ELECTRONICS 365PRACTICAL 2 – FEEDBACK CIRCUITS

31 August – 4September 2020

INTRODUCTION:

Practical 2 is due at midnight (24:00) on Friday 4th September. This practical includes botha spice submission and a practical report.These two submissions must be called 12345678.cir and 12345678.pdf respectively (replacing 12345678 with your student number). The spice submission counts for two marks and the practical report counts for eight marks. The theoretical calculations may be handwritten; however, they must be neat and logically laid out. All other parts of the report must be typed.

For this practical you will need to use Ngspice version 3.2 which can be downloaded fromSUNLearn. The necessary spicefiles provided for this practical are:

- models.cir – contains5 different BJT transistor models.
- Prac2Test1.cir – Testbench for MODEL 1.
- Prac2Test2.cir – Testbench for MODEL 2.
- Prac2Test3.cir – Testbench for MODEL 3.
- Prac2Test4.cir – Testbench for MODEL 4.
- Prac2Test5.cir – Testbench for MODEL 5.
- Feedback_ex.cir –Template forthe spice submission.

AIMS:

This practical is divided into two sections. The first section involves analysis and simulation of a feedback circuit in both open and closed loop configurations.The second section involves the design and simulation of feedback circuitry to meet certain closed-loop voltagegain and closed-loop input resistance requirements.

As in practical 1, there are slightly different variations of this practical depending on your student number. Ensure that you carefully read the practical brief and use the correct values, models and testbench.

SECTION A:

This section of the practical concerns Figures 1 and 2 below. There is no spice submission related to this section and all relevant work should be included in the practical report.

Figure 1 is an open-loop amplifier circuit and figure 2 is that same circuit with feedback resistor R_F and capacitor C_F included. Use Tables 1 and 2 to determine which resistor values you should use for your circuit according to your student number.

EXAMPLE: student number 12345678 must use resistor values from the “SET 1” column in Table 2.

Table 1: Section A Value Assignment

STUDENT NO. ENDING IN	CIRCUIT VALUES
0 OR 6 OR 8	SET 1
1 OR 7	SET 2
2 OR 3	SET 3
4 OR 9	SET 4
5	SET 5

217 50793

Table 2: Circuit Values for Section A

CIRCUIT VALUES	SET 1	SET 2	SET 3	SET 4	SET 5
R1 (kΩ)	75	35	130	50	100
R2(kΩ)	24.5	12	50	21	60
R_{C1}(kΩ)	5	2	5	3	4
R_{C2}(kΩ)	7.5	1	8	4	6.5
R_{E1}(kΩ)	1.5	0.8	1.8	1.7	2
R_{E2}(kΩ)	9	5	7	6.5	8
R_F(kΩ)	2	0.6	2.5	1.4	1.2
R_S(kΩ)	0.75				
R_L(kΩ)	1				
h_{fe}	150				
$V_{BE(on)}$(V)	0.7				
V_A	∞				

All theoretical calculations must be presented in full in the practical report. For any spice simulation work in Section A use this BJT model:

.model BJT1 NPN (BF=150 VAF=1000 IS=1e-15)

NOTE: DO NOT USE THIS MODEL IN SECTION B!!!

SECTION A TASKS:

a) Determine the voltage gain $A_V = V_o/V_S$ of the circuit in Figure 1

b) Determine the input resistance R_{io} of the circuit in Figure 1

c) Simulate the circuit in Figure 1 in spice and compare the voltage gain and input resistance values obtained with those in parts a and b. Include the spice code used to do this in an appendix in your report and any plots necessary to discuss the results.

d) What feedback topology is used in Figure 2 and How did you identify it? Which components form the β-circuit?

e) Determine the closed-loop voltage gain A_{vf} of the circuit in Figure 2

f) Determine the closed-loop input resistance R_{if} of the circuit in Figure 2

g) Simulate the circuit in Figure 2 in spice and compare the voltage gain and input resistance values obtained with those in parts d) and e). Include the spice code used to do this in an appendix in your reportand any plots necessary to discuss the results.

h) Discuss the results obtained in parts a)-f) above. What effect did the feedback have on the input resistance and gain of the circuit? Why would we want to use feedback in a circuit such as this?

