



---

# Electronic Circuits

Lecture 6.1: Pspice Examples

# Pspice Tutorial: Comments & End of File

- **\* TITLE**
- **\* comment as a whole line**
- **; comment after a legal command to the end of the line**
- The first line must be a title to describe the circuit!
- **.END ; End of the simulation file**

# Pspice Tutorial: Sources

## ■ SOURCE, GENERAL

- **Vxxx N1 N2 DC [VALUE]**
- **Vxxx N1 N2 AC [VALUE]**
- **Vxxx N1 N2 SIN(VO VA [Freq][T<sub>d</sub>][D<sub>f</sub>][Phase]]])**
- **Vxxx N1 N2 EXP (V1 V2 [T<sub>d1</sub>][T<sub>d2</sub>][Tau2]]])**
- **Vxxx N1 N2 PWL (T1 V1 {T<sub>u</sub> V<sub>u</sub>}...)**
- **Vxxx N1 N2 PULSE (V1 V2 [T<sub>d</sub>][T<sub>r</sub>][T<sub>f</sub>][PW][Period]]])**
- **Ixxx N1 N2 DC [VALUE]**
- **Ixxx N1 N2 AC [VALUE]**

- **Ixxx N1 N2 SIN(VO VA [Freq][T<sub>d</sub>][D<sub>f</sub>][Phase]]])**

- **Ixxx N1 N2 EXP (V1 V2 [T<sub>d1</sub>][T<sub>d2</sub>][Tau2]]])**

- **Ixxx N1 N2 PWL (T1 V1 {T<sub>u</sub> V<sub>u</sub>}...)**

- **Ixxx N1 N2 PULSE (V1 V2 [T<sub>d</sub>][T<sub>r</sub>][T<sub>f</sub>][PW][Period]]])**

## ■ INDEPENDENT SOURCES

- **Vxxxxxxx N+ N- DC (PULSE,SIN,EXP) AC ; (voltage source)**

- **Ixxxxxxx N+ N- DC (PULSE,SIN,EXP) AC ; (current source)**

# Pspice Tutorial: Dependent Sources

- **CONTROLLED SOURCES (VCVS, ICIC, VCIS, ICVS)**
  - **Exxxxxxx N+ N- NC+ NC- VALUE ; (voltage controlled voltage source)**
  - **Fxxxxxxx N+ N- VNAME VALUE ; (current controlled current source)**
  - **Gxxxxxxx N+ N- NC+ NC- VALUE ; (voltage controlled current source)**
  - **Hxxxxxxx N+ N- VNAME VALUE ; (current controlled voltage source)**

# Pspice Tutorial: R-L-C Elements

- **RLC ELEMENTS**
  - **Rxxxxxxx N1 N2 VALUE ; (Resistor)**
  - **Cxxxxxxx N+ N- VALUE IC ; (Capacitor)**
  - **Lxxxxxxx N+ N- VALUE IC ; (Inductor)**
  - **Kzzzzzzz Lxxxxxxx Lyyyyyy VALUE ; (Transformer)**

# Pspice Tutorial: Semiconductor Devices

- **SEMICONDUCTOR DEVICES**

- **Dxxxxxxx N+ N- DMOD ; (diode)**

- **.MODEL DMOD D DIODE PARAMETER ; (diode model)**

- **Qxxxxxxx NC NB NE QMOD ; (bipolar transistor)**

- **.MODEL QMOD NPN or PNP TRANSISTOR PARAMETERS ; (transistor model)**

# Pspice Tutorial: Analysis Methods

- **ANALYSIS**
  - **.DC SWEEP VARIABLE START STOP INCREMENT (DC)**
  - **.TF OUTPUT VARIABLE INPUT SOURCE ; (transfer function)**
  - **.TRAN TSTEP TSTOP UIC ; (transient)**
  - **.AC DEC or OCT or LIN NPOINTS FSTART FSTOP (AC)**
  - **.IC X(NODE) = VALUE**
  - **.LIB NAME.LIB ; (library statement)**

# Pspice Tutorial: Reporting Outputs

- **OUTPUT**
  - **.PRINT (PLOT) DC OUTVAR1...OUTVAR8**
  - **.PRINT (PLOT) TRAN OUTVAR1...OUTVAR8**
  - **.PRINT (PLOT) AC VM VDB VP VR VI**
  - **.PROBE**



