# ELEN30013 Workshop 03: Bipolar junction transistor

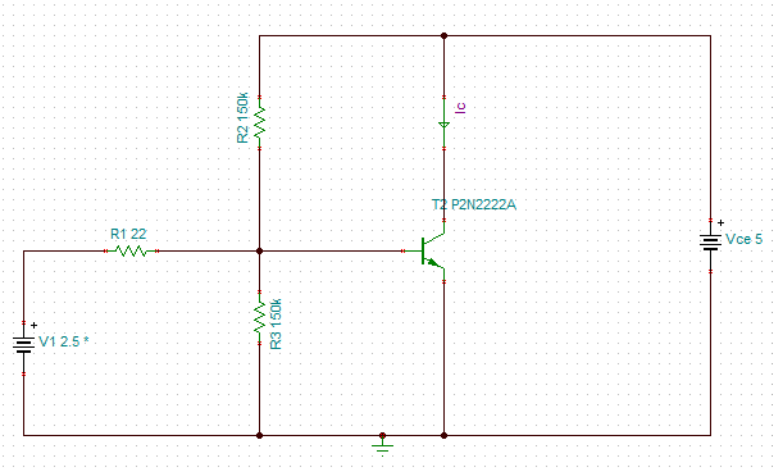
- For demonstrators and students: This workshop will be assessed by the submitted report and demonstration during the workshop.
- Submit report in PDF file by Sunday midnight.
- Attendance (Max 20 points)

## 1. Bipolar Junction Transistor (BJT) Characteristics

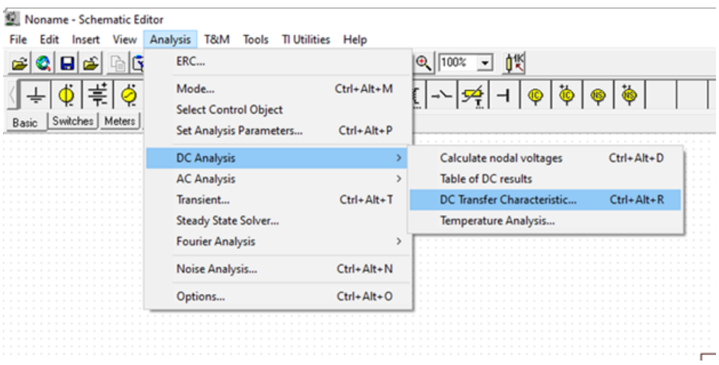
### 1.1 TINA TI Simulation

- **Step 1:** TINA-TI provides all the conventional DC, transient and frequency domain analysis of SPICE and much more. Please download from the link here: https://www.ti.com/tool/TINA-TI

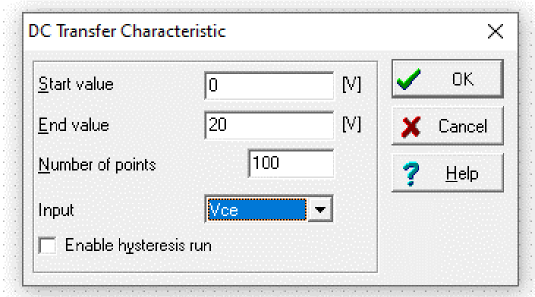
  Construct the following circuit in TINA TI, where the Ic is the amp meter, and the BJT should be P2N2222A. The example circuit is shown in the figure below.

