Electronics Laboratory

Winter semester 2023

Lab 2 – Bipolars

Name	______________________	Reg. Nr.	____________
Name	______________________	Reg. Nr.	____________

Score and comments (only for tutors, please leave blank)

Please fill out this cover sheet and submit it with your lab report.

2 Lab 2 – Bipolars

Introduction

We will begin by studying the operating behavior of a bipolar transistor, deriving the characteristics we have used in lecture by simulation and measurement. Once we understand our bipolar, we will then analyze a common-emitter amplifier, looking at the time and frequency response, and then consider a simple bipolar current source, deriving its operating limits.

Notes

- You will need the SPICE directive `.step` to perform parameter sweeps in the simulation sections. It would be worth your while to refer to the LTSpice manual for a description of how to use this directive and how to define the parameter to be swept.

2.1 Preparation

Please answer the following questions *before* beginning with the simulations or experiments. Then complete the preparation quiz "Preparation 2.1 – Bipolars" on Ilias using your results. The points for this preparation section will only be awarded via Ilias.

2.1.1 Beta

Consider the circuit in Figure 2.2.1 below, where $V_{DD} = 10\,\mathrm{V}$, $R_B = 100\,\mathrm{k\Omega}$ and $R_C = 200\,\Omega$. You apply $V_i = 2\,\mathrm{V}$ and measure $V_o = 9.16\,\mathrm{V}$ at the output. What is β for this transistor? Make standard assumptions.

2.1.2 Early voltage

The measurement of which parameters is typically used to determine the Early voltage?

2.1.3 Operating point

Below are given the output characteristics of a bipolar transistor employed in a simple common-emitter circuit, such as that of Figure 2.2.1.

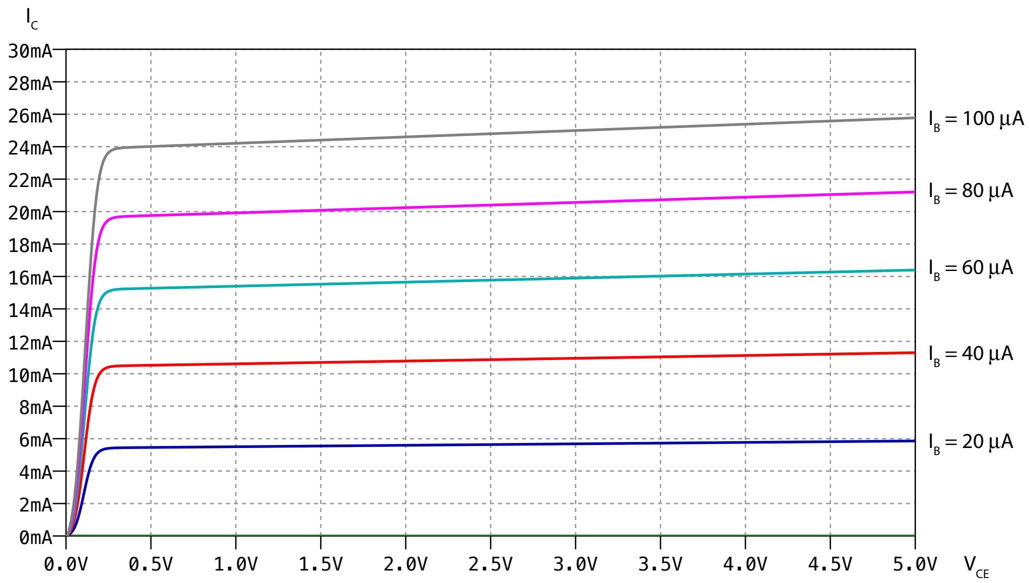

Figure 2.1.1

If $V_{DD} = 5\,\mathrm{V}$ and $R_C = 180\,\Omega$ and you measure an output voltage of $V_o = 3.0\,\mathrm{V}$, what base current flows into the transistor?

2.1.4 Small signal model

For the operating point you determined in Question 2.1.3, calculate the value for the differential input resistance, r_π. Assume room temperature.

2.1.5 Temperature effects

If the temperature of the transistor increases, would you expect any change in the value for r_π?

