Crash Course Electronics

Analogue Electronics Notes

Peter Colls Review

Version 1.1 12/07/2020

Table of Contents

Table of Figures

Bibliography

André LaMothe	"Crash Course Electronics" -
André LaMothe	"Design Your Own Video Game"
André LaMothe	"Common Prefixes"
Mr Carlson Lab	"Types of Capacitors"
Mr Carlson Lab	"Mr Carlson Comments"
David Jones	"Power supply smoothing capacitors
Wikipedia	"Decibels "Table of db Values"

Prologue

This document summarises my study notes of the Analogue component of André LaMothe "Crash Course Electronics", it primarily my own work and not for publication.

To assist with clarity, some headings include a reference to the course lecture number (Lnn), e.g. "Low Pass Filter L52".

Tables and Laws

OHM's Law L2

Volts : V = I . R

Current: $C = \frac{V}{R}$

Resistance: R $= \frac{V}{I}$

Power: P = V . I Or P = $\frac{V}{R}$ Or P = $\frac{V2}{I}$

Rules to Remember

a. **Resistor Wattage – Calculate the Power and then X 2 for safety**
b. **Milli Amps** – Volts / K ohms always results in Milli Amps
c. **Heat** – Devices should be able to dissipate twice the amount of power
d. **Parallel Resistance Calculation** - is the product over the sum, and the result is always less than the smallest resistor.

Common Prefixes

Symbol	Prefix	Scaling Factor 10 to the Minus
a	atto	10-18th
f	femto	10-15th
p	pico	10-12th
n	nano	10-9th
µ	micro	10-6th
m	milli	10-3rd
k	kilo	10-3rd
M	mega	10-6th
G	giga	10-9th
T	tera	10-12th
P	peta	10-15th
E	exa	10-18th

Note:

- Other prefixes are based on multiples of 100, but most commonly in electronics, the multiples of 1000 or 103 are used.
- In pure scientific notation = 3.0x10-4th watts
- Using the kilo (10-3rd) prefix = 30k watts or 30kW
- Using the mega (10-6th) prefix = .03M watts or .03MW
- 005 amps In pure scientific notation = 5.0x10-3rd amps

- Using the milli (10-3rd) prefix = 5.0m amps or 5ma
- Using the micro (10-6th) prefix = 5000µ amps or 5000µa

Electronic Terms and Symbols

V = Volts Measured in V. mV

R = Resistance Measured in Ohm's

I = Current Measured in Amps

C = Capacitance Measured in Farad's

L = Inductance Measured in Henry's

X = Reactance Measured in Ohm's

Q = Capacitor Charge Measured in Coulomb's

Z = Impedance Measured in Ohm's Φ(Phi = Flux) **= Induction Flux**

Mathematical Equations

Resisters in Parallel	PR = $\frac{(R1\ .\ R2)}{R1+R2}$ = 3.33 Ω The result is always less than the smallest resistor in the parallel circuit. – Refer Fig 3 & Fig 5
Voltage Dividers	V_1 = R_1 * $\frac{Vin}{R1+R2}$ = 1.899v V_2 + R_2 * $\frac{Vin}{R1+R2}$ = 3.100v - Refer Fig 4 & Fig 6
Reactance (X)	As frequency goes up, Reactance (X_c) goes down. ∴ As frequency goes down, reactance (X_c) goes up. $X_c = \frac{1}{2\pi.F.C}$ Refer Fig 14
Impedance (Z)	$V1 = 10v$ RMS ∴ $Xc = \frac{1}{2\pi.F.C} = 72.34\Omega$ $Z = \sqrt{R^2 + Xc^2} = \sqrt{100^2 + 72.34^2} = Z = 123.42\Omega$ - Refer Fig 17
Phase Angle	- $\varphi = Tan^{-1}\frac{Vc}{Vr}$ (inverse tan "Tan-1" is the opposite over the adjacent, e.g. V_c over V_a - Refer fig 19
Calculate β	$\beta = \frac{Ie}{Ib}$ = $\frac{56.8}{0.42}$ = 135 ∴ $\beta = 135$ - Refer Fig 48
Inductor in series	L_{tot} = L1 + L2 + L3 - Refer Fig 31
Inductor in Parallel	$L_{ToT} = \frac{1}{\frac{1}{L1}+\frac{1}{L2}+\frac{1}{L3}}$ - Refer Fig 32

Resisters

Resisters in Series

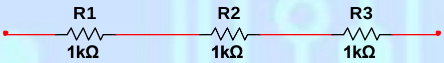


Figure 1 – Resisters in Series

Resisters in Parallel

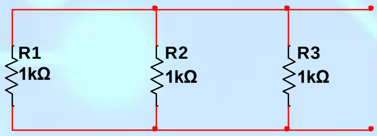


Figure 2 - Resisters in Parallel

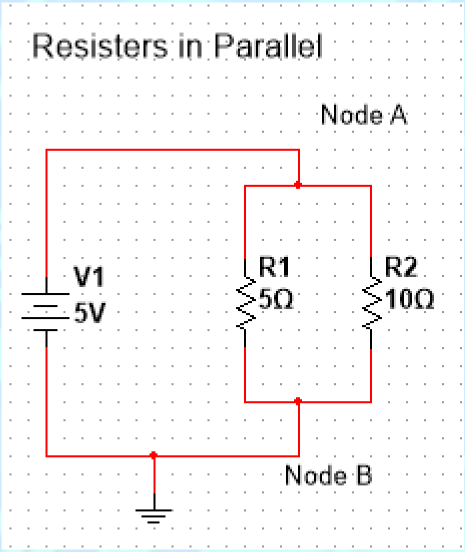


Figure 3 - Resisters in Parallel 2

KCL & KVL

KCL = current into node = 0

KVL = Voltage drops around a loop = 0

∴ $i_1 = \frac{5v}{5r}$ $i_2 = \frac{5v}{10r}$ ∴ p_{tot} = 1.5 Amps

1A + 0.5A = 1.5A

Resistance in Parallel – The result is always less than the smallest resistor in the parallel circuit.

PR = $\frac{(R1\,.\,R2)}{R1+R2}$ = 3.33 Ω

Calc the Voltage from the Resistance:

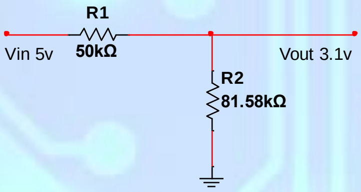


Figure 4 – Voltage Divider

$V_1 = R_1 * \frac{Vin}{R1+R2}$ = 1.899v

$V_2{}^{+} R_2 * \frac{Vin}{R1+R2}$ = 3.100v

Calculate all Voltages and Currents for each of the 4 Nodes:

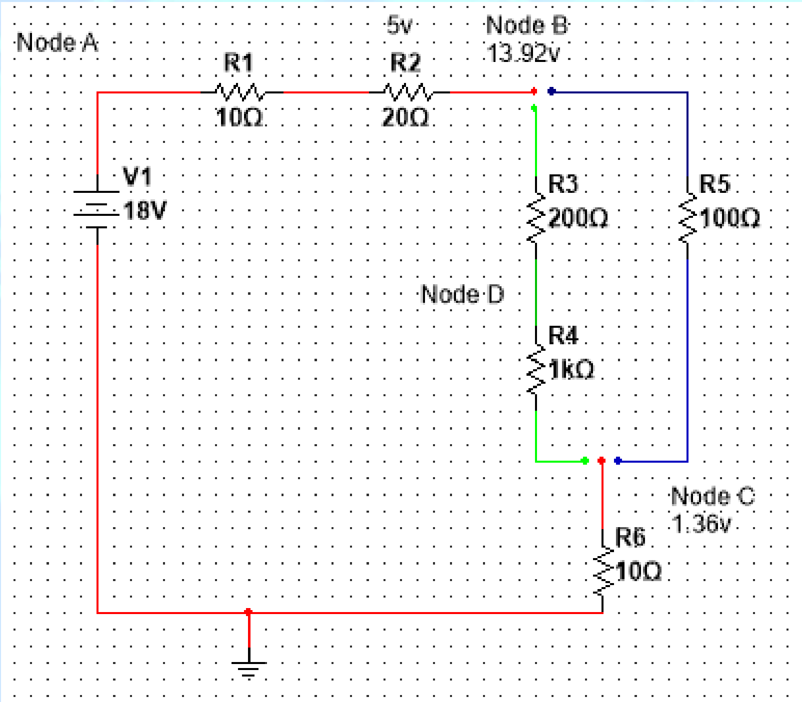


Figure 5 – Node Current Flow

Vs = 18v
R1 + R2 =30 Ω ohms
R3 + R4 = 1.2k Note: These values should be added together when calculating parallel resistance. Refer below.
R5 100 Ω
R6 = 10 Ω

Resistance over R3 + R4 II R5 (Product over the Sum) = $\frac{(R3+R4).R5}{(R3+R4)+R5}$ = $\frac{(200\Omega+1k).100\Omega}{(200\Omega+1k)+100\Omega}$ = 92.3 Ω

∴ R_{tot} = 30 + 10 + 92.3 = a total circuit resistance of 132.3 Ω

Node A

Current = I_1 = $\frac{V}{R} = \frac{18}{132.3}$ = 136mA

Volts = V_1 = I . R = 136.3 . 10 = 1.36v

V_2 = = 136.3 . 20 = 2.72v

Node "B"

Current at "B" = 136.6ma

Volts = Vs – (V1 + V2) = 13.92v

Node "C"

Current = 136.6ma

Volts = V_6 = I . R +136ma * 10 Ω = 1.36v

Note The other reference of R_6 is the ground reference ∴ it is 0v (KVL)

Node "D"

Voltage – The voltage differential between Node B & C is 13.92v – 1.36v = 12.56v

Current – I_5 = current over R^5 = $\frac{V}{R} = \frac{12.56v}{100\ \Omega}$ = 125.6mA

Current - $I_{3,4}$ = 136 – ($I_{3,4}$ +125.6) = 136 – 125.6 = 10.4

∴ $I_{3,4}$ = 10.4ma

Voltage Divider

Voltage Dividers are constructed by having two resistors in series with the second connected to ground. The junction between the two resisters is the reduced voltage value because of the resistance values of the individual two resisters.

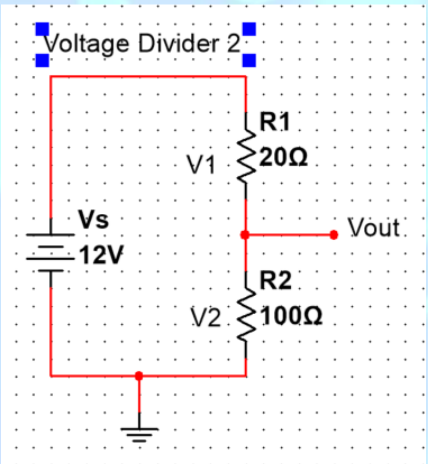


Figure 6 - Voltage Divider 2

$V_{out} = \left(\frac{R2}{R1+R2}\right) . V_{in}$

Voltage divider circuit 3.3v with a 10mA current. – How to work out the R_1 and R_2 Values.

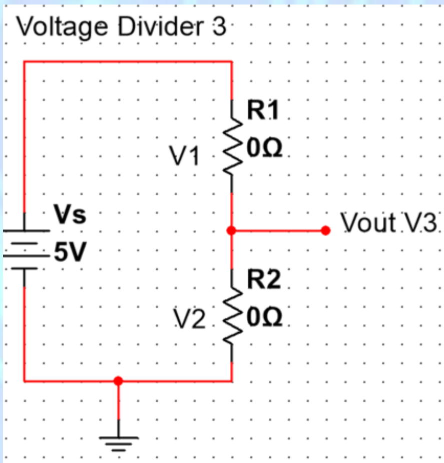


Figure 7 - Voltage Divider 3

1. Work out the current required at S_2
2. We want to set the circuit up for 10mA
3. What is the total resistance needed to get 10mA?
    a. $R_{Tot} = \frac{V}{I} \therefore \frac{5v}{10ma} = 500\ \Omega$

4. Voltage divider equations
    a. $V_1 = R_1{}^{*}\left(\frac{V}{R1+R2}\right)$
    b. $V_2 = R_2 * \left(\frac{V}{R2+R1}\right)$

5. What do we know?
    a. $S_1 = V = 5v$
    b. $S_2 = \quad = 3.3v$
    c. Total Resistance 500mA

6. Use the equation that has R^2 in it?
    a. $3.3v = R_2 \cdot \frac{5v}{500\Omega}$
    b. Do the Algebra and figure out R_2
    c. $3.3v * \frac{1}{\left(\frac{5}{500}\right)} = \frac{5}{500} = 0\ .0\ X^{-1} = 100$
    d. $100 * 3.3 = 330\ \Omega$
    e. $R_1 = 330\ \Omega$
    f. $R_1 = 500\ \Omega\ 330\ \Omega = 170\ \Omega$
    g. $R_1 = 170\ \Omega$

7. Future Parallel Resistance

a. If the load (resistance) on S_2 is 10 X larger than R_2, it will not make a significant difference to R_2 (the voltage on S_2 may drop 0.02v)
b. What is the parallel combination of 330 Ω and 3.3k = 330 II 3.3k. The parallel calculation is the product over the sum ∴ $\frac{330\ .3.3k}{330+3.3k}$ = 300 Ω
c. This will now change R_2 to 300 Ω that will reduce the voltage on s_2 to (V=I.R) S_2 = 10ma . 300 Ω = 3 volts.
d. If we needed the voltage to stay at 3.3v then we could change the R_2 resistance to compensate or change the current to allow a lower the total resistance on R_1 and R_2 thus stiffing the circuit.

RC Circuit

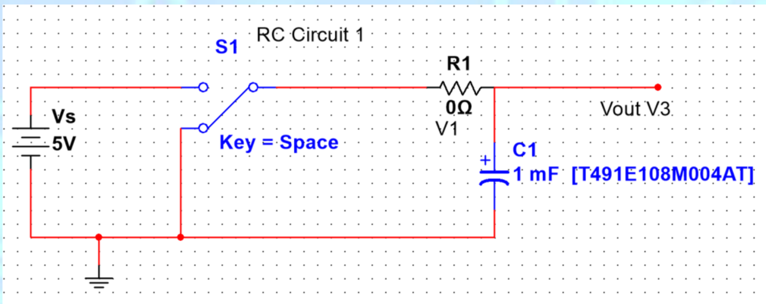


Figure 8 – Cap Discharge 1

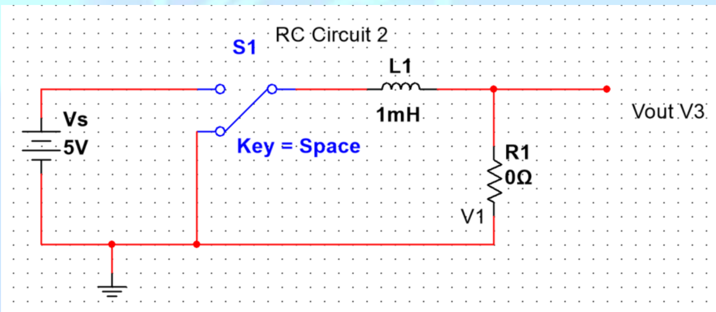


Figure 9 - Cap Discharge 2

Cap Discharging

V_c of (τ) = V_s . $e^{-τ/rc}$

Note: V^s may not be the same voltage as V^c

V = L . $\frac{di}{dt}$ the change in time

Energising

Energising

I of τ = I not . (I – e -τ/l/r)

De-energising

I of τ = I not . e -τ/l/r)

Note: “Not” = Current at any time

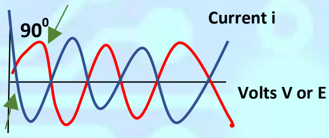
Time Domain

I C E

Current Voltage

Capacitance

Current leads voltage by 90^0



ICE – Current is always at its maximum when it’s on the zero plane

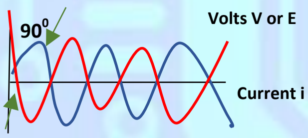
LC Circuit

E L I

Voltage Current

Inductance

Voltage leads Current by 90^0



Frequency Domain

The frequency-domain relates to the Sinusoidal Signals for both frequency and amplitude.

A =Amplitude
F = Frequency
T = Time (in Sec)
Φ= Phase Angle (Phi)
ω = Omega
Volts as a function of time =
$V(t) = A \,.\, \text{Sin}\,(2\pi.\, F.\, T\,) + \Phi$
OR we could use Omega $\omega = 2\pi\, F\, T$
$\therefore A \,.\, \text{Sin}\,.\, \omega + \Phi$
As we only care about Frequency and Amplitude so we can remove time and proceed as follows:

We require 10v RMS (Root Mean Squared). The Peak voltage from 10v rms is $V_1 = \frac{10v}{0.707} =$

14.144PK(peak) = 28.28v PP (point to point)

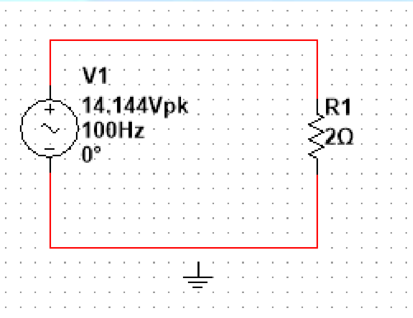


Figure 10 – Phase Circuit 1

Note1: The TVS value is always 0.707 of the peak value.

Note2: Over the resister, both the current and the voltage are in phase.

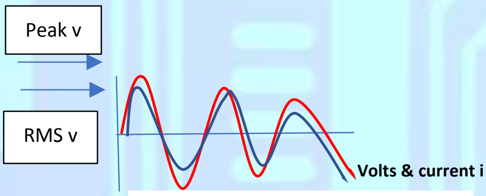


Figure 11 – Phase Shift

V_s =100 hz , 10v rms

$I = \frac{Vs}{R} = I = \frac{10v}{2r} = 5A$ rms

$V_s(t) = 140144 \,.\, \text{Sin}\,(2\pi.\, 100.\, T + \varphi)$

Rms Current $= \frac{V}{R} = \frac{14.144}{2} = 7.07mA$

$I(t) = 7.07mA \,.\, \text{Sin}\,(2\pi.\, 100.\, T + \varphi)$

Phase Diagram

Phaser Diag for Resistance

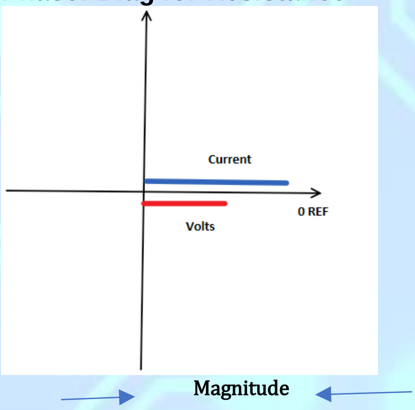


Figure 12 – Phase Diag 1

With Capacitor and Inductor, Current and Voltage are not in phase.