There are 6 marks available for section A. Four marks for the theoretical calculations and two marks for the discussion and comparison of the results obtained.

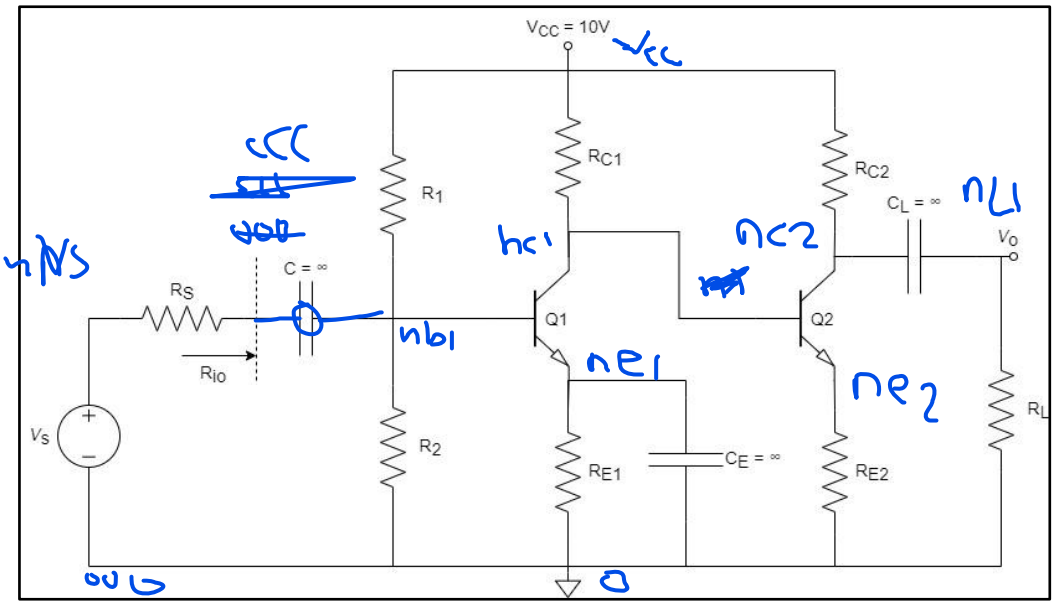


Figure 1: Open-Loop circuit

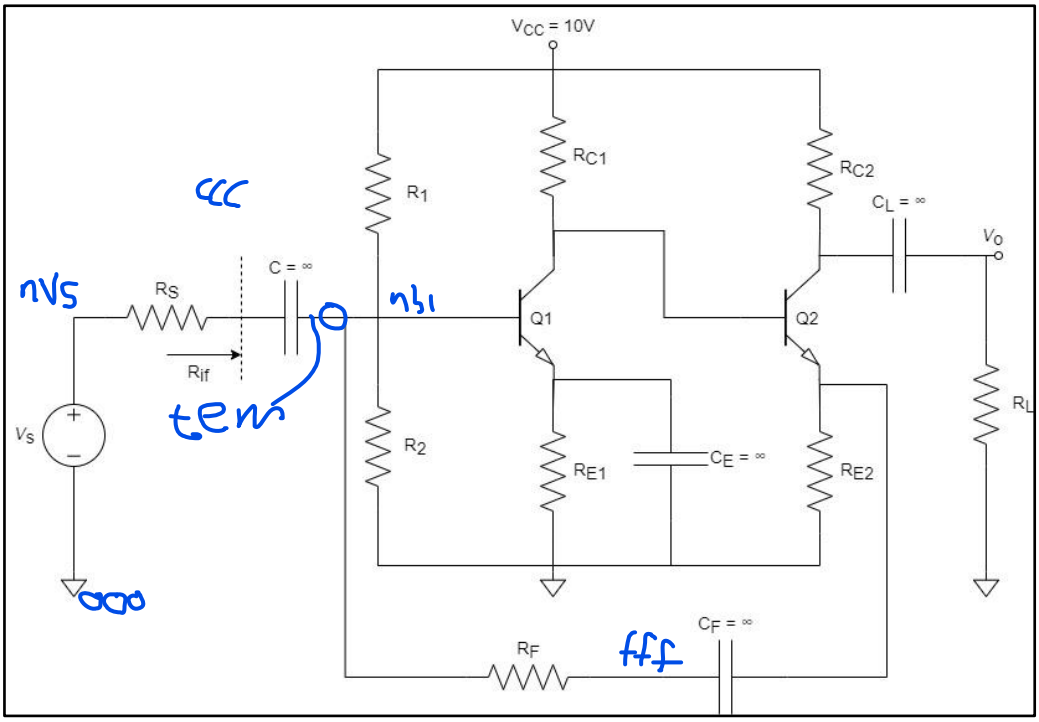


Figure 2: Closed Loop circuit (With Feedback)

SECTION B:

There are five variations of thissection with different closed-loop voltage gain and input resistance requirements. Each variation requires the use of a different model from the models.cir file and has a different testbench.You may only use the transistor model assigned to you.Use Table 3 below to determine which values, model and testbench you should use according to your student number.

EXAMPLE: 87654321 uses MODEL 2, Prac2Test2.cir and designs for A_{vf} = and R_{if} = Ω.

Table 3: Section B Requirements

STUDENT NO. ENDING IN	MODEL	TESTBENCH	A_{vf}	R_{if} (Ω)
0 OR 6 OR 8	MODEL 1	Prac2Test1.cir	$20.0 < A_{vf} < 20.1$	$37.2 < R_{if} < 37.6$
1 OR 7	MODEL 2	Prac2Test2.cir	$18.3 < A_{vf} < 18.4$	$32.5 < R_{if} < 32.85$
2 OR 3	MODEL 3	Prac2Test3.cir	$15.7 < A_{vf} < 15.8$	$24.5 < R_{if} < 24.87$
4 OR 9	MODEL 4	Prac2Test4.cir	$13.5 < A_{vf} < 13.6$	$17.7 < R_{if} < 18.1$
5	MODEL 5	Prac2Test5.cir	$11.8 < A_{vf} < 11.9$	$12.66 < R_{if} < 13.0$

