saving your schematic (.asc file).
- Select an appropriate name and location for the file and click **Save**.



- To open the saved file, click on **File** > **Open** and locate the file.