Manual for

ELECTRO TOOLS V 1.00

Issued 10 November 1999

Table of Contents

<TOC 1 "Introduction">Introduction

To do rewarding (and well paid) things in the world of electronics, you need a kit of sharp ideas and the mathematical tools to play with those ideas.

Electro Tools can do two things for you :

- Draw your attention to a few basic ideas. This manual defines each concept in basic form.
- Give you a tool to apply those ideas. Each worksheet is dedicated to a single concept, and this manual gives you hints on how to apply each concept to real tasks.

Experienced engineers already use concepts like those contained in Electro Tools, often in a sophisticated way. Most engineers have Basic programs, spreadsheets and the like they have developed over the years. Electro Tools is still a useful to engineers because 1) it is prebuilt and 2) it allows gives immediate feedback as variables are tuned. You are not forced to use a “design procedure”.

Technicians can use Electro Tools to get control over basic circuit building blocks. Once you can conjure up component values and predict performance, you can stop using schematics produced by others and take control yourself.

Electro Tools is still in development. Please help shape it with your feedback !   Contact me with criticisms and suggestions.

Email : rogerl@datafast.net.au

Roger Lascelles
88 Lake Avenue
Ocean Grove 3226
Australia

<TOC 1 "Managing Files and Worksheets">Managing Files and Worksheets

Overview

Worksheets

Each design tool is a *worksheet* where you enter values, adjust settings and read results.

There are many types of worksheets such as Rectifier, Pole Zero Plotter, Matching Network, Power & Noise etc.

You can create any number and combination of worksheets. For example, six Matching Networks worksheets could hold the designs for six different networks used in a radio transmitter.

Project Files

When you start a new design task, you create a new *project file* to hold all the worksheets you create during the design process.

Your project worksheets are saved in the project file. When you reopen the project file, all the project worksheets are available, just as you left them.

Using Project Files

Creating A Project File

1. select FILE | NEW.
2. Enter a Title which best describes the new project.
3. Click OK.

Saving A Project File

1. select FILE | SAVE.
2. Enter a file name for the project.
3. Click “Save”.

Opening an Existing Project File

1. select FILE | OPEN.
2. Locate the file.
3. Click “Open”

Project Manager

The Project Manager window lets you create, delete, copy, open, close and title the worksheets in your current project.

From the menu, select PROJECT | MANAGER to display the Project Manager window.

All worksheets in the current project are listed. Names in bold represent worksheets which are already open.

Project Manager Tasks

In the Project Manager window, click *New* to make a new worksheet.

In the Project Manager window, use the mouse or arrow keys to highlight a title from the list of worksheets then :

- Click *Delete* to delete the worksheet .
- Click *Copy* to create a clone of the worksheet .
- Click *Open* or *double click* the worksheet title to view the worksheet.
- Click *Close* to hide the worksheet

Printing Worksheets

ElectoTools Version 1 does not provide inbuilt printing. However, you can use the Windows clipboard to capture and print worksheet output, including graphs.

To copy a worksheet to the Windows clipboard :

- View the worksheet you wish to capture. If part of the worksheet is off-screen, click on the worksheet title bar and move the worksheet so that it is completely visible on screen.
- Click on the worksheet so that it becomes the active window.
- Hold down the Alt key and press the Print Screen key. This copies the worksheet to the Windows clipboard.

To print the image from the windows clipboard :

- Click the Windows Start button, select Run and type WORDPAD. Click OK.
- In WordPad, hold down the Shift key and press the Insert key. The clipboard image will be pasted into WordPad.
- In WordPad, select File | Print to print the image.

Manipulating Worksheets

Worksheets adhere to the usual Windows conventions -

- TAB moves to the next data entry location.
- Shift - TAB moves to the previous data entry location.
- Alt - F4 closes the current worksheet. The worksheet is not lost and can be reopened from the Worksheet | Open menu.
- The menus can be operated by holding the Alt key and pressing the letter key matching the underlined letter in the menu title. In this way you can access the menus at all times, even when the menus are obscured by worksheets.
- Worksheets can be moved by clicking on the caption bar and dragging the sheet.
- Some worksheets can be resized by dragging the sides or corners.