Capacitor – I C E Current leads voltage **by 90deg**

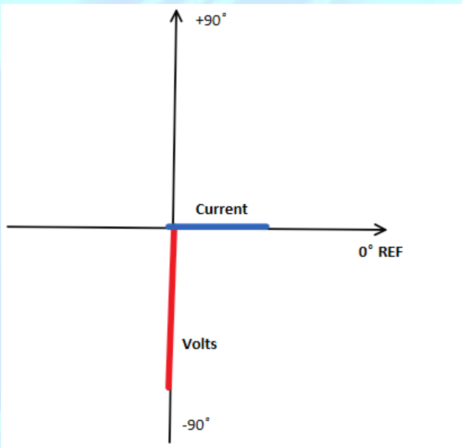


Figure 13 - Phase Diag 2

Inductor – E L I – Voltage leads current by 90deg

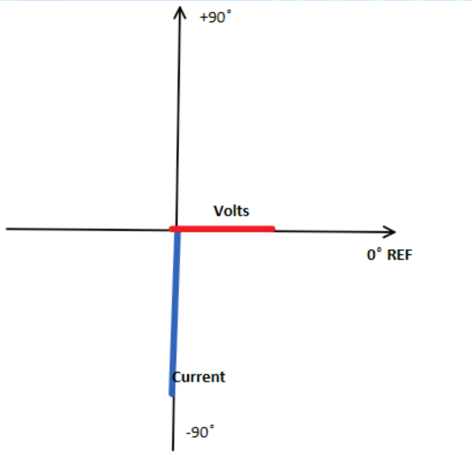


Figure 14 - Phase Diag 3

Phases

Reactance (X)

Reactance is the term for the AC resistance within a capacitor and Inductor.

As frequency goes up, Reactance (X_c) goes down. ∴ As frequency goes down, reactance (X_c) goes up. $X_c = \frac{1}{2\pi.F.C}$

Phaser Diag All Together - Reactance

+90˚

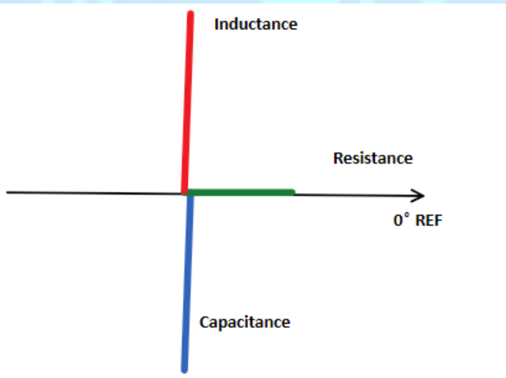


Figure 15 - Phase Diag 4

-90˚

Impedance (Z)

The definition of Impedance is for a component to resist current flow which has both a resistive and a reactive component. The term is used to cover both resistance and reactance components, the resistor is purely resistive, and the Capacitor and the Inductor is strictly reactive. Both are impedances they just happen to be 100% resistive in the resister and 100% reactive in the Capacitor and Inductor. Impedance is referred to when discussing AC signals, although this can include DC when the frequency gets down to 0. Thus:

- Resister is resistive
- Inductor and Capacitor are reactive
- Both are known as Impedances for AC circuits
- Impedance does cover the full frequency spectrum, which includes 0hz (DC) but is referred to in AC circuits.

When considering Impedance, there are always two parts.

- The Real part ($\Re$) and
- The imaginary part (j)
- $Z = \Re + X^{j}$

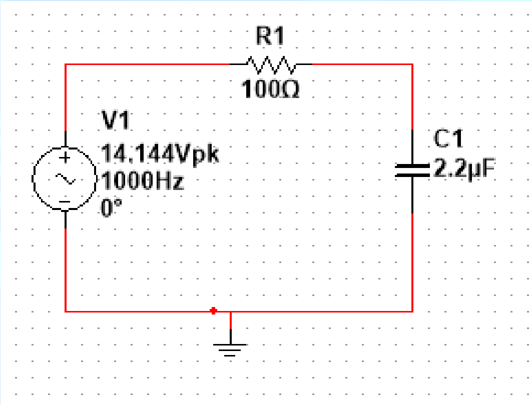


Figure 16 - Impedance Circuit

Polar Diag – Resistance - Magnitude

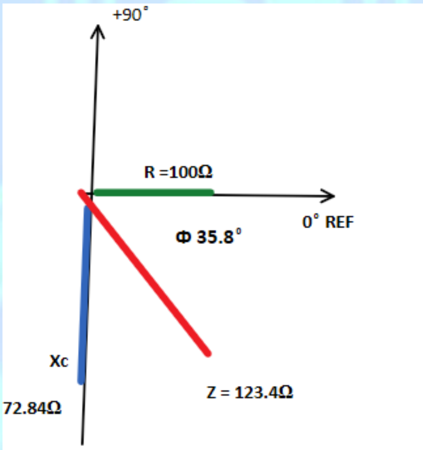


Figure 17 - Phase Diag 5

$V1 = 10v\ RMS \therefore X_c = \frac{1}{2\pi.F.C} = 72.34\Omega$

$Z = \sqrt{R^2 + Xc^2} = \sqrt{100^2 + 72.34^2} = Z = 123.42\Omega$

Current

$I = \frac{V}{Z} = \frac{10v}{123.42\Omega} = 81.0mA$

Volts:

$V = I.R \therefore V = I\,.\,Z$

$V_R = I.Z = 0.081A\,.\,100\ \Omega \quad = 8.1v$

$V_c = I.X_c = 0.081A\,.\,72.34\ \Omega \quad = 5.85v$

Note: $V_r + V_c$ does not add up to 10v due to the phase shift.

Polar Diag – Volts - Magnitude

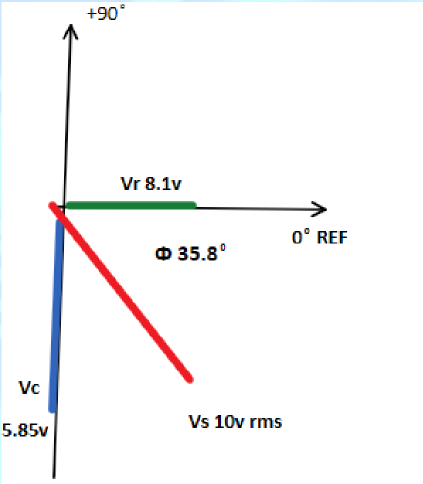


Figure 18 - Phase Diag 6

$\varphi = \text{Tan}^{-1}\frac{Vc}{Vr} = \text{Tan}^{-1}\frac{5.85v}{8.1A} = 35.8°$

Or

$\varphi = \text{Sin}\frac{Vc}{Vs} = \text{Tan}^{-1}\frac{5.85v}{10v} = 35.8°$

$\varphi = \text{Cos}\frac{Vr}{Vs} = \text{Tan}^{-1}\frac{8.1A}{10v} = 35.8°$

Inductor Note: All the above math is the same for Inductors except $X_L = 2\pi\,.\,F.\,L$

Polar / Phaser Format

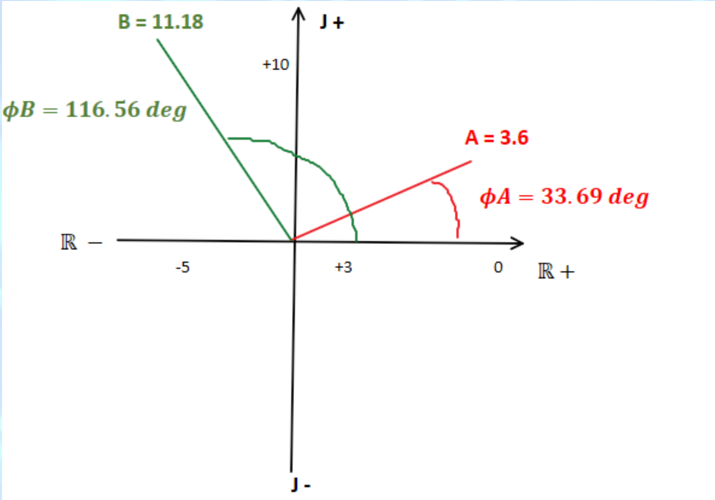


Figure 19 - Phase Diag7

Refer to Wikipedia **Imaginary Unit** to refer to Imaginary numbers.

$i^{-1} = -I$, $\frac{1}{i} = -I$, $i^2 = -1$

Impedance Z = $\mathfrak{R}$. (X . j) the "$\mathfrak{R}$" is the real number and the "x.j" is the imaginary number.

Complex numbers are a combination of real and imaginary numbers and defined by using an "*" superscript. E.g. A*

$A^* = a + (b.j)$

$B^* = c + (d.j)$

Magnitude – "II" is used to represent a magnitude variable, eg IC*I = $\sqrt{e^2 + f^2} = \text{Length}$

Angle of φ - φ $= \text{Tan}^{-1}\frac{Vc}{Vr}$ (inverse tan "Tan^{-1}" is the opposite over the adjacent e.g. V_c over V_a

Length of A (the length of the hypotenuse) - $\text{IAI} = \sqrt{e^2 + f^2}$

Polar Phaser Method

$A^* =$ IaI $\angle \varphi_1$

$B^* =$ IbI $\angle \varphi_2$

$\mathfrak{R}$ J "$\mathfrak{R}$" real and "J" imaginary parts

To add using Rectangle version, A* + B* = (a+b) + (b+d) . J

To multiply using Polar form multiply the magnitude and add the angles

A* X B* = (IAI . IBI) $(\angle\varphi_1 + \varphi_2)$

To divide using Polar form divide the magnitude and minus the angles A* / B* = $\frac{IAI}{IBI}$...... ($\angle\varphi_1 - \varphi_2$)

Calculating the diagram above, we see that A = a magnitude of 3 on the $\Re$ axis and B has a magnitude of -5 and on the "J" axis A = 2 and B=10.

Hence:
A= 3 + 2j
B = -5 + 10j
---- ----
$\Re$ Imaginary No.
To calculate all phase angles and voltages for the dig above - We require:

Requirement	Variable	Answer
Add A & B	A+B	
Length of A	IAI	
Length of B	IBI	
Angle of A	φa	
Angle of B	φb	
Multiply A.B using Phaser format	A . B	

A = 3 + 2j so on the $\Re$ line 3 graduations and the imaginary line 2 graduations.
B = -5 + 10j – same as above but on the imaginary line.

Length of A – IAI = $\sqrt{3^2 + 2^2}$
9 + 4 = 13
Square Root of 13 = 3.60

Length of B – IBI = $\sqrt{(-5)^2 + 10^2}$ = 11.180
Note1: Drop the minus sign
Note2: Using the Casio calculator, it's easier to calculate the numbers, then the square root. E.g. $A^2 + B^2$ = then SquRoot Ans.

Calculate ϕa The opposite over the adjacent = 2 over 3.
∴ ϕa = Tan-1 $\frac{2}{3}$ = 33.69˚

Calculate ϕb ϕb = Tan-1 $\frac{10}{-5}$ = -63.4˚
Note:3 Use the minus 5 in this case as the phase angle is on the negative side.
Note:4 Because ϕb is on the negative side, we need the plus side angle, so 180˚- 63.4˚= 116.56˚ refer diag above.
A* = 3.6 ∠ 33.69˚

B* = 11.8 ∠ 116.56˚

A . B = (3.6 . 11.18) ∠ 33.69 + 116.56 = 40.2 ∠ 150.25

A/B = ($\frac{3.6}{11.18}$) ∠ 33.69 - 116.56 = 40.2 ∠ 150.25

Polar Calc 2

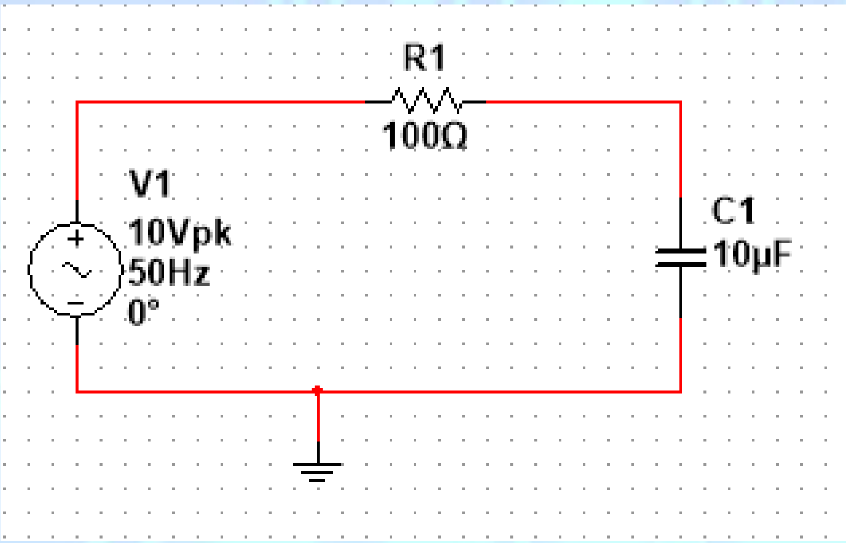


Figure 20 – Polar / Phasor Circuit

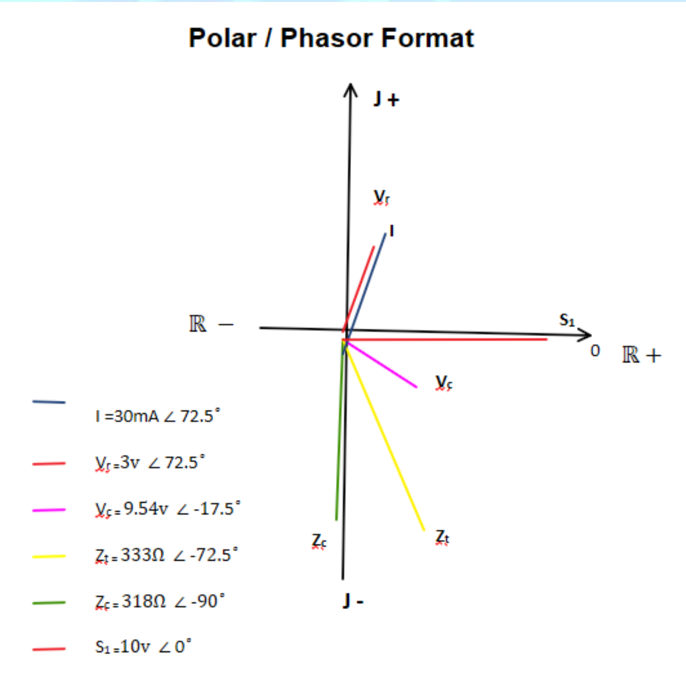


Figure 21 - Phase Diag 8

Capacitive Reactance

$X_c = \frac{1}{2\pi.F.C} = X_c = \frac{1}{2\pi.50.10\mu F} = 318.3\Omega$

Some Impedances (Rectangular form)

Z_c = 0 -j . X_c = 0 -j . 318 = -j318.3Ω
$Z_r = 100 + j0 = 100\Omega$ (Resistances do not have an imaginary number)

Polar Form

To convert Rectangular to Polar form, take the square root of the sum of the squared values.

$Z_r = \sqrt{Zr^2 + 0^2} = Z_r = 100\ \Omega\ \angle 0^\circ$ (Resistances do not have an imaginary number)

$Z_c = \sqrt{0^2 + Xc^2} = Z_c = \sqrt{0^2 + 318.3\Omega^2} = Z_c = 0\ \angle 318.3\ \Omega$ at -90˚

Now work out Volts and Current

V = I . R = I . Z for impedance.

Total Impedance = $Z_t = Z_c + Z_r$ = 0 -j318 + 100 j0

Because Z_c does not have a real part and Z_r does not have an imaginary part – Then:

Z_t = 100 -j318

The voltage source V_s has a phase angle of 0˚ as the reference point for the circuit.

∴ $V_s = 10v\ \angle 0^\circ$ or 10 +j0

I = $\frac{V}{Zt}$ = $\frac{10v\ \angle 0^\circ}{333\Omega\ \angle -72.5^\circ}$ = $\frac{10}{333}$ = 30mA ∠ 0 – (-72.5) = 72.5

∴ $I = 30mA\ \angle 72.5^\circ$

To calculate the magnitude (II) of Z_t = I Z_tI = $\sqrt{Zr^2 + (Zc^2)}$ = I Z_tI = $\sqrt{100^2 + (-318^2)} = 333\Omega$

Note: forget the minus as -318 becomes +318 or 318Ω.

Now the phase angle. The imaginary part over the real part = $Tan^{-1} \frac{-318}{100}$ = -72.5˚

∴ I Z_tI = 333Ω ∠ -72.5˚

The voltage and impedance angle over the capacitor = (multiply the real numbers and add the imaginary numbers)

V_c = I . Z_c (i∠ + Z_c ∠) = V_c = (30mA ∠ 72.5˚) . (318Ω ∠ -90˚)

∴ (30mA . 318Ω) (∠ 72.5˚ + ∠ -90˚) = V_c = 9.54v ∠ -17.5˚

V_r = I . Z_r ($i\angle + Z_r\angle$) = V_r = (30mA ∠ 72.5˚) . (100Ω ∠ 0˚)

∴ (30mA . 100Ω) (∠ 72.5˚ + ∠ 0˚) = V_r = 3v ∠ 72.5˚

Power – via the resistor = I. V_r = 30mA . 3v = 0.09w or 90mW

Note: Real power is not dissipated, but reactive power is.

Note: There is zero power being dissipated over the Capacitor because the current and the voltage are 90˚ out of phase, and therefore no power can be dissipated – impossible.

Note: look up Phasor on Wikipedia.

In summary:

I = 30mA ∠ 72.5˚ - Blue

V_r = 3v ∠ 72.5˚ - Red

V_c = 9.54v ∠ -17.5˚ - Purple

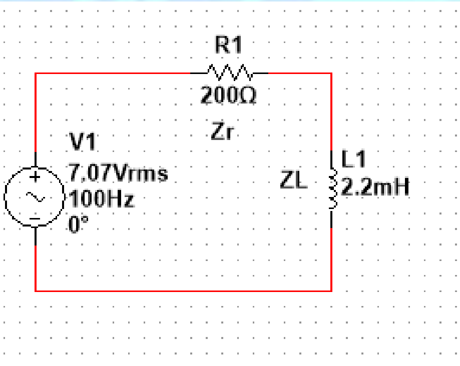
Z_c = 0 ∠ 318.3 Ω at -90˚ -

Z_r = 100 Ω ∠ 0˚ - on the zero line

I Z_tI = 333Ω ∠ -72.5˚ - Yellow

V_s = 10v ∠ 0˚ - Red

Note: the angles are turned around because we have mixed the components e.g. both a Resister and a Capacitor.



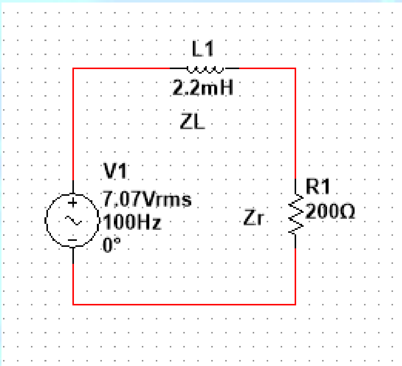


Figure 22 – Inductor Reactance Circuit

V^s = 7.07v RMS – 100 Htz

If we look at the above circuits from the Impedance point of view, they could be considered a Voltage Divider, e.g. $V_2 = V_1 . \frac{ZL}{ZR+ZL}$

Inductor Reactance (X)

Inductors are mostly measured in Micro “µ” or Milli “m” Henry’s and it’s difficult to design circuits that do anything at low frequencies. The 200 Ω resister was picked because at a frequency of 100 Hz the calculated inductance will be acceptable for low frequencies, but useless for higher frequencies.