2.1.6 Amplification

Consider the CE amplifier in Figure 2.3.1 below, which uses the bipolar transistor whose output characteristics are shown in Figure 2.1.1. Take $R_o = 0$, $R_C = 680\,\Omega$ and $V_{DD} = 10\,\text{V}$. If your bias network generates $I_B = 40\,\mu\text{A}$, calculate the small-signal voltage gain $A_V = v_o/v_i$.

2.1.7 Current source

What are the relationships between currents and voltages in a current source?

2.2 Bipolar characteristics

We will simulate and experimentally characterize a bipolar transistor using a simple common-emitter configuration.

2.2.1 Simulation

Consider the common-emitter circuit shown in Figure 2.2.1.

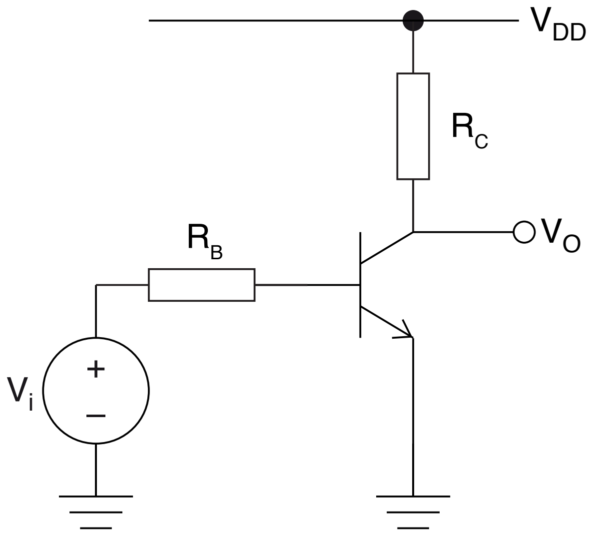

Figure 2.2.1

(a) Set up a SPICE simulation of this circuit using a BC547B bipolar and setting $R_B = 100\,\mathrm{k\Omega}$ and $R_C = 200\,\Omega$.

(b) Set $V_{DD} = 10\,\mathrm{V}$ and apply a ramp for V_i of $0\,\mathrm{V}$ to $10\,\mathrm{V}$ (use the `.dc` directive).

(c) During the ramp, measure I_B, I_C as well as V_{BE}.

(d) Determine β for this transistor at low currents.

(e) Plot I_B as a function of V_{BE}.

(f) Plot I_C and β as a function of I_B on a single graph. Is β constant? Comment on this.

(g) Now, again using the SPICE directive `.dc`, set a ramp for V_{DD} from $0\,\mathrm{V}$ to $10\,\mathrm{V}$ while stepping V_i from $0\,\mathrm{V}$ to $5\,\mathrm{V}$ in $1\,\mathrm{V}$ steps.

(h) Record and plot I_C as a function of V_{CE} for the different I_B, not forgetting to properly label your plot with the I_B values.

(i) Determine the Early voltage for each I_C/V_{CE} trace; how do the values compare?

2.2.2 Measurement

In the "BIPOLAR" section of the electronics board, we will use the *BJT characteristics* circuit, shown in Figure 2.2.2.

