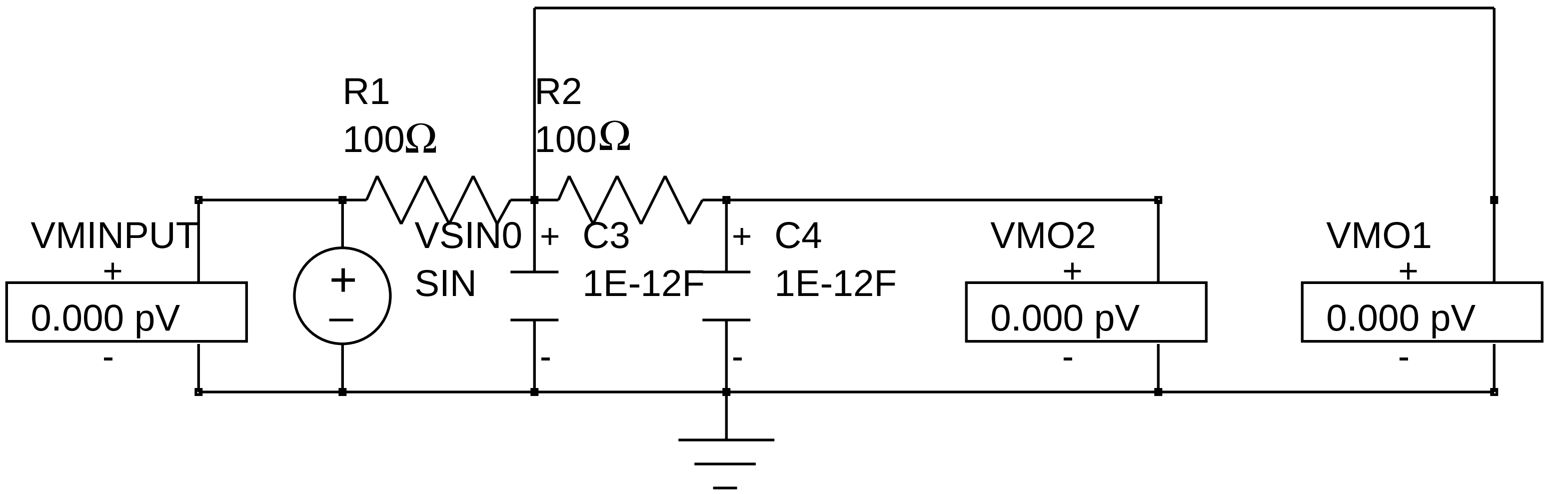
SPICE

- Circuit simulation tools (e.g. SPICE, APLAC) can also be used to generate frequency plots
- SPICE (Simulation Program with Integrated Circuit Emphasis), and its many derivatives (HSPICE, Spectre, P-SPICE, IS-SPICE, B^2SPICE...), is the most widely used.
- SPICE structure:
    - (Schematic Capture)
    - Input Parser
    - Simulation Engine:
        - Device modeling
        - Numerical solution of linear and non-linear systems of network equations
    - Output Formatting
    - (Graphical Post-processor)

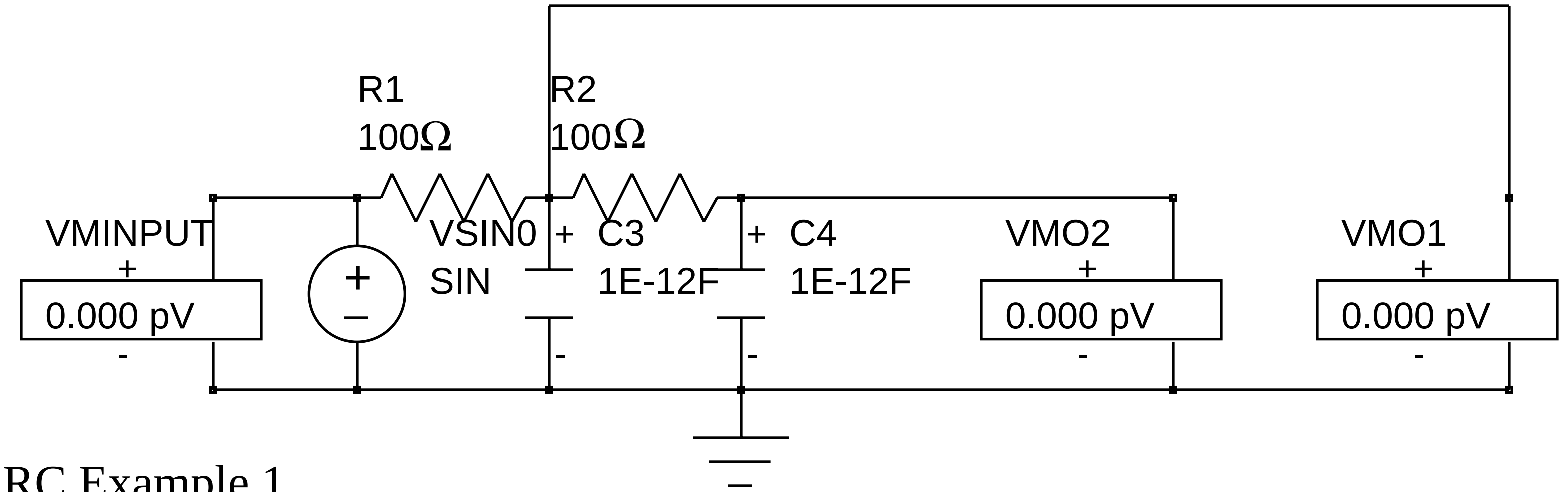
SPICE

- Here's an example from B^{2}SPICE from Beige Bag Software which also includes a schematic capture package



- An ac analysis can be used to generate frequency domain plots of magnitude and phase of the transfer function

