Laboratory exercise: the Bipolar Transistor

Semiconductor Physics 2015

Lab meeting point – k-space at Solid State Physics

This exercise consists of two experimental parts and one simulation part.

In the first experimental part you will be measuring the *IV*-characteristics of a Si *npn*-transistor in the common emitter mode and analyse the gain of the transistor in a so called Gummel plot.

In the second experimental part you will be using a transistor to amplify a small current.

In the simulation part you will use commercial software which is used for industrial design of semiconductor devices. We will use the software to study physical parameters like the electric field and the minority carrier concentration inside the device. This information is difficult to obtain experimentally, but it can easily be simulated.

As a preparation for this lab we recommend you to study the handout and the chapter about the bipolar transistor in the text book (chapter 4 in Sze 3rd edition, chapter 5 in the 2nd edition).

2015-09-14

Device layout

In the simulation software, it may take some effort to identify the emitter, collector and base regions of the device. Look at the screenshot below:

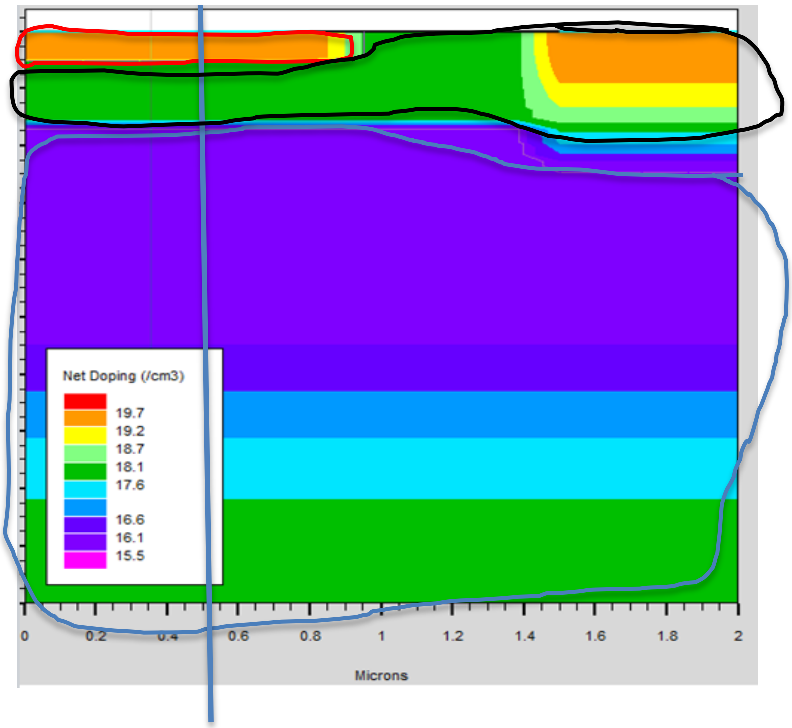


Figure 1: Screenshot from ATLAS.

The emitter is the highly doped region on the top left (orange, outlined in red).

The base is below it (green, outlined in black). The region of base doping extends out to the right hand side where there is a heavily doped area (orange) under the top base metal contact.

Below the base is the lighter doped collector region (blue, green, also outlined in blue), taking up the majority of the structure above.

To picture the emitter-base-collector structure, look along a vertical line such as the one inserted in the figure. Then we have a heavily doped emitter, a more lightly doped base, and then an even lighter doped collector.

Experimental Part 1:

A. IV-characteristics

We will start the experimental part by measuring the *IV*-characteristics of a Si npn bipolar transistor in the common emitter mode. These measurements will be performed by the measurement set-up shown below where we vary the base current as well as the emitter-collector bias V_{CE} and measure the current I_C through the collector. These measurements are performed for a set of different base currents I_B and hence we get the *IV*-characteristics for a bi-polar transistor in common-emitter mode (fig 10 p. 137 in Sze 3rd edition and fig 10, page 145 in the 2nd edition).

Connect the instruments as shown below:

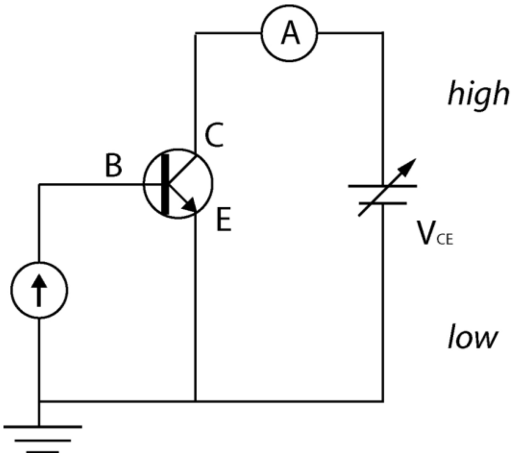


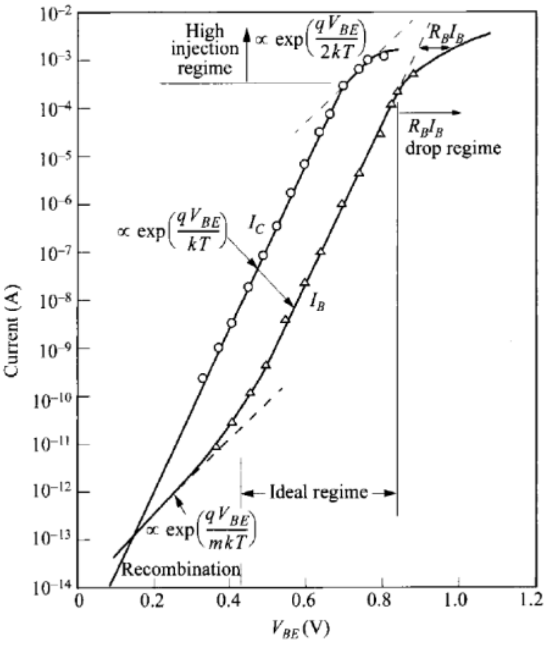
Figure 2 Experimental set up for *IV* characterisation in common emitter mode.

- Measure the *IV* characteristics. Use input parameters:
    - $V_{start} = 0V$, step = 0.1V, $V_{stop} = 6V$,
    - Start base current = 0A, Stop base current = 100µA, step = 20µA)
- Measure I_{CE0}
- The *IV* curves have a certain slope. Explain the reason!

B. Gummel plot

The gain of a transistor is a key parameter and we will be measuring it in a so-called Gummel plot, where I_C and I_B are plotted as a function of V_{BE} (see figure 3)

We will study the base- and collector current as a function of the base-emitter bias at a fixed collector voltage (2V). The relation of these two currents (I_C/I_B) will give the common emitter gain, β, of the device. The measurement set-up is given in figure 4.



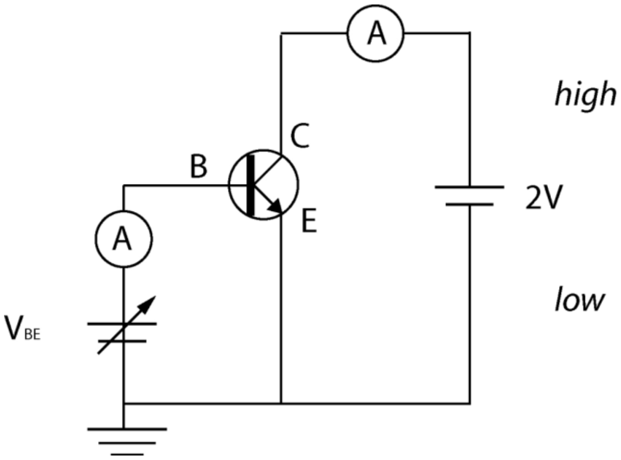


Figure 4 Experimental set up to obtain a Gummel plot. The base and collector currents are measured for varying base-emitter voltages.

- Connect the instruments as shown above
- Measure the Gummel plot (base and collector current and sweep the base voltage. Use input parameters:
    - $V_{start} = 0.1V$, $V_{stop} = 1.1V$ and $V_{step} = 0.05V$.
- Plot I_C and I_B vs V_{BE}.
- Estimate β, and calculate the common base gain, α.
- How does gain (β) vary with collector current? Why does the gain saturate?
- Discuss the problems of obtaining a high gain in the device. Is it, for instance, possible to combine a high gain and a high current level?
- If the transistor is used in common collector configuration instead, would you expect a higher or lower gain?

Simulation Part

The simulation program numerically finds the solution to a set of coupled differential equations, namely the Poisson equation and the continuity equations. The solutions to these equations, with proper boundary conditions, determine all intrinsic parameters like the potential, the electric field and the carrier concentrations. The program uses a finite element method, where the device geometry is divided into a fine mesh. Numerical iteration is then performed for each mesh point in order to determine the solution to the differential equations.

About the software:

We are using a commercial software, ATLAS by Silvaco, for the simulations. The program basically consists of a numerical solver, which finds a solution to by iterations as described above.

The program is controlled via a set of tools for graphical interaction. In this exercise, we will be using a module called **Deckbuild**, into which we will load a code defining the device geometry. The software then deduces solutions for various boundary conditions (biases or current levels) and saves the solutions to output files.

The second module we will use is a graphical interface called **Tonyplot**, which displays the calculated values graphically. We can for instance obtain 2D plots of the device geometry and of physical parameters, like the electrical field and the carrier concentration. In addition to the 2D plots we can also make cuts through the device geometry and study 1D cross-sections of the device. These cross-sections will be similar to the images in the textbook. Finally, Tonyplot may also be used to plot the calculated IV-characteristics.