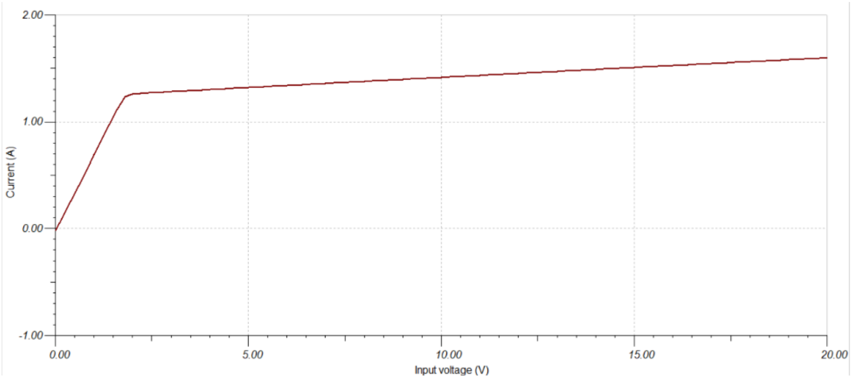



- **Step 2:** We would like to sweep Vce and measure the collector current with amp meter Ic. Go to the Analysis tab and select DC Transfer Characteristic as shown in the figure below.

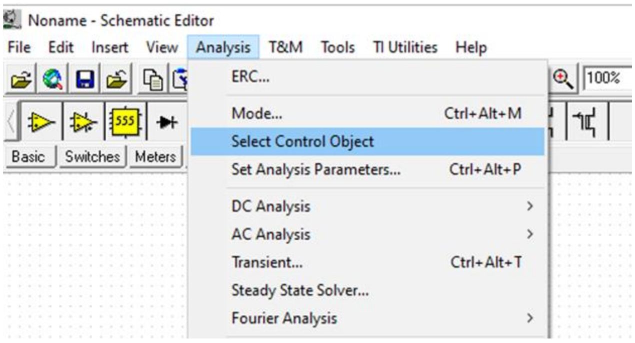


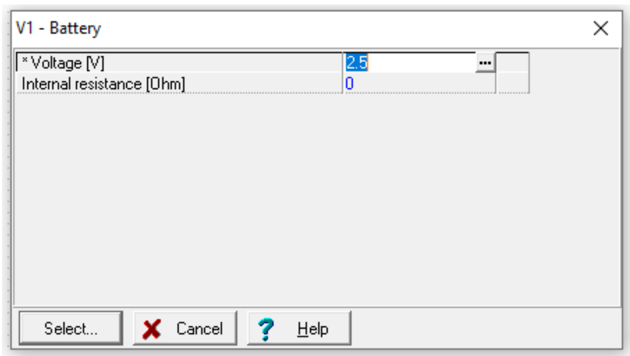
- **Step 3:** The following message box in the figure below should show up. I have called my dc voltage source Vce, so that is the one I want to select. Once you press "OK" the simulation will run, and you should be able to get a single Ic vs Vce curve, as shown in figure below.

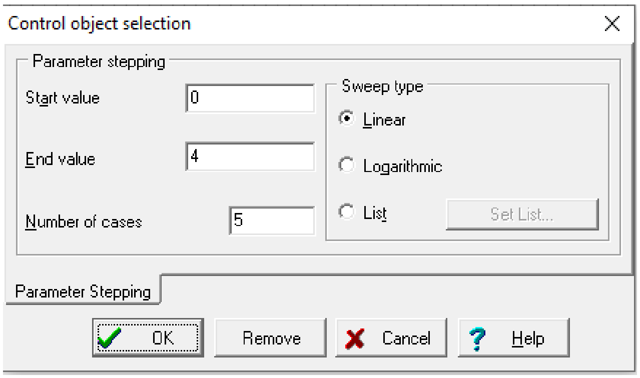


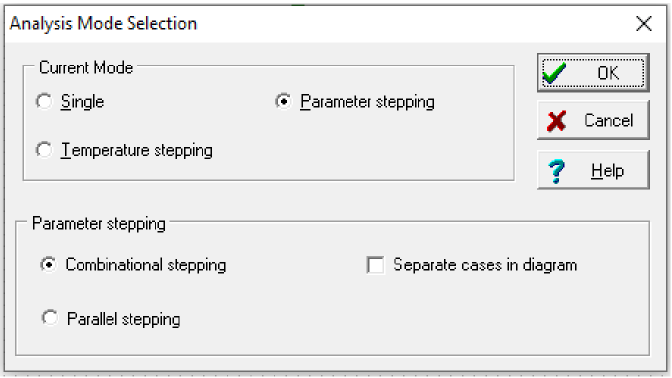

- **Step 4:** Now go to the analysis tab once more but this time select the "Select Control Object" menu option. Your mouse pointer should change to a resistor looking icon. Go ahead and select battery V1 and the following message box in the figure below should show up. Press selects and you should see the following set up message box. In the Control object selection message box, you can set up the start value, end value and number of cases.






