SPICE Exercise Number 1

SPICE Introduction

Install PSpice AD Lite Edition on your personal computer. Start PSpice, and prepare a SPICE input file using the text editor (File -> New -> New Text File). Another method to enter the circuit is by drawing the circuit schematic. Type the following and save it as **example.cir**

```
Resistor bridge
VIN 1 0 10
R1 1 2 2
R2 1 3 1
R3 2 0 1
R4 3 0 2
R5 3 2 2
.end
```

This circuit description file corresponds to the circuit shown below:
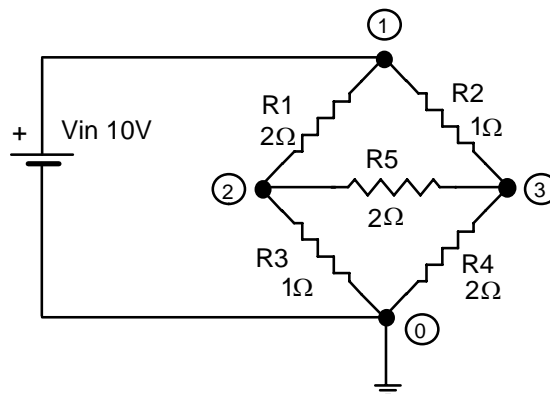
All SPICE input files should start with a line containing the title (or name) of the circuit; in this case it is `Resistor bridge`. The SPICE file should also end with a `.end` (dot-end) line. The lines between the first and last lines describe the elements in the circuit and their interconnections.

Note how the voltage source and resistors have been defined:

```
element_name from_node to_node element_value
```

For example the line:

```
R2 1 3 1
```

states that the element R2 is connected from node number 1 to node number 3 and has a resistance of 1 Ohm.

Note also that there should always be a node numbered 0, and this is used as the ground reference.

Prepare the SPICE simulation for this circuit (File -> Open Simulation; choose Circuit Files (*.cir); select example.cir). Run the simulation using (Simulation -> Run example). After the simulation is finished, examine the output file (View -> Output File) and look for the following lines:

```
****    SMALL SIGNAL BIAS SOLUTION     TEMPERATURE =  27.000 DEG C


*********************************************************************************


 NODE  VOLTAGE    NODE  VOLTAGE    NODE  VOLTAGE    NODE  VOLTAGE

(  1)  10.0000  (  2)   4.0000  (  3)   6.0000



    VOLTAGE SOURCE CURRENTS
    NAME       CURRENT

    VIN     -7.000E+00

    TOTAL POWER DISSIPATION  7.00E+01  WATTS
```

SPICE has calculated the DC voltages at nodes 1, 2, and 3 (10 V, 4 V, and 6 V, respectively). It has also calculated the current of the voltage source (-7 A, negative because the current is flowing *out* of the source), and its total power dissipation (70 W). This mode of operation, where the DC node voltages and DC source currents and powers are calculated, is the default operation mode in SPICE.

Use SPICE to analyze the circuit shown below.
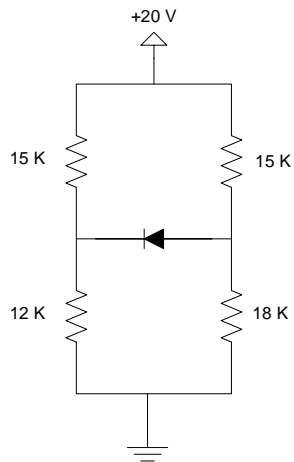
The circuit description file should be as follows:

```
Diode circuit
VIN 1 0 20
R1 ...
R2 ...
R3 ...
R4 ...
D1 2 3 MyDiode
.MODEL MyDiode D (IS=1e-12 , n=1)
.OP
.end
```