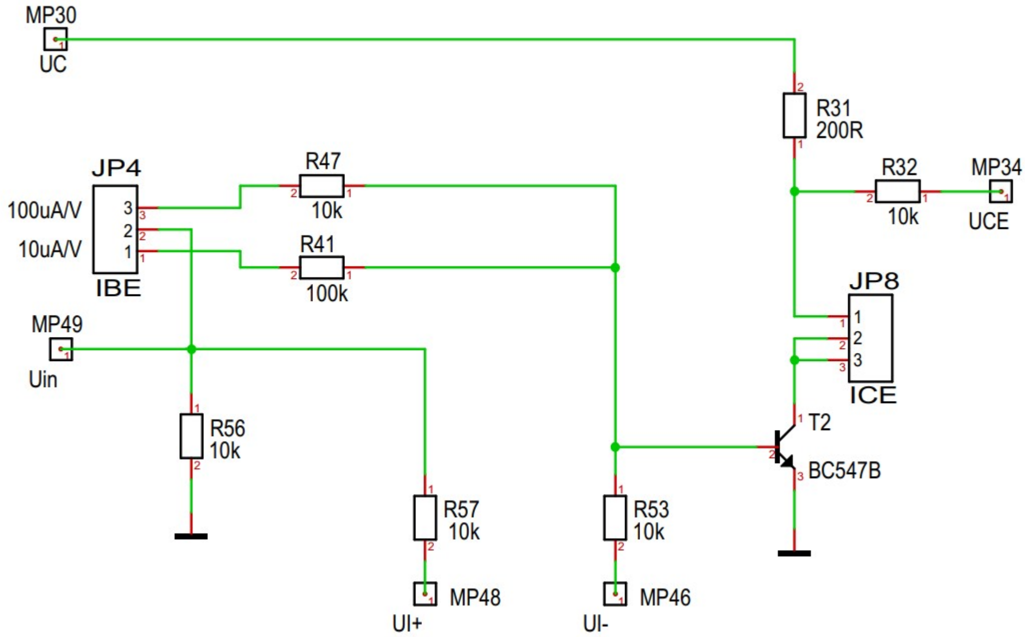

Figure 2.2.2: Schematic of the *BJT characteristics* circuit

We will undertake three measurements: the I_B/V_{BE} characteristic (a –g); the current gain β (h – o); and the I_C/V_{CE} characteristic (p – w).

$\mathbf{I_B/V_{BE}}$ characteristic

(a) Short-circuit pins 1 and 2 of the jumper *JP8*.

(b) Connect R_{41} via jumper *JP4*.

(c) Connect the two measurement points *MP48* and *MP46* to the differential amplifier and use this to measure the voltage drop across R_{41} (the current I_B can be calculated based on this voltage).

(d) Set the measurement point *MP30* (V_{DD}) to 10 V and measure the applied voltage to verify the value.

(e) Apply a voltage ramp (Triangle, 0 V $\rightarrow$ 5 V, 50 %, 50 Hz) to the measurement point *MP49*. Set a proper offset and amplitude to have a well-defined triangle input voltage.

(f) Measure V_{BE} directly at the measurement point *MP46*.

(g) Measure and plot I_B as a function of V_{BE}. Save your data.

Current gain

(h) Connect the handheld multimeter to measure the current I_C between pins 1 and 2 of the jumper *JP8*.

(i) Again connect R_{41} via jumper *JP4*.

(j) Again connect the two measurement points *MP48* and *MP46* to the differential amplifier to measure the voltage drop across R_{41}.

(k) Set the measurement point *MP30* (V_{DD}) to 5 V and measure the applied voltage.

(l) Apply a dc voltage at measurement point *MP49*, increasing in value stepwise from 0 V to 2 V in 0.1 V steps, thereby increasing I_B. Tune your input voltage using the *Waveforms* software until you measure the defined voltage. This procedure applies to any DC input voltage from the signal generator.

(m) Measure I_C as a function of I_B.

(n) Calculate β as a function of I_B.

(o) Plot I_C and β as a function of I_B on a single diagram.

$\mathbf{I_C/V_{CE}}$ characteristic

(p) Short-circuit *pin 1* and *pin 2* of the jumper *JP8*.

(q) Again connect R_{41} via jumper *JP4*.

(r) Connect the measurement point *MP30* and pin 3 of the jumper *JP8* to the differential amplifier to measure the voltage drop across R_{31} (you can calculate the current I_C based on this value).

(s) Apply a voltage ramp (Triangle, 0 V $\rightarrow$ 10 V, 50 %, 50 Hz) to the measurement point *MP30* (V_{DD}).

(t) Apply a dc voltage at measurement point *MP49*, increasing in value stepwise from 0 V to 2 V in 0.5 V steps, thereby increasing I_B.

(u) Measure V_{CE} directly at the measurement point *MP34*.

(v) Measure and plot I_C versus V_{CE} for the different I_B values. Use your results from the previous experiment for the labeling of I_B.

(w) Determine the Early voltage for each trace and compare the values.

2.3 Common emitter amplifier

Based on the bipolar we characterized in the first section, we will now simulate and experimentally characterize a bipolar common-emitter amplifier.

2.3.1 Simulation

Consider the common-emitter amplifier shown in Figure 2.3.1. We will model the amplifier twice, once in time domain and once in the frequency domain, and compare the results.