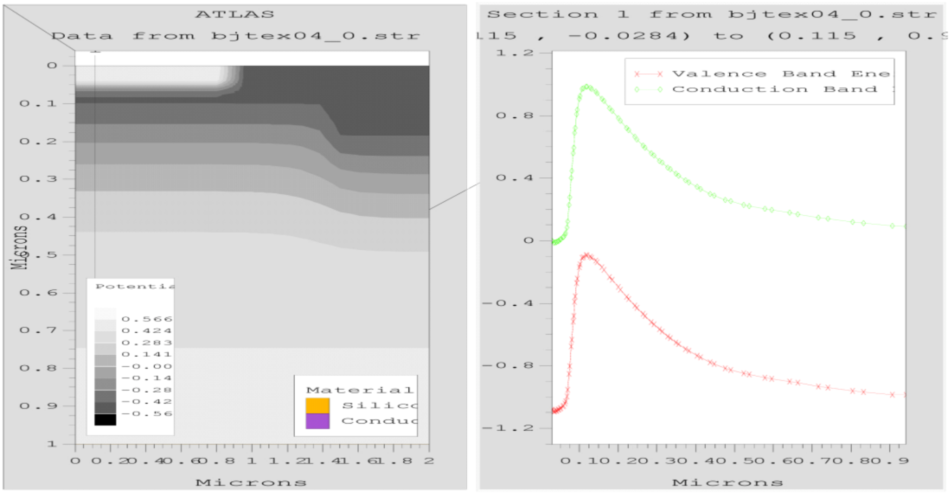


Figure 5. Example of output from the Silvaco modelling system on a npn bipolar transistor.

Start the simulation tool `Silvaco Deckbuild.` A shortcut is located on the desktop. The computer must be connected to the licence server via LU Web-logon.

The device that we will study is a npn bipolar transistor. The structure layout is similar to the picture on p. 49 in the handout.

(A) Initiation and basic plots

Load `bjtex04.in` located in c:\SilvacoWork\Bipolar Transistor\

The `.in` file is a set of instructions specifying the device structure and what properties to extract. For now, just press start (green arrow) to run the simulation.

After some simulation time you should have a number of plots on the screen. Play around with Tonyplot to see what information you can extract. Especially the Plot/display../contours command and the Tools/cutline tool are very useful.

(B) Reading the code and observing different bias conditions

Study the .in file. The first part sets the geometry and doping levels of the device. You could in principle change parameters as you like in order to change the specifics of your device. The second part solves for carrier concentrations, band structure etc. and creates output files. There is no need for you to understand each command but the overall structure of the program should be clear.

Follow the code downwards. As the program proceeds with calculating the Gummel plot and the IV-curve, it makes several outputs of the cross-section under different bias conditions. See if you can locate these in the program code. You can find the output files in the Silvaco BJT folder with names bjtex04_?.str where the ? is a number. Study these plots and read the bias conditions from the code.

What can you observe? What can you say about the injection? Can any high injection be observed etc.?

Experimental Part Two:

This part aims at giving a short introduction to how to use a BJT in an electrical circuit. Your job is to amplify the small current from a photoresistor so that it can drive a computer-fan. The idea is that the light in the room should control the speed of the fan. You should use the transistor in common-emitter mode.

Start by examining the parts:

- Measure the resistance of the photo resistor in darkness and in bright light. Let's assume that the maximum current (a higher current causes it to break) in the photo resistor is approx 5 mA. What is the maximum voltage it can be connected to?
- How much current does the fan draw?
- What is the maximum current and voltage the transistor BC548B, can take? (Datasheet will be available.) Maximum power dissipation? What is the maximum inside (junction) temperature of the transistor?

Make a sketch of how you would like to realize the circuit and show it to the supervisor.

Do some calculations:

- What is a suitable base current in order to make the fan go at full speed?
- What is the minimum total resistance you need to connect at the base in order to not go above the previously calculated base current? Assume that $V_{BE} = 0.6$ V.

Remember – there is nothing limiting the current into the base of a BJT. It will blow up immediately if you connect it directly to the supply! Use a resistor!

Now connect the components on the prototype board and connect the supply. Check for smoke and smell of hot components. Check if the circuit is working as supposed.

Do some measurements on the circuit:

- What is the voltage drop over the fan at high and low illumination?
- What is the base-emitter voltage at high illumination?

For the lab report:

a) Experimental part one: plot the measured data of the transistor (IV characteristics and Gummel plot) and answer the questions in the text.
b) Simulation part: show some results of the simulation and explain what you see. Answer the questions in the text.
c) Experimental part two: show a sketch of the circuit. Answer the questions in the text.