The diode is described in the SPICE input file using the D element (D1 is the diode in the circuit), together with a model that lists the diode parameters. The model (MyDiode in this case) is described using the .MODEL statement, and lists the values of I_S and n for the diode. Other parameters may also be listed, but they are not of interest to us at this stage.
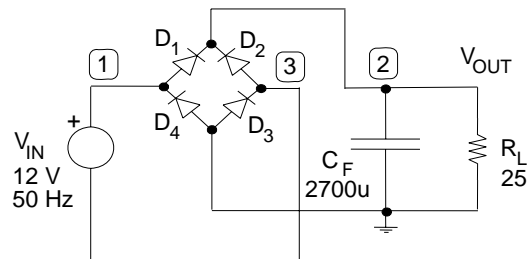
The .OP line instructs SPICE to show the operating point (currents, voltages, etc.) for the devices in the circuit.

Complete the missing entries for the four resistors in the circuit and save the file as **diode.cir**. Run the simulation and extract the values of diode voltage and current from the output file.

SPICE Exercise Number 2
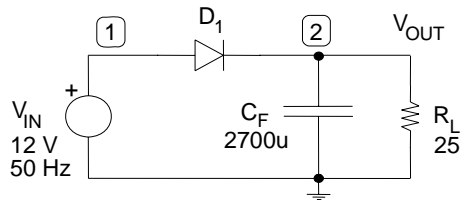
Circuit 1
Single Diode

Circuit 2
Diode Bridge

The SPICE circuit description file for Circuit 1 (a half-wave rectifier with capacitor filter) is the following

```
Single Diode
VIN 1 0 SIN(0 12V 50Hz 0 0 0)
D1 1 2 DM
CF 2 0 2700uF
RL 2 0 25
.MODEL DM D (IS=0.01nA)
.TRAN 1ms 80ms 20ms
.OPTIONS reltol=1e-6
.PROBE
.END
```

The second line in the file:

```
VIN 1 0 SIN(0 12V 50Hz 0 0 0)
```

states that the voltage source VIN is connected between node 1, the positive node, and ground. It is a sine wave with an amplitude of 12 V and a frequency of 50 Hz. The other four parameters in the SIN function are set to zero. See the PSpice Reference Manual (part of the Online Manuals) for the exact meaning of the parameters of the SIN waveform.

Lines 4 and 5 define the capacitor and resistor, which are both connected from node 2 to ground (node 0):

```
CF 2 0 2700uF
RL 2 0 25
```

Note how the capacitance value has been set using the letter 'u' to designate 'micro', so that uF means micro Farads. Other letters that can be used in this fashion are 'k' for kilo, 'm' for milli, and 'n' for nano.

The .TRAN statement tells SPICE to run a transient simulation from t = 0 to t = 80 milliseconds with print increments, starting at 20 ms, of 1 ms. During transient analysis, all node voltages and source and device currents are calculated *as functions of time*.

The .OPTIONS line reduces the relative tolerance limit of PSpice to 10^{-6} in order to get more accurate results (the default tolerance limit is 10^{-3}, or 0.1%).

The line next to last contains a .PROBE command which automatically prepares the data for graphical display using Probe. You will use Probe to examine the various voltage and current waveforms in the circuit.

PART A:

Edit a text file with the above SPICE circuit description, and save it as `rectify.cir`. Prepare the SPICE simulation for this circuit (File -> Open Simulation; choose Circuit Files (*.cir); select rectify.cir). Run the simulation using (Simulation -> Run rectify). After the simulation is finished, plot the voltage at node 1, using Trace -> Add Trace (or by pressing the Insert key) and selecting `V(1)`.

Add another trace corresponding to the output voltage, which is `V(2)`

Add a third trace, which is the voltage across the diode; this is equal to the difference between the voltage at node 1 and that at node 2, so enter next to "Trace Expression":
`V(1) - V(2)`

You should now have three waveforms displayed in the Probe window: the input, output, and diode voltages.

Answer the following questions

A-a)   What is the magnitude of the ripple voltage?

A-b)   What is the peak inverse voltage of the diode?