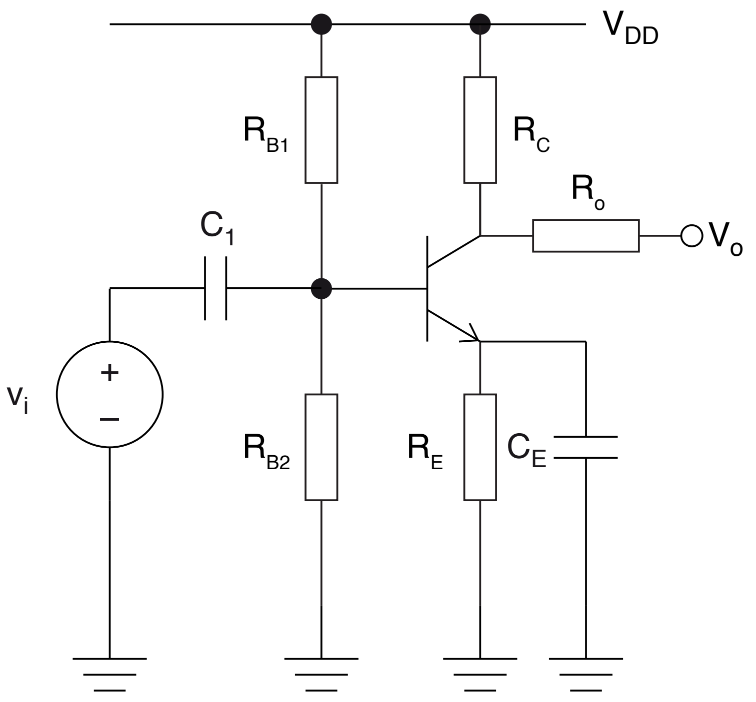

Figure 2.3.1

(a) Set up a SPICE implementation of the circuit in Figure 2.3.1, using the following component values found on the electronics board: R_{35}, R_{42}, R_{37}, R_{52}, C_{14}, and C_{16} (see Figure 2.3.2 below). The transistor model is already included in LTSpice.

(b) For R_E, set a parameter sweep between the values of R_{48} and R_{51} using the `.step` directive.

(c) Use a supply voltage of $V_{DD}(= V_{MP38}) = 10\,\text{V}$.

(d) Add an RC high pass filter to the output after $R_{52}(= R_o)$ with $C_{HP} = 120\,\mu\text{F}$ and $R_{HP} = 10\,\text{k}\Omega$. From there, measure the output signal V_o.

(e) For the time domain simulation, set the input signal V_i as a sine wave with an amplitude of $15\,\text{mV}$ and a frequency of $10\,\text{kHz}$.

(f) Using the `.tran` directive, record the input and output signals for this circuit over 500 µs. From your data, determine the gain (in dB) and phase shift between input and output for both emitter resistor values.

(g) For the frequency domain simulation, replace the `.tran` directive with `.ac`, defining a frequency range from 1 kHz to 100 MHz. Plot both amplitude and phase, again for both emitter resistor values.

(h) Compare the four simulation results for gain and phase shift for a frequency of 10 kHz.

2.3.2 Measurement

In the "BIPOLAR" section of the electronics board, we will use the *BJT amplifier* circuit, shown in Figure 2.3.2.

