**4400:360-001 PHYSICAL ELECTRONICS**

**Design Project 1**

**Purpose:**

To become familiar with some of the types of analysis available in SPICE, a circuit simulator, while investigating the design of an inverting amplifier.

**Background:**

SPICE is a circuit simulator that was originally written as a completely text-based system. There are two ways to use it. One is to interact with SPICE through a graphical user interface (GUI).The other is to use the text-based system directly. You may do either for this course, although these instructions are geared more towards the GUI version.

**Task1:** Design an inverting amplifier with a DC input resistance of 10Kohms and a gain of 20V/V. Build the amplifier in SPICE, using the “uA741” operational amplifier as the base. (The “uA741” operational amplifier appears in the parts list under the evaluation library. This op-amp has two extra terminals marked OS1 and OS2; leave these disconnected. Resistors are found in the analog library. Voltage sources are found in the source library.)Use DC power supply voltages of +15V and -15V.

**Task2:** Hook a small DC voltage source (using the VDC component) up to the input of the amplifier and run a transfer function analysis. This will put information about the DC transfer function and the input and output resistances in a SPICE output file. Verify that SPICE computes the correct values for DC gain, RIN, and ROUT.

**Task3:** Replace the DC voltage source with a sinusoid of small amplitude and slow frequency (using the VSTIM component; when you double click on this component, the stimulus Editor Window will pop up, so that you can specify the sinusoid’s amplitude and frequency). Run a transient analysis. Use the probe tool to plot the input and output voltage waveforms, and show that they are what you expect. Print out the waveforms for inclusion in the report; show at least a few cycles of the sinusoids.

**Task4:** Increase the amplitude of the sinusoid used in Task 3 enough so that the output will saturate. Again, run a transient analysis and plot the input and output voltage waveforms. Print the waveforms out for the report. Note the clipping on the output and from the plot determines what L+ and L-(the limits on the output voltage) are.

**Task5:** Replace the sinusoidal voltage source with an AC voltage source of small magnitude (using the VAC component).Run an AC sweep analysis, and use the results to make a Bode plot. Remember to plot both magnitude and phase, and plot enough of a frequency range to see all the pertinent parts of the plot. Print out the full Bode plot for your report. Does your Bode plot differ from what you expect when you analyze your circuit by hand, assuming an ideal op-amp? If so, explain the discrepancies. From the Bode plot, find the corner frequency and unity gain bandwidth for your inverting amplifier circuit. Use the results to compute the unity gain bandwidth for the uA741 operational amplifier alone.

**Basic Steps for using SPICE through the GUI:**

* **Enter the circuit as a schematic in the MicroSim Design Center.**

To do this, start up the MicroSim Design Manager, and select File … New Workspace. Type in a name for the workspace. Start up the Schematic Editor by selecting Tools…Schematics. Get a listing of the available components for use in circuits (resistors, capacitors, voltage sources, etc.) by selecting Draw…Get New Parts. Highlight the component you want, select place, and place it on the drawing. After you have placed a number of components, you can use the pencil with the thin line on the tool bar to draw wires interconnecting the components. Each component is automatically labeled with a name and one or more attribute values (for example, resistors have an attribute for their resistance, set by default to 1K.) You can change the values by clicking on them and typing a new value in to the dialog box that pops up. Use the BUBBLE part for ports (inputs and outputs of your circuit), giving each port a name. You can also label wires with names to make your circuit easier to trace; just double –click on the wire, and then type the name in.

When the schematic is done, save it by selecting File…Save. Then run Analysis…

Electrical Rule Check to check for obvious errors. Run Analysis…Create Netlist.

This creates a text file version of your schematic, in SPICE format. You can see this text

file version by running Analysis…Examine Netlist, if you wish.

* **Tell SPICE what type of analysis you want to do.**

In the Schematic Editor, select Analysis….Setup. Click the “enable” box next to the type

of analysis you would like to do. For most types of analysis, you must also click on the

box that says the type of analysis as well; this brings up a dialog box where you can set

the parameters for the analysis. For example, AC sweep is used to calculate the small-

signal response of a circuit as a function of frequency. In the dialog box, you specify

what specific frequencies to include in the analysis, by specifying a start frequency, an

end frequency, whether frequencies in between should be spaced out linearly or

logarithmically by octave or decade, and how many frequencies to include. This type of

analysis can be used to supply data points for a Bode plot. As another example, Transfer

Function is used to calculate the transfer function of a system; in the set-up dialog box,

you specify which transfer function you want by specifying which source you are using to

drive the input, and which signal you consider to be the output of the system.

