

# EE321 Project 2: Transistor Simulation

## Introduction

In this project you will use **Mentor Graphics** to solve problem 4.78 of the course text (pg. 342). The circuit to be simulated will use an NPN bipolar junction transistor.

## Entering the Circuit

Log into one of the HP workstations and, from the pop-up menu, and start **Mentor Graphics**. Once **Design Manager** loads, maximize the **Design Manager** window. You will need to input the new circuit using **Design Architect**. Start **Design Architect** by double clicking on the `design_arch` icon in the Tools window. If you can't see the icon, use the scroll bars or the pop-up menu command `Update Window` to bring it into view. Once you have started **Design Architect**, you can minimize the **Design Manager** window. When **Design Architect** has loaded fully, its window will maximize itself.

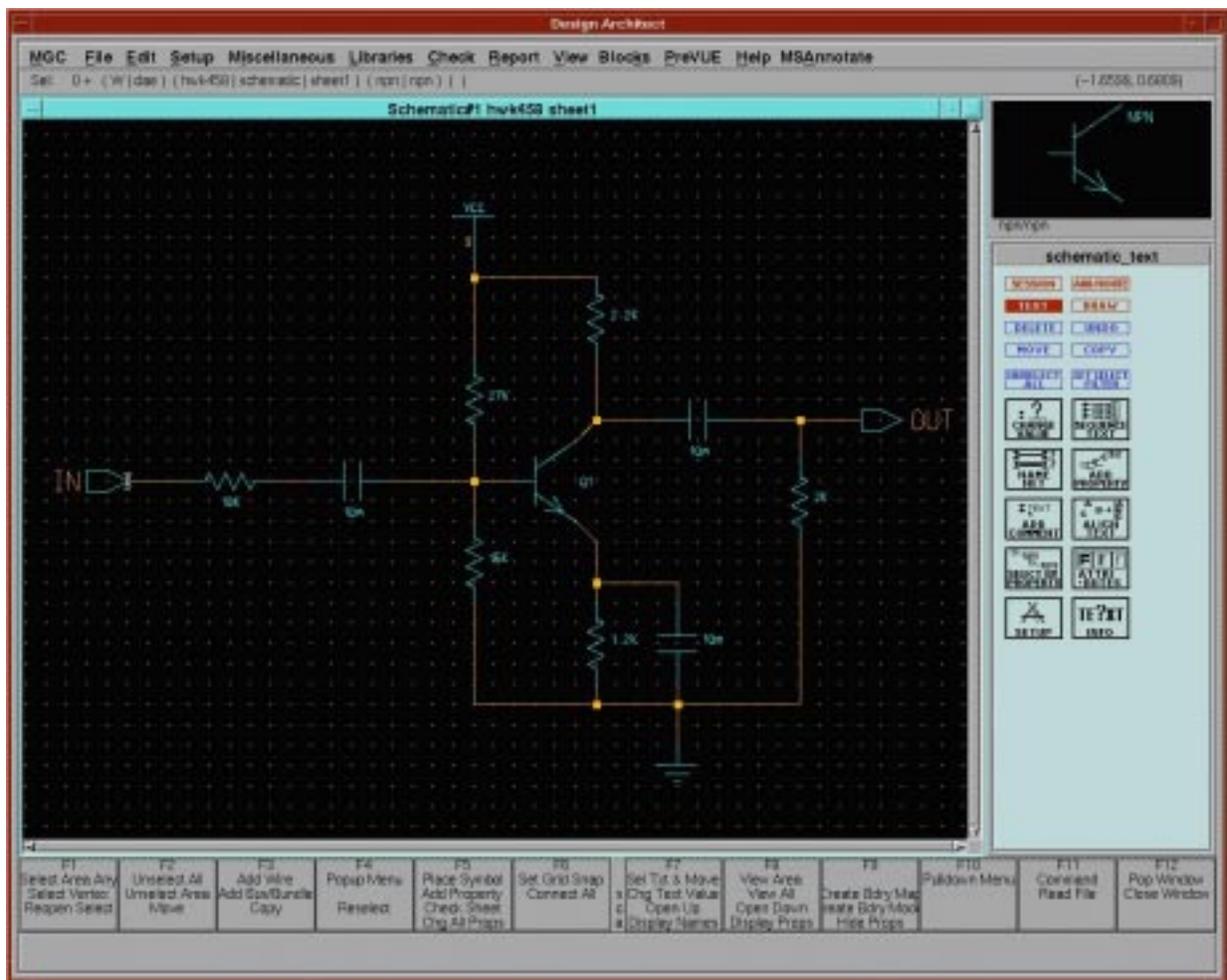
Create a new sheet by clicking on the `OPEN SHEET` icon in the palette area. In the form, click in the `Component Name` text window and add `/hwk478` to the existing text. OK the form. When the new, blank sheet appears, you might want to maximize the inner window the sheet is in and zoom out a little so you have some room to draw your circuit. The circuit you will be entering is Figure P4.78 of the course text (pg. 343). Because **Mentor Graphics** uses slightly different symbols for `Vcc`, inputs and outputs, a copy of the schematic entered in **Design Architect** can be seen in Figure 1 of this handout.