- **Step 5:** Once that is set up, you can go to the analysis tab once more, but this time select Mode from the menu. A message box displaying the current analysis mode should show up, and it should have "Parameter stepping" selected as the simulation mode.

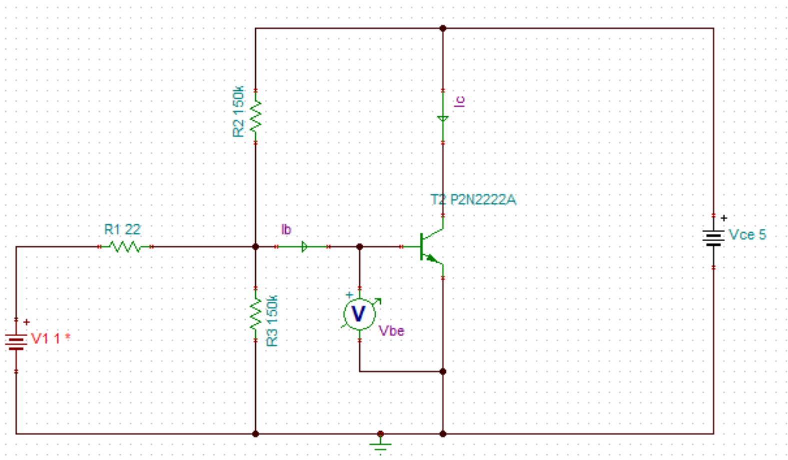



- **Step 6:** At this point the parametric sweep is set up and ready to go. Now if you go back to steps 1 and 2 and run a DC sweep you should be able to see Ic vs Vce vs V1.

## 1.2 Questions

- **Question 1.1:** Capture the output plot in step 5, identify the linear (active) region, the cut-off region, the saturation region. **(15 points)**

- **Question 1.2:** Once you figure out the linear (active) region in question 1, take any 5 data points in the linear region (Vce, V1). And change your simulation circuit to the corresponding voltage, for example, in figure below, I pick (Vce=5V, V1=1V). Then click Analysis, DC analysis, Table of DC results, then you could get the results of Ib, Ic and Vbe under different bias conditions. Fill in those measurement values in the table **(10 points).** Then calculate the current gain of the transistor **(5 points)**.




| Ib | Ic | Vbe | ß |
|---|---|---|---|
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |

- **Question 1.3:** Compare the results you simulated with the datasheet. Do the measurement and calculation correspond to the datasheet recommend value? Give some comments. **(5 points)**