Inductor Reactance Calculations

$X_L = 2\pi \,.\, F \,.\, L = X_L = 2\pi.100\text{Hz} \,.\, 2.2\text{mH} \quad = 1.31\ \Omega$
$\quad\text{@ 1k Hz} \quad = 13.8\ \Omega$
$\quad\text{@ 10k Hz} \quad = 138\ \Omega$
$\quad\text{@20k Hz} \quad = 276\ \Omega$

Complex Impedance (Z)

$Z_r = 200\ \Omega + 0j$ OR $200\ \angle 0^\circ$
$Z_L = 0 + X_L \,.\, j = 0 + 1.38j$ OR $1.38\ \Omega\ \angle +90^\circ$

Calculate Current and Voltage

V = I . R OR I . Z

$= \frac{V}{Zt} = \frac{7.07v\ \angle 0^\circ}{200\Omega\ \angle 0.4^\circ} = 35.35\text{mA}\ \angle 0.4^\circ$

Note: 0.03535 = 35.35mA

$Z_t = Z_r + Z_L = 200 + 0j + 0 + 1.38j$
$= 200 + 1.38j$
$=200\ \angle 1.38\ \Omega$

Find the length

$$\text{Len} = \sqrt{\text{Zr}^2 + \text{ZL}^2} = \sqrt{200^2 + 1.38^2} = 200.004$$

$$\varphi_1 = \text{Tan}^{-1}\frac{1.35}{200} = 0.39$$

The above result indicates that due to the low frequency, the Inductor is not doing anything because the Inductor at this frequency has an Impedance of 1.38 Ω. Remember, the Inductor is a frequency dependant Resister, and its Impedance goes up linearly with the increase in frequency.

Voltage over the Resister

$V_r = I \,.\, Z_r = (35\ \angle -0.4) \,.\, (200 \angle 0)$
$V_L = I \,.\, Z_L = (35\ \angle -0.4) \,.\, (1.38 \angle 90)$

$V_r = (35\text{mA} \,.\, 200\ \Omega)\quad (-0.4 + 0) = 7v \angle 0^\circ$
$V_L = (35\text{mA} \,.\, 138\ \Omega)\quad (-0.4 + 90) = 48.3mv \angle 89.6^\circ)$

Decibels

The tables below describe the db levels from 100db to -100db and display the significant differences in power and amplitude ratios between the two 100 and -100db levels.
The second table describes the power and voltage gain equations.

DBM = Power 1mw
DBV = Voltage 1v rms
DB = The Ratio

The 20 log rule below states that the voltage gain and the power gain are the same if (and only if) the input and output impedances are the same. This means that if the output impedance of one electronic circuit section has the same Impedance of the input section, then the power will be at "Maximum Power Transfer".

To compute power, use the 10 Log db scale and for voltage use the 20 Log db scale (detailed below).

dB	Power ratio	Amplitude ratio
100	10 000 000 000	100 000
90	1 000 000 000	31 623
80	100 000 000	10 000
70	10 000 000	3 162
60	1 000 000	1 000
50	100 000	316.2
40	10 000	100
30	1 000	31.62
20	100	10
10	10	3.162
6	3.981 ≈ 4	1.995 ≈ 2
3	1.995 ≈ 2	1.413 ≈ $\sqrt{2}$
1	1.259	1.122
0	1	1
−1	0.794	0.891
−3	0.501 ≈ ½	0.708 ≈ $\sqrt{½}$
−6	0.251 ≈ ¼	0.501 ≈ ½
−10	0.1	0.316 2
−20	0.01	0.1
−30	0.001	0.031 62
−40	0.000 1	0.01
−50	0.000 01	0.003 162
−60	0.000 001	0.001
−70	0.000 000 1	0.000 316 2
−80	0.000 000 01	0.000 1
−90	0.000 000 001	0.000 031 62
−100	0.000 000 000 1	0.000 01

An example scale showing power ratios x, amplitude ratios $\sqrt{x}$, and dB equivalents $10 \log_{10} x$.

Figure 23 – Decibel Table

Logarithmic units and decibels [edit]

Power gain [edit]

Power gain, in decibels (dB), is defined as follows:

$$\text{gain-db} = 10\log_{10}\left(\frac{P_{\text{out}}}{P_{\text{in}}}\right)\ \text{dB},$$

where P_{in} is the power applied to the input, P_{out} is the power from the output.

A similar calculation can be done using a natural logarithm instead of a decimal logarithm, resulting in nepers instead of decibels:

$$\text{gain-np} = \frac{1}{2}\ln\left(\frac{P_{\text{out}}}{P_{\text{in}}}\right)\ \text{Np}.$$

Voltage gain [edit]

The power gain can be calculated using voltage instead of power using Joule's first law $P = V^2/R$; the formula is:

$$\text{gain-db} = 10\log\frac{\frac{V_{\text{out}}^2}{R_{\text{out}}}}{\frac{V_{\text{in}}^2}{R_{\text{in}}}}\ \text{dB}.$$

In many cases, the input impedance R_{in} and output impedance R_{out} are equal, so the above equation can be simplified to:

$$\text{gain-db} = 10\log\left(\frac{V_{\text{out}}}{V_{\text{in}}}\right)^2\ \text{dB},$$

$$\text{gain-db} = 20\log\left(\frac{V_{\text{out}}}{V_{\text{in}}}\right)\ \text{dB}.$$

This simplified formula, the 20 log rule, is used to calculate a **voltage gain** in decibels and is equivalent to a power gain only if the impedances at input and output are equal.

To Recap Filters

To compute Gain

Transfer Function: $H(_f) = \frac{1}{1+j\,(2\pi.F.R.C)}$

Magnitude: $IH(_f)I = \frac{1}{\sqrt{1^{2}+j^{2}\ (R^{2}+Xc)^{2}}}$

Cut-off Frequency: F = of $\frac{1}{2\pi.R.C}$

If we make R=1 then we get $\frac{1}{\sqrt{2}}$ = 0.707 = -3db

Logarithms

Basic info for Logs and Exponents

- 10^3 where 10 = the base and 3 = the exponent = 10.10.10, 2^8 = 256, 2^0 = 1
- $10^{-3} = \frac{1}{10\uparrow3} = \frac{1}{1000} = 2^{-1} = \frac{1}{2}$
- Add: $10^3 + 10^5 = 10^8$ Add the exponents.
- Divide: $10^9 / 10^5 = 10^{(9-5)} = 10^4$ Subtract the exponents.

Logs

10^3 = 1000 Log (base 10) 1000 = 3 ∴ $\text{Log}\ y = x\ = 6^x = y$

Voltage gain: $\text{db} = 20\text{Log}\ \frac{Vout}{Vin}$

$20\text{db of gain} = \text{db}\ 20\ \text{Log}\ \frac{Vout}{Vin} = 20\text{db}$

$= \frac{20}{20} \text{ Log } \frac{Vout}{Vin} = \frac{20}{20} = \text{Log } \frac{Vout}{Vin} = 1\text{db}$

$= \text{Log (base 10) } \frac{Vout}{Vin} = 1\text{db } = 10^1 = 10 = \frac{Vout}{Vin}$

From above: 1 db ≡ X and $\frac{Vout}{Vin}$ ≡ Y and 10 ≡ Base ∴ $10^1 = 10$ ∴ the voltage gain =10

For -3 db

Gain db = -3db = 20 Log $\frac{Vout}{Vin} = \frac{-3db}{20} \text{ Log } \frac{Vout}{Vin} = \frac{-3}{20} \text{ Log (base 10) } \frac{Vout}{Vin} =$

$10^{(-3/20)}$ = Voltage gain of $\frac{Vout}{Vin} = 10^{(-3/20)} = 0.707$

$\equiv G = \frac{1}{\sqrt{2}} = (-0.707)$ same as the gain of the Low Pass Filter = $\frac{1}{(2\pi.R.C)}$

CAPACITORS

Mr Carlson Comments

Name		+/- PPM/0C	Temp
NP0 (zero) Negative Positive (or C0G)	Extremely Stable found in Oscillators, RF and Coupling circuits – Does not move with temp changes.	0-30	-55c +125c
Mica	Very Stable over a wide temp range, same circuits as above, they rarely fail.	50	Really good temp stability
Polystyrene	OK if Stable, found in Tuners, not good for RF oscillators – Very poor with temp. They will change when hot and the stay at the changed setting		Temp Sensitive
Ceramic – X5R	Disc Type - Great for Bypass, Never use for Oscillator or tuned circuits, or Audio signal path – Very poor with temp.		-55 - +85
Ceramic – X7R	Ditto – Not good if vibrations e.g. Audio		-55 - +125
Ceramic – Y5V	Ditto		-30 - +85
Ceramic – Z5U	Ditto		+10 - +85
Polypropylene	Good for Audio and RF stage coupling – Not as Stable as above. Never use for Oscillator or tuned circuits. Good vibration tolerance.		Reasonably good for heat tolerance
Tantalum	Bad for Audio (Non-Linear) very good for bypass or timing e.g. 555 circuits		Temp Stable
Paper	No good at all		

Notes: Tantalum – Tantalum Caps would be better than Electric if you could afford them.

What is power decoupling

A decoupling capacitor's job is to suppress high-frequency noise in power supply signals. They take tiny voltage ripples, which could otherwise be harmful to delicate ICs out of the voltage supply. ... Decoupling capacitors are used to connect between the power source (5V, 3.3V, etc.) and ground.

Coupling and decoupling

While decoupling capacitors are connected in parallel to the signal path and are used to filter out the AC component.
Coupling capacitors, on the other hand, are connected in series to the signal path and are used to filter out the DC component of a signal. They are used in both analogue and digital circuit applications.

Why is decoupling necessary?

Decoupling capacitors help to provide a local instantaneous charge source that prevents the voltage source from dipping and a bypass path that dampens ringing. Noise on the PCB is also locally damped, helping the local circuit remain unaffected by ripple on the power plane that could otherwise disturb the circuit.

Capacitor Charging and Discharging

Charging

Current: I = C. - $\frac{dv}{dt}$ Where dv = small voltage charge and dt = small change in time.

If C = 100uF, V = 1v. τ = 1mSec Then I = 100uF . $\frac{1v}{1uS}$ $\therefore \mathrm{I} = 100\mathrm{A}$

Note: $\tau = \mathrm{Tau}$ *or RC Rate of Charge in relation to time.*

In most cases, 1 x τ = 63% of the charge relating to time. Therefore 5 x τ would = 99.99% of the charge.

$\tau = 2.2 \,.\, 10^{-3}$ Sec = 1 x RC time = approx. 63%

For charging = v = v^{i} x (1 -e - τ /RC)

Note: Caps can pull a high current (Current Rush) if there is not a resister to limit the input current in place, above 10A.

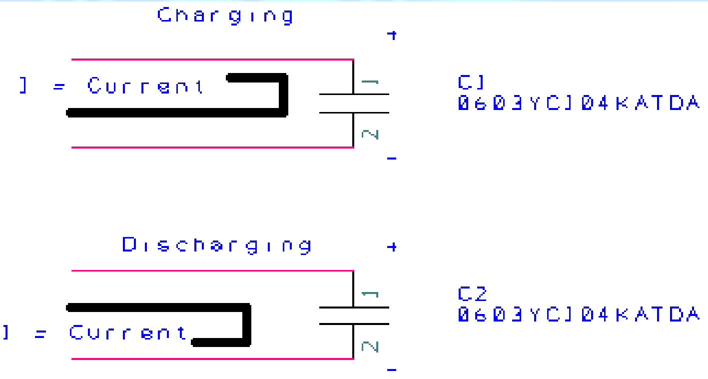


Figure 24 – Current Flow Circuits

When charging current flows from Positive to negative But when discharging current flows from negative to positive.

ESR – Equivalent Series Resistance

ESR is always an AC resistance, it is measured at specified frequencies, 100 kHz for switched-mode power supply components, 120 Hz for linear power-supply components, and at its self-resonant frequency for general-application components. Additionally, audio components may report a "Q factor", incorporating ESR among other things, at 1000 Hz. – DC Voltage has a frequency of 0 hz.

Parallel Capacitors

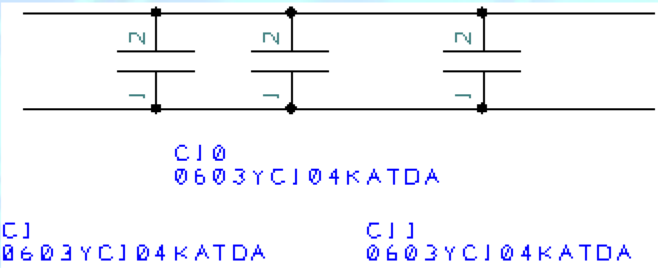


Figure 25 – Parallel Capacitors

The math for parallel capacitors is the same as series resistors.

Total C = C1 + C2+ C3

Series Capacitors

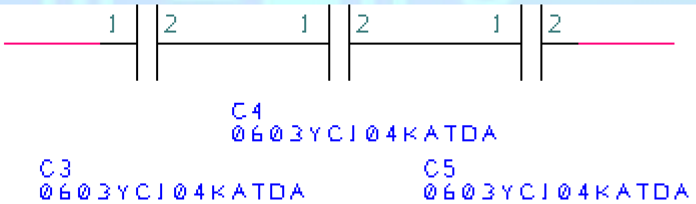


Figure 26 – Serial Capacitors

The math for series capacitors is the same as parallel resistors.

$$C = \frac{1}{\frac{1}{100} + \frac{1}{47} + \frac{1}{100}}$$

OR (1/100) + (1/47) + (1/100) x^{-1}

Typical Frequency noise bypass circuit

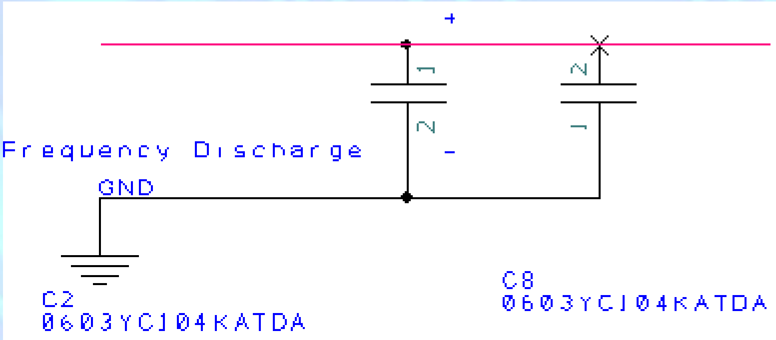


Figure 27 - Frequency noise bypass circuit

RC Filters

Filter tools

Filter design and analysis - http://sim.okawa-denshi.jp/en/Fkeisan.htm
Also held under "Calculators."

The impedance and Reactance Symbols

Z_R = Resister Complex Impedance
Z_C = Capacitor Complex Impedance
X_L = Inductor Complex Impedance
X_C = Capacitor Complex Reactance
X_L = Inductor Complex Reactance
H = Transfer Function

Low Pass Filter

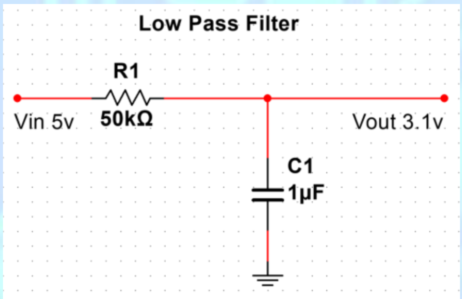


Figure 28 – Low Pass Filter Circuit

The impedance for R_1 = $Z_r = R_1 + 0j$

The impedance for C_1 = $Z_c = 0 – X_c . j$

Where $X_c = \frac{1}{2\pi.F.C}$

Note Capacitor reactance

With Capacitors, as the frequency goes up, the reactance goes down.
F ↓0 Then X_c ↑∞
F ↑∞Then X_c ↓0

Compute the voltages

From a mathematical point of view, the filter circuit is very similar to a voltage divider circuit. This is also known as a Transfer function. Hence:

$V_{out} = \left(\frac{Zc}{Zc+Zr}\right) . V_{in}$

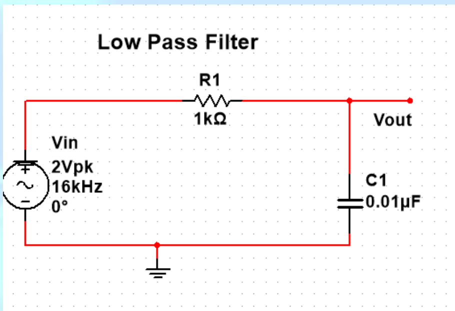


Figure 29 - Low Pass Filter Circuit 2

Low pass filter data

V_{in} = 2v p-p 16k Htz
R = 1k Ω
C = 0.01mF

$X^c = \frac{1}{2\pi.F.C}$

Resistors are real and do not have an imaginary part. Complex impedance $Z_r = R + 0j$
Capacitors and Inductors are reactive and only have an imaginary part and do not have a real part. Complex impedance $Z_c = 0 \;\; X_c .j$

$X_L = 2\pi.$ F. L. (for reference only)

As above $X_c = \frac{1}{2\pi.F.C}$ and V_{out} = V_{out} = $\left(\frac{Zc}{Zc+Zr}\right) . V_{in} . V_{in}$ Then to calculate the Transfer Function:
$V_{out} = \left(\frac{Zc}{Zc+Zr}\right) . V_{in} = \frac{Vout}{Vin} = \left(\frac{Zc}{Zc+Zr}\right) = H_{(f)}$ - The only variable in the impedance calculation is Frequency ∴ $H_{(f)}$ = The Transfer Function. After some Algebra, we have $H_{(f)} = \frac{1}{1+j\,(2\pi.F.R.C)}$
∴ Can we get rid of the "j" and calculate the magnitude (the square root of the real and imaginary parts) $|H_{(f)}| = Z = \sqrt{R^2 + Xc^2}$
∴ $|H_{(f)}| = \frac{1}{\sqrt{1^{2+j2}\;(R^2+Xc)^2}}$ at the frequency of $\frac{1}{2\pi.R.C}$ then the output of this $|H_{(f)}|$ function = 0.707 or 70%. This is known as the cut off frequency when the magnitude of $H_{(f)}$ is at 70%.

Set Filter for a specific frequency

$F = \frac{1}{2\pi.R.C}$ We need to make a filter that cut in at 2khz ∴ $2kHtz = \frac{1}{2\pi.R.C}$ but as "R" and "C" are two variable numbers, we should pick one. As Resister values are plentiful, we will choose the value of the Capacitor and calculate the Resister. The Capacitor value will be 0.01mF. So now

$2kHtz = \frac{1}{2\pi.R.0.01mF}$ ∴ $R = \frac{1}{2\pi.2khz.0.01mF} = 795\ \Omega$. But if we only have a 1k Ω Resister, then the frequency would be $F = \frac{1}{2\pi.1k.0.01} = 1591$ Htz.

Passive filters can have 1,2 or 3 poles to make them stiffer (rate of drop off), if more poles are required, they would need to be Active Filters.

High pass filter L51

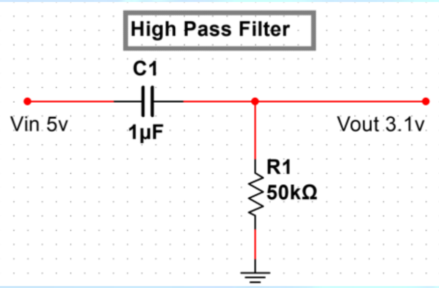


Figure 30 - High Pass Filter Circuit

Cap Reactance $X^c = \frac{1}{2\pi.F.C}$

Cap Impedance $Z_c = 0 \;\; -X_c$ J Note: $-J = \frac{1}{j}$

Complex Impedance for $R = Z_R = 0 \; + 0J$

voltage - $Vout = Vin\ .\ (\frac{Zr}{Zr+Zc})$ Just like a Voltage Divider.

