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.)

Analog simulation:

9. Attach an AC voltage generator to the input. Set voltage to an appropriate value for measuring the gain. Note that voltages and phases have to be set in two places: one for operating the circuit and one for running simulations .

10. Get the Q-point bias values by "Analysis" --> "DC Operating Point". In the window that comes up, select all the net names that you want to appear in the output. You can print out the resulting table for your notebook.

11. Use “single” sweep on scope to get full trace when it stops, “Reverse” to get white background. But the screen outputs of the instruments also appear on pages in the Grapher window, and you have a lot more flexibility here for fixing the display before you copy them, so this is usually recommended.

12. "Analysis" —> "AC Frequency" to make Bode Plot (gain and phase vs frequency). Use log-log scales. This takes over the frequency settings of any AC voltage sources you have in the circuit. Set the initial frequency range to cover 1 Hz to 100 MHz. You can adjust the later to cover the interesting features. Set “Sweep Type to “decade”. Ten or twenty points per decade is plenty.

13. The Grapher window will come up by itself, but if you’ve closed it, "View" --> "Grapher" (or click tiny graph icon near center) to see results. DC bias point is still on first tab (old results are kept even if you close the window – you can press “delete” key twice to delete them). **Try the icons from the little grid through the third magnifying glass to see what they do. You will want to leave the grids on – need to put on both graphs. You also need to switch background to white if it is black.** You can resize graph by edges or corner, and move the legend around and reshape it. Right click on graph →“Properties” lets you change scales, colors, etc. On “file” → “print preview” (from top bar of the grapher window) select pages you want to print. Don’t forget to Save! You can arrange amplitude and phase plots one above the other, amplitude is normally on top. If you wish, you can set it up to plot Vout/Vin directly, so you don’t need to calculate the gain.

14. Look at the output and see if you can explain the low- and high-frequency behavior (remember that the major high-frequency factor is the collector-base miller capacitance in the transistor). The generic transistor is "ideal" and won't show any high frequency effects — be sure you have the transistor properties set to 2N3904.

Digital simulation (you don’t need the for lab 13, but it might be useful for your digital project)

Right click on schematic sheet → “Place component”. Use TTL →74LS series flip flops and gates. For the DAC use “Mixed” → “ADC_DAC → VDAC. A good comparator is “Analog” → “Comparator” → LM311N. It needs all the connections, including power supplies – look at the data sheets or your notebook if necessary.

The "Word generator" can generate an arbitrary timing diagram on up to sixteen lines. Each 4-hex-digit "word" in the list drives the sixteen output lines (four bits per digit), and the generator advances through the list of words at the clock rate that is set in the generator. If you use just the lsb of however many of the hex digits you need, it will be easier to “read” your table (or you can set the format to binary).

The logic analyzer clock should be set about 64x word generator clock rate to give good resolution, then set "clocks per division" to about 64 for a reasonable display. (Double-click logic analyzer to open its display, then click on the "set" box under "clock" to set rate.) If you “color code” the wires going into the logic analyzer, the corresponding traces on the analysis plot will come out in the same color. This can be handy. Use a four-channel oscilloscope to look at the comparator inputs and output. **Push ‘reverse’ button to set white background before copying.**

Cycling the power (0-1) rocker switch in the upper right corner resets all the plots to start at the left edge