

MultiSim Hints & Kinks

Note: This is a brief summary of how to use a small part of a powerful and flexible tool. Hopefully, it will explain most of what you need for the first couple of multisim labs. For further guidance on specific points, or to see what the program can do, you can download a pdf of the Multisim *Manual* and *Tutorial* found under Laboratory Software: Multisim Documentation on the Physics 321 home page. There is also very good online help within the program under the Help menu: Multisim Help. This has a search facility, table of contents, and detailed index.

0. Create a folder on the desktop with your name.

1. Start multisim. There is an icon at Desktop\Lab Software\623\multisim. “File” → “New” if you don’t start up with a blank schematic sheet.

2. “File” --> “Save As” : find the folder with your name, save the circuit with a reasonable name (such as "diffamp". Don't put any "." extension on it).

3. Click on any of the small “place component” icons (at left end of 2nd row under menus) too bring up the “Select a Component” window. “Database:” should be “Master Database”. Select a “Group” with the component you want. This is not always obvious. But ground points, power supplies, and adjustable DC voltage sources and AC signal generators are all under “Sources”. Resistors, Capacitors, and Inductors are under the “Basic” group. Simple voltmeters and ammeters are under “Indicators”. There are fancier instruments under the “Simulate” menu: Instruments. After placing one of these, you have to double-click on it to adjust the settings. There is even a Tektronix oscilloscope similar, but not identical, to the ones in the lab (but the generic oscilloscope and 4-channel oscilloscope are much easier to use). You can get rid of any unwanted components in your schematic area by selecting them and hitting the “Delete” key (not “Backspace”). Right click → “Horizontal flip” to get mirror image for one.

4. For resistors and capacitors, you don’t need to select the particular values you want to use – it’s very easy to change the values later for any component (right-click on the component and chose “properties”, then edit whatever you need to. Watch the units!). Save often! (This is Windows, remember?)

5. You can copy any object (or collection of objects) on your schematic and paste it as many times as you want. If you’ve altered properties first, you’ll get the same thing in the copies.

6. Right-click component --> "rotate" or "flip" to reorient.

7. Wiring is easy: just click on a component lead and then move to where you want to connect it and click there. It will usually auto-route pretty intelligently through one bend, but to make it go where you want you can click at any intermediate point to force it to turn there. You can terminate the wire on another wire just by clicking a point somewhere along the existing wire. So you should generally try to go from a component lead to a wire. If you have to start on a wire, you will need to use ctrl-j and click on the wire to put an almost invisible “junction dot” on it. You can then click on this to start the wire. The escape key will terminate wire routing if you don’t want to finish it. And selecting any segment of a wire and hitting the “Delete” key will delete just that segment. Briefly look through the tabs of “Global preferences” under the “Options” menu. This will give you some idea of what is going on (and the opportunity to change defaults if you wish).

8. “Nets” are a network of connected wires that go to different components; since all the wires are connected, a “net” is always at the same potential everywhere. You can give important nets (such as inputs and outputs) unique names and colors. (Nets are just numbered in order created and colored red by default.) To do this, double click the net, and in the properties box that comes up, select a unique color, type in a name (such as “Input-1”) and check the box that says “show net name”. This will do several good things for you. (When you start doing real simulations, net names show up on DC bias point lists and as labels on plots. The plots will also color code the lines the same as the net color by default. You will be able to plot the voltage on any net, or the current through any component, without having any meters or instruments in your schematic.)