A-c)   Erase all the waveforms, then add a new trace, that of the current in the diode: `I(D1)`. What is the peak diode current? What is the average diode current? Note that the AVG() function in Probe does *not* give an accurate answer in this case.

PART B:

Write the circuit description file for Circuit 2. Repeat the steps that were preformed for Circuit 1, and answer questions a) to c) of PART A. Note in this case that the negative terminal of the source is *not* connected to ground. Use `V(1)-V(3)` to display the input voltage.

Spice Exercise Number 3
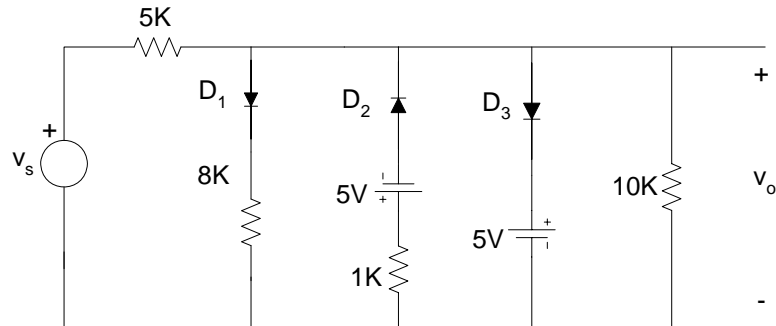
Plot, using SPICE, the transfer characteristics of the circuit shown. Assume that when conducting in the forward direction, diodes drop 1 V.

The diodes can be modeled in SPICE using a D element with $n = 0.001$ (the small n value makes the D element behave as an ideal diode: `.model Dideal D (n=0.001)`), in series with a 1 V battery to model the voltage drop when the diode conducts.

The V_S source is set initially to -20 V, and then changed in steps of 0.1V to +20V, using the `.DC` statement:

```
VS 1 0 DC -20
.DC VS -20 20 0.1
```

SPICE Exercise Number 4

a) Design the amplifier shown in the figure below to achieve a small-signal voltage gain $A_v = v_o/v_i$ of -10 for $R_L = 47$ KΩ, and to have $R_{in} = 1$ MΩ. Assume the Q-point is chosen at $I_D = 0.1$ mA and $V_{DS} = 3$ V. For the MOSFET, $V_t = 1$V, $k'(W/L) = 1$ mA/V^2 and $\lambda = 0.02$ V^{-1}. Note that for this amplifier, $R_{in} = R_1 \,//\, R_2$. Assume that $V_{DD} = 9$ V, and that all capacitors are very large.
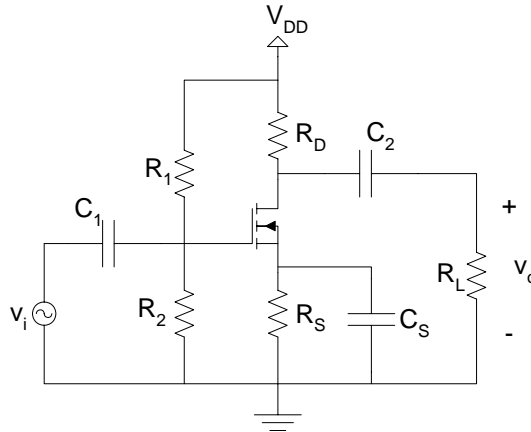
b) Find the largest possible input signal that results in:

1. MOSFET entering cutoff: i_D becomes equal to zero.
2. MOSFET entering linear: v_{DS} becomes equal to $v_{GS} - V_t$
3. Distortion: $v_{gs(peak)}$ equal to $(V_{GS} - V_t)/25$.