# Pspice Tutorial: Prefixes and Suffixes

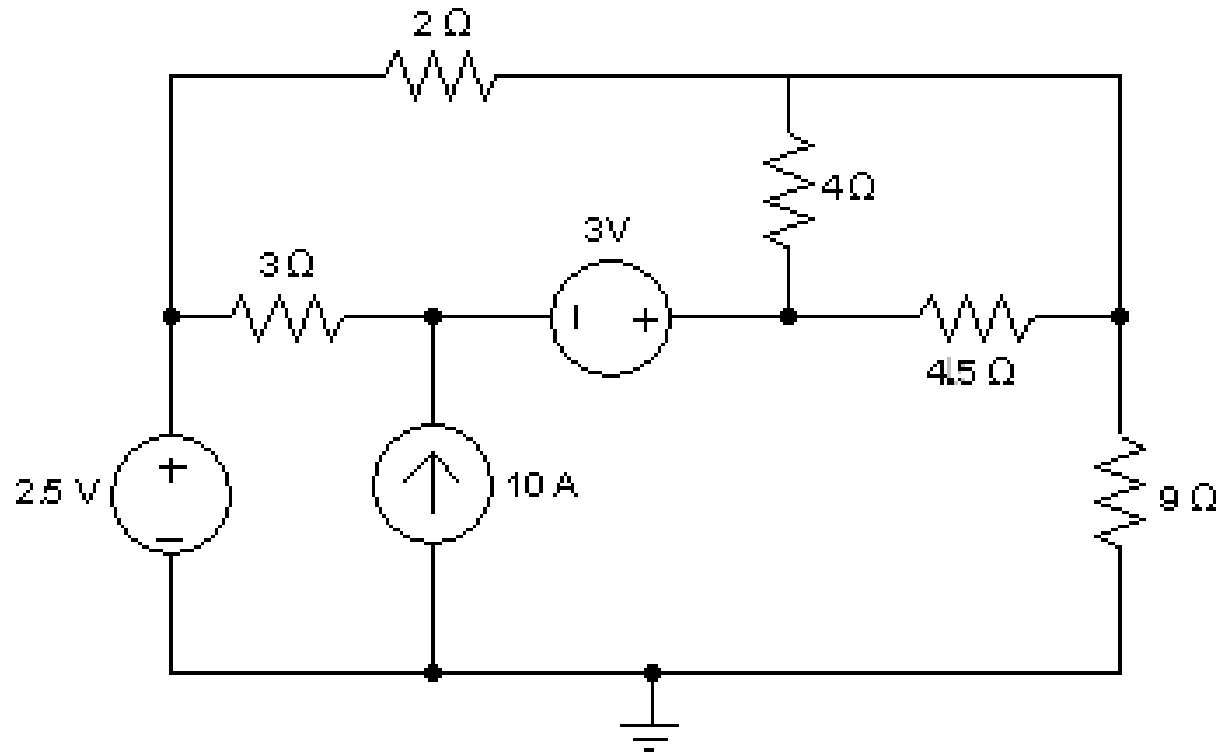
- F fento-  $10^{-15}$
- P pico-  $10^{-12}$
- N nano-  $10^{-9}$
- U micro-  $10^{-6}$
- M milli-  $10^{-3}$
- K kilo-  $10^{+3}$
- MEG mega-  $10^{+6}$
- G giga-  $10^{+9}$
- T tera-  $10^{+12}$
- MIL (0.001")  $25.4 \times 10^{-6}$

# Pspice Tutorial: Magnitude and Phase

- **VM(n)** - magnitude of  $n^{\text{th}}$  node voltage
- **VP(n)** - phase of  $n^{\text{th}}$  node voltage in degrees

Example 1: Find the current across each branch and voltage across each element.