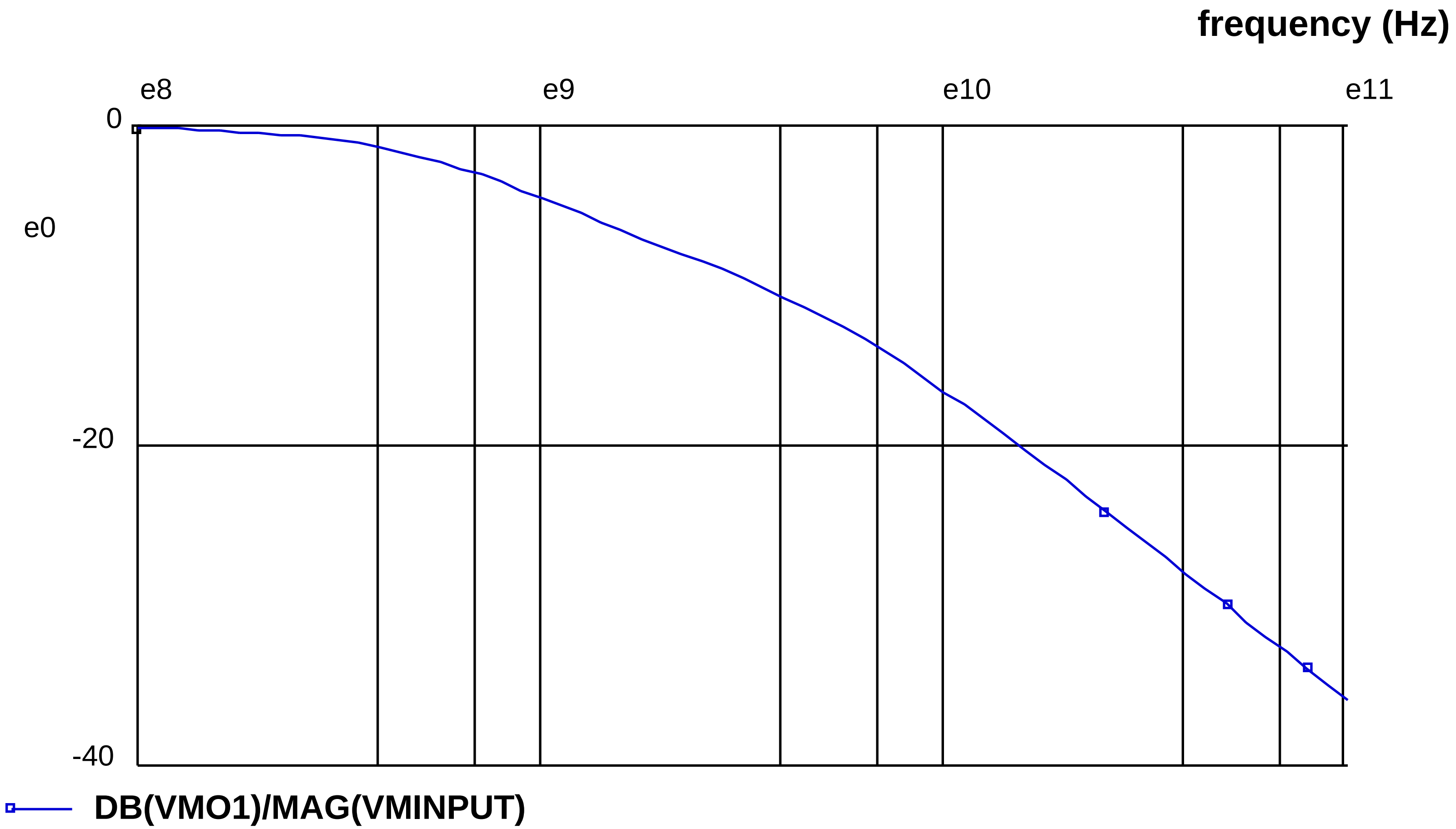
SPICE input file




```
RC Example 1
* B2 Spice default format (same as Berkeley Spice 3F format)
***** main circuit
R1 1  2  100
R2 2  3  100
C3 2  0  1e-12
C4 3  0  1e-12
Vin 1  0 SIN( 1 1 1meg 0 0) AC 100

.OPTIONS
.AC  dec  30  1e7  1e12
.PRINT AC LOG(V(2)) LOG(V(3))
.END
```