What should $v_{i(peak)}$ be limited to in order to avoid all the above conditions?

c) Find the worst-case (smallest), and the best-case (largest) values of voltage gain when V_t varies in the range 0.8 V to 1.2 V, $k'(W/L)$ varies in the range 0.9 mA/V^2 to 1.1 mA/V^2, and λ varies in the range 0.015 to 0.025 V^{-1}.

d)
```
MOSFET Amplifier
*
* DC supply
VDD 1 0 DC 9V
*
* Signal source
vi 2 0 SIN(0 1mV 1KHz)
C1 2 3 100uF
*
* MOSFET
M1 4 3 5 5 menhn W=100u L=10u
.model menhn nmos (kp=100u vto=1 lambda=0.02)
*
Cs 5 0 100uF
C2 4 6 100uF
.
```

```
.
.
.
.
.Tran 10us 8ms 5ms 10us
.OP
.Probe
.End
```

Complete the circuit description file shown above, and simulate the circuit in PSpice to verify the results of part (a). Note that the signal source (connected between nodes 2 and ground) is specified to have a 1 mV amplitude and a 1 KHz frequency:

```
vi 2 0 SIN(0 1mV 1KHz)
```

A MOSFET is defined in a SPICE file as follows:

```
Mxxx drain_n gate_n source_n body_n model_name L=length W=width
```

where `Mxxx` is the transistor name (for example Mampli or M123), `drain_n`, `gate_n`, `source_n`, and `body_n` are the node numbers to which the transistor drain, gate, source, and body are connected, respectively; `model_name` is the name of the transistor model, and `length` and `width` are the gate length (*L*) and gate width (*W*), respectively. For example, in the circuit above, the MOSFET line

```
M1 4 3 5 5 menhn W=100u L=10u
```

states that MOSFET M1 has drain connected to node 4, gate connected to node 3, source connected to node 5, and body connected to node 5. M1's model is "`menhn`", which should be defined somewhere in the SPICE file. The gate length of M1 is 10 micrometers (`u` in SPICE means micro), and its width is 100 micrometers, for a *W/L* ratio of 10.

The MOSFET model is defined using a .model statement:

```
.model model_name nmos (model_parameters)
```
for NMOS transistors

or

```
.model model_name pmos (model_parameters)
```
for PMOS transistors

For example

```
.model menhn nmos (kp=100u vto=1 lambda=0.02)
```

states that MOSFETs of type "menhn" are NMOS, with:

$k' = 100\ \mu A/V^2$ (`kp`), $V_t = 1$ V (`vto`), $\lambda = 1/V_A = 0.02$ V^{-1} (`lambda`)

The "large" capacitors in the circuit are specified to have a capacitance of 100 μF. At the signal frequency, their impedance is $1/(2\pi \times 1\ KHz \times 100\ \mu F) = 1.6\ \Omega$, which is quite negligible when compared with the other resistance values in the circuit.

Measure, using Probe, the voltage gain, by measuring the peak value of V(6). Extract from the output file the values of I_D, V_{GS}, V_{DS}, g_m, and $r_o = 1/g_{ds}$. Compare with the values calculated or specified in part (a).

Include a printout of the Probe waveforms for v_i and v_o.

Verify using PSpice, by running a *Monte Carlo analysis*, the best-case and the worst-case gains when the MOSFET parameters vary.

In order to specify MOSFET parameter variations in PSpice, modify the MOSFET model as follows:

```
.model  menhn  nmos  (kp=100u  DEV=10u  vto=1  DEV=0.2  lambda=0.02
DEV=0.005)
```

The DEV modifier causes the parameter value to be any number in the range NOMINAL-DEV to NOMINAL+DEV. The NOMINAL value is the default value specified for the parameter (e.g. 100u for kp). Monte Carlo analysis simulates the circuit a large number of times, each time choosing a random value for the parameters in their specified ranges.

To run a Monte Carlo analysis, add the line

```
.MC 100 TRAN V(6) YMAX OUTPUT ALL
```

to the PSpice file. Run the simulations (which may take a while, since we are asking for 100 simulations on the .MC line) and plot in Probe the output voltage V(6). Zoom in on the range of peaks for the resulting waveforms.