  *datasheet: https://www.onsemi.com/pdf/datasheet/pn2222a-d.pdf

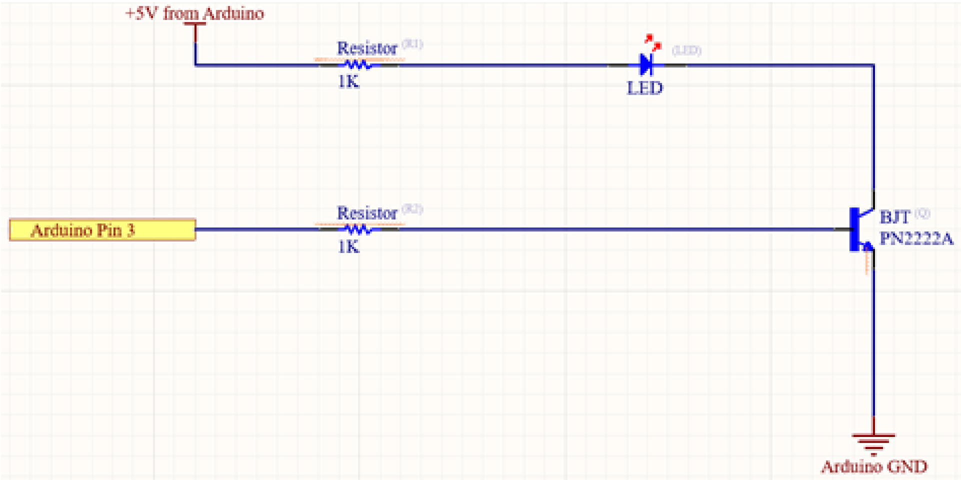
## 2. BJT Switch Circuit

### 2.1Simple Switch Circuit Construction

- **Step 1:** Prepare the component in the list below:

  LED×1; 2N2222A BJT×1; Arduino; 1k ohms Resistor ×2

- **Step 2:** Construct the circuit in the breadboard as shown in the figure below:




- **Step 3:** Run and build the Arduino code below:

```
const int transistorPin = 3;

void setup()
{
  pinMode (transistorPin, OUTPUT);
}

void loop()
{
  digitalWrite (transistorPin, HIGH);
  delay(1000);
  digitalWrite (transistorPin, LOW);
  delay(1000);
}
```

- **Step 4:** **(10 points)** Show the circuit (LED blinking) to demonstrator to check marked.

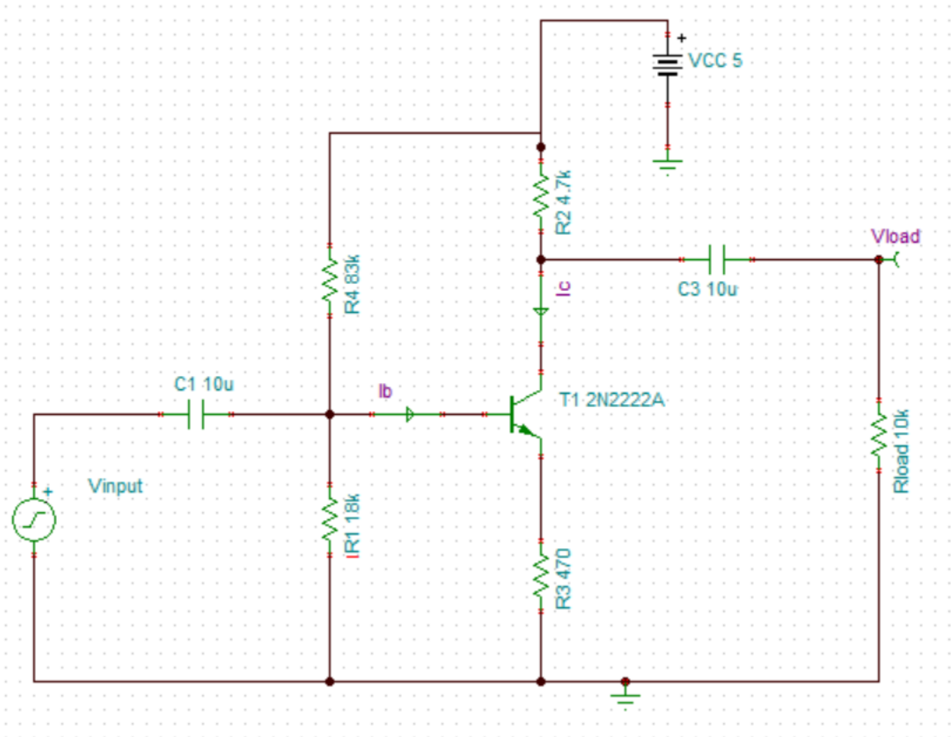
## 2.2 Questions

- **Question 2.1:** Please identify which region is the transistor working at when the LED (Light Emitting Diode) is on? And off? **(5 points)**
- **Question 2.2:** Calculate the blinking frequency. **(5 points)**
- **Question 2.3:** Modify the code, change the blinking frequency to 50 Hz. Include and submit your Arduino code. **(5 points)**