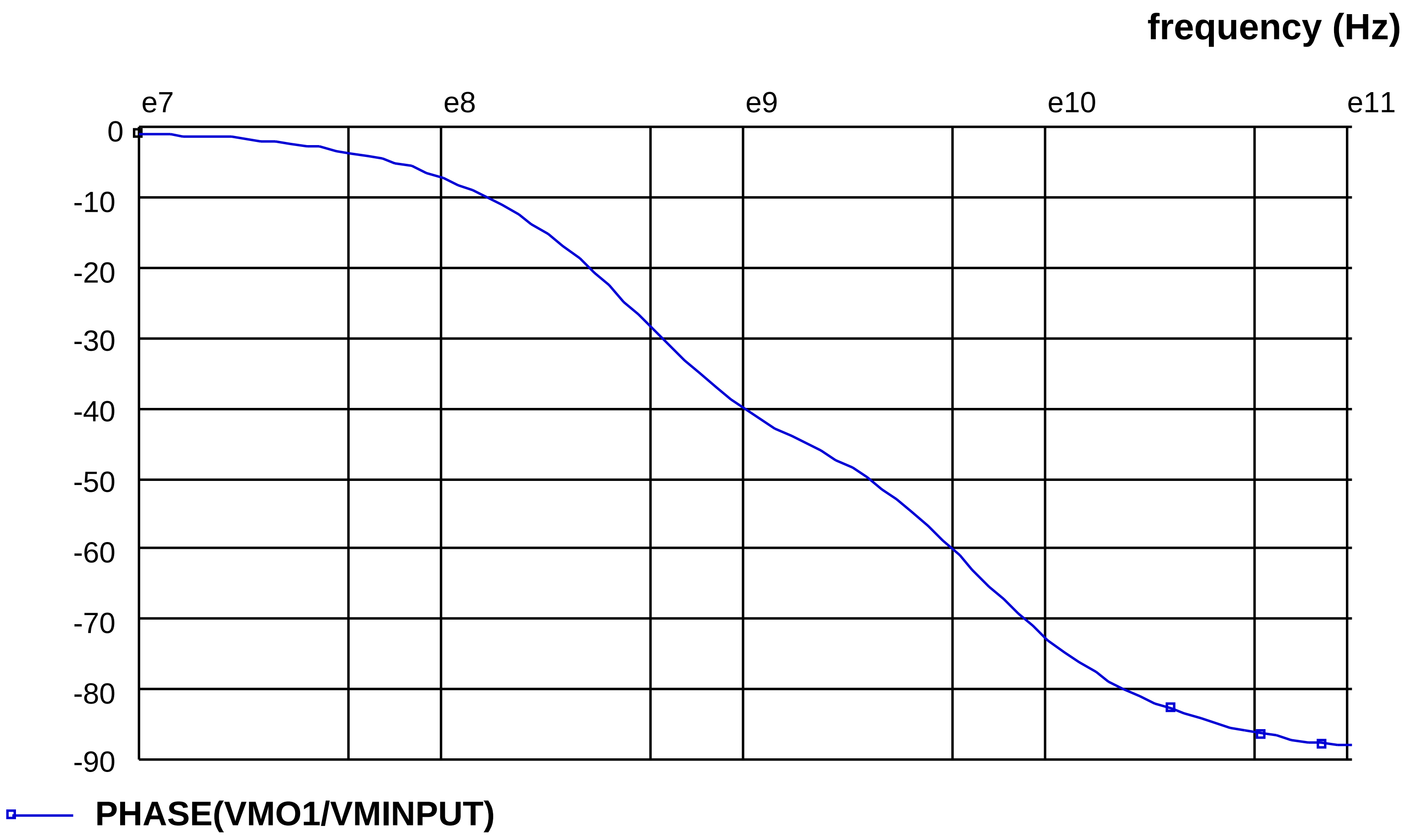
SPICE



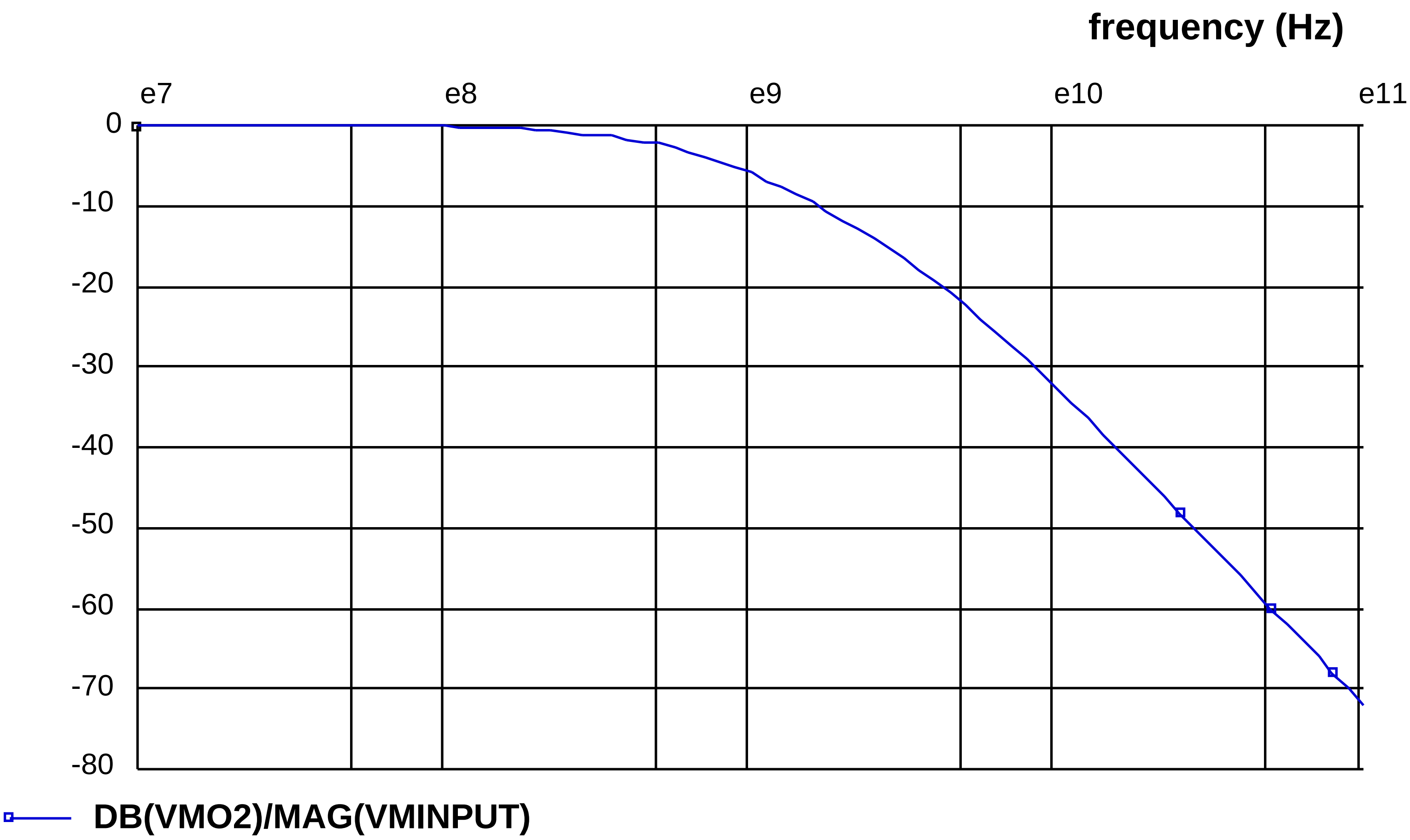
- Note that the frequency is in Hertz in SPICE, so it must be scaled by 2π for comparison with our Bode plot