What is the range of values for the voltage gain, as obtained from Monte Carlo simulations?
How does this compare with your calculations in part (c)?
Include a printout of the Probe waveforms for *the peaks* of v_o (zoom in on the voltage range 6 mV to 12 mV, and on the time range 5.5 msec to 6 msec).

SPICE Exercise Number 5

A useful feature of PSpice is the ability to specify the gain of a dependent source as a Laplace transform function. We will use this feature to investigate the amplifier transfer function. The syntax used in PSpice for a Laplace voltage-controlled voltage source is:

```
Ename n+ n- Laplace {controlling variable} = {Laplace Transform}
```

As an example, the circuit description file for this assignment is:

```
Bode plot
Vin  1 0 AC 1
Rin  1 0 1
Eamp 2 0 Laplace {V(1)}={ 1e6*s / ( (s + 125.664)*( s + 9.42478e4) )}
Rout 2 0 1
.AC DEC 20 1 1e5
.Probe
.End
```

We use an AC voltage source (Vin), and an E element (Eamp) to model the amplifier transfer function. The two resistors (Rin and Rout) do not affect the voltages at the input (node 1) and at the output (node 2), but they are needed to close the loops in SPICE.

The analysis type is AC (**.AC** DEC 20 1 1e5), which sweeps the frequency with 20 points per decade (.AC **DEC 20** 1 1e5) between the frequencies of 1 Hz and 100,000 Hz (.AC DEC 20 **1 1e5**).

To obtain the magnitude plot in Probe, add a trace for V(2) in dB using: db(v(2))
To obtain the phase plot in Probe, add a trace for the phase of V(2) in dB using: p(v(2))

SPICE Exercise Number 6

For the circuit shown below, the BJT has $\beta = 100$. Assume that $V_T = 25$ mV and that all capacitors are very large.
a) Find the DC values of I_B, I_C, and V_{CE}.
b) Find the small-signal parameters g_m, r_π, and r_e.
c) Find the small-signal voltage gain of the amplifier, v_o/v_i.
d) Find the largest allowable signal swing (peak-to-peak) at the collector to satisfy the two conditions: BJT remains active, and v_{be} limited to 5 mV. What is the corresponding allowable peak-to-peak input voltage v_i?

The SPICE circuit description file for the circuit is shown below. The source is a 10 mV peak, 1 KHz sinusoidal voltage. C_1 is the capacitor that couples the source to the transistor, and its value is 100 μF (this is very large, as the corresponding impedance is 1.6 Ω at 1 KHz, which is negligible). C_2 is the capacitor between the two 50K resistors, and C_3 is the capacitor that couples the load to the transistor. In the circuit description file, some lines are missing, and you need to complete them.
Note that the BJT is described in the SPICE file as follows:

```
Qxyz collector_node base_node emitter_node model_name
```

where Qxyz is the name of the BJT. The model of the BJT in the circuit is QNPN; we define the parameters I_S and β of this model in the statement:

```
.model QNPN npn bf=100 is=5e-15
```

which states that the QNPN model is an NPN transistor with a β (bf) value of 100, and an I_S value of 5 fA.

```
Common-Emitter Amplifier
*
* DC supply
*
Vcc 1 0 DC 12V
*
* Source
*
vi 2 0 SIN(0 10mV 1KHz)
Rs 2 3 4.7K
C1 3 4 100uF
*
* BJT
*
Q1 6 4 0 QNPN
.model QNPN npn bf=100 is=5e-15
*
C2 5 0 100uF
C3 6 7 100uF
.
.
```

```
.
.
.Tran 10us 8ms 5ms 10us
.OP
.Probe
.End
```