Transfer Function - $H(f) = (\frac{Zr}{Zr+Zc}) \; = \frac{R}{R+(-Xc\ .j)}$ ∴ $H(f) \; = \frac{J.2\pi.R.C}{1= J.2\pi.R.C}$

Magnitude or Gain – real part = Rp & imaginary part Jp = $IH(f)I \; = \sqrt{Rp^2 + Jp^2}$

∴ $JpIH(f)I \; = \frac{2\pi.F.R.C}{\sqrt{1^{+} \; (2\pi.F.R.C)^2}}$

High Pass Cut off Frequency – $F(hp) = \frac{1}{2\pi.R.C}$

Note: Both High and Low pass filters have the same Cut Off formula. This is the charging constant of the circuit; an RC circuit does not care about the "R" and "C". The charge is going to happen the same way. However, the action of the Current is going to be different depending on where you take the Voltage. If we take the Voltage reading from Vout in the High Pass Filter, it will be different to the Voltage reading from Vout in the Low Pass Filter, because the "C's" are changing as a function of frequency.
At a frequency of 0hz the gain for the High Pass Filter is 0 and for a Low Pass Filter it is 1.0. As F goes to ∞ then, the gain goes to 1.

Note: R – 1k and C = 0.1uF

The frequency for the High Pass Filter = $F(hp) = \frac{1}{2\pi.R.C} = F(hp) = \frac{1}{2\pi.1k.0.1uF} = 1591Hz$

Band Pass Filter

The combination of both the High and Low pass filters will result in a Band Pass Filter, and when placed in series, their filter curves become multiplied.

Inductors

Inductors are the opposite of capacitors, as the frequency goes up the reactance goes up, and as the frequency goes down, the reactance goes down. Inductors can be thought of as a linear frequency resister. $X_L = 2\pi \cdot F \cdot L$

An Inductor also called a coil, choke, or reactor. It is a passive two-terminal electrical component that stores energy in a magnetic field when electric current flows through it.[1] An inductor typically consists of an insulated wire wound into a coil around a core.

When the current flowing through an Inductor, the change in the time-varying magnetic field induces an electromotive force (e.m.f.) (voltage) in the conductor. Described by Faraday's law of induction. According to Lenz's law, the induced voltage has a polarity (direction) which opposes the change in current that created it. As a result, inductors oppose any changes in current through them.

An inductor is characterised by its inductance, which is the ratio of the voltage to the rate of change of current. In the International System of Units (SI), the unit of inductance is the “Henry” (H) named for 19th century American scientist Joseph Henry. In the measurement of magnetic circuits,

The Capacitor pushes back current, and the Inductor pushes back voltage. For Lenz’s law and Faraday’s law of Induction refer Wikipedia. Note the Induction Right Hand Rule, check this out on Wikipedia and why we need to know?

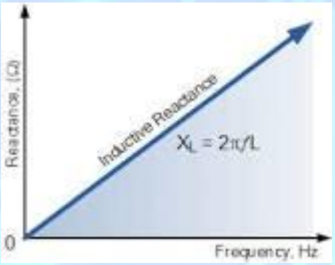
The symbol for Inductance is Φ(Phi = Flux) $\Phi = L * I$ or Inductance * Current.
Capacitor Charge (Coulomb’s) – The symbol for charge is Q = $Q = C \cdot V$

Voltage and current used to charge the Inductor and Capacitor – Transient Response (time domain)

Inductor = V = -(L . $\frac{di}{dt}$) Capacitor = I = (C . $\frac{dv}{dt}$)

Inductor	**Capacitor**
Energise = Inductance * Current $\Phi = L * i$	Charge = Capacitance * Volts Q = C . V
The Derivative	*The Derivative*
$V = \frac{d\Phi}{dt} = L \cdot \frac{di}{dt}$ *OR* $\frac{\Delta\Phi}{\Delta t} = L \cdot \frac{\Delta i}{\Delta t}$	$i = \frac{dq}{dt} = C \cdot \frac{dv}{dt}$
Inductor current change in unit by time	*Capacitor voltage change in unit by time*

- **Inductors Block high frequency**
- **Capacitors block Low Frequencies**



Inductive Reactance:

$$X_L = 2\pi f L = 2\pi \times 50 \times 0.15 = 47.12\Omega$$

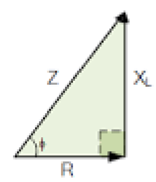
Current:

$$I = \frac{V}{X_L} = \frac{100}{47.12} = 2.12A$$

$$X_L = 2\pi f L$$
$$X_L = 2\pi.50.0.5$$
$$X_L = 157\Omega$$

$$Z = \sqrt{R^2 + X_L^2}$$
$$Z = \sqrt{30^2 + 157^2}$$
$$Z = 160\Omega$$

$$V = I \times Z = 4 \times 160 = 640V$$



Impeadance Triangle

$$Z = \sqrt{R^2 + X_L^2}$$

$$\tan\Phi = \frac{X_L}{R}$$

$$\sin\Phi = \frac{X_L}{Z}$$

$$\cos\Phi = \frac{R}{Z}$$

Inductive reactance is the name given to the opposition to changing current flow. This Impedance is measured in ohms, just like resistance. In Inductors, voltage leads current by 90 degrees.

Capacitive reactance (in ohms) decreases with increasing AC frequency. Conversely, inductive reactance (in ohms) increases with increasing AC frequency. Inductors oppose fast-changing currents by producing higher voltage drops; capacitors oppose faster changing voltage drops by allowing greater currents.

Reactance $= \mathrm{Xl} = 2\pi . F . L$

Inductors in Series

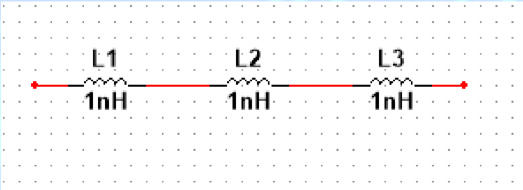


Figure 31 - Inductors in Series

$L_{tot} = L1 + L2 + L3$

Inductors in Parallel

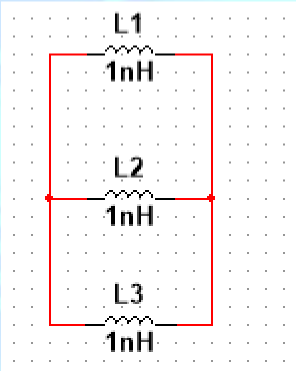


Figure 32 - Inductors in Parallel

$L_{ToT} = \frac{1}{\frac{1}{L1}+\frac{1}{L2}+\frac{1}{L3}}$

LC Filters

Low pass filter L52

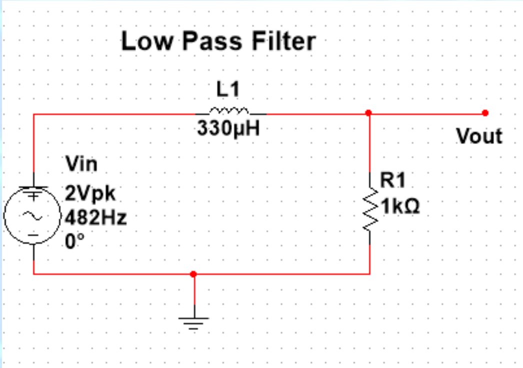


Figure 33 – LC Low Pass Filter Circuit

RC and RL circuits both for Capacitance and Inductance are the opposites of each other. Looking at the transfer function, or the reactance of the Inductor, compared to the Capacitor for each circuit. They are the inverse of each other:

Eg Reactance - $X_L = 2\pi . F. L$ and $X_c = \frac{1}{2\pi.F.C}$

Impedance – $Z_L = 0 + I.X_L$ and $Z_R = R + 0J$

Volts - Vout = Vin . $\frac{Zr}{Zr+ZL}$

Frequency – H(f) = $\frac{R}{R+(J.XL)}$ =

Divide by R - H(f) = $\frac{R/R}{R/R+(J.XL)}$ = H(f) = $\frac{1}{1+(2\pi.F.L)/R}$ =

Transfer Function = H(f) = $\frac{1}{1+(2\pi.F.L)/R}$

Magnitude - $\text{IH(f)I} = \frac{1}{\sqrt{1^{+}\ (2\pi.F.R.L/R)^2}}$ - At DC or F – 0hz – At F = 0 then IH(f)I = 1

Cut of Frequency = $\frac{1}{\sqrt{1+?}}$ = ? = $(2\pi.F.R.L/R)^2 = 1$

Take the square root of both sides = $(2\pi.F.R.L/R) = 1 =$

$F = \frac{R}{L.2\pi}$ - R = 1k , L = 330uH note: 330uH = 330e-6

Frequency Low pass = $F(_{LP}) = \frac{1k}{330uH.2\pi} = 482.28Hz$

Inductors have an extremely high “Q” (quality factor) which makes the Inductor very pure. Capacitors have a lot of extraneous components. E.g. resistance, capacitance and inductance and a small amount of reactance, Therefore the Inductor is a much purer part.

High pass filter L53

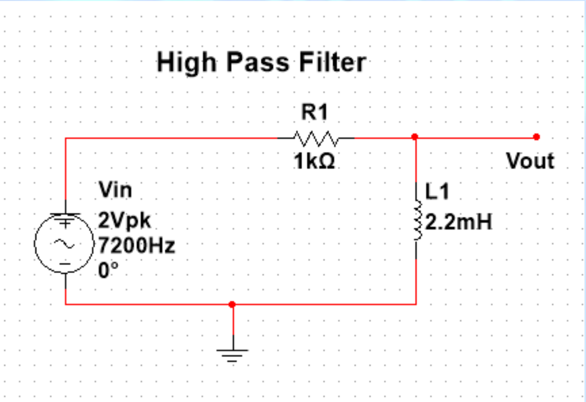


Figure 34 – LC High Pass Filter Circuit

Inductors have an extremely high "Q" (quality factor) which makes the inductor very pure.

Capacitors have a lot of extraneous components

E.g. Reactance - $X_L = 2\pi . F. L$

Impedance – $Z_L = 0 + I.X_L$ and $Z_R = R + 0J$

Note: RC section above, with X_c the :J" is -J because the inductive reactance are in opposite directions.

Volts - Vout = Vin . $\frac{ZL}{Zr+ZL}$ This can also be expressed as : $\frac{Vout}{Vin} = \frac{J.2\mu.F.L}{R+J.2\pi.F.L}$

Divide by j.2µ.f.L - H(f) = H(f) = $\frac{1}{1+R/(2\pi.F.L)}$ = H(f) = $\frac{1}{1-J.R/(2\pi.F.L)}$

Transfer Function = H(f) = $\frac{1}{1+(2\pi.F.L)/R}$

Magnitude - $\mathrm{IH(f)I} = \frac{1}{\sqrt{1^{+}\ (\mathrm{R}/(2\pi.\mathrm{F.R.L}))^2}}$

We now need to make the term "(R/(2π. F. R. L))2" $= 1$

1- $(\frac{R}{2\mu.F.L})^2 = \sqrt{1}$

2- $\frac{R}{2\pi.F.L} = 1$

3- $\frac{2\mu.F.L}{R} = \mathrm{R}$

4- $F = \frac{R}{2\pi.L}$ = Cut Off Frequency

Note: To have multiple filter poles, you would need to calculate the transfer function of each one and then compute their magnitude and phase angle. When using numerous poles, the transfer functions are multiplied together e.g. TF1 + TF2 + TF3 Etc. This will allow the extra pols to be added, resulting in a steeper curve. But Active filters would be more efficient for more than two poles.

Frequency – F = $\frac{R}{L} = \frac{1k}{2\pi.2.2mH}$ = 72.343 kHz

For more reading refer RC Circuits in Wikipedia.

Diodes L55

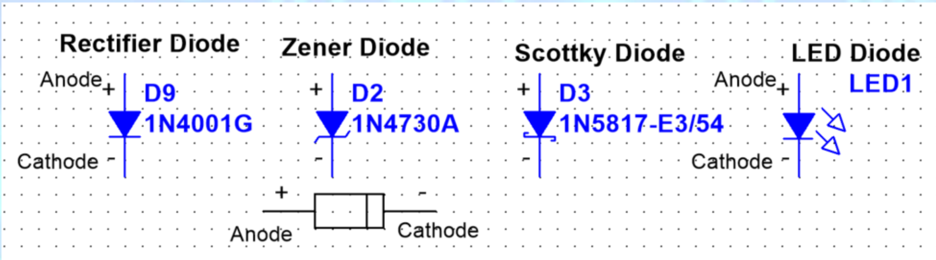


Figure 35 – Diode Identification

LEDs – In most cases, the longer lead connects to the Anode. If a round LED has a flange around its base, a flat spot on the base will be closest to the cathode side of the component.

Diodes allow current to flow in one direction.

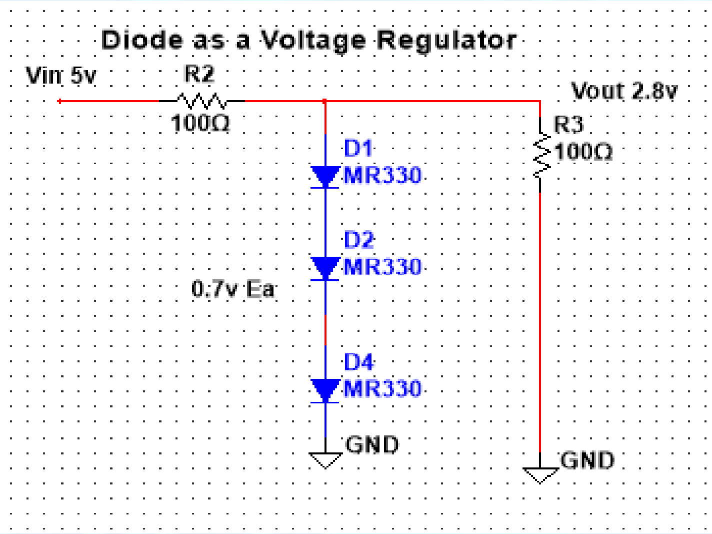


Figure 36 – Diode as a Volt Regulator Circuit

The silicone diodes total forward voltage of 2.1v and the diodes will maintain this voltage droppage subject to the amount of forward current available. Diodes use a different forward voltage drop depending on the current output; this should be verified in the datasheet.

R2 is a current limiter, and R3 is the load for 100Ω, 1k and 10k

When calculating current in this type of circuit, it is difficult to manually calculate an accrete amperage because the diode voltage and current needs to stabilise before an accurate value can be calculated. This is best done using Circuit Simulation. However, we can get close:

I = $\frac{V}{R}$ = $\frac{5v-2.1v}{100\Omega}$ = 29mA when there is no load and the current is only going over the diodes

At 100 Ω Load. = I = $\frac{V}{R}$ = $\frac{5v-2.1v}{100\Omega}$ = 29mA

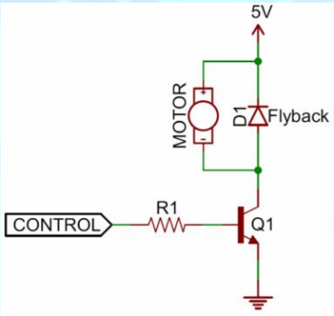
At 1k Ω Load. = I = $\frac{V}{R}$ = $\frac{5v-2.1v}{1000\Omega}$ = 2.1mA = 29-2.1 =26.9mA

At 10k Ω Load. = I = $\frac{V}{R}$ = $\frac{5v-2.1v}{10000\Omega}$ = 0.21mA = 29 – 0.21 = 28.79mA

Flyback Diodes and Voltage Spike Suppression

Diodes are very often used to limit potential damage from unexpected large spikes in voltage. Transient-voltage-suppression (TVS) diodes are speciality diodes, kind of like Zener diodes -- lowish breakdown voltages (often around 20V) -- but with very large power ratings (usually in the range of kilowatts). They're designed to shunt currents and absorb energy when voltages exceed their breakdown voltage.

Flyback diodes do a similar job of suppressing voltage spikes, specifically those induced by an inductive component, like a motor. When the current through an inductor suddenly changes, a voltage spike is created, possibly a very large, negative spike. A flyback diode placed across the inductive load, will give that negative voltage signal a safe path to discharge, actually looping over-and-over through the Inductor and diode until it eventually dies out.



Switching Signal Diode – Switching diodes are just like standard diodes except that they are designed around the premise that they must change from conduction to non-conduction very quickly for use in switching circuits such as digital electronics.

The 1N4148 is a standard silicon switching signal diode. It is one of the most popular and long-lived switching diodes because of its dependable specifications and low cost. Its name follows the JEDEC nomenclature. The 1N4148 is useful in switching applications up to about 100 MHz with a reverse-recovery time of no more than 4 ns.

Reverse Current Protection

Ever stick a battery in the wrong way? Or switch up the red and black power wires? If so, a diode might be to thank for your circuit still being alive. A diode placed in series with the positive side of the power supply is called a reverse protection diode. It ensures that current can only flow in a positive direction, and the power supply only applies a positive voltage to your circuit.

This diode application is useful when a power supply connector is not polarised, making it easy to mess up and accidentally connect the negative supply to the positive of the input circuit.

The drawback of a reverse protection diode is that it'll induce some voltage loss because of the forward voltage drop. Therefore, the Schottky diode is an excellent choice for reverse protection diodes.

Rectifier Diode – The standard rectifier diode is what you will use most of the time you use a diode. You will design the diode to operate as a "switch" in the forward bias direction. There isn't a colour code or anything for diodes; you just need to know the number of the part and what its characteristics are.

The standard garden variety diode is the 1N400X series. Various versions that can handle more current, higher breakdown voltages and current ratings, have incrementing numbers like 1N4001, 1N4002, etc. Also, the packaging changes based on power absorption; hence, a power diode might be in a larger package.

Zener Diodes – Zener diodes are fascinating devices. The symbol is shown in Figure 35. They are designed to operate in the reverse bias direction or in breakdown mode portion of the operational curve, that is you purposely reverse bias a Zener diode into the breakdown region and then it becomes a constant voltage drop with variable current. Hence, they are beneficial as voltage regulators (devices that supply varying current at a consistent voltage level). Of course, Zeners wouldn't be very useful if you had to apply a breakdown of 100-200V, so they have much smaller reverse voltages on the order of a fraction of a volt to many volts.

Zener's are designed to have an exact breakdown voltage, called the Zener breakdown or **Zener voltage**. When enough current runs in reverse through the Zener, the voltage drop across it will hold steady at the breakdown voltage.

Taking advantage of their breakdown property, Zener diodes are often used to create a known reference voltage at exactly their Zener voltage. They can be used as a voltage regulator for small loads, but they're not made to regulate the voltage to circuits that will pull significant amounts of current.

Zeners are special enough to get their circuit symbol, with wavy ends on the cathode-line. The symbol might even define what, exactly, the diode's Zener voltage is. Here's a 3.3V Zener diode acting to create a reliable 3.3V voltage reference:

Zener's are also used to clamp voltages and as above always used in the reverse direction.

Zener as a Regulator L56

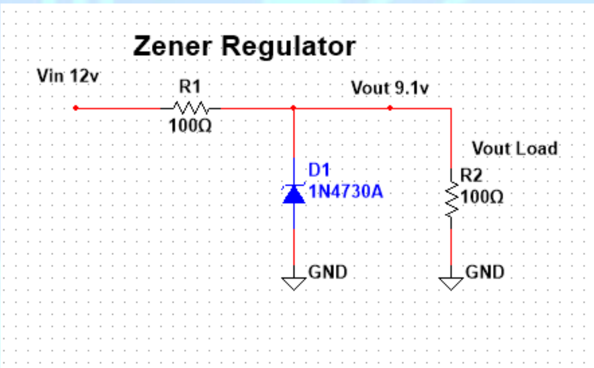


Figure 37 – Zener Regulator

Vin = 12v, Zener = 9.1v, Vout = 9.1v, R = 1k

Current = I = $\frac{V1-V2}{100\Omega}$ = $\frac{12-9.1}{100}$ = 29mA – the minimum current for the Zenner is 25mA which is high, this is not an efficient circuit.