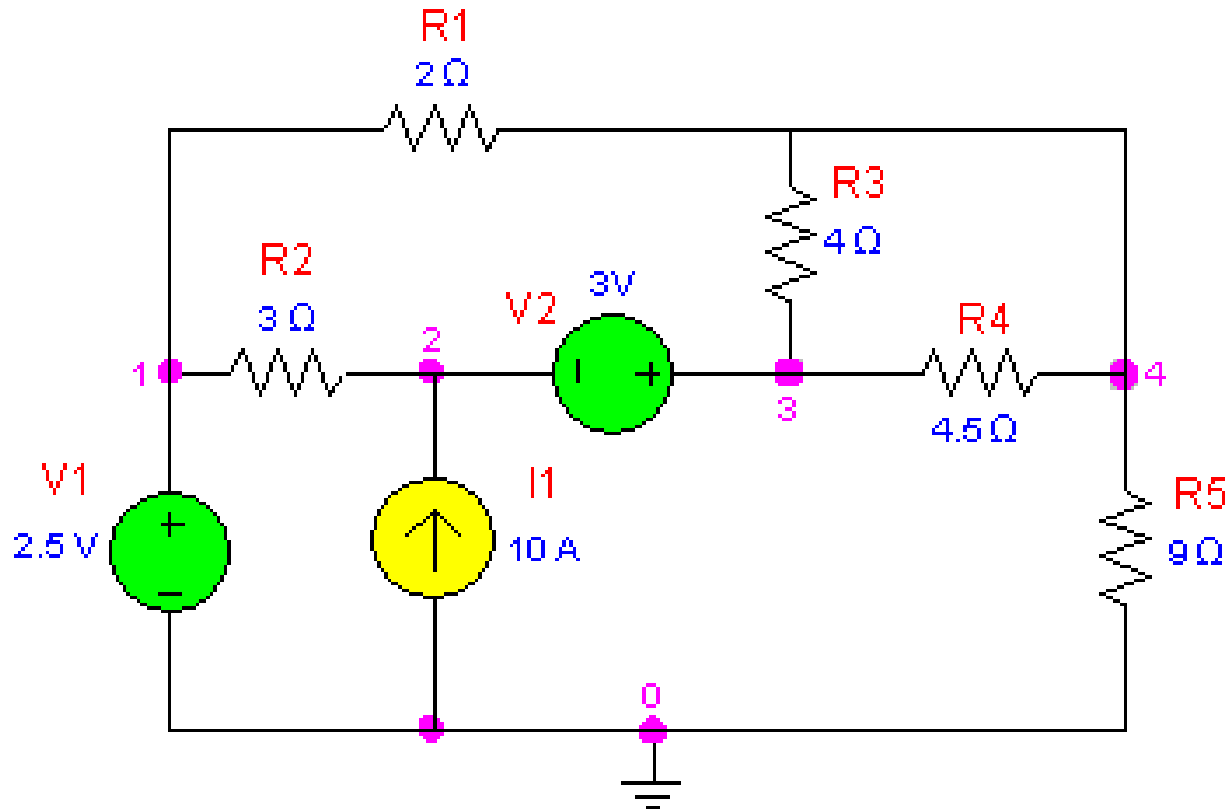
Given Circuit:



- After associating ALL nodes (including non-essential ones) with numbers, with node 0 as the reference or ground node, the circuit elements must be labeled/associated as well by PSpice standard symbol representations. After doing this, the netlist which PSpice can understand can be written and simulated.
- So as you can see, one MUST know how to understand how circuit elements are arranged and assembled (according to polarity and direction) in a PSpice Netlist.

# Example 1: PSpice solution

## PSpice Circuit:



\*BME206 - Example 1

```
R1 1 4 2ohm
```

```
R2 1 2 3ohm
```

```
R3 3 4 4ohm
```

```
R4 3 4 4.5ohm
```

```
R5 4 0 9ohm
```

```
V1 1 0 2.5V
```

```
V2 3 2 3V
```

```
I1 2 0 10A
```

```
.DC V1 10 10 1 ; analysis method
```

```
*To find ALL current and voltages across
```

```
.PRINT DC V(R1), V(R2), V(R3), V(R4),
```

```
V(R5), V(V1), V(V2), V(I1), I(R1), I(R2),
```