To begin entering the circuit, use the method you learned in the first **Mentor Graphics** project. You will use the `Generic Parts` library for the circuit (it appears when you select `Libraries⇒MGC Analog Libraries` from the menu bar and click on `Generic Parts` in the palette area). For this circuit you will need six (6) resistors, three (3) capacitors, an NPN transistor, a `PORTIN`, a `PORTOUT`, a `Vcc` symbol and a ground symbol. If you run out of room, use the view commands to zoom out or `View⇒All`. (View commands are found in the menu bar, the pop-up menu and the function key definitions.) NOTE: Be sure **not** to overlap any parts, especially their terminals.

Once you have placed all of the parts, go back to the schematic palette by using the pop-up menu in the palette area. Use the `ADD WIRE` tool to draw all the nets (wires) in the circuit. See Figure 1. Remember: use one click to start a wire, one to add a bend and two clicks to end a wire. If you make a mistake, backspace undoes the last click. Be careful when drawing the nets connecting to the base of the transistor. The safest way is to draw the wire between the base terminal and the capacitor first. Then, draw the wire from each of the bias resistors to that wire. This will

help ensure that the bias resistors are connected to the base terminal. When you are done wiring, cancel the ADD WIRE tool in the lower left corner. If you haven't done so, you may want to unselect everything now.

To finish the circuit, you need to change the component values to match those in the problem statement. First, bring up the Text palette by clicking on its brown button. Make sure nothing is selected, then, holding down the shift key, click on a component's value (for example, we'll use the upper bias resistor). The value should turn white. Next, click CHANGE VALUE on the Text palette. A prompt bar should come up. Enter 27K (the new value for the resistor) and press ENTER or click OK. Then, Unselect All and repeat the process for every part in the circuit.



**FIGURE 1.** Circuit to be analyzed.

See Figure 1 or the text for the values to use. For the infinite capacitances, use a value of 10m (10 millifarads); this will seem infinite to the simulator. The Vcc value needs to be changed from 5 to 9. Also, the ASIM\_MODEL component value of the transistor needs to be changed to Q1

(the process is the same as that of changing the resistance values). Don't forget to rename the ports to IN and OUT. When you are finished modifying component values, check the circuit using the `Check⇒Sheet` menu command. If there are any errors, use the cross-selection feature to locate the errors; then correct them. When the sheet checks with no errors, save it using `File⇒Save Sheet`. At this point, you can close **Design Architect**.

## Modeling a Transistor

A transistor has many parameters that define it. The simulator needs to know the values of these parameters in order to simulate the circuit containing the transistor. Most of these parameters have reasonable default values; you won't have to change them. However, there are a few parameters that you will need to change almost every time you use a transistor in a circuit (such as the beta value).

In this circuit, you told Mentor to look at a model file called Q1 for the transistor's model information (this is similar to the way SPICE works). To create the file, go into **Design Manager** and double click on the `Editor` icon (in the Tools window on the left). A blank text editor window will appear where the Tools window was. Enter the following line:

```
.model Q1 NPN (BF=100 VA=100)
```

This will tell the simulator that transistor Q1 has a forward beta of 100 and an Early voltage of 100 volts. To save the file, close the editor window. A form will pop up asking if you want to save the file. Answer YES and enter `q1.net` for the file name. OK the form. Now, the circuit is ready to simulate.

## DC Operating Point Analysis

Select your circuit in **Design Manager**. Use the pop-up menu to start **AccuSim** on the circuit. While **AccuSim** loads, you can minimize the **Design Manager** window. You'll probably want to maximize **AccuSim**'s window when it's finished opening, and do a `View⇒All` on the schematic window so you can see everything clearly. The transistor's model file is not automatically loaded. To bring it in, select `File⇒Auxiliary Files⇒Load Netlist File...` from the main menu. Enter `q1.net` in the filename box and click OK.