SPICE

- The phase is plotted in degrees here

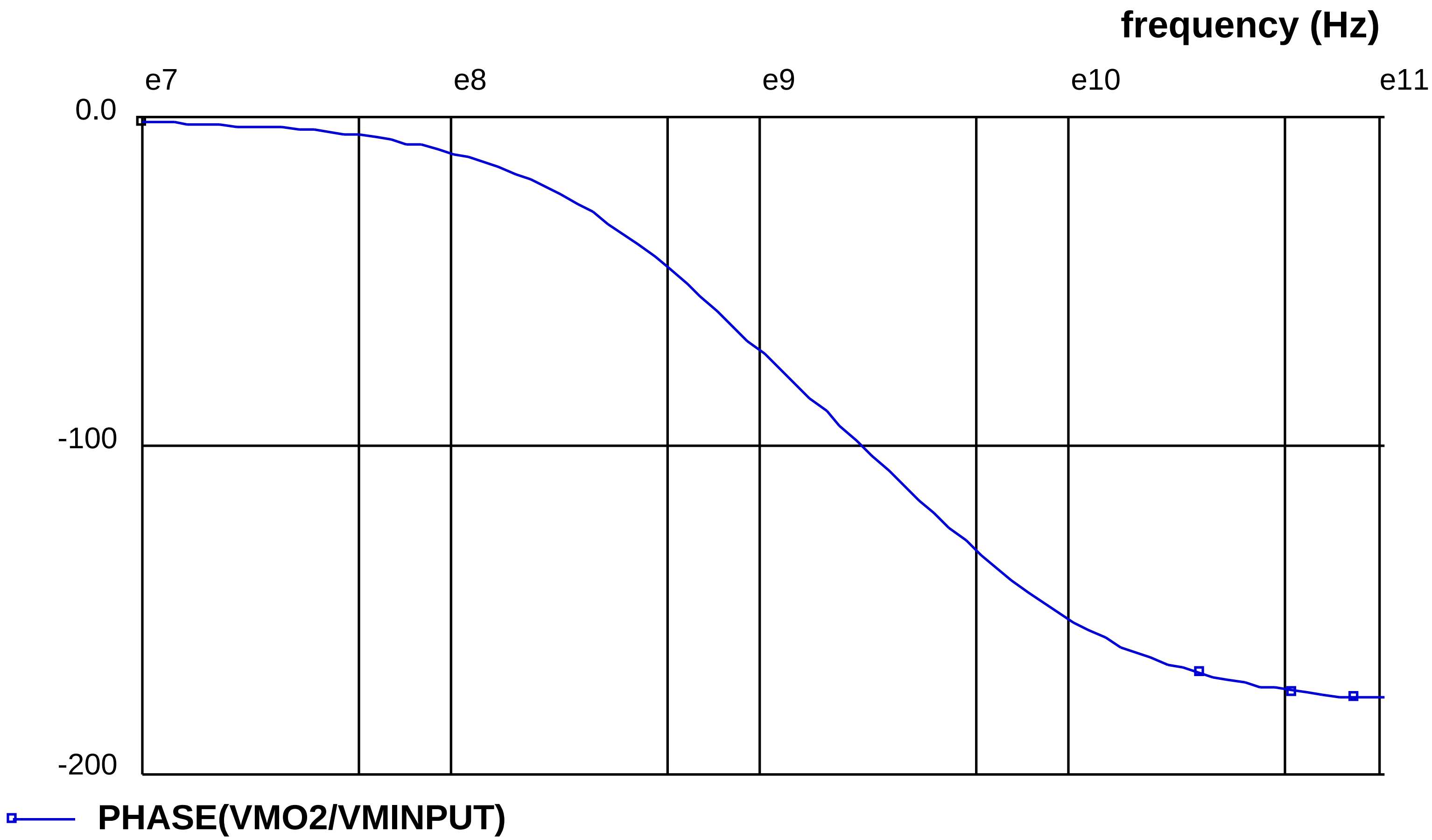


SPICE



frequency (Hz)
e7
e8
e9
e10
e11
0
-10
-20
-30
-40
-50
-60
-70
-80
DB(VMO2)/MAG(VMINPUT)

SPICE

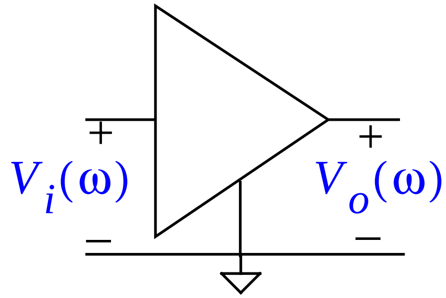


SPICE vs. Bode Plots

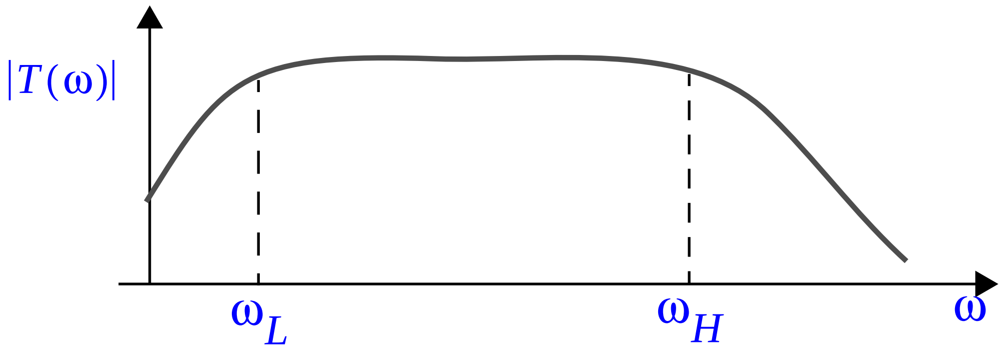
- Bode plots were introduced before software tools such as SPICE were available, and to some extent, SPICE makes them unnecessary
- But Bode plot theory helps us to understand the relation between the circuit poles and zeros and the frequency response
- This is helpful during design --- modify a certain parameter because we know that it moves the pole or zero in a particular way
- For example: design an amplifier to have a certain gain over the required frequency range of operation by modifying the smallest pole (largest time constant) in the circuit

Amplifiers: Frequency Response and Transfer Function

- A good amplifier would have a frequency response that is flat over the range of frequencies that it operates --- why?



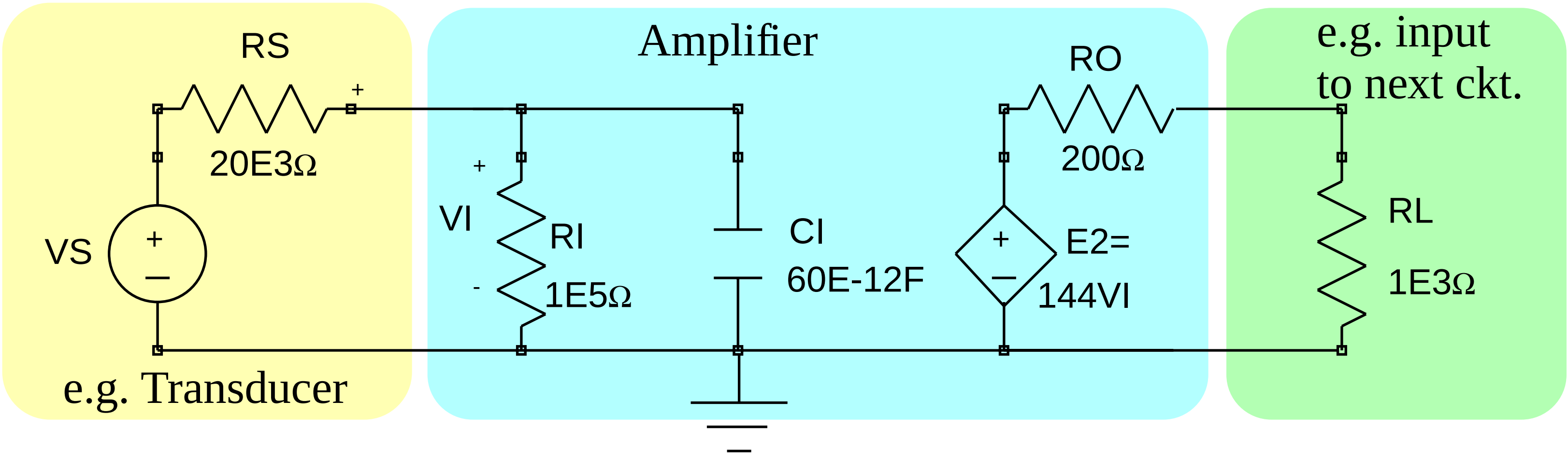
$$T(\omega) = \frac{V_o(\omega)}{V_i(\omega)}$$



- The high-frequency roll-off is due to amplifier internal capacitors (parasitics)
- The low frequency roll-off is generally due to ac coupling
- The bandwidth is defined as $\omega_H - \omega_L$