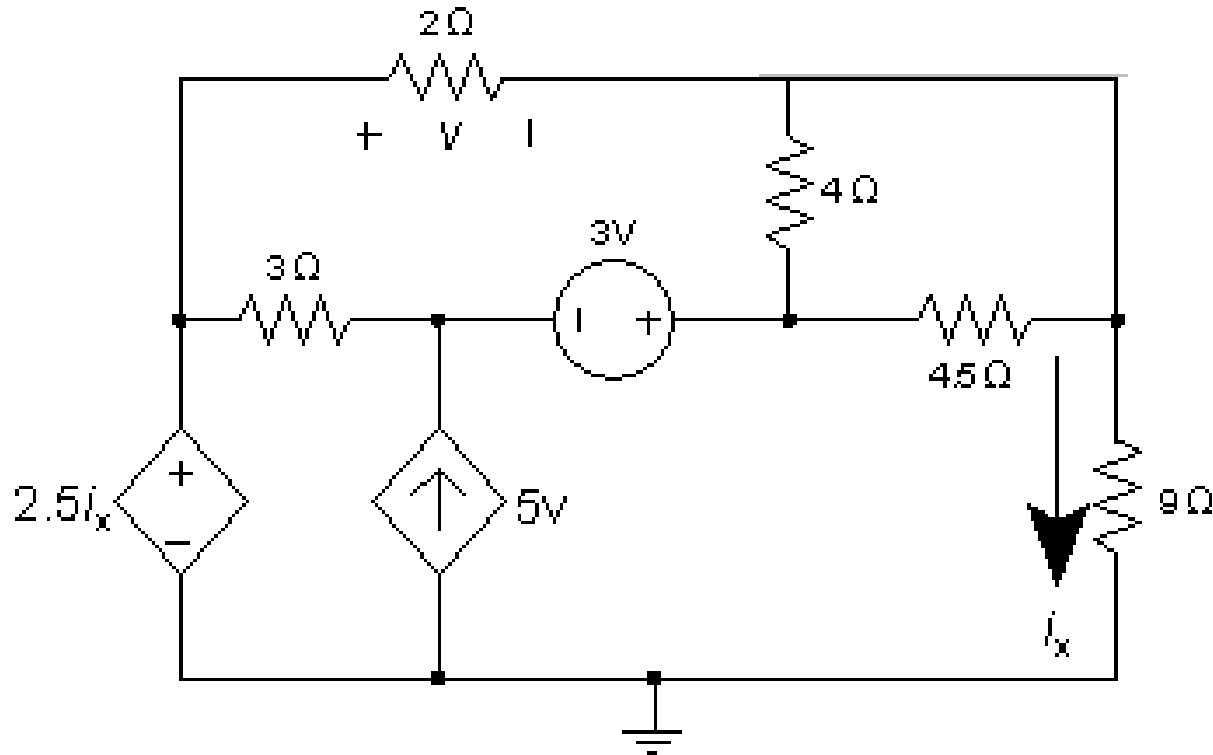
```
I(R3), I(R4), I(R5), I(V1), I(V2), I(I1)
```

```
.OPTIONS NOPAGE ; to minimize the output
```

```
.END ; must have to end netlist.
```

Example 2: Find the current across each branch and voltage across each element.

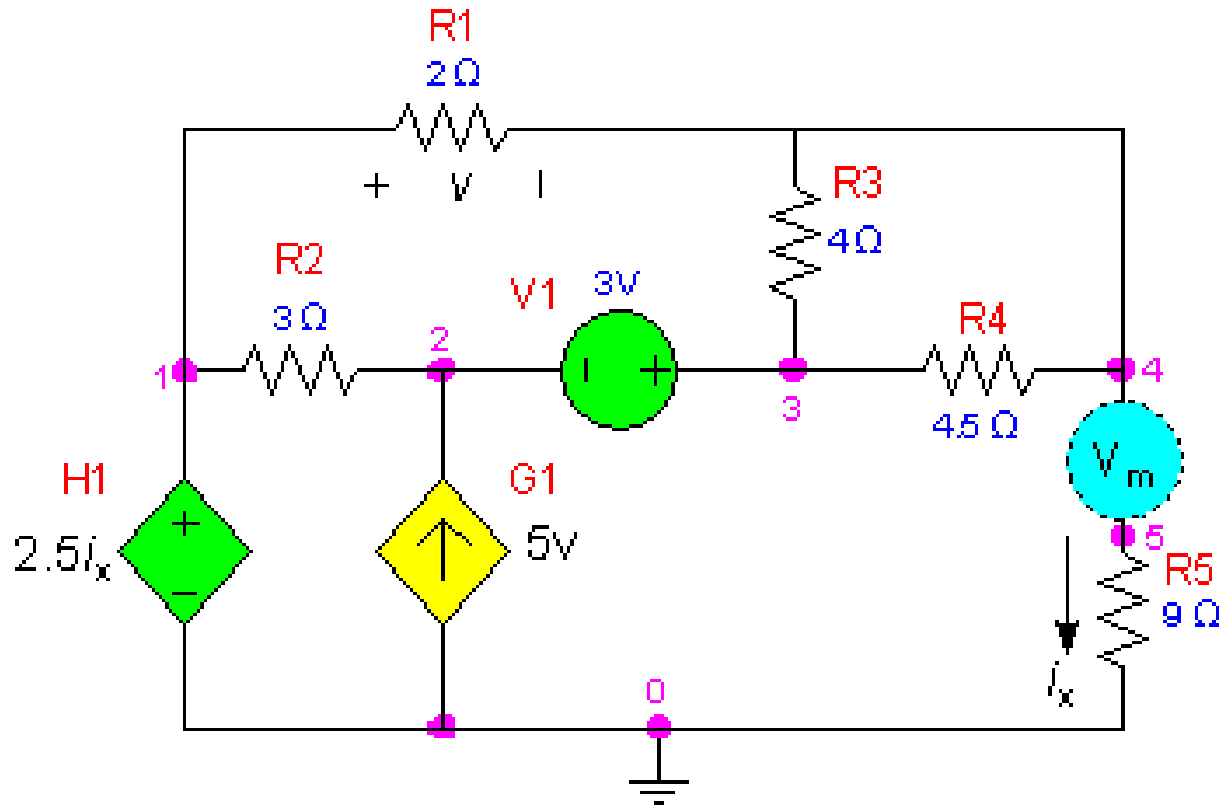
Given Circuit:



- The only difference between the examples is that two of the sources are now dependent sources. And since H1 is a current dependent voltage source, a voltmeter is required along the controlling branch which will let us extract the current through the resistor and use it through PSpice.

## Example 2: PSpice solution

### PSpice Circuit:



\*BME206 - Example 2

R1 1 4 2ohm

R2 1 2 3ohm

R3 3 4 4ohm

R4 3 4 4.5ohm

R5 5 0 9ohm

Vm 4 5 ;Voltmeter used for H1

V1 3 2 3V

H1 1 0 Vm 2.5

G1 2 0 1 4 5

.DC V1 10 10 1 ; analysis method

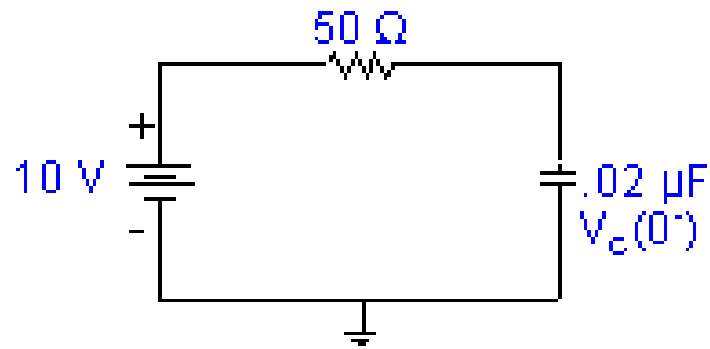
.PRINT DC V(R1), V(R2), V(R3), V(R4),  
V(R5), V(V1), V(H1), V(G1), I(R1), I(R2),  
I(R3), I(R4), I(R5), I(V1), I(G1), I(H1)

.OPTIONS NOPAGE

.END

Example 3: Find the voltage  $V_c(t)$  in the RC circuit below.

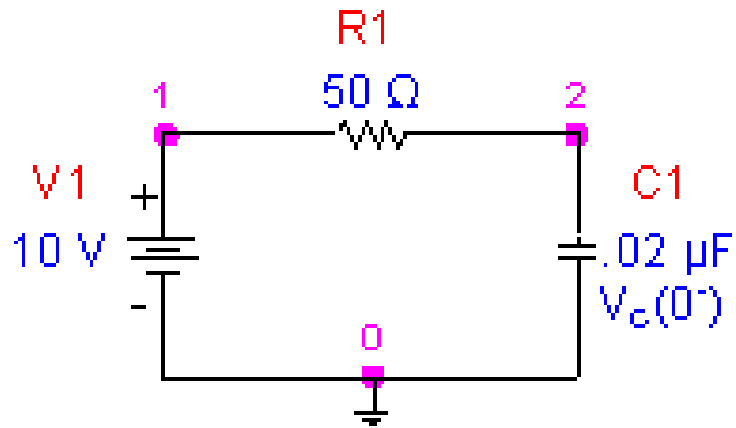
Given Circuit:



- Note that the "use initial condition" command is used in the TRAN statement and that the initial condition for the capacitor is specified. In this case, the applied voltage was assumed to be a step function, i.e.  $V1=10V \ t>0$ ; thus a simple DC source was used to represent the input voltage. In other cases, a time dependent source such as a sinewave, pulse, etc., may be required.