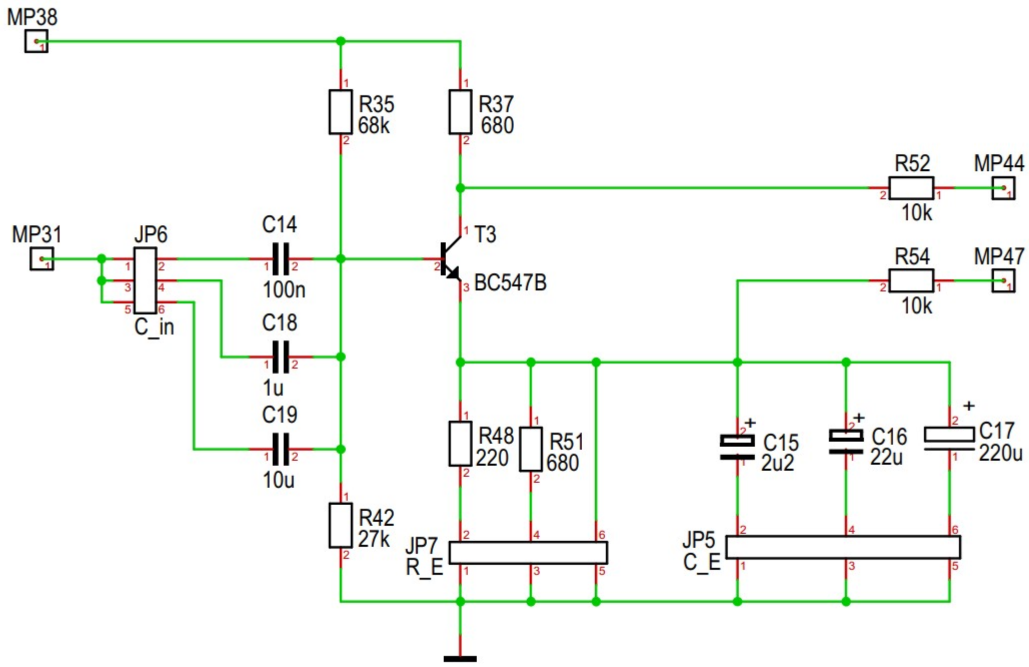

Figure 2.3.2: Schematic of the *BJT amplifier* circuit

(a) Configure the amplifier using *C14*, *C16* and *R48*.

(b) Set the measurement point *MP38* to 10 V and measure the actual applied voltage.

(c) Measure the output via the measurement point *MP44*. Set the switch on the oscilloscope to *AC* to add a high-pass filter to the input and thereby filter out the offset.

(d) Apply a sine wave (Sine, ±15 mV, 50 %, 10 kHz) to the measurement point *MP31* and measure it directly (switch the oscilloscope to *AC*).

(e) Measure the input and output characteristics; save your data.

(f) Measure the time offset of both signals by means of two X-cursors and calculate the phase shift from these values.

(g) Let WaveForms display the phase difference of both signals (*Waveforms* / *Scope* / *Measurements* / *Add* / *Custom Global* / *Add*). The default setting calculates phase difference.

(h) Using your measured characteristics, calculate the gain (in dB) for the given system.

(i) Perform a spectral analysis. To do this, open the *Spectrum* menu in *WaveForms*. Make a *Single* recording of both inputs. Measure both peaks at 10 kHz using an X-cursor. From this, calculate the gain (in dB).

(j) Now perform a small signal analysis. To do this, open the *Network* menu in *WaveForms*. In the header, set the frequencies from 1 Hz to 100 kHz (at 16 samples). Set 15 mV as amplitude on the right. Perform a *single* recording of the output signal.

(k) From this measurement, determine the gain (in dB) and phase shift at 10 kHz and compare with your spectral analysis and simulation results.

(l) Repeat the tasks a – k using the emitter resistor R_{51}.

(m) Plot the small signal analysis characteristics (gain in dB and phase) of both emitter resistors in a single plot.

2.4 Current source

We will now analyze, design and characterize a bipolar-based current source.

2.4.1 Simulation

Consider the simple bipolar current source configuration shown in Figure 2.4.1.

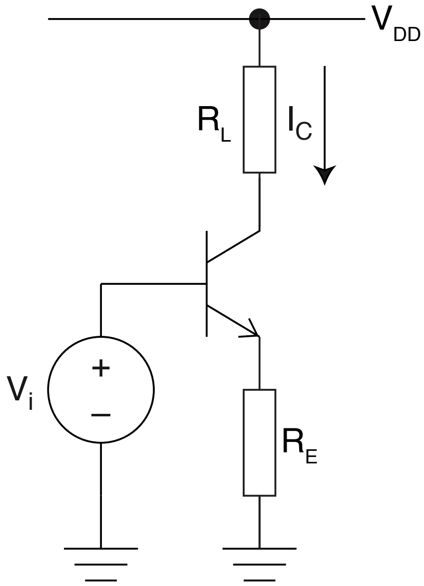

Figure 2.4.1

(a) Set up a SPICE simulation of this circuit using the bipolar transistor *T4* employed on the electronics board.