Amplifier Circuit Example

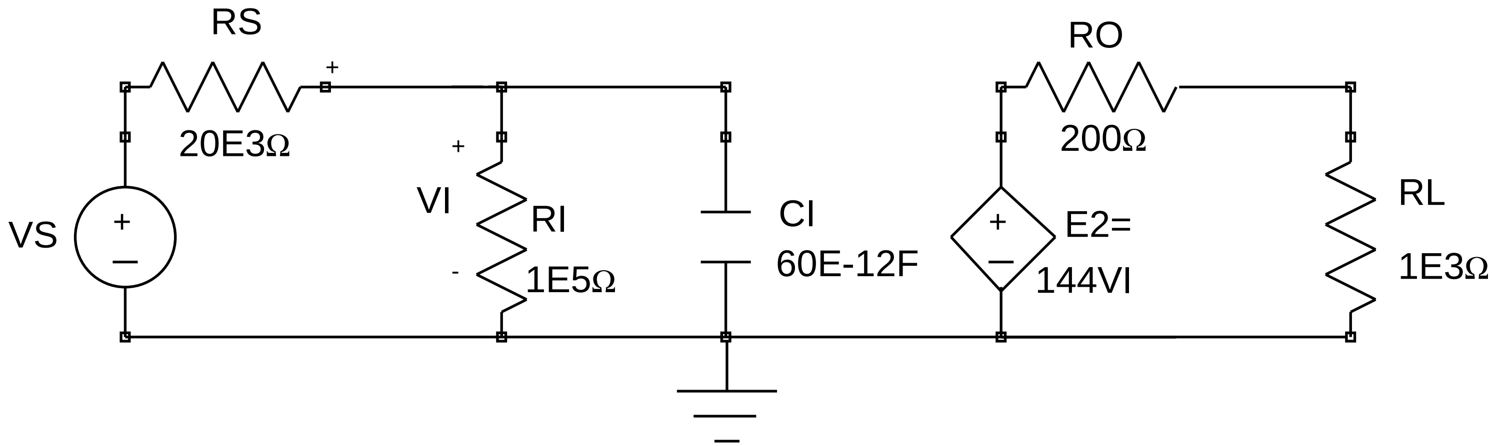
- An amplifier macromodel must include parasitic resistance and capacitance to see this frequency domain behavior



- What are the values of ω_H and ω_L for this single-time-constant (STC) circuit?
- What do these values indicate for this design?

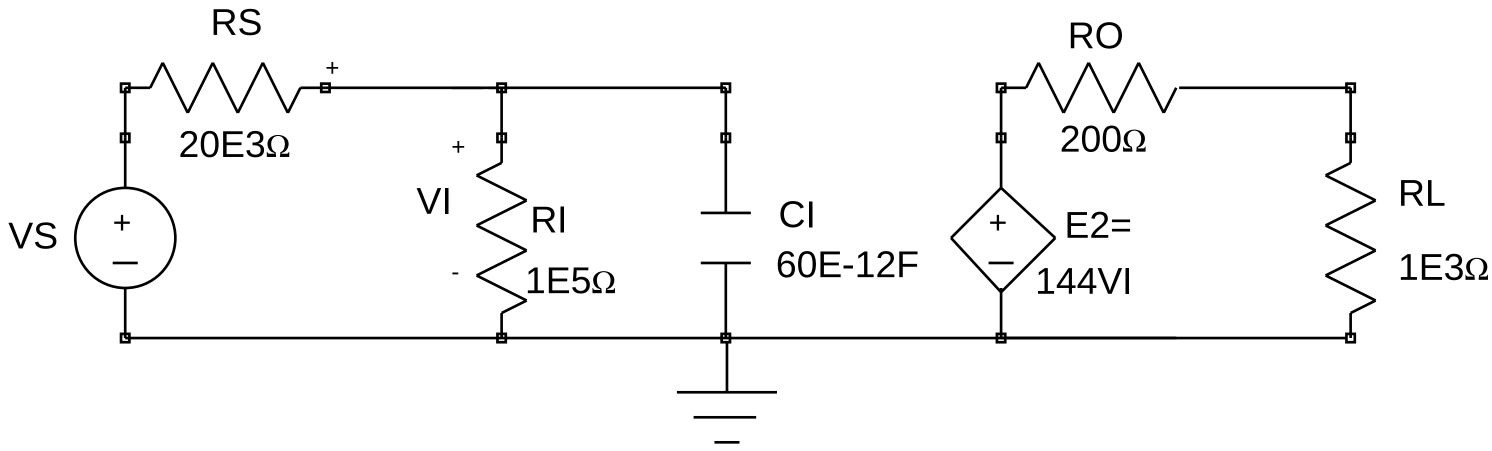
Amplifier Circuit Example

- This is obviously a low-pass filter
- The 3dB point, ω_H occurs at the pole value

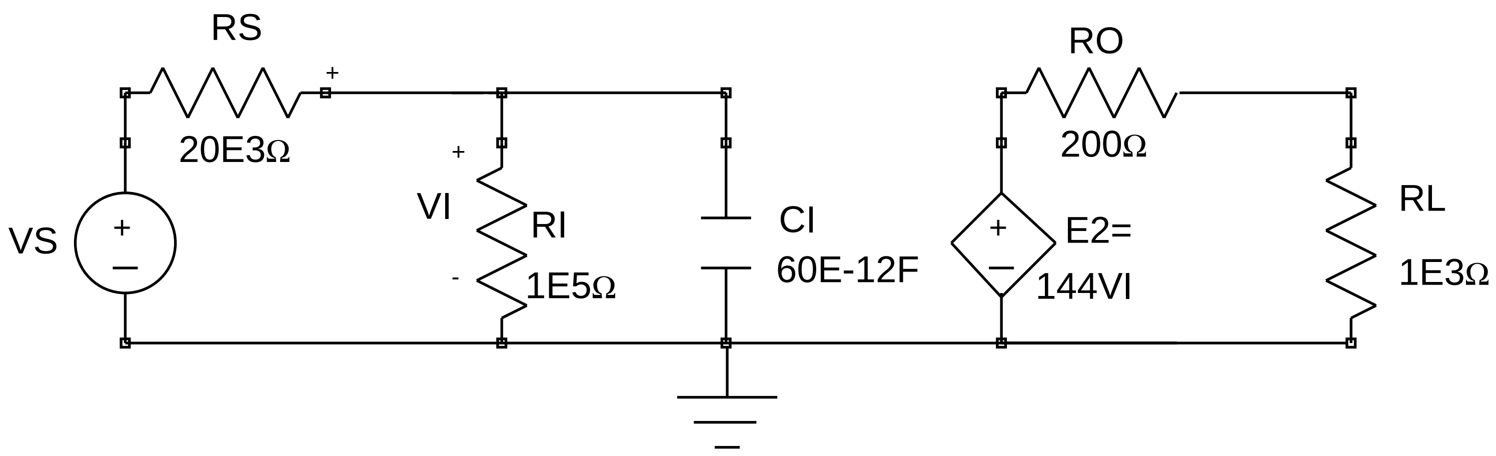


- What does the transfer function look like, and how do we solve for it?