## Example 3: PSpice solution

PSpice Circuit:



```
*BME206 - Example 3
```

```
R1 1 2 50ohm
```

```
V1 1 0 10V
```

```
C1 2 0 0.02UF IC=0V ; cap w I. condition
```

```
.TRAN 0.05US 5US UIC ; analysis method
```

```
.PROBE ; PROBE graphics output, V(t)
```

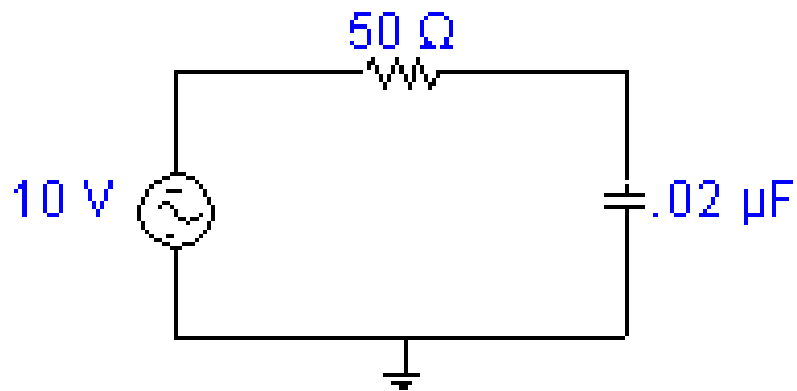
```
.OPTIONS NOPAGE
```

```
.END
```



Example 4: Find the voltage  $V_C(t)$  in the RC circuit below.

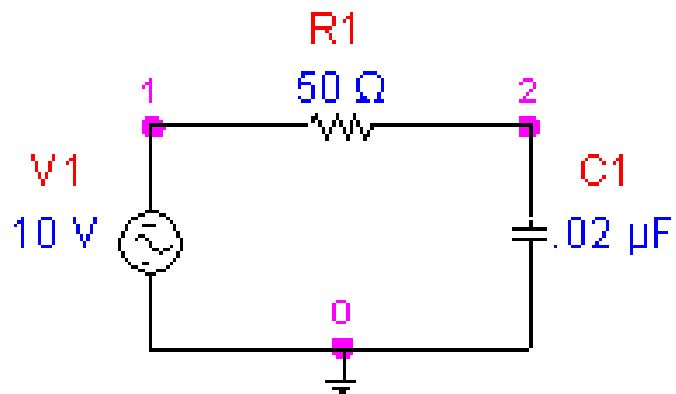
Given Circuit:



- Find the current  $i_C(f)$ ,  $V_R(f)$ , and  $V_C(f)$ .
- AC analysis using PSpice is phasor analysis: the source is an AC/Phasor source of amplitude 10V and phase of zero degrees.