(b) Set $V_{DD} = 10\,\mathrm{V}$.

(c) Add the SPICE directive `.op` .

(d) Perform a parameter sweep for the load resistor R_L (again using the `.step` directive) from $1\,\mu\Omega$ to $10\,\mathrm{k}\Omega$ with ten sample points per decade.

(e) Run the simulation for input voltages $V_{in1} = 2.5\,\mathrm{V}$ and $V_{in2} = 5\,\mathrm{V}$ and emitter resistances $R_{E1} = 100\,\Omega$ and $R_{E2} = 200\,\Omega$ (thus four simulations).

(f) Measure and plot I_C as a function of R_L for all combinations of V_{in} and R_E in a graph with a logarithmic abscissa.

(g) Determine the maximum current $I_{C(max)}$ of the current source for each case.

(h) Also determine in each case the maximum load ($R_{L(max)}$) at a maximum deviation of $5\,\%$ from $I_{C(max)}$.

(i) Analyze and interpret any linear or non-linear characteristics which you determine for all parameters.

2.4.2 Measurement

In the "BIPOLAR" section of the electronics board, we will use the *BJT current source* circuit, shown in Figure 2.4.2, as well as the *diode characteristics* circuit (Figure 2.4.3) and *LED characteristics* circuit (Figure 2.4.4) from the "DIODES" section.

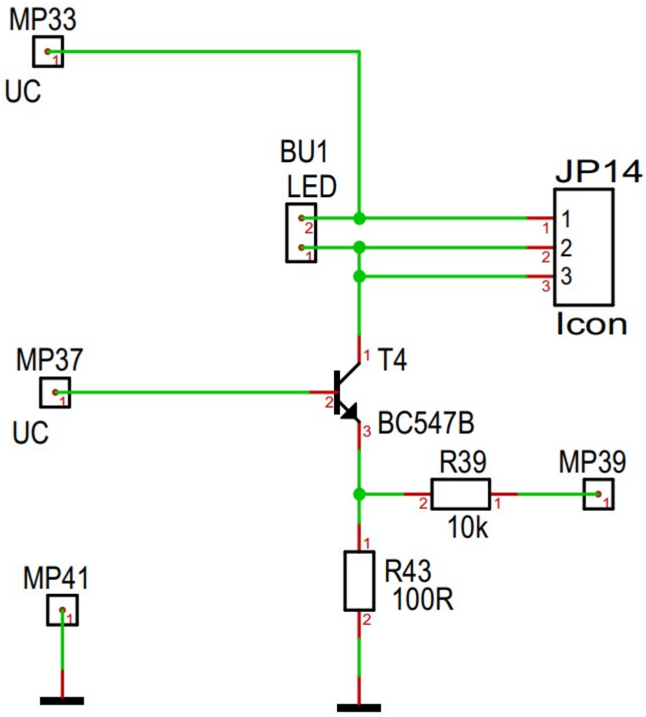

Figure 2.4.2: Schematic of the *BJT current source* circuit

(a) Set the measurement point *MP33* to 10 V and measure the actual applied voltage.

(b) Connect the handheld multimeter to measure the current between *pin 1* and *pin 2* of the jumper *JP14*.

(c) Successively connect DC voltages of $V_{in1} = 2.5\,\text{V}$ and then $V_{in2} = 5\,\text{V}$ to the measurement point *MP37*.

(d) Measure the current I_C as a function of the input voltage; save your data.

(e) Now connect the handheld multimeter to measure the current between *Pin 1* of the jumper *JP14* and *MP23*. Also connect *Pin 3* of the jumper *JP14* to *MP26*.

(f) Connect a DC voltage of $V_{in3} = 3.9\,\text{V}$ to the measurement point *MP37*.