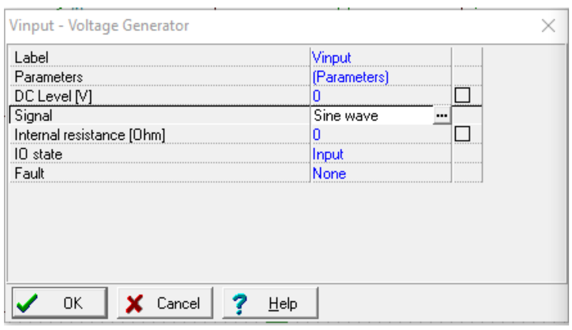
# 3. BJT Amplifier

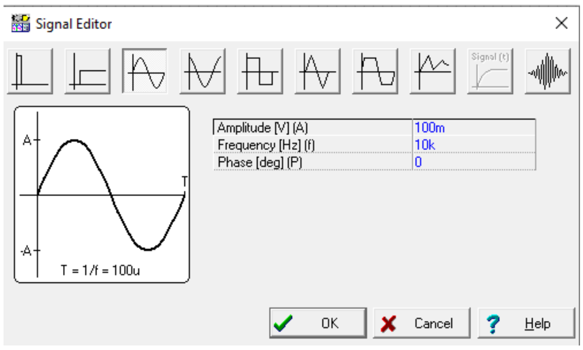
## 3.1 Simulation in TINA TI

- **Step 1:** Connect and build the circuit in TINA TI as shown in figure below:

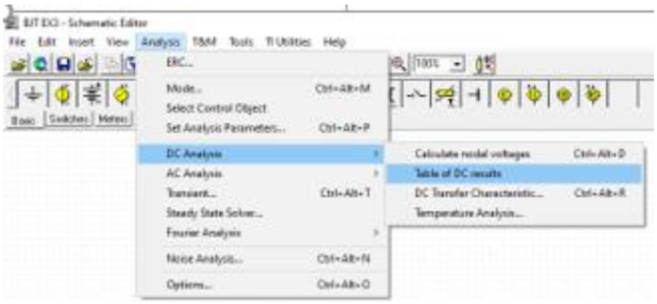



- **Step 2:** Right click Vinput -> Properties, and it will pop out a box as shown in figure below, change the input voltage properties to be the same with the figure. In “Signal” settings, change it to a 100m Amplitude, 10k HZ frequency Sine wave.

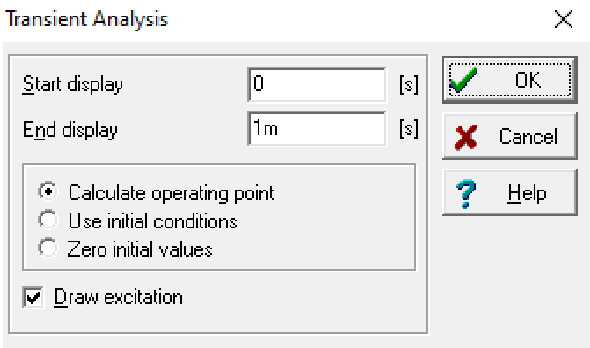


- **Step 3:** Go Analysis -> DC Analysis -> Table of DC results. Take a screenshot of the result. And answer question 3.1.




- **Step 4**: Go Analysis ->AC Analysis -> Table of AC results. Take a screenshot of the result. And answer question 3.2.
- **Step 5:** Go Analysis -> AC Analysis -> Transient. And change the settings as figure indicated below, save the screenshot of the result.