Light Emitting Diodes – LEDs are diodes that are enclosed in a transparent plastic package so that the P-N junction is open for view. LEDs are a diode that has been doped and designed so that some of the electrons that pass through the depletion region cause photons of light to be emitted. LEDs come in various sizes, power ratings, and colours such as red, green, yellow, orange, blue, and even white! LEDs are the indicators of choice in digital electronics and computer front panels. The more, the better. Their forward voltage is 1,7v for Red and Yellow, and 3v for Blue and Green.

Schottky Diodes – Very similar to silicone diodes but only use one semiconductor Junction "P" or "N", the "N" type is very typical, The forward voltage is low 0,2 – 0.3 volts and is very fast, but they can't tolerate high reverse voltages. They stop the current from flowing back to sensitive components.

The semiconductor composition of a Schottky diode is slightly different from a standard diode, and this results in a much smaller forward voltage drop, which is usually between 0.15V and 0.45V. They'll still have a substantial breakdown voltage though.

Schottky diodes are especially useful in limiting losses when every last bit of voltage must be spared. They're unique enough to get a circuit symbol of their own, with a couple of bends on the end of the cathode-line.

Schottky's are used in switching circuits, Boost & Buck converters, and for polarity protection. Refer below.

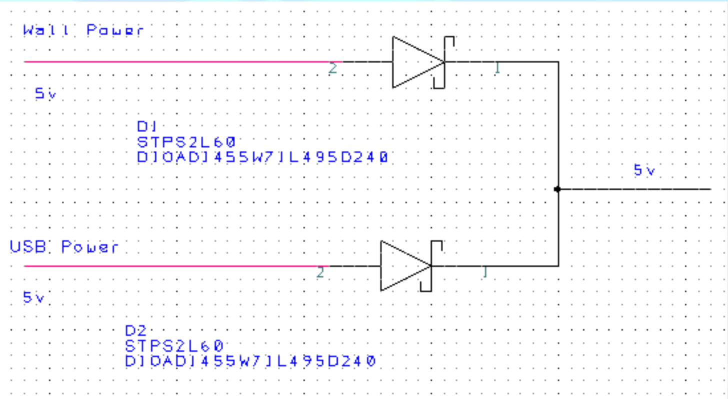


Figure 38 - polarity protection 1

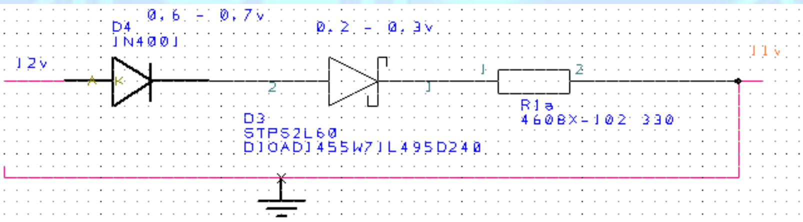


Figure 39 - polarity protection 2

- Both the rectifier and the Schottky diodes provide a similar function but with different voltage drops.

- Zener Diodes are used in a reverse direction for voltage clamping
- The 5.1v Zenner is clamping the line voltage to 5.1 volts. Note the db is reverse Biased.

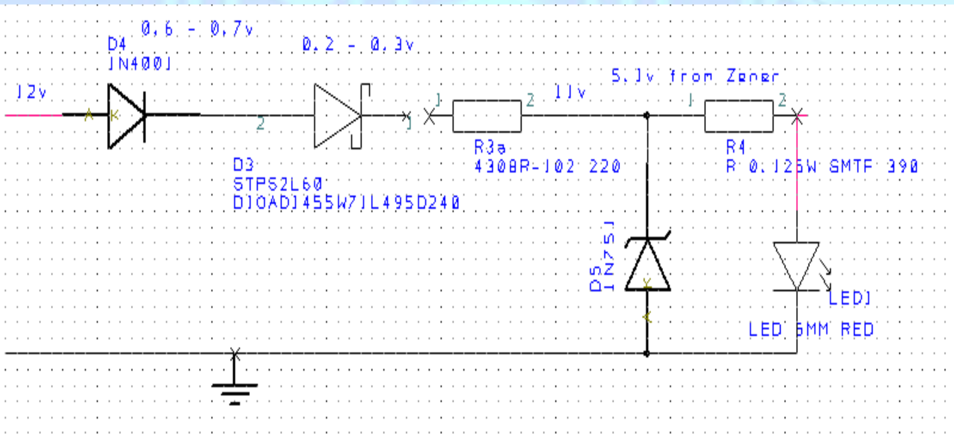


Figure 40 – Zener Voltage Clamp

In Summary

If you want to:

1- Make current go in one direction – use a Schottky
2- Rectify, and you don't care about the voltage drop – use a Silicone.
3- Regulate or clamp a voltage – use a Zenner.

Voltage Regulators and Power Supply Design L60

The 5v Schottky diode power regulator is a simple way of regulating the voltage but depending on the load can be expensive on current

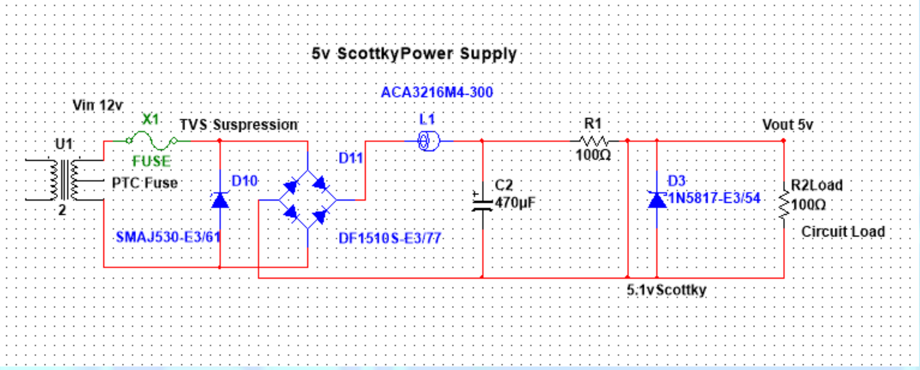


Figure 41 - Schottky diode power regulator

A basic design Linear power supply using the 7812 Voltage regulator.

The 7812 voltage regulator requires an input voltage which is several volts higher than 12 in order to function properly (see data sheet). The difference between that minimum voltage and 12V is called "dropout". The 7812 is a reasonably high dropout regulator. As long as the input voltage is sufficient (the minimum dropout current is supplied), the regulator can provide a smooth DC supply, in which the input ripple from the bridge rectifier is reduced by around 80 dB (check the exact ripple reduction decibel figure in the data sheet).

If you measure the voltage on the Capacitor you will see that it charges to a higher voltage than 12v. The secondary winding of the transformer is 12V, but that's a nominal RMS AC voltage. The peak voltage is higher, and the peak voltage is what charges the Capacitor. If the secondary windings operate at 12V RMS, then the capacitor will charge to a peak of about 17V. Thus, at the peak, there is 5V of dropout.

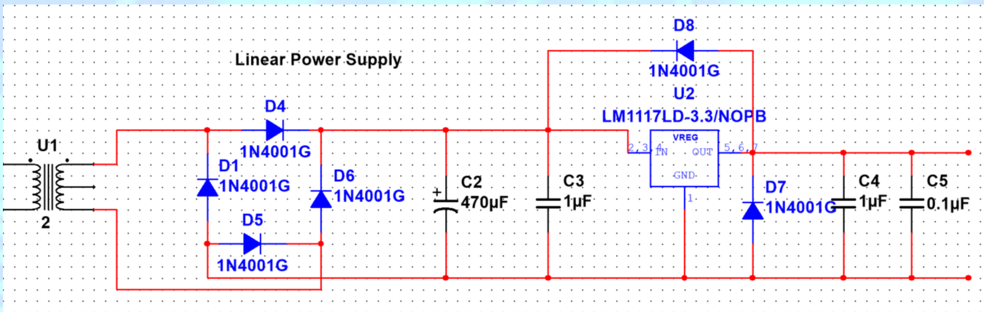


Figure 42 – Basic Linear Power Supply

As the AC cycle changes the polarity changes through the Bridge Rectifier the circuits below detail the voltage flow for each of the two cycles.

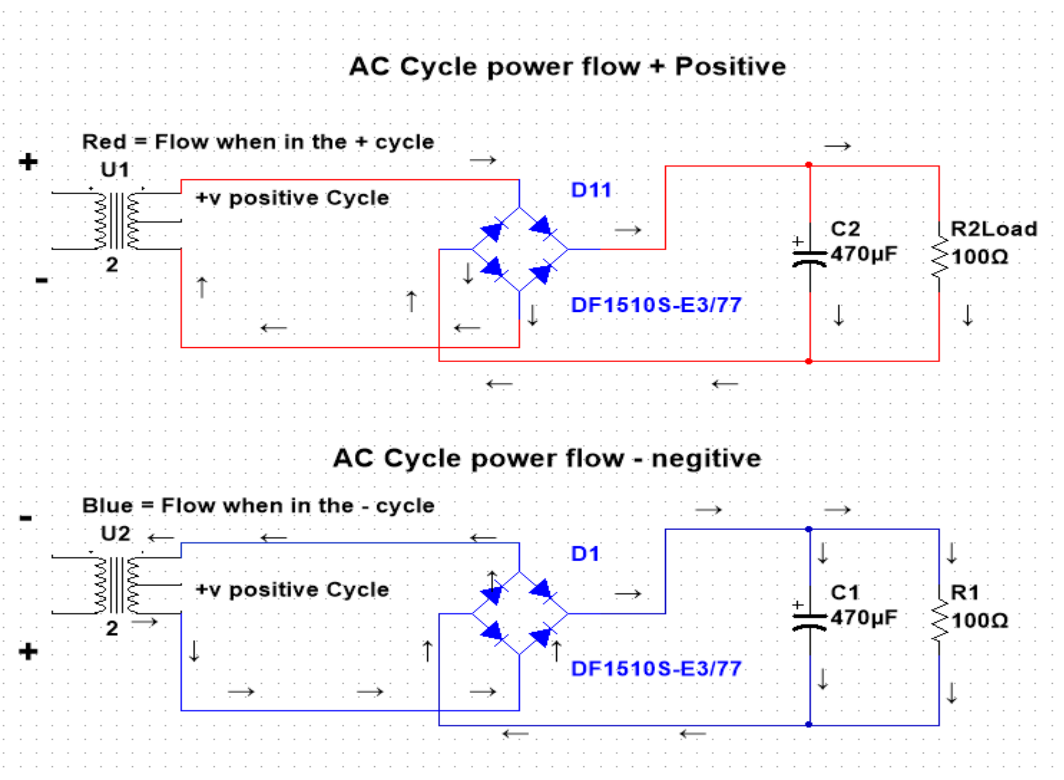


Figure 43 – AC Power Flow

Power supply smoothing capacitors

On each cycle, the Capacitor charges to the peak voltage. Then, it discharges as the regulator draws current from it. The Capacitor must be large enough that when the regulator draws current from it between the charge cycles, the voltage will not drop below the minimum voltage specified for that regulator.

The minimum voltage must be maintained under the worst-case load for the regulator when it draws the most current. Beyond satisfying the worst-case current draw, if you further increase the Capacitor to a larger value, the only benefit it provides is that it reduces the peak-to-peak ripple. This is a minor benefit since the regulator is actively reducing that ripple by 80 to 90 decibels. If the ripple is 0.5V peak to peak at the input of the regulator and is reduced to 80 dB, it becomes 50 µV peak to peak at the output. If you reduce the input to 0.3V peak to peak with a larger capacitor, the output ripple goes from 50 µV to 30 µV. Both these values are small and possibly insignificant to the circuit.

If the circuit needs less ripple, by far a better way to get it is to use a better regulator with more decibels of ripple rejection, rather than making the Capacitor larger. A regulator that improves rejection from 85 dB to 110 dB will make the same difference as a huge and impractical capacitor substitution.

A Capacitor which is too large stresses the transformer and rectifier diodes when power is applied. Because the bigger the Capacitor, the bigger and more sustained is the inrush current.

The equation to calculate the size of the Capacitor is: $C=I*(\Delta V)/(\Delta T)$ Where I is the current you want to output ΔV is the Maximum amount of Voltage Ripple (Peak to Peak Ripple of the Capacitor Voltage) that your circuit can safely handle. The minimum peak should be above your voltage regulators Desired input which is usually 3-volts above your regulated voltage. ΔT can be calculated taking the Minimum Peak To Peak Voltage and dividing it by the maximum Peak Voltage and that value being = X and the solving arcsin(x). The angle you get in degrees must have 90° subtracted from it. 360° occurs in Δt=1/(50-hertz) =.02 so ΔT=(arcsin(x)-90°)*.02/360°

If I=1Amp, ΔV=5% and Vmax=15-volts then Vmin=14.25 and x=14.25/15=0.95 So ARCSIN(0.95)*.02/360=3,989uF and the nearest standard capacitor value would be 4,700uF for the Filter Capacitor size.

A safe rating for components (such as the 25V on the Capacitor) is two times the working value (12V in this case). This part is serving as a bulk capacitor, and other values such as 100uF or 1000uF would work as well.

A second method for capacitor size calculation:

HCT = Half Cycle Time
AVD = Accepted Voltage Drop
50Hz AC = HCT = 10mS
60Hz AC = HCT = 8.3mS
Peak Voltage after rectification = 70.5v DC
Minimum acceptable voltage (AVD or Ripple) 65v DC
Required Current = 10A
Current x Half Cycle Time / Accepted Voltage Drop = Capacitor Microfarads
Cap uF = $\frac{I.HCT}{AVD} = \frac{10\ .8.3}{5.5}$ = 15090uF

Robust Power Supply Design L65

The two circuits below detail the design for a 5v and 3v power supply where the 3v supply is powered by the 5v supply. Some of the primary design efficiencies from the above circuits are as follows:

1- The voltage Regulator – Low Drop Out (LDO) regulators are used. (LD29150 and MC33269). LDO's are more efficient than the old linear regulators and can provide a drop out voltage between 0.2 – 0.5v
2- Circuit Protection – The modern LDO regulators have inbuilt circuit protection, but the circuits below also offer:
    a. TVS (Transient Voltage Suppressor) this diode is like a Zenner that can stand high voltage and works in the Nano or Picosecond range.
    b. PTC Fuse (Positive Thermal Coefficient) this fuse will melt at the cut off amperage and then cool and reconnect providing the amperage is below its limit.
3- Noise – Extra filter capacitors and ferrite beads have been used.

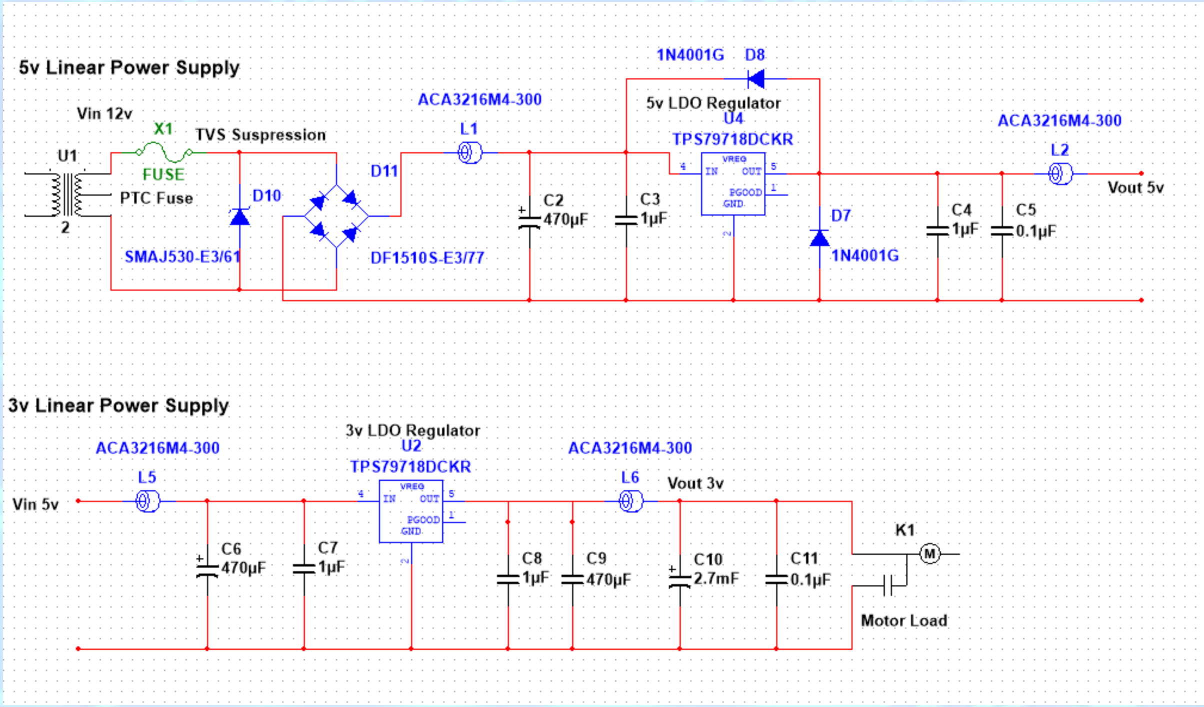


Figure 44 – Robust Power Supply

Power Supply Noise

One of the noisiest components is a motor attached to the power supply. The above circuit has been modified with capacitors C10, C11 and Ferrite Beads L5, L6 to help reduce the noise.
To reduce large amounts of noise (300 – 500 mV) a capacitor multiplier can be used. (Refer to Figure 45 – Capacitance Multiplier)

Capacitance Multiplier

The Multiplier used a combination of a RC Filter and an Emitter Follower to amplify the current and the filtering effect of the Capacitor.
The circuit uses a standard NPN BJT transistor and a small capacitor. The current will bypass R_1 allowing the small Capacitor to be used.

For a reduction of -3db $= \frac{1}{2\pi.F.C}$

With a 1k Resistor and a 10nF Capacitor the cut-off frequency should be 59kHz, but because the circuit generates a β or Gain value of 100, the cut off frequency will be:

$1k\Omega + (100nF \text{ x } 100) = 10uF \therefore \text{ at -3db } F = 15.9Hz$

If a higher current gain is required a Darlington pair of MOSFET transistor could be used.

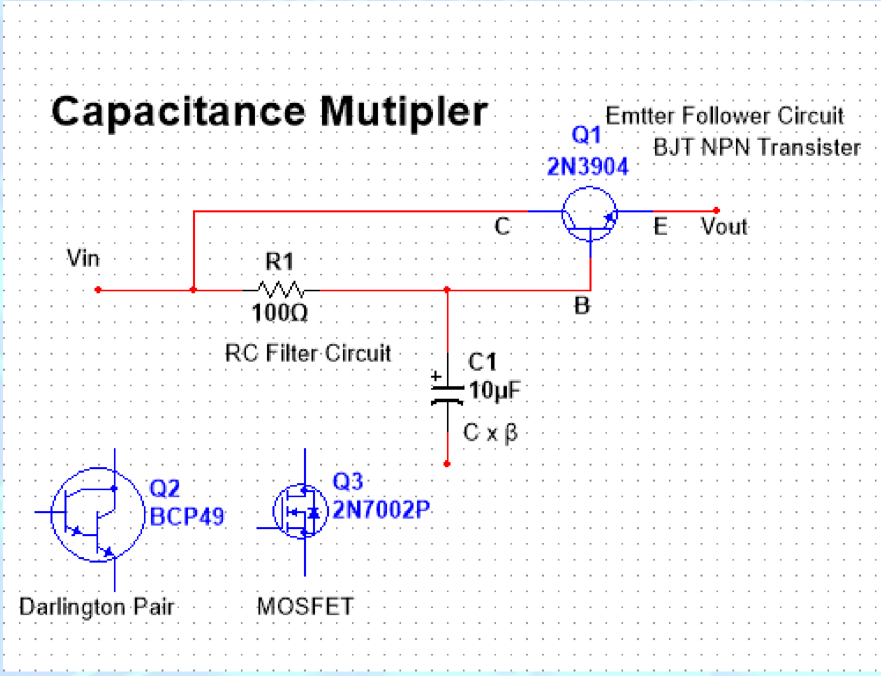


Figure 45 – Capacitance Multiplier

Transistors L67

NPN	Opposite	**PNP**
Arrow → Out		Arrow ← In
Positive Volts at Base		Negative Volts at Base
Current flows from the collector to the emitter.		Current flows from the emitter to the collector
B E C		B E C

Voltage drop – 0.6 – 0.7v over the emitter and the voltage on the collector needs to be 0.2. or > than the emitter.

