We’ll use MultiSim for circuit simulation!

As an exercise, we’ll simulate voltage clamp in Lecture 01. Let’s assume the input is a 1 kHz sine wave (peak = 6V).
Start MultiSim...

To place components, you can either:

1) Click on the appropriate icon ...

2) ... or click on Place >> Component

3) ... or use CTRL+W
For this example, we want to use a 1N4148 “switching diode”.

1) Click on the “diode” icon
   • Group = Diode
   • Family = Switching Diode
   • Component = 1N4148

2) … or Select Place >> Component
   • Group = Diode
   • Family = Switching Diode
   • Component = 1N4148

NOTE: The “Select a Component” window stays open after you place a component. If you want to close the window, you must click “Close” or the “X” button (top right).

NOTE: To rotate a component, press CTRL+R.
To place a resistor you can either:

1) Click on “resistor” icon
   - Select “RESISTOR” in “Family” menu
   - Type in the desired value in “Component” box
   - Press “Enter”

2) … or Select Place >> Component
   - Group = Basic
   - Family = RESISTOR
   - Type in the desired value in “Component” box
   - Press “Enter”

NOTE: To rotate a component, press CTRL+R
To place grounds, you can either:

1) Click on “Sources” icon
   - Select “POWER_SOURCES” in “Family” menu
   - Select “GROUND” in “Component” box
   - Press “Enter”

2) … or Select Place >> Component
   - Select “Sources” in the “Group” menu
   - Select “POWER_SOURCES”
   - Select “GROUND”
   - Press “Enter”
To place a sine wave signal source you can either:

1) Click on “Sources” icon
   - Select “SIGNAL_VOLTAGE_SOURCES”
   - Select “AC_VOLTAGE”
   - Press “Enter”

2) ... or Select Place >> Component
   - Select “Sources”
   - Select “SIGNAL_VOLTAGE_SOURCES”
   - Select “AC_VOLTAGE”
   - Press “Enter”

After placing the source on the schematic, double-click on it and adjust the following:
   - Frequency = 1 kHz
   - Voltage (pk) = 6V
To place a DC voltage source you can either:

1) Click on “Sources” icon
   • Select “POWER_SOURCES”
   • Select “DC_POWER”
   • Press “Enter”

2) … or Select Place >> Component
   • Select “Sources”
   • Select “POWER_SOURCES”
   • Select “DC_POWER”
   • Press “Enter”

After placement, double-click on the source and set “Voltage (V)” to 5V
Wiring the components together is easy!

To make a wire, just click on the start and end points.
Use File >> Print to print the schematic (full page printout with nice border)

However, for prelab and lab reports you can save paper by doing the following:

1) CTRL+A (select all)
2) CTRL+C (copy)
3) Paste the figure into Word or PowerPoint or Paint ...
4) Re-size/rearrange multiple schematics to fit on a page.
We want the simulation to produce waveforms for the input and output voltage (above the diode).

One way to do this is to place a “voltage probe” on the desired location in the circuit.

This is easily done by clicking “Place >> Probe >> Voltage”.

Double-click on the voltage probe to change its name.
Now we need to configure the simulation! We’ll use a “transient” simulation to make plots that resemble an oscilloscope waveform:

1) Simulate >> Analyses and Simulation
2) Select “Transient” from left side menu
3) Set the “End Time (STOP)” = .002 s
4) Click on “Output” tab
5) You should see “V(vin)” and “V(vout)” appear in the right-side list.
6) Press “Run”
The “Grapher View” should pop up with a plot of the input and output voltages.

1. If you want, you can edit the plot titles by double-clicking on them.
2. To save printer toner, change the plot background from black to white by clicking on the icon just below the “Cursor” menu.
3. You can copy and paste the graph into Word, PowerPoint, etc.
You can measure the actual voltage by using the “Cursors”. Two ways to do this:

1. Cursors >> Show cursors
2. Or click the “Show cursors” icon.

Then you can drag one of the cursor lines to the desired location. The little pop-up window will show you the x and y value of each cursor.