## Homework 2: First SPICE Assignment

SPICE (Simulation Program with Integrated Circuit Emphasis) is a large circuit analysis program originally developed at UC Berkeley about 40 years ago. It has become the *de facto* standard for such programs and the continued development of device models and computation algorithms is still coordinated through Berkeley. It is sold or in some cases given away by at least a dozen companies. In this class I am experimenting with using a free version of SPICE called LTSpice XVII that is given away by the Linear Technology Division of Analog Devices Corporation on their web site: https://www.analog.com/en/design-center/design-tools-and-calculators.html .

We have installed it on the computers in the lab and you can download it for yourself. I am told that the Mac version may be unstable at times.

This assignment is a simple introduction to the use of SPICE based on a problem with the voltage doubler rectifier circuit shown below. It will give you a chance to think about diodes! Because the circuit is intrinsically non-linear, we make use of SPICE's ability to do time domain rather than frequency domain analysis. Razavi has an analysis of voltage doublers on pp. 105 - 110. I have modified the basic circuit to give the source a non-zero resistance and to add a resistive load. This kind of circuit is often used with a square-wave source at much higher frequency with smaller capacitors as an on-chip higher volt-age source, for example, for programming flash RAM. It is also the basis of the Cockroft - Walton multiplier that was used in the first "atom smasher." That work won Mssrs. Cockroft and Walton the Nobel Prize in Physics in 1951. (If you have a taste for history, try the web - it comes with circuit diagrams.)
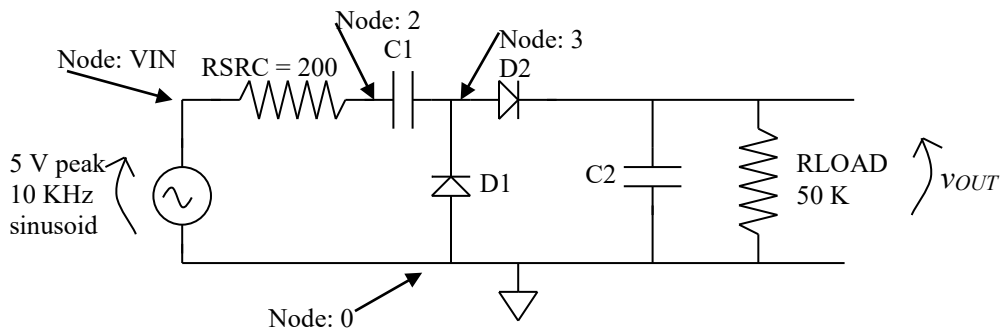


This circuit is simple to analyze by hand if there is no load but is difficult to do with any accuracy with a load. SPICE makes quick work of it, allowing you to do component op-timization under load. Here I ask you to use SPICE for the full analysis and to compare that analysis to what you can do by hand.

To use any version of SPICe, you have to build a text file with information on the netlist, the device models, and the desired calculations. The SPICE engine itself is a text input

batch processing program. To do this, you need some knowledge of SPICE syntax and naming conventions. Appendix A in your textbook has most of that information.  I have put the SPICE appendix from an earlier textbook on the class website under Course Materials/Sedra and Smith on SPICE Syntax.  Finally there are innumerable sites on the web with such introductions, including http://web.stanford.edu/class/ee133/handouts/general/spice_ref.pdf, and http://www.ecircuitcenter.com/SPICEsummary.htm . I recommend staying away from descriptions of PSPICE because that proprietary version of SPICE has many non-standard features that may not be supported in LTSpice XVII or many other versions of SPICE.

There are two ways to create the required text file. First, you can draw the circuit by hand, label its nodes, and then use a text editor like Notepad or emacs to write the file.  I recommend this method for this problem because it helps you get the ideas behind the process. This circuit is small enough that this is not much work. Should you ever get to doing serious design at the transistor level, a knowledge of the structure of the SPICE deck will be valuable.

Second, LTSpice has a simple schematic entry tool built-in.  Instructions on how to use it are included under the Help menu and it really is easy to learn and use. I use this capture tool only for circuits I will not actually build either as an integrated circuit or as a printed circuit board.