Impedance Reflection

Looking from E to B the resistance can be large
Looking from B to E the resistance is low.
Looking from E to C the Impedance is ∞

NPN Transistor

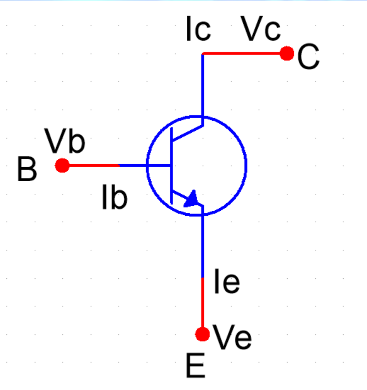


Figure 46 - NPN Transistor

Note: The Eber-Moll This model is more accurate.

Transistor Modelling Equations

HFE = β or Gain

H(FE) = DC voltage

H(fe) = small AC voltage

β (Beta) or Gain = 50 – 250 fold average 100

I_c = H(FE) . I_b ∴ $I_c = \beta . I_b$

$I_e = I_C + I_b = \beta . I_b + I_b = I_b = I_b . (1 + \beta)$

$\therefore I_e = I_b (1 + \beta)$

$I_C = I_b . \beta$

Set Current

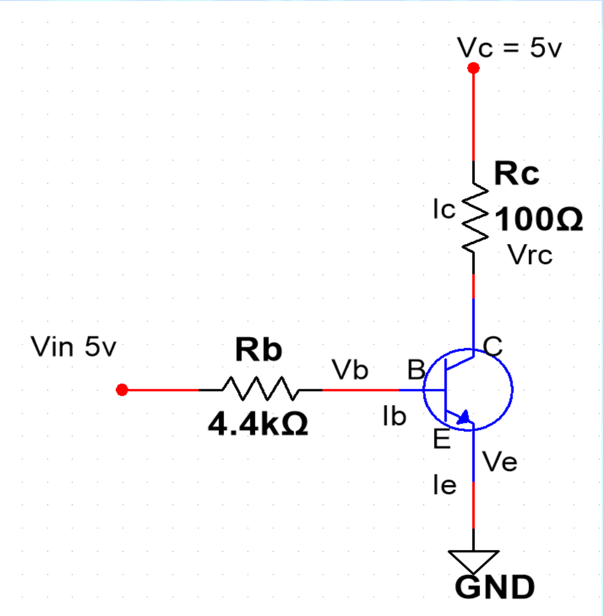


Figure 47 - Set Current Transistor

Set Current

We want I_e to equal 100mA with a transistor voltage drop of 0.6v

The voltage over R_b $= 5v - 0.6v = 4.4$ v

$I_c = \beta \,.\, I_b$ and we want I_c to $= 100mA$

How much base current do we need to turn this on?

$\therefore$ $100mA = 100 \,.\, I_b$ Divide by 100 $= \frac{100}{\beta\,(100)} = I_b = \frac{100}{100} = 1mA$

Now figure out the resistance for R to get 1mA?

$R = \frac{V}{I} = \frac{4.4v}{1mA} = 4.4k\ \Omega$

With I_e = 100mA, what's the voltage drop over R_C ?

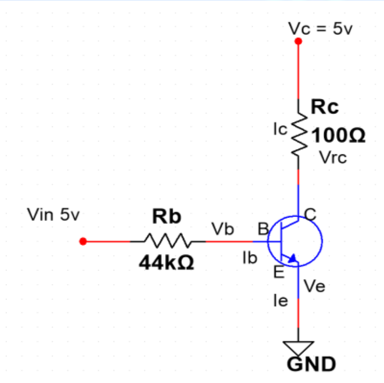
V_{RC} = I . R = 100mA . 100 Ω = 10V

Note: The above Emitter Follower transistor circuit allows us to input a voltage and get the same voltage out with current amplification.

However, as Vin is 5v the circuit does not have 10v, so the above design will not work. The circuit needs to be changed as follows:

Reset Current Transistor

Figure 48 - Reset Current Transistor



Reset Current

We want I_e to equal 10mA with a transistor voltage drop of 0.6v

The voltage over R_b = 5v – 0.6v = 4.4 v

$I_c = \beta \cdot I_b$ and we want I_c to = 10mA

How much base current do we need to turn this on?

$\therefore$ 10mA = 10 . I_b Divide by 10 $= \frac{10}{\beta\ (100)} = I_b = \frac{10}{100} = 0.1\text{mA}$

Now figure out the resistance for R to get 0.1mA?

$R = \frac{V}{I} = \frac{4.4v}{0.1mA} = 44\text{k}\ \Omega$

With I_e = 10mA, what's the voltage drop over R_C ?

V_{RC} = I . R = 10mA . 100 Ω = 1V

Result = V_{RC} – 1 v, I_b = 0.1mA and R_b goes up to 44kΩ

So, by changing the required I_e to be 10mA we can reduce the V_{RC} voltage demand to 1v. and the circuit will now work.

To Calculate β

Beta is calculated by dividing the emitter current over the base current =
I_e = 56.8mA and I_b = 0.42mA - Refer **Error! Reference source not found.**

$\beta = \frac{Ie}{Ib} = \frac{56.8}{0.42} = 135 \therefore \beta = 135$

Current Source

Current Source Single Resister

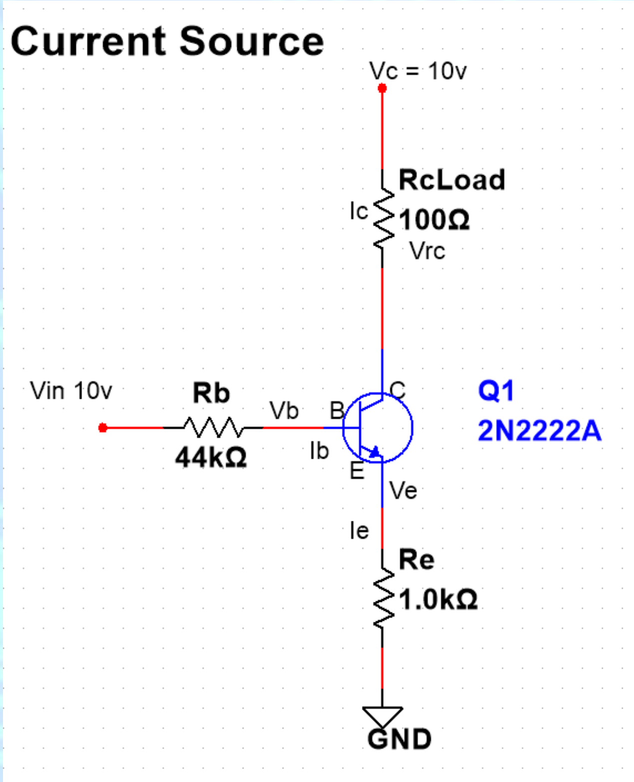


Figure 49 - Current Source

The circuit above (Figure 49 - Current Source) uses the transistor as a Current Source were we can program transistor always to deliver the same current.

How do we get 1mA at R_E? What we know:

1- The transistor uses 0.6v, so if we make V_{RB} 1.6v then V_{RE} will = 1v. Then, $R_E = \frac{V}{I} = \frac{1v}{1mv} = 1k \therefore$ $V_{RB} = 1k\ \Omega$.

2- We know I_B must be $\frac{1}{100}$ of I_E, so $I_E = \beta \cdot I_B \therefore I_B = \frac{Ie}{\beta} + 1 = 0.01mA$

3- $V_{RB} = 10\text{-}0.6 = 9.4v \quad R_b = \frac{V}{I} = \frac{8.6}{0.1} = 940k\ \Omega$

If we set the load R_{CLOAD} to 10Ω, 100Ω, 1kΩ and 5kΩ and test I_E stays at 1.mA – What's the voltage drop over the load for the above loads. Using the Proteus and Multisim simulations (refer Fig 50 Current Source Simulations) below, the Simulations demand R_B to have a much higher resistance.

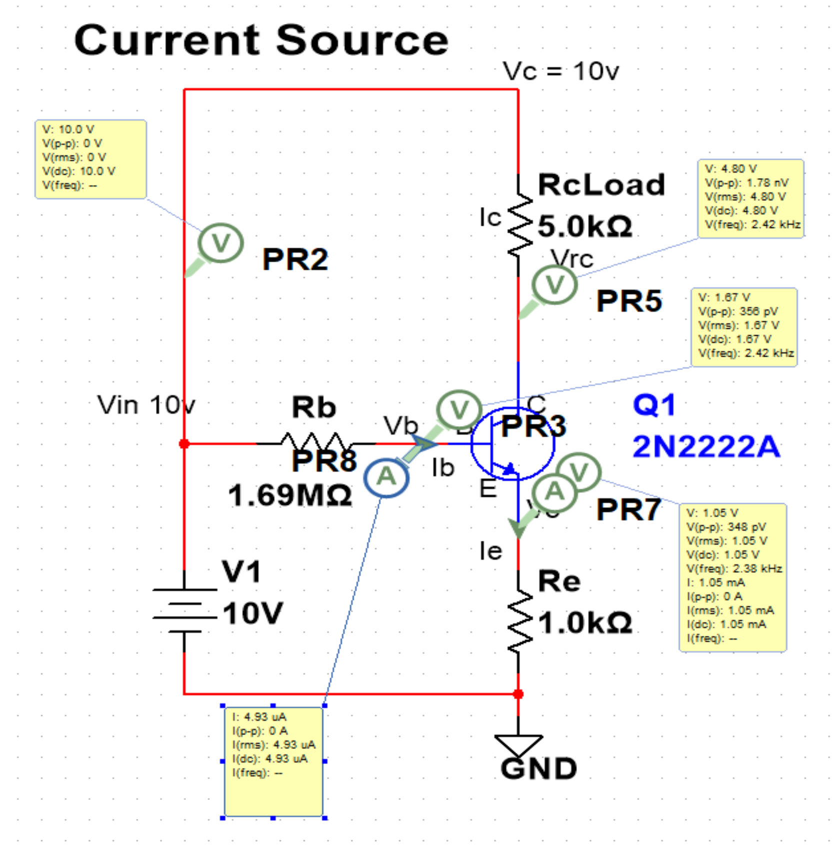
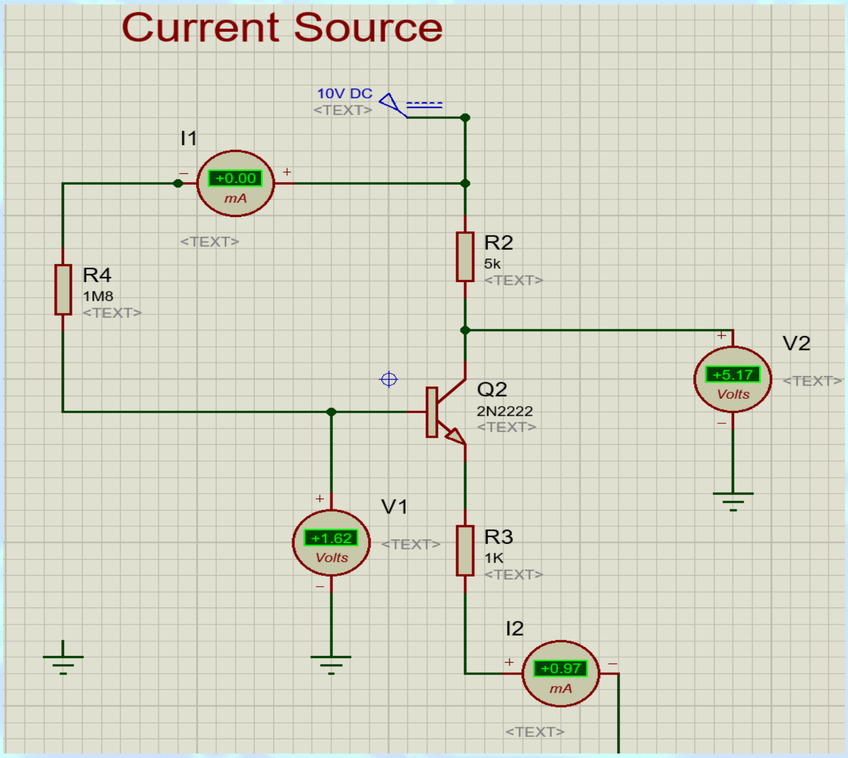


Figure 50 - Current Source Simulations



R2 represents the circuit load and can vary from 0 – 4kΩ without changing the 1A current at R3. R2 at 5k will lower the current to 0.78mA.

Current Source Voltage Divider

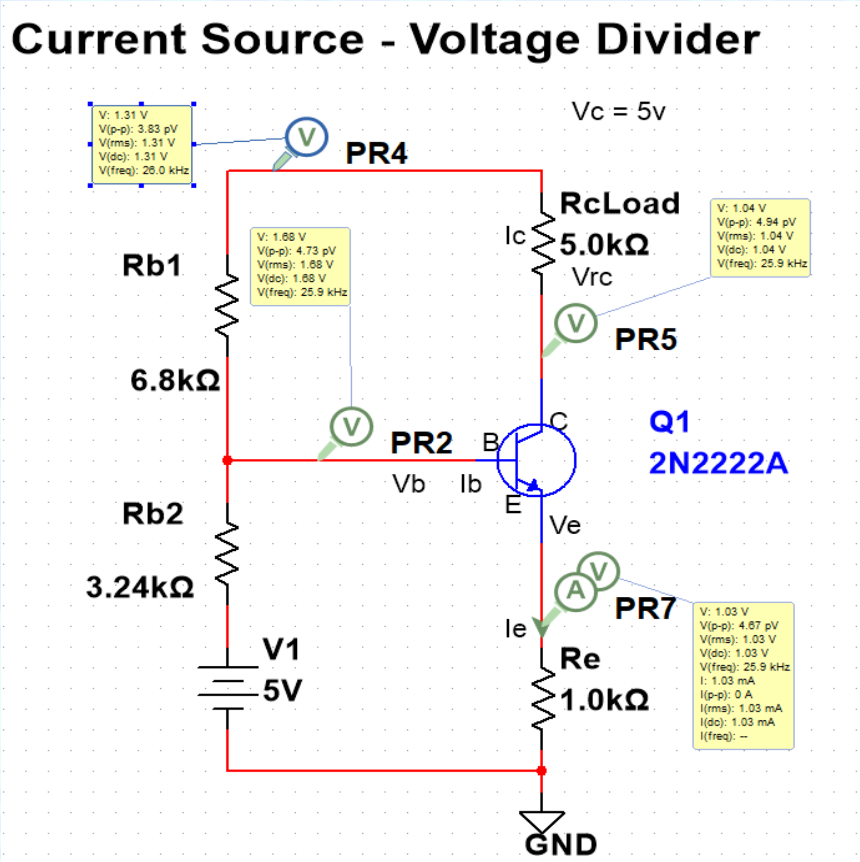


Figure 51- Current Source Voltage Divider

A voltage Divider is a step more stable to provide the transistor bias voltage than using a single Resister. To work out the resister values, we can use a percentage value between the Vin and the voltage required at VB . Referring to fig 51 above, we know that VB needs to be 1.6v, the typical voltage divider formula is RB2 = $\frac{Rb2}{Rb2 + \ Rb1}$. Vin, however, a quick estimation can be used by calculating the percentage of VB from Vin. RB2 = $\frac{1.6v}{5v}$ = .32 or 32%. That makes RB2 = 32% and RB1 = 68%. $\therefore if\ we\ use\ a\ total\ of\ 10k\Omega$ then, $\frac{10k}{.68}$ = 6.8 = RB1 = 6.8kΩ and RB2 = 32%. ∴ $\frac{10k}{.32}$ = 3.2 = RB2 = 3.2kΩ.

The Proof:

V_B = ($\frac{R2}{R1+R2}$) . Vin = V_B = ($\frac{3.2}{6.8+3.2}$) . 5v = 1.6v

As RB2 is in parallel with RE due to the Impedance on the transistor. Then the value of RB2 the will change as a result of the resistance in parallel rule. RB2 II RC * (*=RE as seen through the base known as the Reflective Impedance) = RB2 II (RE . β) where β = 100 or 100kΩ ∴ RB2 * = 3.2kΩ II 100kΩ = Parallel Resistance = “The product over the Sum” = $\frac{3.2k\ .\ 100k}{3.2k+100k}$ = 3.1kΩ , thus the effect of the Reflective Impedance reduces RB2 by 100Ω . If the resisters are small, we will use more current, but the circuit will be stiffer and not subject to much change due to other resistance or Impedance.

So, it is suggested to change the resisters from 6.8k and 3.1k to 680Ω and 330Ω.

Current Source – Zener Diode

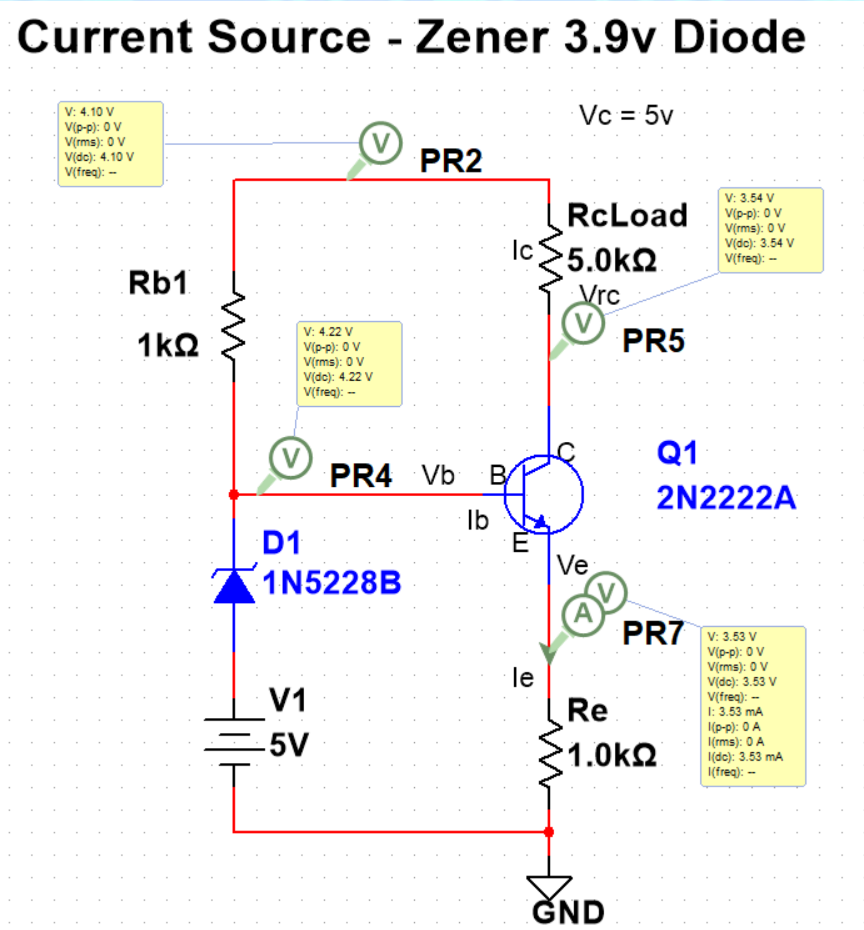


Figure 52 - Current Source – Zener Diode

To set the transistor bias by using a Zener Diode (refer fig 50 above), we will change I_E from 1mA to 3mA and use a 3.6v Zener diode to set the voltage on $V_{B.}$

The resistance on R_{B1} must be set small enough to all the current to maintain Zener regulation. If R_{B1} is set to 1k them $\frac{1k}{5v}$ = 5 mA, (note volts / k – mA). This 5mA needs power both the Zener regulation and the transistor bias. If we use a 3.6 Zener then V_E will 3v and for I_E to have 3mA we need to calculate a resistance. $R_E = \frac{V}{I}$ = $R_E = \frac{3v}{3mA}$ = 1kΩ.