* **Run the SPICE simulator to perform the analysis.**

In the Schematic Editor, Select Analysis…Simulate. If any errors occur, error messages will pop up in a dialog box. At the end of the simulation, a probe window will pop up.

* **Look at the results of the analysis.**

In the Probe window, select File…open and then click on your circuit. The Probe tool is used to plot the results of the simulation. Probe will try to set things up for you based on the type of analysis you have just run. To add a curve to the plot, select Trace….Add. A dialog box will pop up showing all the quantities you might plot (currents and voltages for all the nodes in your circuit). You can plot them individually by clicking on one. You can also combine them using math formulas, which is useful for plotting things like gain in decibels (20\*log10 (abs (V (out)/V (in))).You can also get the magnitude of an expression by using the M () function, and the phase angle by using P ().

Some results that aren’t typically graphed, like input and output resistance and power consumption, are in a output file. You can see these by selecting Analysis…Examine Output from within the Schematic Editor.

**Basic Steps for using text-based SPICE:**

* **Enter the circuit directly as a test file.**

Follow the examples in the Tuinenga text (or get a SPICE book from the library). You may use the text editor of your choice. Be sure that the first line of the file is either a comment (starting with \* in the first column) or blank.

* **Tell SPICE what type of analysis you want to do.**

This is done directly in the SPICE text file. For example, the ‘.AC’ statement corresponds to an AC sweep, and is used to calculate the small-signal response of a circuit as a function of frequency. The format of the statement is:

.Ac {LIN|DEC|OCT} points start frequency end frequency

where the parameters specify whether the frequencies to be analyzed should be spaced out linearly with points points total, logarithmically with points points per decade, or logarithmically with points points per octave. As another example, the .TF statement is used to calculate the transfer function of a system; the format is:

.TF output\_variable input\_source

where input\_source specifies which source you are using to drive the input, and output\_variable specifies which signal you consider to be the output of the system.

* **Run the SPICE simulator to perform the analysis.**

Start up the Microsim Design Manager and select File…New Workspace. Type in a name for the workspace. Select Tools…PSpiceA/D. The PSpiceA/D simulation window will pop up. In that window, select File…Open, and then browse until you find your circuit (it will have a .cir suffix).Select your circuit. The SPICE simulator will run, letting you know if there were errors.

* **Look at the results of the analysis**.

Go back to the Microsim Design Manager, and select Tools…Probe. The Probe window will pop up. In that window, select File…open, and then browse until you find the simulation data for your circuit (it will have a .dat suffix). Select it. Use the probe tool to plot the data of interest (see the Probe description under the GUI instructions).Some results are also in a SPICE output file; this file has a .out suffix.

**Report Requirements**

The design project requires a written report with the sections outlined below.

**Title page:**

This page should have your name, the course name with semester and year of enrollment, and the title for the project at the top. The bottom should have the following clause with your signature underneath:

This report represents the work of the author and is solely the work of the author. Submitting the work of others and representing that work as your own constitutes plagiarism and is punishable by failure of the course for the slightest infraction.

**Purpose:**

This section should be a paragraph describing the purpose of the project in your own words. Don’t simply list the steps that you took.

**Discussion:**

The section should summarize the work done for the project, task by task, including any pertinent circuit diagrams, hand calculations, SPICE outputs, etc. Be sure to answer all questions asked in the handout. Also discuss any difficulties you had in doing the project, and the steps you took in overcoming those difficulties.

**Summary:**

This section should be one or two paragraphs summing up the work done. Do not simply restate the steps that you took; this should be a “big picture” view of what the project was about, and what the goal of the project was, and to what degree the project achieved that goal.

**Appendix (optional):**

This section should contain any additional information that you think will contribute to a better understanding of the report and the work you did.