e) Complete the circuit description file, and run the transient analysis. From the Probe plots of v(2) and v(7), calculate the gain v_o/v_i. Also, calculate the values of v_o/v_b and v_b/v_i from the AC components of v(7), v(4), and v(2).
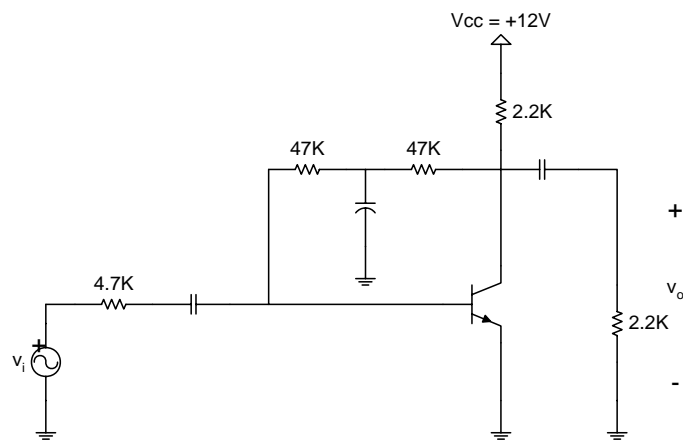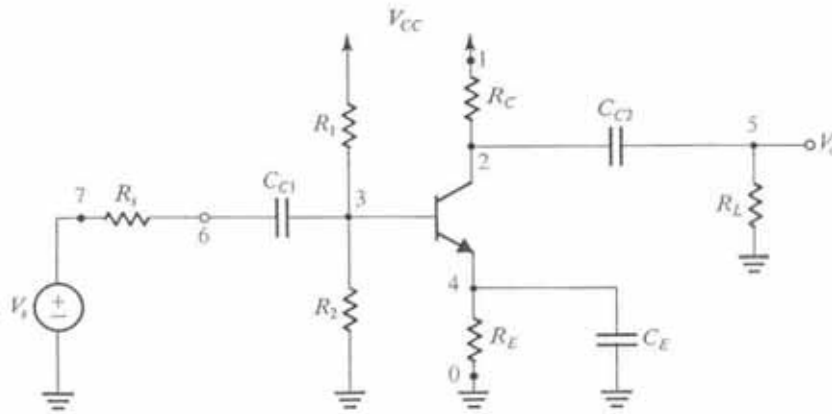
f) Verify, using SPICE, the results of part d).

Figure 4

SPICE Exercise Number 7

Consider the common-emitter amplifier shown in the figure below.

Use $V_{CC} = 12$ V, $R_1 = 100$ KΩ, $R_2 = 39$ KΩ, $R_S = 4.7$ KΩ, $R_C = 4.7$ KΩ, $R_E = 2.2$ KΩ, $R_L = 4.7$ KΩ, $C_{C1} = 2\mu F$, $C_{C2} = 1$ μF, $C_E = 100$ μF.

The BJT parameters are: $\beta = 160$, $V_A = 78$ V, $C_\pi = 55$ pF, and $C_\mu = 4$ pF.

Assume for hand analysis that $V_{BE} = 0.7$ V and $V_T = 26$ mV.

Estimate A_M in dB for this amplifier using:

    a. The small-signal equivalent circuit at midband.
    b. PSpice (use .AC DEC 20 10Hz 1000KHz)

Estimate f_L for this amplifier using:

    c. AC analysis in PSpice.
    d. Verify, using transient analysis at f_L, that the amplitude and phase of the output match with your expectations. Use an input such as
        vs **input_node** 0 SIN(0 5mV **f_L**)   with the following transient analysis:
        .tran 150us 240ms 150ms 150us

Estimate f_H for this amplifier using:

    e. Miller's theorem.
    f. AC analysis in PSpice.
    g. Verify, using transient analysis at f_H, that the amplitude and phase of the output match with your expectations. Use an input such as
        vs **input_node** 0 SIN(0 5mV **f_H**)   with the following transient analysis:
        .tran 0.05us 0.08ms 0.05ms 0.05us

In PSpice, use the `Q2N2222` model from the "`eval.lib`" library for the BJT. You can use the statement
`.lib "eval.lib"`
to include the library, which defines, among other models, the `Q2N2222` BJT model.