The first step in analyzing the circuit is performing a DC operating point (bias point) analysis. You'll need to provide an appropriate input and set the analysis options. To provide the input, select the IN port on the schematic, then click on `ADD FORCE` in the palette. The form that pops up should say that you are forcing a DC voltage on the signal / IN referenced by //ground. Enter a value of 1 for the magnitude, and OK the form. To verify that the force was added, select `Report⇒Forces` from the menu bar. The top line of the report will list the force ID (probably

d\$0). Make a note of this force ID; you will need it in a minute. You can close the report window now.

Next, click the `SETUP ANALYSIS` button in the palette. Select the `DCOP` analysis type. We'll also do a transfer function analysis at the same time, so click the "Off" box next to `Transfer Function` to turn it on, then click the `Setup . . .` button to the right. Make sure that `V(Net)` (a single net transfer function) is selected. Enter `/OUT` in the Net Name box (make sure to include the slash). Enter the Force ID from above in the Input Source Name box, then OK both forms.

At this point, the simulator knows everything it needs to simulate the circuit. Click the blue `RUN` button in the palette. OK the "Keeps" form that pops up. There may be a few messages about "no DC path to ground" - they can be ignored. The status window in the upper left of the screen will tell you when the analysis is finished.

After the analysis is complete, the palette changes to the `RESULT` palette. There are several options that can be used to view the results. The information can also be seen using the `Results⇒DCOP` menu item. You will probably want to look at the outfile and the DCOP report; both can be accessed from the palette. Use these tools to obtain any data you need to explain the DC bias of the circuit. The windows can be printed using `MGC⇒Setup⇒Printer` and `File⇒Print⇒Active Window` from the menu bar. Be careful not to print a window whose output would be many pages long; it could be harmful to your print quota.

## AC Analysis

An AC (frequency sweep) analysis is set up in a similar way to a DC operating point analysis. The transistor's model file (`q1.net`) is still loaded, so it doesn't need to be loaded again. First, get to the AC analysis palette by clicking the brown `FREQ MODE` button. A form should pop up to let you set the analysis parameters (if not, click `SETUP ANALYSIS` and select a type of AC). Use a sweep type of "decade" with a "points per decade" value of 20. Enter `1k` into the Start Freq box, and `500k` in the Stop Freq box. With only the input signal `IN` selected, click `ADD FORCE`. Select an AC force type with a magnitude of `0.3`, phase of `0` and DC offset of `0`. OK the form and click `RUN` to perform the analysis. When the simulator is finished, select the output port and click the `TRACE` palette button. This will show a plot of the output signal versus frequency.

To calculate the gains and impedances needed for the solution to the problem, you'll need to find the ratios of some waveforms. This can be done automatically by **AccuSim**, using the Waveform Processor tool. To use it, click the `OPEN WF PROC` box in the palette. For example, we'll plot the sum of the output voltage and the input voltage.

In the `WF PROC` window, click on the diamond above the `X`, then click in the white text window above it, and enter `freq`. This will place the frequency along the `X` axis. Now, click on

the diamond above the  $Y$ , then click on the  $V$  next to the **CHART** button. (To plot a current instead of a voltage, you would click the  $I$  instead). Move the mouse cursor into the schematic window (it'll become a set of red crosshairs). Click on the point you want to measure the voltage of (in this case, the input port). The internal name for that point will appear in the white text box. Click in that box and add a  $+$  to the end of the text. The waveform processor also understands subtraction, multiplication ( $*$ ) and division ( $/$ ). Now, click on the  $V$  button, and select the output port. Click the **CHART** button to create the plot, and **OK** the form that pops up.

When the Chart window is active, moving the cursor around will display the coordinates of its position in the upper-left corner of the chart window. The **Chart** menu item on the pop-up menu has several options to use; a handy one is the **Add Cursor** selection. It adds a graphical marker to the plot that can be moved around, and labels the coordinates of the point where it's placed. When this menu item is selected, a prompt bar will appear. Enter a name to identify the point you're labeling. Enter an X-axis location in place of 0 and **OK** the prompt bar, or click on the crosshair symbol (next to the word "click") and click in the chart, to place the cursor.

Using these tools and the outfile information (if necessary), make some printouts that provide data to verify your hand solution of the problem. At this point, **AccuSim** can be closed (**Without Saving**). Make sure to close **Design Manager** as well before logging out.