Most serious integrated circuit design packages include tight coupling to some more powerful version of SPICE. I am thinking about how to have you work with such a tool on a final SPICE assignment near the end of the course.

To encourage you to type in your first file, here is a file that has much of the circuit entered in the correct form.  An electronic copy is available on the class web site under "Course Materials/First SPICE File".

```
* Voltage Doubler with Load - this is always a title line
*
* Input voltage source 5 volts peak sinusoid at 10 KHz with source
* resistance of 200 ohms
*
VIN vin 0 sin(0 5 10K)
RSRC vin 2 200
*
* The doubler circuit itself
*
C1  2  3  0.1u
D1  0  3  D1N4448
D2
C2
RLOAD vOUT 0
```

```
*
* Diode model courtesy of the Fairchild Semiconductor website
*
.model D1N4148  D(Is=5.84n N=1.94 Rs=.7017 Ikf=44.17m Xti=3 Eg=1.11 Cjo=.95p

+            M=.55 Vj=.75 Bv=100 Ibv=100u Tt=11.07n)

*
* Control commands
*
* Time domain analysis reported at 2 microsecond intervals to 5 milliseconds
* of simulation time.
*
.TRAN  2u
*
* Display some data
*
.PRINT V(vin) V(2) V(3) V(vOUT) I(VIN)
*
*Save all node voltages and branch currents just in case
*
.PROBE
.END
```

I have removed some of the connections and command arguments from this deck and you
need to restore them to make it work.  I strongly suggest running this by yourself rather
than getting someone else to make it work for you.  SPICE is insensitive to case.  Its syn-
tax is fairly simple for most things.

What you need to do:

1.) Run LTSpice to calculate the circuit operation and print a plot with both the output
voltage and the source voltage as a function of time for the first 3 milliseconds of opera-
tion.

2.) How many cycles of the input voltage does it take to reach equilibrium?  Define equi-
librium as when the peak output voltage minus the final value is less than 1 % of the final
value.   What is the equilibrium DC output voltage? The ripple voltage peak-to-peak?
The ripple frequency?

3.) Similarly plot the currents in D2 and $R_{LOAD}$ on the same time scale as part (1). In
equilibrium, what is the output DC current and the peak current through diode D2?  Is
their ratio in agreement with the numbers I quoted in class for rectifiers with filter caps?

4.) Do an analysis by hand by assuming there is no load, that is, that RLOAD is infinite.
Assume that the diodes can be modeled with a large-signal model having a forward volt-

age $V_F = 0.75$ volts. The series resistance would be on the order of 10 ohms but that is not really very important. Find the equilibrium DC output voltage.

5.) Assuming that the peak output voltage is the equilibrium value from part 4, find the output ripple when the 50 K load is reconnected. What is the DC output after compensating just for the ripple voltage? How do the DC output and ripple voltages compare to what SPICE calculates?

6.) This circuit is a variant on a half-wave rectifier and such circuits normally have a high ratio of the peak source current to IDC. Assume that ratio is 10:1 and estimate a correction to your hand calculated value of $v_{OUT}$. Is this an improvement? The simplest approach is to assume further that the peak source current and peak source voltage occur at the same time, effectively reducing the peak source voltage.

7.) SPICE makes it easy to examine more complicated problems. Do one of the following additional SPICE runs:

- Replace the sinusoidal input with a square wave of the same frequency. (Hint: make VIN a PWL source and set it to give a piecewise linear square wave with say 1 microsecond rise and fall times and 10 KHz repetition rate.) This type of source is often used on-chip to generate voltages for such purposes as programming floating gate transistors. What change is there in output voltage and current from the results of part (1)?

- Add an additional two diodes and two capacitors to make a voltage tripler. Reconnect the load resistor to the new output terminal and change the value to 75 K before rerunning SPICE.

Please turn in the SPICE plots and the answers to the questions. Make the answers to questions a coherent presentation so I can read it easily. The questions are aimed at seeing how well you can get the "right" answers, that is, the answers SPICE gives, when you do the analysis by hand. Arrange them so the comparison comes through clearly. The agreement can be pretty good.