Amplifier Circuit Example

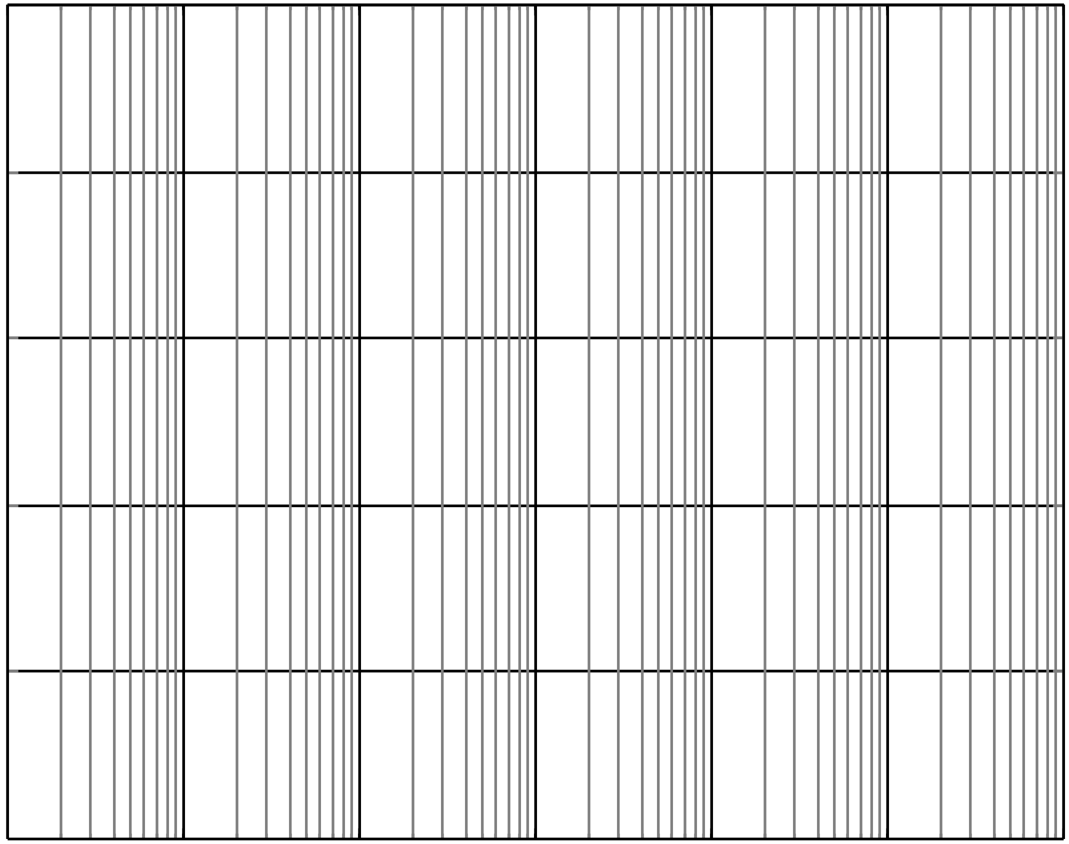


Amplifier Circuit Example



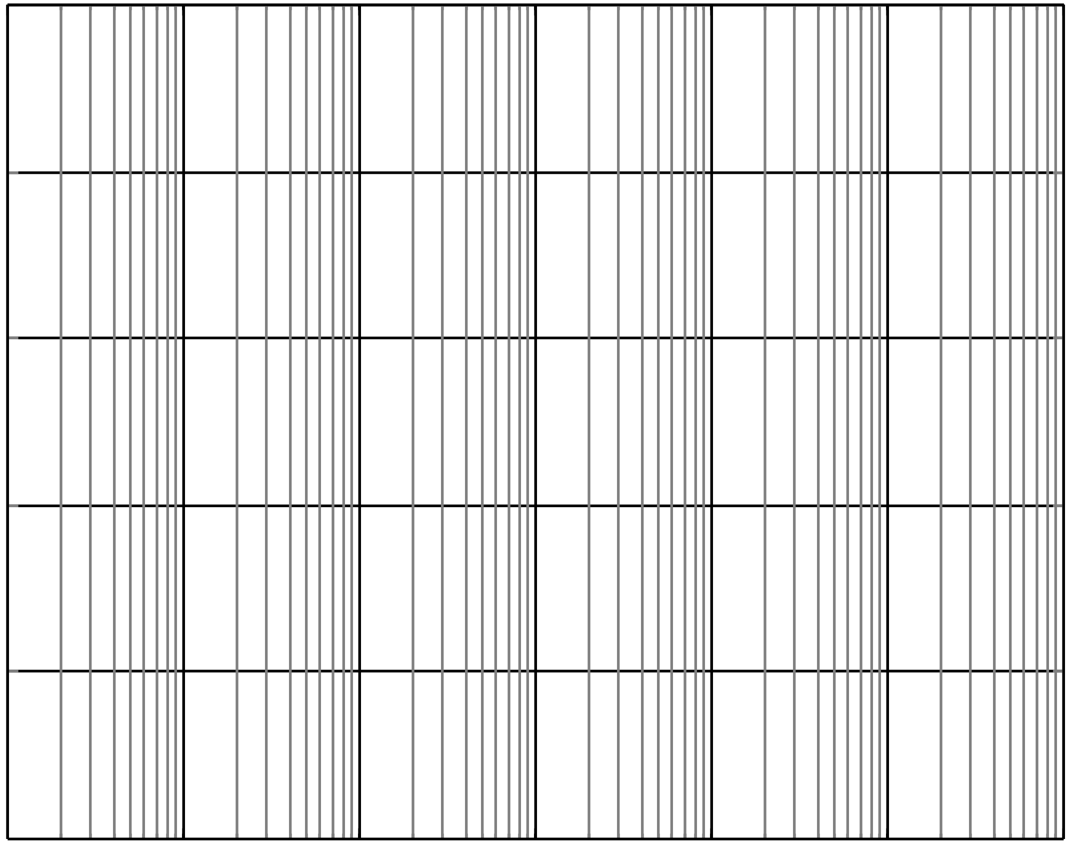
Amplifier Circuit Example

- What happens when the gain crosses below the unity gain (0dB) frequency point?