All necessary information about the transistors is included in the .model statements in the models.cir file (no datasheets are required for the design of the circuit).You may assume that $V_A \rightarrow \infty$ for simplification purposes. The testbench provides a power supply of Vcc = 10 V to the circuit as well as any input voltage required to test the circuit. You may not include any voltage components in your spice submissions besides those specified in Figure 3 below.

SECTION B TASK:

For this section, you are required to design a feedback resistor R_f to achieve certain closed-loop voltage gain and input resistance requirements. The circuit used is the one in Figure 2 from Section A with the values from Table 4 below. You may not change any of the values given in this table. You will be required to submit a spice file containing the circuit with your chosen feedback resistor included. This spice file will be run through the specified testbench.

Table 4: Section B Circuit Values

CIRCUIT VALUES	
R1 (kΩ)	65
R2 (kΩ)	30
R_{C1} (kΩ)	5
R_{C2} (kΩ)	7
R_{E1} (kΩ)	2
R_{E2} (kΩ)	5
R_S (kΩ)	0.5
R_L (kΩ)	10
$V_{BE(on)}$ (V)	0.7

A template for the spice file submission is given on SUNLearn (Feedback_ex). Please use this template and replace only the highlighted sections with your own code:

```
*-------------------------------------------------------------------------------------------------------------------
* SPICE model for feedback amplifier
* ---------------------------------------
* Surname: Your surname
* First name: Your first name
* Student number: Your student number
* ---------------------------------------

.include models.cir

* Define feedback amplifier as a subcircuit
.subcktfeedamp in out nvcc
```