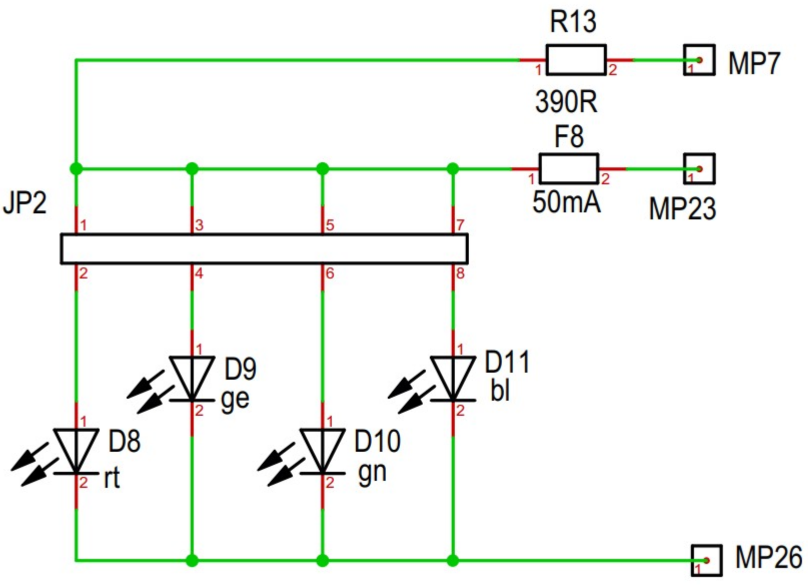

Figure 2.4.3: Schematic of the *diode characteristics* circuit

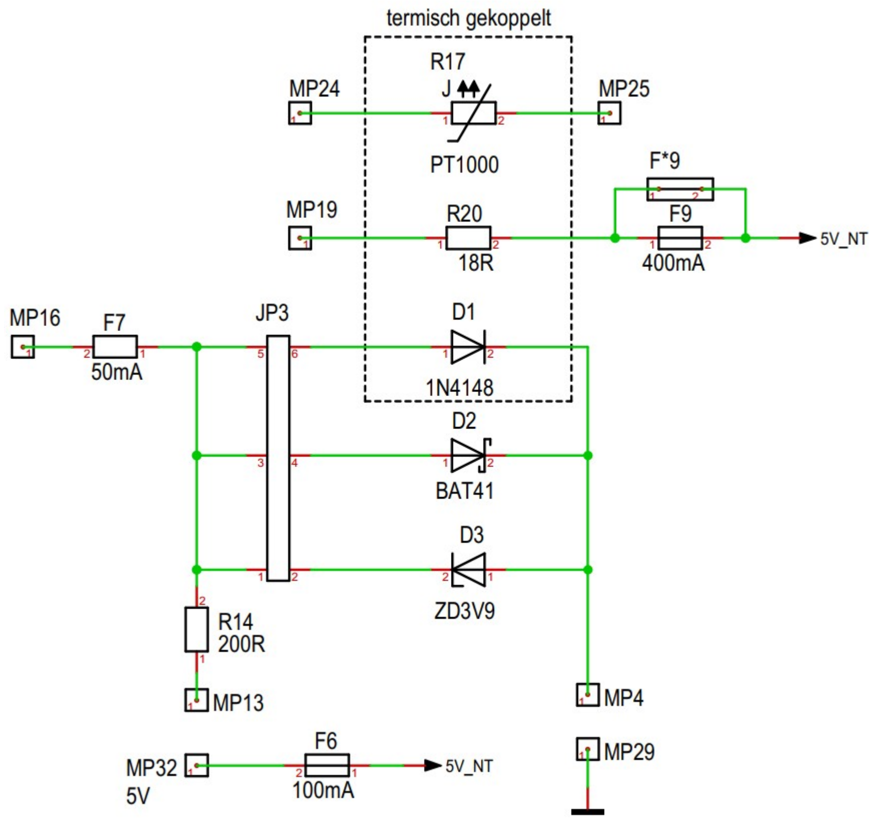

Figure 2.4.4: Schematic of the *LED characteristics* circuit

(g) Connect all the LEDs one at a time to the current source using jumper *JP2* and determine I_C in each case. Observe the light-emission behavior of the LEDs and note down your (subjective) observations.

(h) Remove the input voltage from *MP37*. In its place, connect *MP13* to *MP33*, *MP16* to *MP37* and *MP4* to *GND*. Also connect the Z-diode to the current source using jumper *JP3*.

(i) Again connect all the LEDs one at a time to the current source using jumper *JP2* and determine I_C in each case. Again observe the light-emission behavior of the LEDs and note down your (subjective) observations.