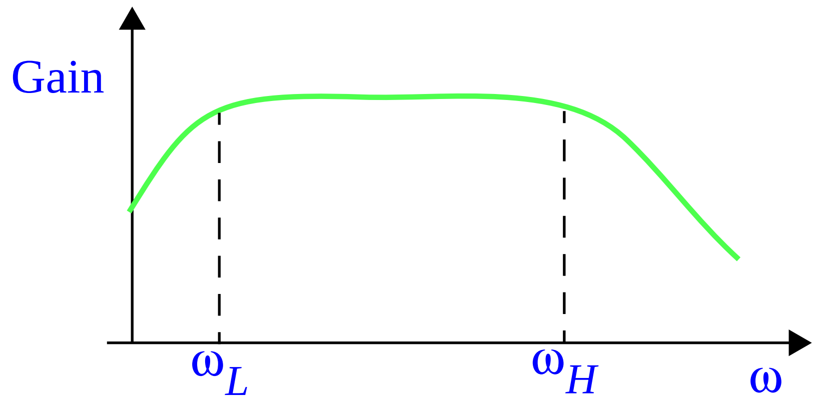
Amplifier Circuit Example

- Phase plot

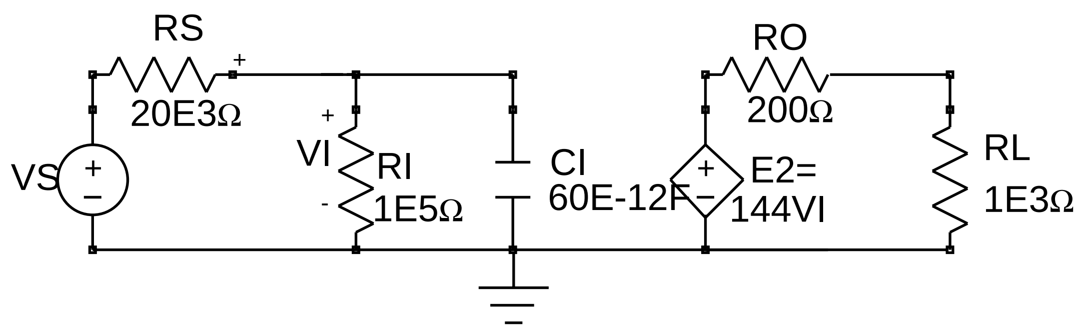


Low Frequency 3 dB Point

- The low frequency 3dB point for an amplifier, if it exists, is due to ac coupling

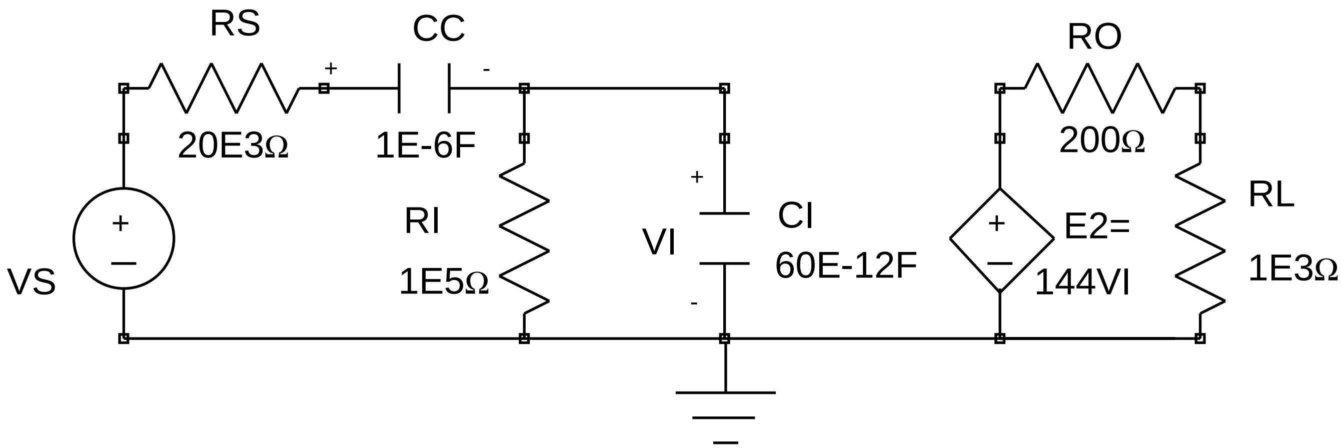


- ac coupling is generally used to block dc bias signals from entering, and saturating, the amplifier
- Example: If VS has a dc component that is much larger than the ac components, it may cause the amplifier output to saturate to the supply voltage



Amplifier Circuit: SPICE Schematic

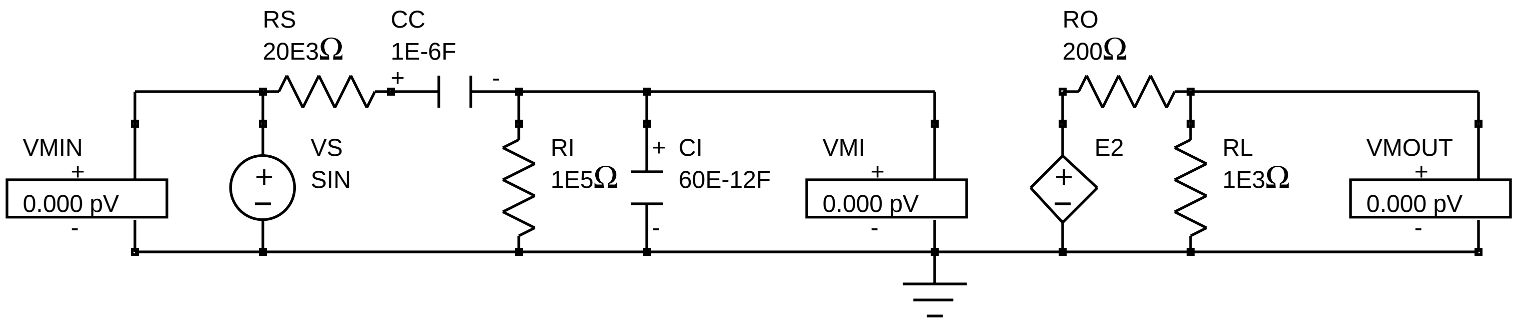
- So we can add a series capacitor to block the dc, but we want to make this CC as large as possible so that it does not affect the bandwidth of interest



- This capacitor adds a zero (at zero frequency) to the transfer function which we can observe from a SPICE simulation, or calculate and Bode plot