Using simulation V_B was shown to 3.54v, as this is low we could lower R_{B1} to 470Ω or change the Zener to be 3.9v. However, if we also chance R_{B1} to 470Ω we can get more current on the circuit.

Refer fig 52 above for live data.

Current & Voltage Regulator

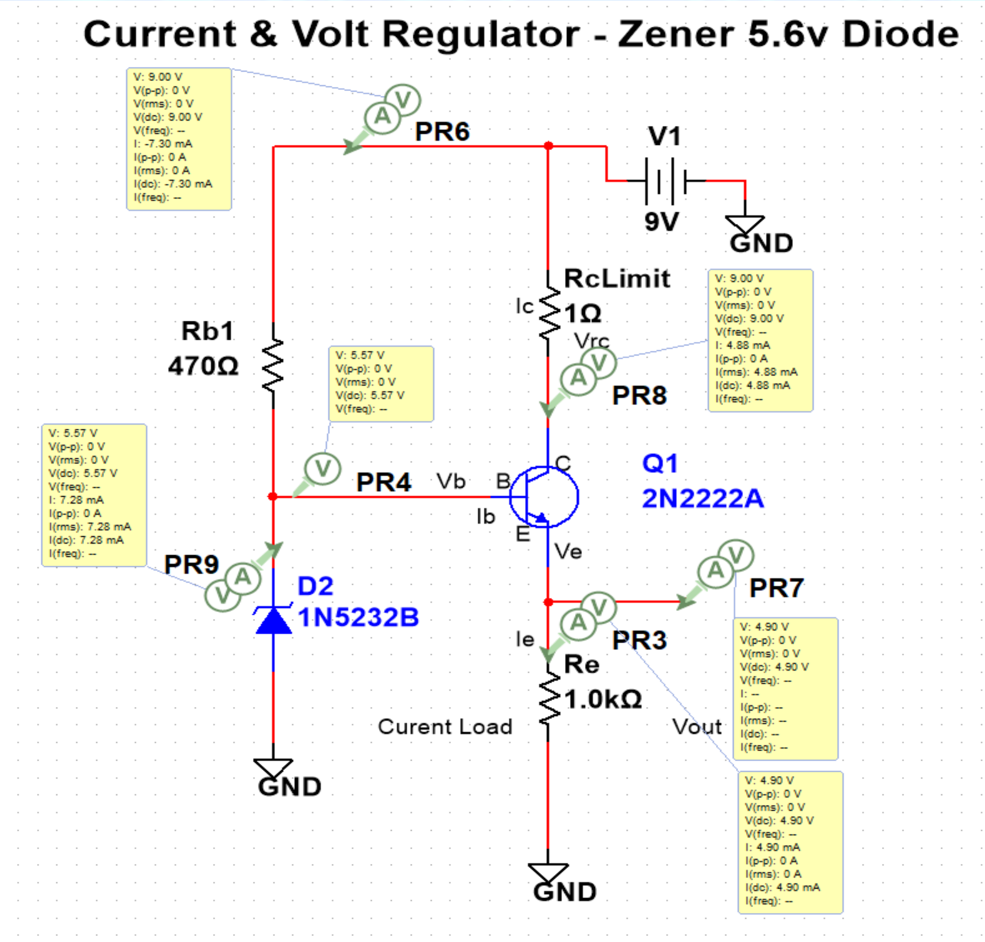


Figure 53 - Current & Voltage Regulator

We could make some small changes to the fig 53 circuit and include a 5v voltage regulator as well as providing a 3mA current source. The 5v supply should be maintained irrespective of the current load.

The circuit changes: Change R_C to Rc Limit of 1Ω, add a 5v junction at V_e, Change Vin to 9v. (refer to figure 53 above.

Emitter Follower Current Amplifier

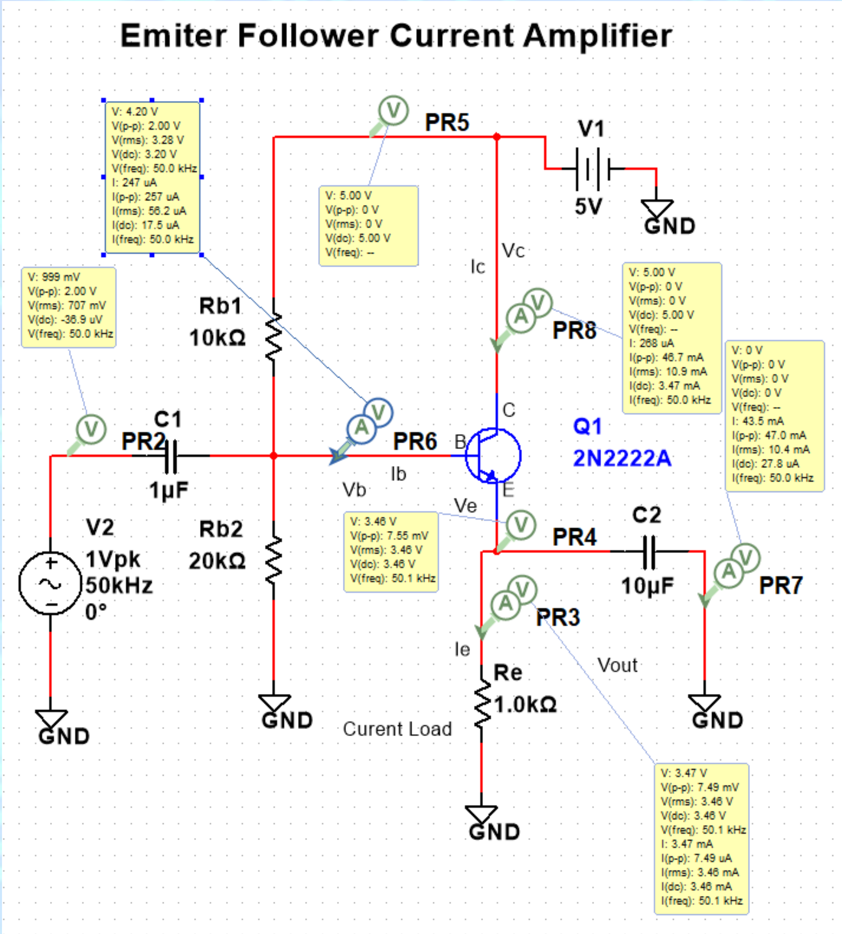


Figure 54- Emitter Follower Current Amplifier

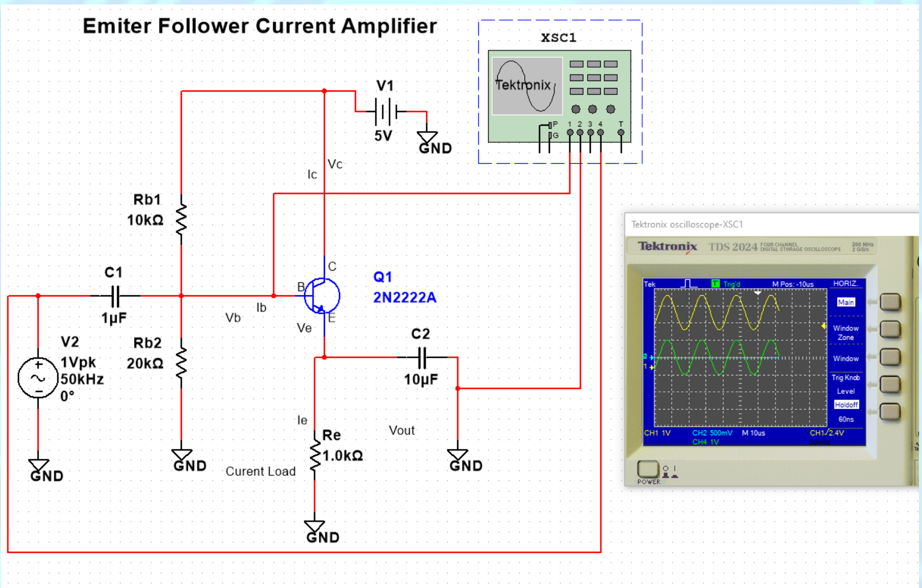


Figure 55 - Emitter Follower Current Amplifier2

The above circuits (fig 54 – 55) describe a Transistor Emitter Follower Current Amplifier. The term "Transistor Emitter Follower" Because of this behaviour, the common-collector amplifier circuit is also known as the voltage-follower or emitter-follower amplifier, because the emitter load voltage follows the input so closely. ... The output is the same peak-to-peak amplitude as the input.

By placing a small AC signal at Vin (Amplitude = 1v Freq = 50Hz) and receiving the AC signal at Vout, the frequency "Wiggles" the transistor that assists with the amplification of the current. The AC ground signal is raised the by 2.5v and sits on top of the DC supply, with an 2.5v off set. There is also a 0.6v drop from Vin to Vout due to the bias requirement of the transistor.

The circuit is set up to get the maximum amount of output swing. Given Vout = 2.5v (half of the 5vDC supply) and use a 1kΩ for R_E then $I_E = \frac{Ve}{Re} = \frac{2.5v}{1k\Omega}$ = 2.5mA. this 2.5mA is the quiescent or steady-state current supply from the circuit.

As Vin is set to 2.5 via the voltage divider and V_B need to be (2.5 + 0.6) = 3.2v.

Input Impedance (Z)

What is the input impedance om Vin? Note: Whenever we have an output, we want to input the signal with a high impedance so that we that the circuit does not drag a lot of current and drop the voltage and sag the circuit. We would like ∞ Impedance unless we are concerned about Maximum Power Transfer when the signal passes between several stages, which we are not in this case.

To calculate the Impedance using R_{B1} at 10kΩ and R_{B2} at 20kΩ for the voltage divider, looking through Vin we can see the reflective Impedance of β. If β = 100 and R_E = 1kΩ then the reflective impedance is 100kΩ.

With R_{B2} at 20kΩ so the Impedance now = 20kΩ II 100kΩ = 20kΩ in Parallel with 100kΩ (II = Product over the sum) $\therefore Z = \frac{20k\Omega \,.100k\Omega}{20k\Omega + 100k\Omega}$ = 16.64kΩ Note: we know that when calculating parallel resistance the result is always less than the value of the smallest resistor in the calculation. So if we get a result bigger than this, we have made a mistake.

We need to take into account R_{B1} at 10kΩ as this affects the reflective impedance calculation, ∴ 10kΩ II 16.64kΩ = $\frac{16.64k\Omega \;.\; 10k\Omega}{16.64k\Omega + 10k\Omega}$ = 6.24kΩ so Z = 6.24kΩ.

The voltage at V_B = $V_B = (\frac{Rb1}{Rb1=Rb2})$. Vcc = $(\frac{20k\Omega}{20k\Omega+10K\Omega}) \,.\, 5v$ = 3.3v. This is a little more than the 3.2v required, but as we have not taken into account the parallel effect of Z (6.24kΩ) the actual v_E voltage should be slightly less.

The AC signal is coupled in by using the two capacitors C1 and C2. The inclusion of the two capacitors is called capacitor coupling. If we look at C1 and R_{B2} we have a High Pass Filter and we want to design the circuit to pass an AC signal close to 20Hz. To calculate this $F = \frac{1}{2\pi \,.Z \,.C1}$ Note: we are using Z and not R_{B2} as it is the impedance resistance that is required in this case. ∴ if we use an estimated capacitor value of 1µF and the Impedance on RB2 then $F = \frac{1}{2\pi \,.6.24k\Omega \,.1\mu F}$ = 25Hz. If we made C1 10 X bigger the F value would fall to 2.5Hz, and if we made C1 smaller, the F value would increase.

Very important note: – The resistance of RB2 in the case of the High Pass Filter is the parallel impedance value of 6.24kΩ not 20kΩ the actual value of the resistor. The voltage divider recognises the 20kΩ but the high pass filter does not.

To calculate C2 we also have a high pass filter in the circuit between R_E and C2 as R_E is going to earth. $\therefore F = \frac{1}{2\pi . Re . C2}$ Parallel Impedance does not affect this calculation, we know R_E is 1kΩ, and we used 1µF for C1, but as R_{B1} is 6.24k we should compensate by setting C2 to 10µF, now we have $F = \frac{1}{2\pi . 1k\Omega . 10\mu F} = 15.9Hz$.

Common Emitter Transistor Voltage Amplifier

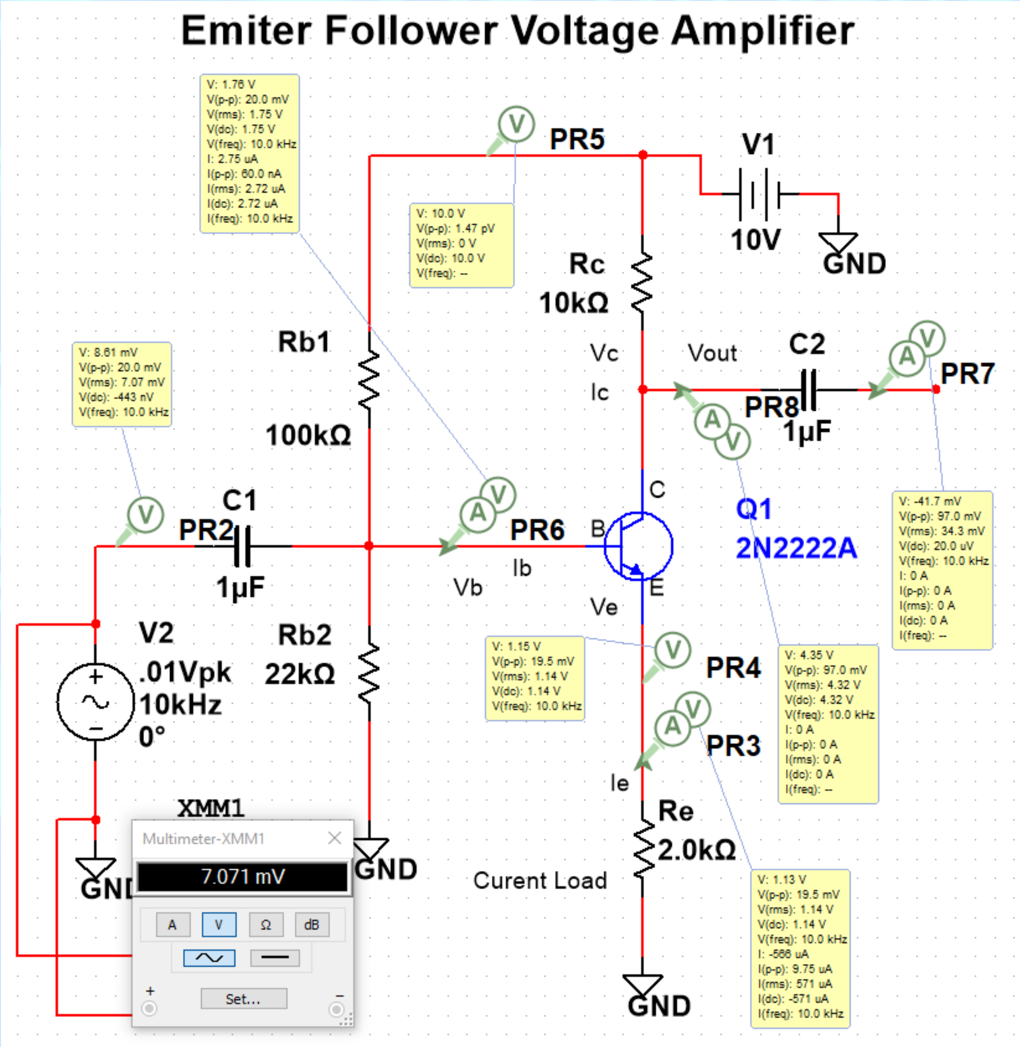


Figure 56 - Common Emitter Transistor Voltage Amplifier

The above circuit is powered by a DC 10v source (Vcc) and a small-signal AC source 10mV amplitude with a frequency of 10kHz (Vin).

The circuit is designed to provide a constant 5v supply at Vout irrespective of the current load on I_E.

$I_C = \beta . I_B$ *

$I_E = I_B + I_C$

$I_E = I_B . (1 + \beta)$ *

I_C and I_E are always very close

- * = Important equations
-

We have set E_B to 2KΩ as a starting point; therefore, the current on I_E is as follows:
$I_E = \frac{IV}{2k}$ = 0.5mA If 0.5mA is going through I_E then 0.5mA is also going through I_C.
To get 5v on Vout we have 10v at Vcc and 0.5mA on I_E ∴ $R_C = \frac{5v}{0.5mA}$ = 10kΩ ∴ R_C = 10kΩ.

Now figure the voltage divider resistance for R1 and R2. In order to get 1v at V_E and we have a 0.6v bias on the transistor, then the voltage on V_B must be 1.6v. ∴ $V_B = (\frac{R2}{R2+R1})$. Vcc = 1.6v

One way to estimate the R1 and R2 resistance values is to calculate their percentage values first then transpose the percentage values to resistance. If we start by setting R1 to 100kΩ and calculate the percentage values between V_B = 1.6v and Vcc = 10v then we get R2 = $\frac{1.6}{10}$ = 16% R2 = 16% ∴ 100% - 16% = 84% ∴ R1 resistance of 100kΩ must = 84% of the total voltage divider resistance. ∴ R2 = $\frac{100}{84}$. 16 = 19.047kΩ ∴ R2 = 19.047kΩ The Proof = $V_B = (\frac{19.047}{19.047+100})$. 10v = 1.599v

Due to the reflected Impedance from R_E and β, we know the total will be 200kΩ due to β at 100. ∴ 19kΩ II 200kΩ = $\frac{19k\Omega\ .\ 200k\Omega}{19k\Omega + 200k\Omega}$ = 17.35kΩ so Z = 17.35kΩ If we set R2 a little higher say 22kΩ then this will compensate for the reflective Impedance. The proof 22kΩ II 200kΩ = $\frac{22k\Omega\ .\ 200k\Omega}{22k\Omega + 200k\Omega}$ = 19.81kΩ.

We need to take into account R_{B1} at 100kΩ as this affects the reflective impedance calculation, ∴ 100kΩ II 19.81kΩ = $\frac{19.81k\Omega\ .\ 100k\Omega}{19.81k\Omega + 100k\Omega}$ = 16.53kΩ so Z_{total} = 16.53kΩ.

The AC signal is coupled in by using the two capacitors C1 and C2. The inclusion of the two capacitors is called capacitor coupling. If we look at C1 and R1 we have a High Pass Filter, and we want to design the circuit to pass an AC signal. To calculate the Frequency, F = $\frac{1}{2\pi\ .Z\ .C1}$ Note: we are using Z and not R1 as it is the impedance resistance that is required in this case. ∴ if we use an estimated capacitor value of 1µF on C1 and the Impedance on R2 then F = $\frac{1}{2\pi\ .\ 16.53k\Omega\ .1\mu F}$ = 9.62Hz ∴ F_{C1} = 3db at 9.62Hz.

