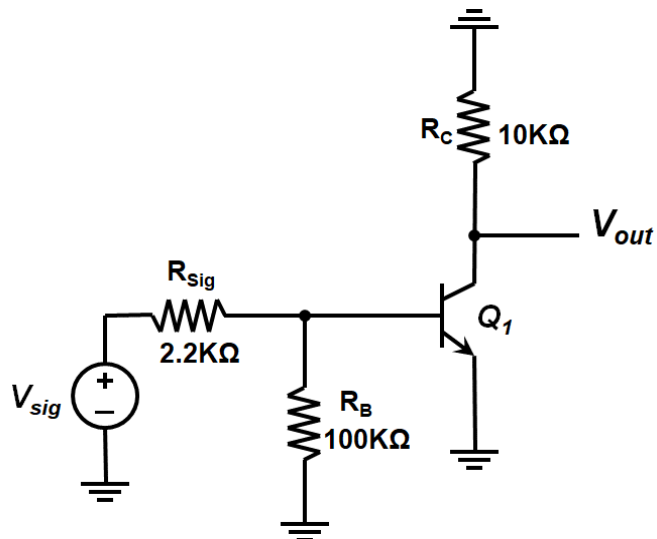


University of New Mexico
Department of Electrical and Computer Engineering

ECE 523/421- Analog Electronics (Fall 2025) Homework #9

Due in class on Monday Dec.1, 2025

1. The circuit below is a small signal equivalent circuit of a simple common emitter amplifier. Assume that $g_m=10\text{mA/V}$, $\beta=100$, $C_{\pi}=10\text{pF}$, and $C_{\mu}=2\text{pF}$ for the transistor.
- (a) Calculate the mid-band voltage gain (V_{out}/V_{sig}).
 - (b) Calculate the high frequency bandwidth, f_H .
 - (c) Use TSPICE to verify your answers in parts a and b.
 - (d) Calculate % of error between your hand calculations and SPICE simulations for bandwidth and gain.



Hint: Your SPICE file should look like the following:

* Homework 9 – Problem 1 SPICE file

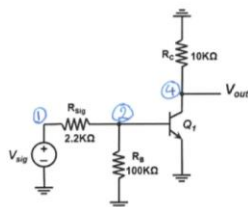
```
Rsig 1 2 2.2K
RB 2 0 100K
RC 4 0 10K
```

```
Vsig 1 0 AC 1V
```

```
XQ1 4 2 0 BJT gm=10m rp=10K
```

```
.subckt BJT C B E
Gm C E B E 'gm'
Rp B E 'rp'
Cp B E 10pF
Cm C B 2PF
.ends
```

```
.ac dec 1K 100 10MEG
.measure ac maxgain max vm(4)
.measure ac f3db when vm(4)='maxgain*0.7071'
.end
```



2. To improve the bandwidth of the circuit in problem 1, a degeneration resistor of $R_E=1K$ is used as shown in the new circuit diagram below. Use the same transistor parameters in problem 1.
- (a) Calculate the mid-band voltage gain (V_{out}/V_{sig}).
 - (b) Calculate the high frequency bandwidth, f_H .
 - (c) Use TSPICE to verify your answers in parts a and b.
 - (d) Calculate % of error between your hand calculations and SPICE simulations for bandwidth and gain.

Hint: For your SPICE simulations, use similar sub-circuit for Q1.