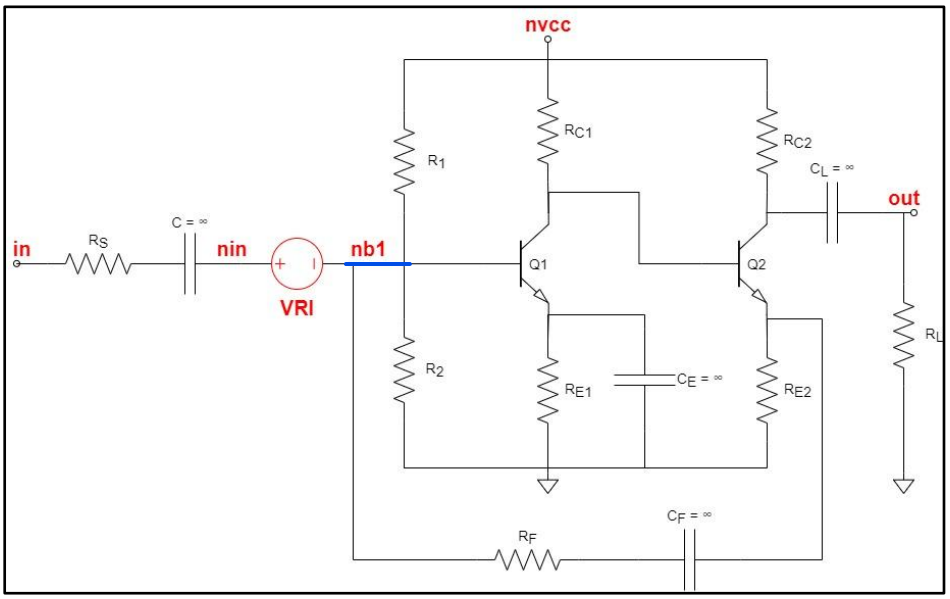
```
* -------- Start of student code --------

R1      nvcc   out   5k
Q1      out    nb1   0    MODEL1
VRI     nin    nb1   DC   0
C       in     nin   1

* --------- End of student code ---------
.ends
```

*----------

The testbench requires certain nodes in your circuit to have specific names. If the nodes are not named correctly, the testbench will fail to run. In addition, you need to include a voltage source VRI in your circuit to allow for measurement of the current into your circuit (**VRI nin nb1 DC 0**). These nodes and the required voltage source are shown in red in Figure 3. The testbench includes the line ".includeyourstudentnumber.cir". In order to test your circuit, replace "yourstudentnumber" with your student number (the name of your spice file) and run the testbench in Ngspice.



There are 4 marks available for section B. Two marks for the theoretical calculations and two marks from the testbench results. If no logical theoretical calculations for this section are included in thereport, you will receive 0 for the testbench results.

IMPORTANT NOTES:

- If your spice submission does not run through the testbench you will receive 0 for it.
- DO NOT INCLUDE ANY VOLTAGE SOURCES IN YOUR SUBMISSION. The required voltages will be supplied by the testbench. You will receive 0/5 for the spice submission if you include your own voltage sources.
- DO NOT INCLUDE THE .MODEL STATEMENT IN YOUR SUBMISSION. The necessary models areprovided in the separate models.cir file that is included in your submission with the line ".include models.cir". You will receive 0/5 for the spice submission if you include the .model statement.

- If the spice submission does not correspond with the report – specifically if differences between theoretical values and spice values aren't addressed – then the report mark will be penalized.
- PLEASE ENSURE YOU USE THE CORRECT CIRCUIT VALUES, MODEL AND TESTBENCH OR YOU WILL BE PENALISED.
- YOU MUST USE THE METHODS TAUGHT IN CLASS FOR THIS SECTION NOT THOSE USED IN NEAMEN.
- THIS PRACTICAL IS NOT OPTIONAL. If you do not submit a valid spice files you will receive 0 for the report as well.
- START EARLY.