Very important Note: – The resistance of R2 in the case of a High Pass Filter is the parallel impedance value of 6.24kΩ not 20kΩ the actual value of the resistor. The voltage divider recognises the 20kΩ, but the high pass filter does not.

To calculate C2 we also have a high pass filter in the circuit between R_E and C2 as R_E is going to earth. ∴ F = $\frac{1}{2\pi\ .Re\ .C2}$ Parallel Impedance does not affect this calculation; we know R_E is 1kΩ and if we use a 1µF for C1 now we have F = $\frac{1}{2\pi\ .10k\Omega\ .1\mu F}$ = 15.9Hz. ∴ F_{C2} = 3db at 15.9Hz.

Compute the Gain

The gain is the amount of AC small-signal amplification between V2 and Vout.
Objective: Gain = $\frac{Vout}{Vin}$ -(IC . RC)

- DC objects are denoted in uppercase e.g. VB
- AC (Small Signal) objects are denoted in Lowercase e.g. vB

Equations:

VC = Vcc – (IC . RC)

IE = $\frac{Ve}{Re}$

Taking VE = BB – 0.6v then looking at the AC small signal, what happens when we put the small wiggle on VB?

If we are talking about ΔVE and ΔVB and we are only looking at the difference, then Ve is following VB – 0.6v. Then the small signals of vE and vB are effectively the same. So, we have:

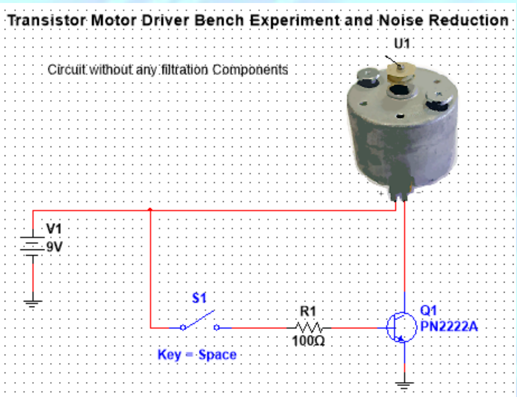
vE = vB (Result 1), iE = $\frac{ve}{RE}$ (Result 2), vC = -iC . RC (Result 3) ∴

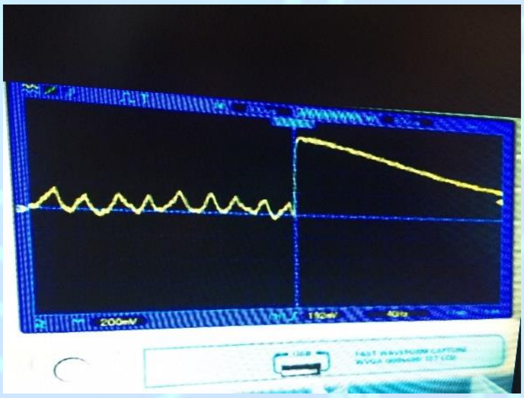
$\frac{vc}{ve}$ = -$\frac{Rc}{Re}$ = Gain = -$\frac{10}{2}$ = an Inverse Gain of 5.

Transistor Motor Driver and Noise Reduction L47

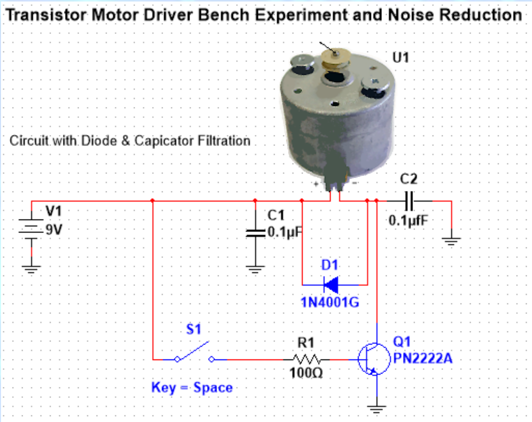
This circuit uses a transistor switch with a PN222A PNP transistor (“P” stands for plastic) to study and eliminate the transient noise. The maximum current through the transistor (I_c) is 500mA. The following is a set of circuits and oscilloscope images that show the circuit and the motor running noise and on /off transients.

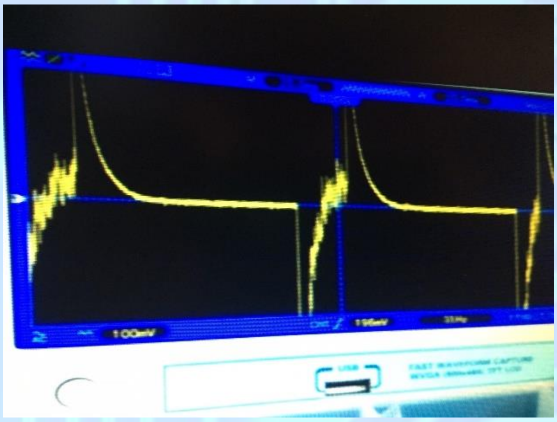
Other circuits are presented with extra filtration components added and corresponding scope images.



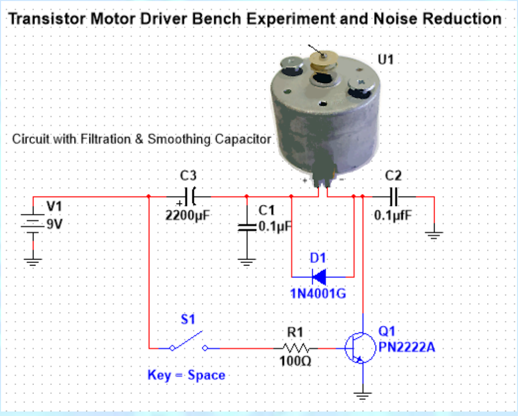


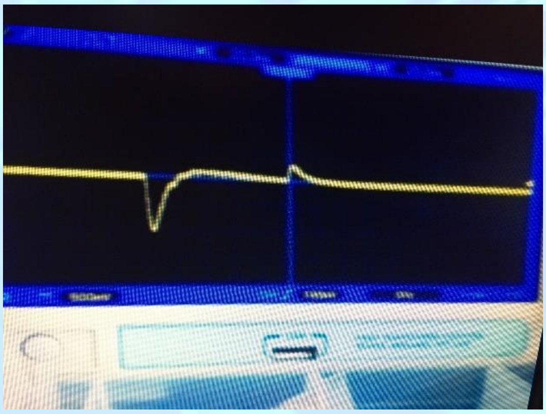
The scope is set to 200mV without any filtering; there is substantial running noise and a large turn off spike.





Changing the scope to 100mv and using the diode and small caps as a filter, the running noise has diminished, but the on / off spikes remain.





The scope is set to 500mv and using a 2200mF electrolytic cap for smoothing we are now getting smaller spikes and killed the noise.

If we use a FET the switch would give us a very small “On” resistance, they are just like mechanical switches. Further reading, look at Motor Driver Circuits and Snubber Circuit Design Calculators.

Oscillators

ac Small Signal

In order to use the Small ac Signal without using a signal generator refer Fig 56 above, an Oscillator circuit needs to be developed. The three basic circuits below have been included as suggestions. As there are many ways to generate an Oscillation signal, the reader should only use the circuits below as a reference and research other circuits to solve specific needs.

Colpitts Crystal Sinusoidal Oscillator

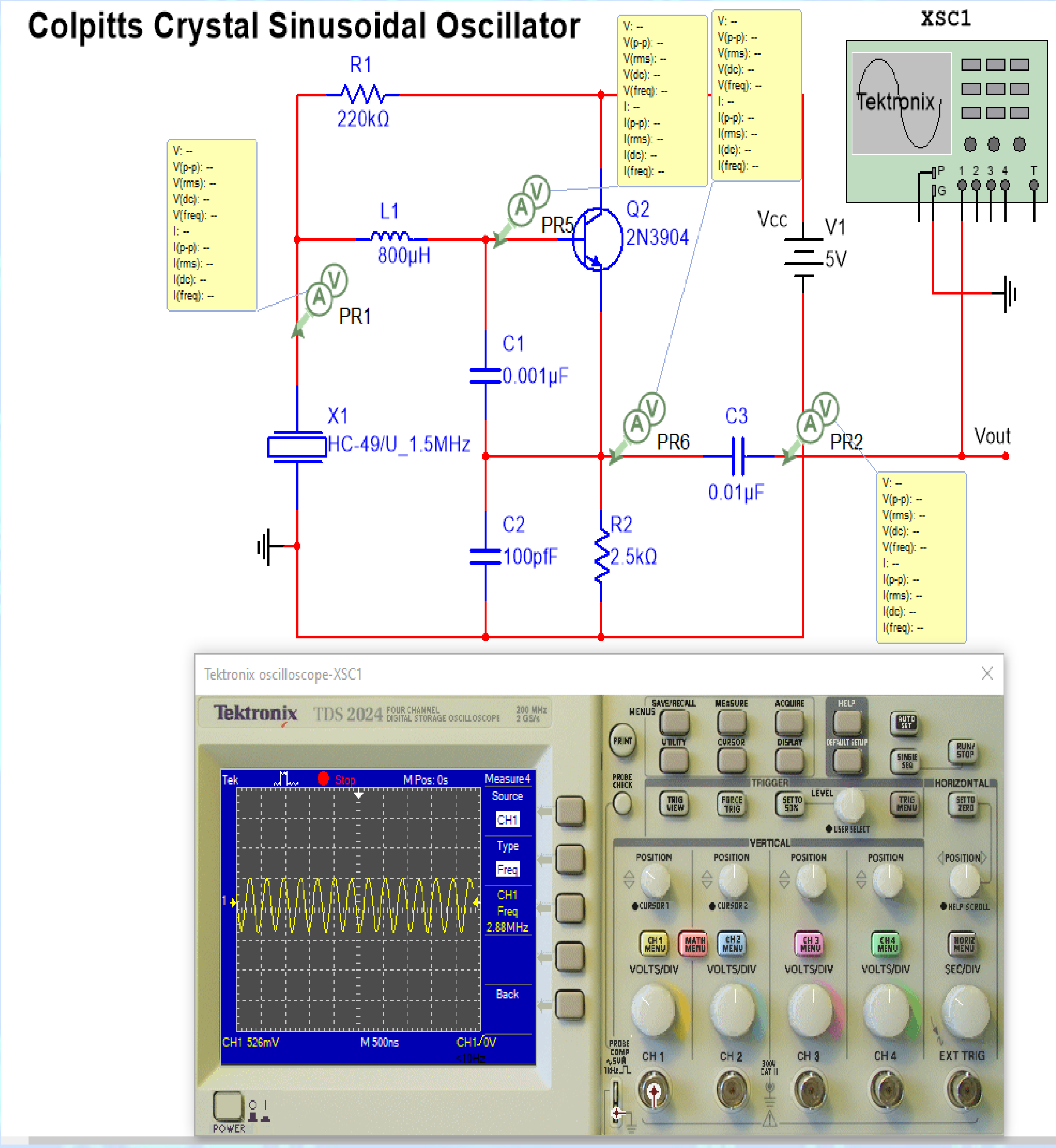


Figure 57-Colpitts Crystal Sinusoidal Oscillator

Quartz Crystals are often used to set the frequency of an oscillator because of their precise frequency and stability. The above circuit is very stable with a frequency range of approx—100kHz to 40MHz. The output is a sine wave with a slight distortion.

Colpitts Square Wave Oscillator

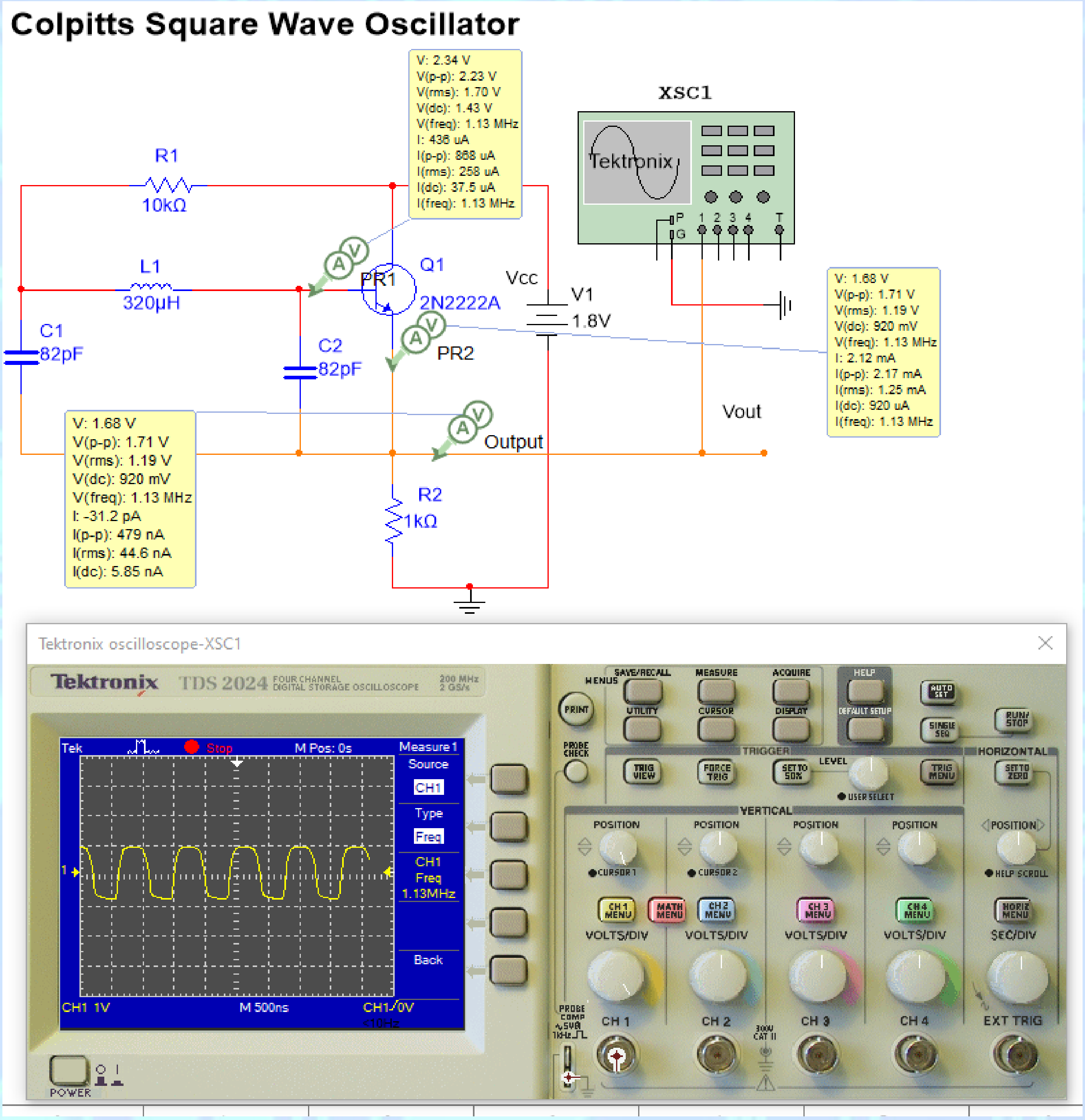


Figure 58-Colpitts Square Wave Oscillator

The Colpitts Oscillator was developed by Edwin H. Colpitts in the year of 1918. This oscillator is a combination of both inductors and capacitor. The features of the Colpitts Oscillator are the feedback for the active devices, and they are taken from the voltage divider and made up of two capacitors which are in series across the inductor.

Square Wave and Filter Oscillator

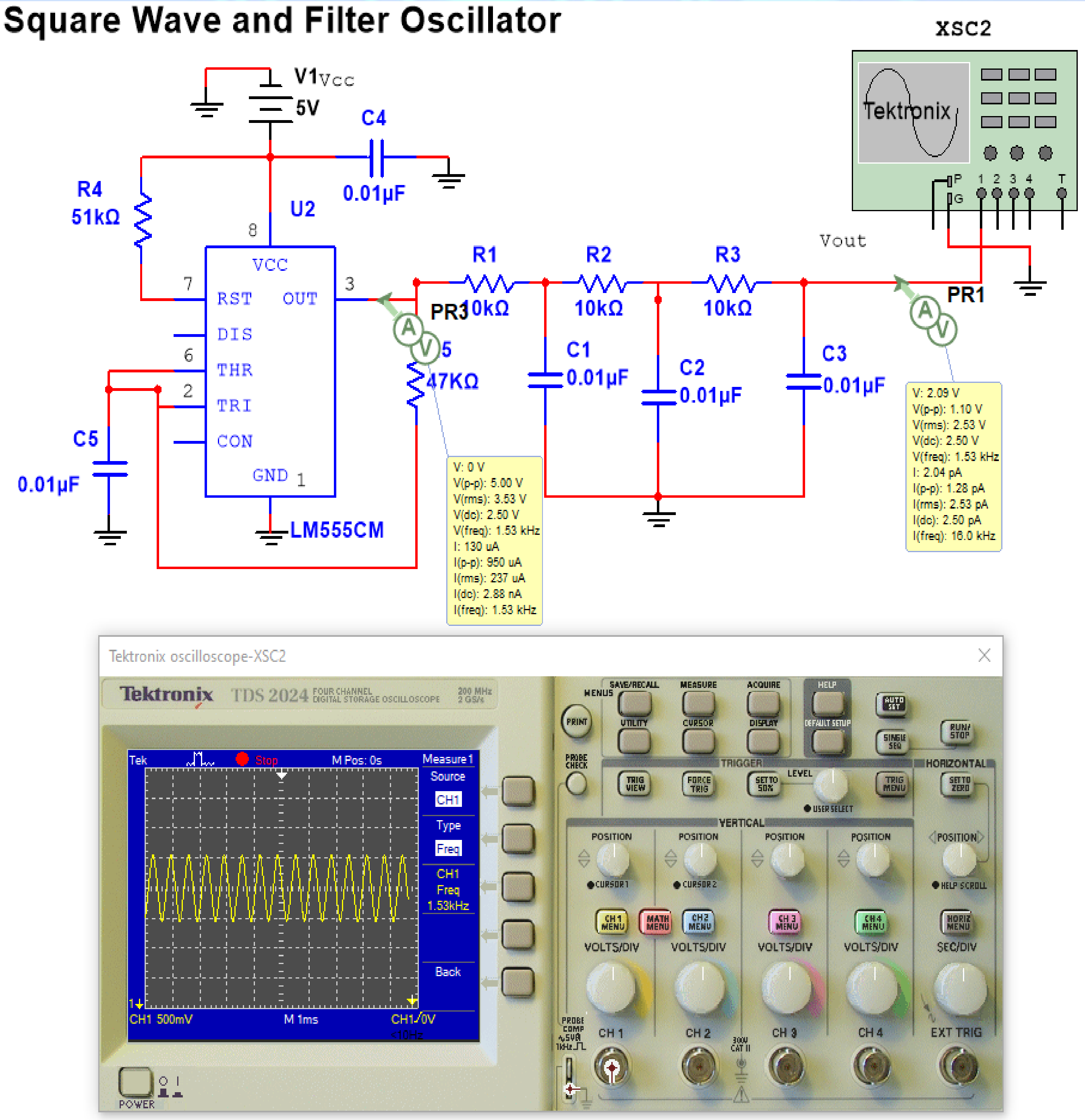


Figure 59- Square Wave and Filter Oscillator

A CMOS 555 IC produces a 50% duty cycle square wave. Its output is sent to the Low Pass Filter that filters out most of the harmonics, leaving the fundamental sine wave. If required, the reader may design a more selective filter to improve the sine wave quality.

Index