

CENG 241

PSPICE Analysis of Common Emitter Transistor Circuit.

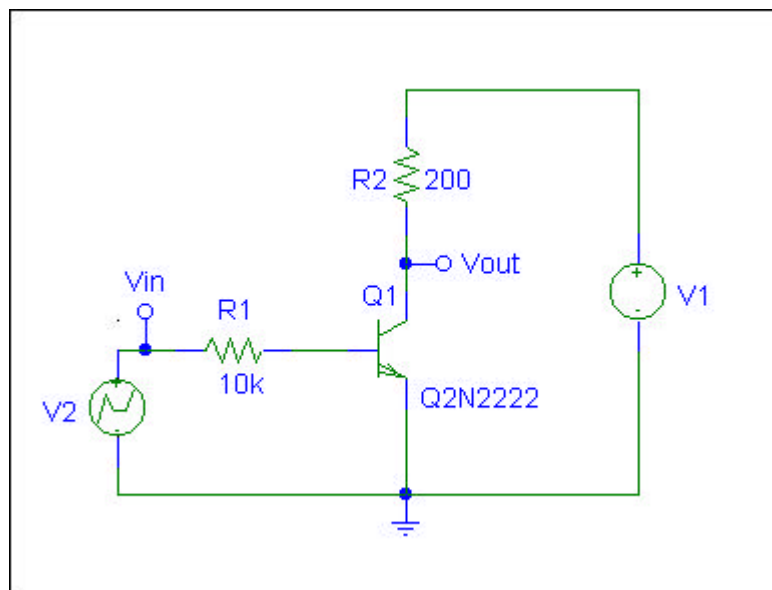
Introduction

PSPICE is a program that can simulate the operation of electrical and electronic circuits. It is a very useful tool that can help in the analysis and design of circuits. The program itself comprises 3 major components:

1. a graphical use interface for constructing circuits and setting various parameters (Schematics Editor)
2. a computational engine that actually calculates the node voltages and currents for each component (PSpiceAD).
3. a graphical interface for displaying the results (PROBE).

PSPICE (evaluation version 8) is available on some of the computers in EP307 (installed on the hard drive), or you may download the evaluation copy at http://www.orcad.com/products/pspice/eval_f.htm. We also have the full version of PSPICE installed on some of the computers in the department, in particular the NT workstations in EP339. The problem in this assignment can be solved using the evaluation version, which although limited in the number of components that can be included in a circuit, is quite powerful program. It is assumed that you will be using the evaluation versions available in ep307.

Circuit



Goal

To make graphs of output voltage, power dissipated in the transistor, and current gain (β) versus input voltage.

Method

1. Run the schematic editor (Start, Programs, Dept, EE Dept, DesignLab Eval8, Schematics)
2. Place a transistor on the schematic (Draw, Get New Part, Libraries, Eval, Q2N2222, OK, Place & Close). Click the left-mouse to actually place the transistor. You will see the black outline of the transistor; you can place additional transistors if you wish. Click right mouse to make the outline go away.
3. Place the base and collector resistors (Draw, Get New Part, Libraries, Analog, R, OK, Place & Close). Use Ctrl r to rotate the resistor as needed. Place the resistor near the transistor so that it can be connected later. Resistors come with a default value of 1k. You can change the value by double clicking on the 1k and then editing the value. Use the values that we used in Lab 8 (200 ohms for R2 and 10k for R1.)
4. Place a ground on the emitter of Q1 (Draw, Get New Part, Libraries, Port, Gnd_Earth, OK, Place & Close). Important, PSPI CE wont run without a ground!
5. Place a DC voltage supply near the collector resistor (Draw, Get New Part, Libraries, Source, VSRC, OK, Place & Close). Double click on the V1 symbol and set the DC value to 5.
6. Place a piecewise linear voltage source near the base (Draw, Get New Part, Libraries, Source, VPWL, OK, Place & Close). Double click on the V2 symbol and edit the values so that $t1=0$, $v1=0$, $t2=5$, $v2=5$. Don't worry about the other values $t3$, $v3$, etc. Each time you edit a value, make sure you save it by clicking save attribute.
7. Connect up the circuit using wire (Draw, Wire); a pencil outline appears, click on the start point, and drag the dotted line to the stop point - if the stop point is not another component terminal, then the dotted line will continue, allowing you to turn corners. Make sure you connect the negative side of both supplies to ground. You can drag components and stretch wires. Anytime two crossing wires connect with each other, a small blue circle appears to identify the connection. (Two crossing wires without a blue circle are not connected.) Blue connection dots do not usually appear when a wire connects with a component terminal, although if this does happen, it is ok.
8. To help with identifying the voltages of interest, add bubbles to the input voltage generator and the transistor collector (Draw, Get New Part, Libraries, Port, Bubble, OK, Place & Close). Use Ctrl r to rotate the bubble as necessary. Double click the bubble and edit the labels to V_{in} and V_{out} . This is not necessary in general, but it will help us to identify the points of interest among the many variable names that appear in the simulation output.
9. At this point, the circuit should appear to be similar to the one shown above. It is now necessary to setup the simulation parameters (Analysis, Setup, Transient). Set the final

time to 5 (seconds), click OK, then make sure the transient analysis check box is checked, then close the Analysis Setup box.

10. You have completed the schematic, it is necessary to save it (use your DOS disk) before going further.
11. You are now ready to run the simulation (Analysis, Simulate). You should see the PSpiceAD window pop up, which informs you of the progress of the simulation. The simulation will proceed quite quickly (because of its simplicity), if you have connected the circuit correctly, and set up the simulation properly. The PSpiceAD window will then report the transient analysis has finished. Otherwise, the message viewer will pop up telling you that an error has occurred. Double clicking on the error message will highlight the offending component in the schematic editor, to help you debug the circuit.
12. Assuming that the analysis proceeded correctly, the display interface, PROBE, should have started. Initially, this window will contain no traces, unless you placed markers (circles with arrows sticking out) on the schematic. In PROBE, select (Trace, Add) and you will see a list of all of the voltages and currents in the simulation. Select V(Vout), (or type this expression into the trace expression window). You should see a green trace appear representing the output voltage as a function of time. Actually, because we made the input voltage be equal to time, this is also a plot of output voltage versus input voltage, but we can change the x-axis as follows (Plot, X Axis Setting, Axis Variable, V(Vin), OK). Make a copy of this graph by printing it out. Make an accurate record of the input voltage that corresponds to cut-off, linear and saturation regions. Estimate the value of β for this transistor.
13. Delete the trace by highlighting V(Vout) in the PROBE window and then pressing delete. Now add a trace that shows the power dissipated in the transistor as a function of Vin: (Trace, Add) then type/select $V(Vout) * IC(Q1)$ in the trace expression line. This curve should show that the transistor dissipates quite a lot of power in the linear region, no power in the cut-off region, and a small amount of power in the saturation region. This is one reason why we like to ensure the transistor is saturated when operating as a switch. Print this graph.
14. Delete the trace you made in part 13, and make a graph of β , by plotting $IC(Q1) / IB(Q1)$ versus Vin. The curve should be flat in the linear region and roughly equal to the value of β you calculated in part 12. print out this graph.
15. If you are interested, try a temperature simulation, (Analysis, Setup, Temperature) then type in 0, 27,50 in the temperature box. Rerun the simulation, and plot all 3 Vout curves versus Vin. You should see some variation due to the dependence of β on temperature. Double click on the trace symbol (e.g. triangle) to get info on the trace.

Conclusion Summarize your results and include a copy of your schematic and the 3 graphs you printed.