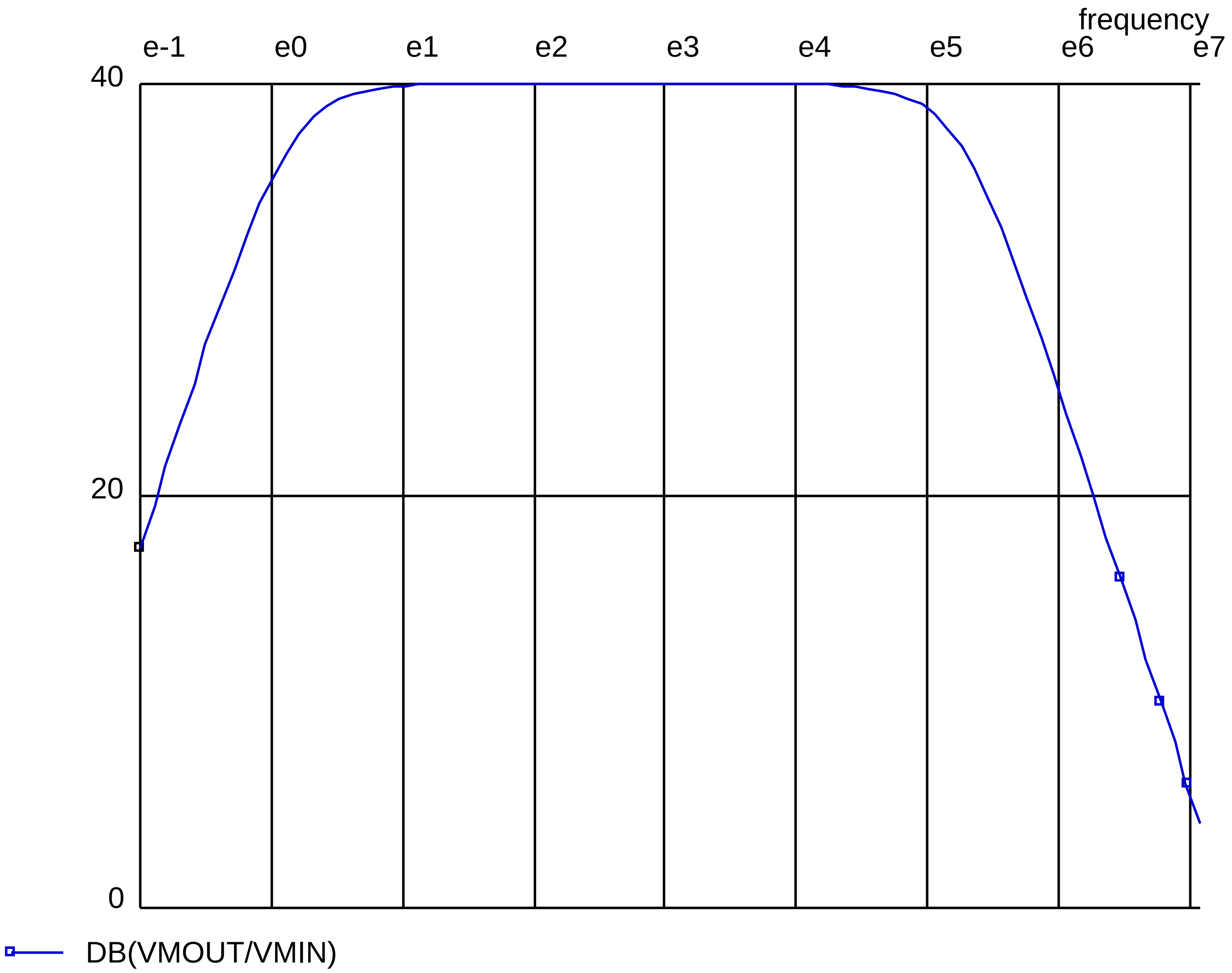
Amplifier Circuit: SPICE Schematic

- The input circuit to B^2SPICE is:

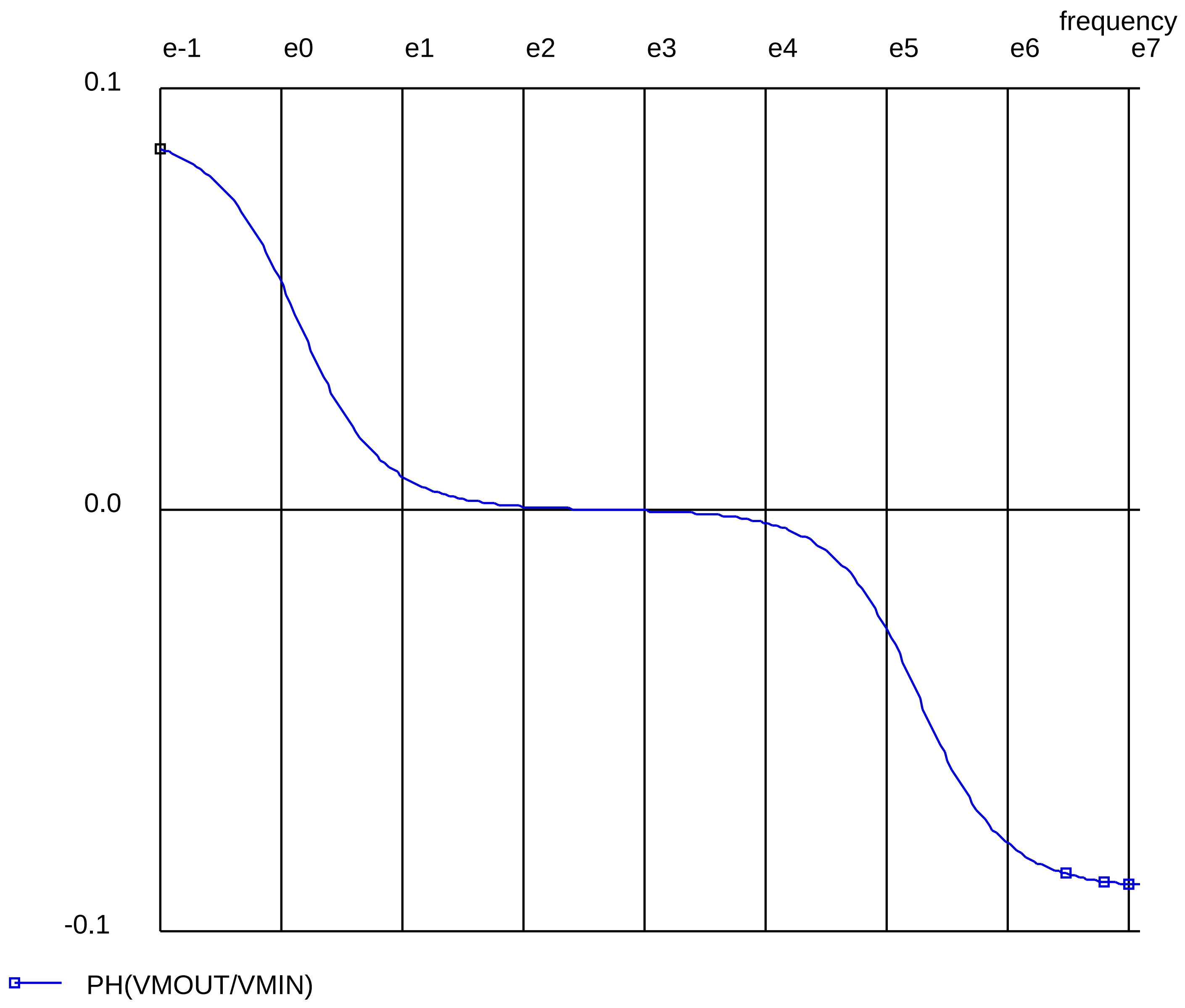


Amplifier Magnitude with Cc

- f_H= 1.58e5Hz, which corresponds to 0.99e6 radians/sec
- f_L= 1.35Hz, which corresponds to 8.48 radians/sec



Amplifier Phase with Cc



Points to remember

- Circuit simulation tools are only as good as the modeling / computational methods they use.
- Always try to understand qualitatively what you expect from simulation. Basing on “common sense”, be critical to simulation results.