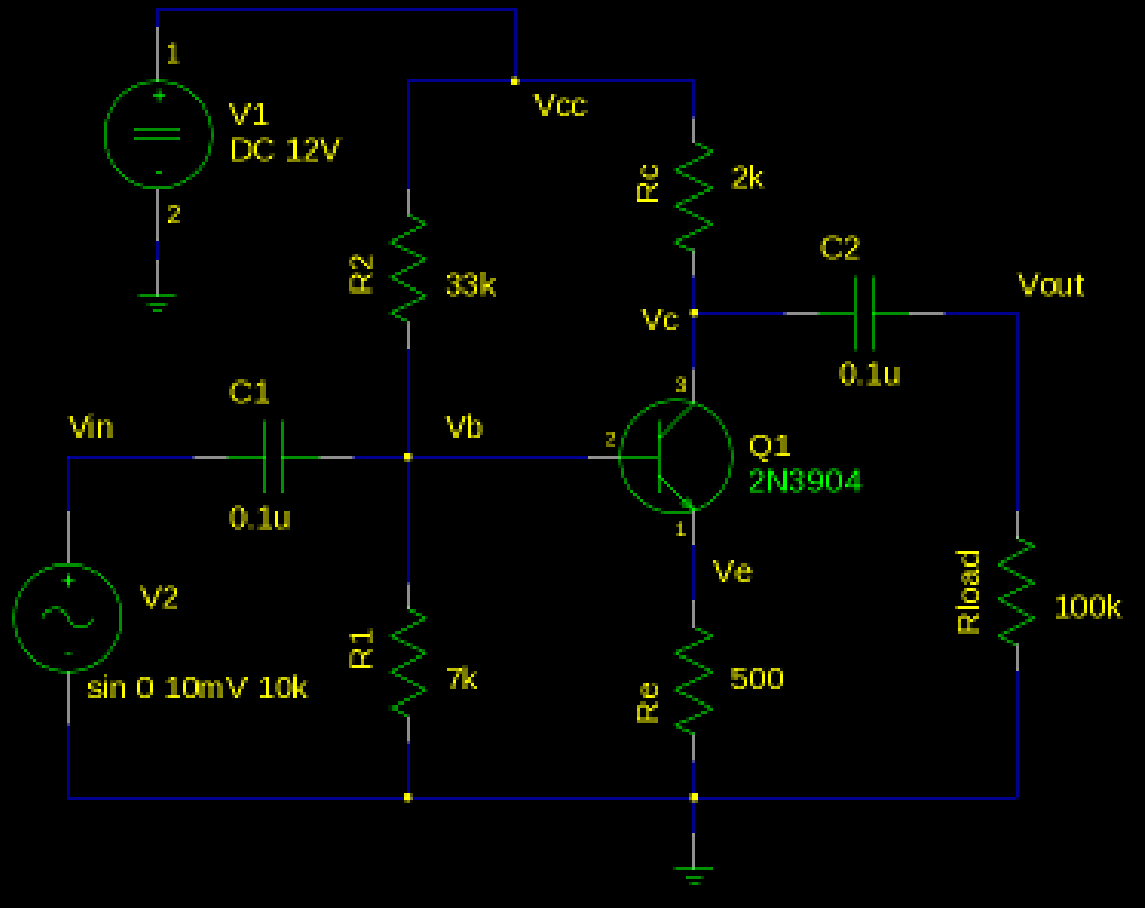
## Example 4: PSpice solution

PSpice Circuit:



```
R1 1 2 50ohm
V1 1 0 AC 10V 0 ; 10 volts, 0 degrees phase
C1 2 0 0.02UF ; capacitor w initial condition
.AC DEC 20 10KHZ 1MEGAHZ ; method
.PROBE ; PROBE graphics output, trace V(t)
.OPTIONS NOPAGE
.END
```

# Example 5: Common Emitter Amplifier



\* Common emitter amplifier

```
.model 2N3904 NPN(Is=6.734f Xti=3 Eg=1.11 Vaf=74.03 Bf=416.4 Ne=1.259
+       Ise=6.734f Ikf=66.78m Xtb=1.5 Br=.7371 Nc=2 Isc=0 Ikr=0 Rc=1
+       Cjc=3.638p Mjc=.3085 Vjc=.75 Fc=.5 Cje=4.493p Mje=.2593
+       Vje=.75 Tr=239.5n Tf=301.2p Itf=.4 Vtf=4 Xtf=2 Rb=10)
*
```

```
V1 Vcc 0 DC 12V
```

\* need extra 0 at "4th" connection of transistors to indicate the reference point

```
Q1 Vc Vb Ve 0 2N3904
```

```
Rc Vcc Vc 2k
```

```
Re Ve 0 500
```

```
R1 Vb 0 7k
```

```
R2 Vcc Vb 33k
```

```
C1 Vin Vb 0.1u
```

```
C2 Vc Vout 0.1u
```

```
Rload Vout 0 100k
```

```
V2 Vin 0 ac 10mV sin(0 10mV 10kHz)
```

```
.tran 1u 5m
```

```
.ac oct 5 100 100Meg
```

```
.end
```

\* .model is the part definition, you may not understand all parameters given in the model, no problem 😊

\* You can name Re, instead of using numbers only as R2, etc.

\* Here, Vin is the node name !



Thanks for  
listening 😊

YALÇIN İŞLER

Assoc. Prof.