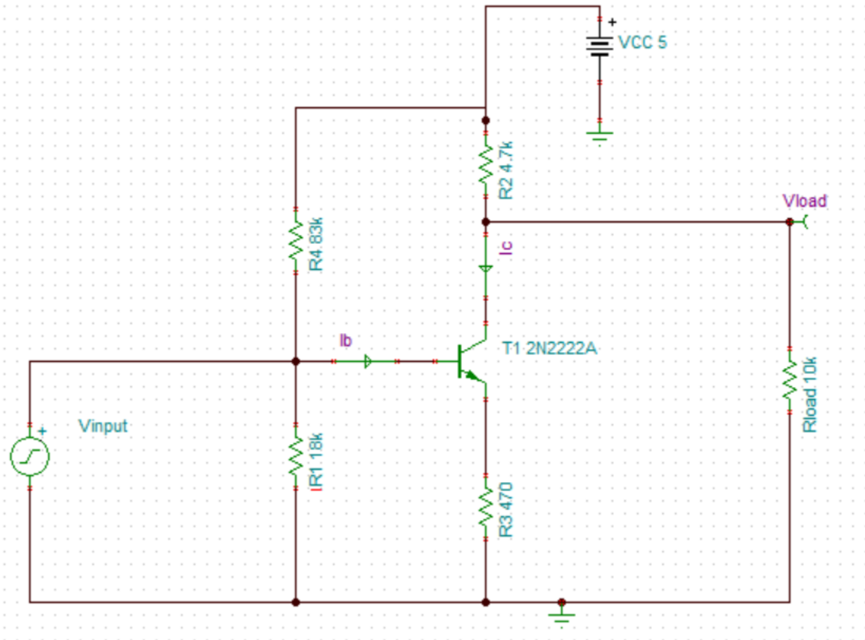



## 3.2 Construct Circuit on Breadboard

AD2 start guide: https://reference.digilentinc.com/test-and-measurement/analog-discovery-2/start?redirect=1

Waveforms Reference Manual: https://reference.digilentinc.com/software/waveforms/waveforms-3/reference-manual

- **Step 1:** Build the circuit on a breadboard as follows, choose the resistance value as close as possible.




- **Step 2:** Give the same input signal: Sine wave, 100 mV amplitude, 10k Hz frequency.
- **Step 3:** Use AD2 channel 1 and channel 2 measure the input and output waveform respectively, take a screenshot of the result and answer Question 3.4.

## 3.3 Questions

- **Question 3.1:** Identify the BJT operation region under this bias point. **(5 point)**

Voltages/Currents

| | |
|---|---|
| I_R1[1,0] | 0A |
| I_R2[3,Vload] | 340.14uA |
| I_R3[4,0] | 3.55pA |
| I_R4[3,1] | 60.24uA |
| I_Rload[Vload,0] | 340.14uA |
| I_Vinput[1,0] | 60.24uA |
| Ib | -1.03pA |
| Ic | 4.59pA |
| V_Ib[1,5] | 0V |
| V_Ic[Vload,6] | 0V |
| V_R1[1,0] | 0V |
| V_R2[3,Vload] | 1.6V |
| V_R3[4,0] | 1.67nV |
| V_R4[3,1] | 5V |
| V_Rload[Vload,0] | 3.4V |
| V_VCC[3,0] | 5V |
| V_Vinput[1,0] | 0V |
| Vload | 3.4V |
| VP_1 | 0V |
| VP_3 | 5V |
| VP_4 | 1.67nV |
| VP_5 | 0V |
| VP_6 | 3.4V |
| VP_Vload | 3.4V |

Show
Nodal Voltages Currents
Other Voltages Outputs
Cancel Help

- **Question 3.2:** Calculate the AC gain. **(5 points)**

- **Question 3.3:** Explain why we need the capacitor C1 and C2? (Hint. Compare the differences between simulation and breadboard). **(5 points)**

- **Question 3.4:** Calculate the breadboard circuit AC gain and compare it with the simulation result. Give some comments. **(5 points)**