Worksheets do not have a "Recalculate" button. When you TAB or click another data entry location, the worksheet recalculates. If you want to recalculate without leaving a data entry location, press the ENTER key.

On some worksheets you can use the UPARROW and DOWNARROW keys to move about in addition to the TAB keys.

<TOC 1 "Attenuator">Attenuator

Overview

The Attenuator worksheet calculates resistor values for several popular attenuator networks.

Ladder Attenuator

The ladder network consists of a chain of identical stages and is convenient for switched attenuators. The ladder can be extended as desired by inserting more series - shunt resistor pairs.

The advantages of the ladder network are :

- uses a minimum of resistors.
- keeps a low impedance to ground at all points along the ladder.
- distributes attenuation over many stages, providing useful performance up to 200 MHz or so with ordinary resistors at 50 ohm.

The worksheet shows a voltage source Vs driving the ladder, while the output is taken from the tapping points, but it is possible to drive the tapping points and take the output from one end of the ladder.

The resistance shown in series with the source is that required to make the taps all present the correct output resistance. In many applications, this resistance is omitted or otherwise differs from the correct value. Whatever the driving arrangements, the other resistor values are always the correct ones to use.

The source resistor produces an initial attenuation at the 0 db tap which may not be acceptable.

The network has the disadvantage that the resistance seen by the source changes as the load tap point is changed.

Bridged Tee Attenuator

The bridged tee lets you change attenuation by varying only two resistances. It is often used for PIN diode or other continuously variable attenuators.

Pi and Tee Attenuators

These are the classic workhorses. Attenuators built with ordinary resistors are accurate to 200 MHz or so, at a resistance of 50 or 75 ohms, provided the attenuation value is 10 db or less. Higher value attenuators can be made by cascading multiple low value attenuators.

<TOC 1 "Bipolar Bias">Bipolar Bias

Overview

The Bipolar Bias worksheet predicts the voltage and current values for a bipolar transistor connected in one of six common bias circuits.

Concept

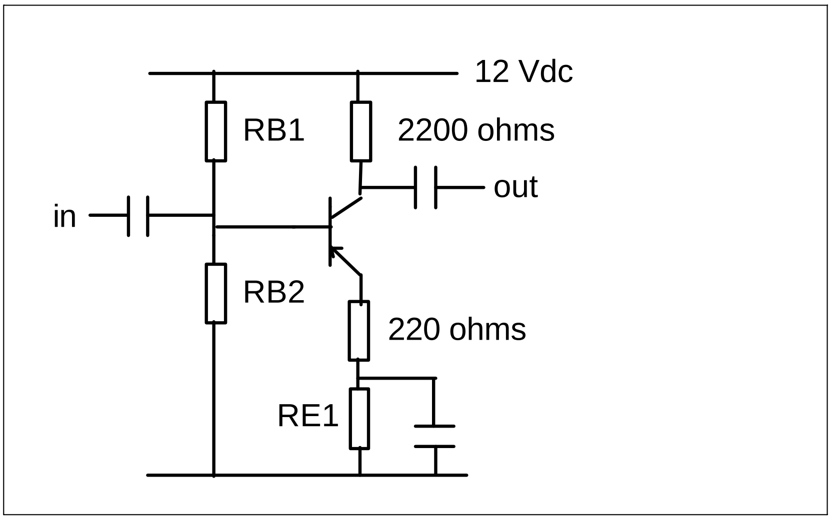
The worksheet provides an exact solution for each bias configuration assuming -

$I_C = H_{FE} \times I_B$
$I_{CBO} = 0$ (collector-base leakage is negligible)
$V_{BE} = \text{constant}$
$H_{FE} = \text{constant}$

Where

I_C = collector current, Amps.
I_{CBO} = collector base leakage current.
V_{BE} = base-emitter voltage.
H_{FE} = dc current gain.

Example



I want a small signal amplifier with a gain of 10 set by the 2200 and 220 ohm resistors. What values should I use for RB1, RB2 and RE?

In the Bipolar Bias worksheet I select *Standard Bias* configuration. I decide to aim for 2 Vdc at the emitter, since this allows about 10V for collector swing. The 2 Vdc is much larger than any variation in V_{BE} in order to assist bias stability.

I set *Vcc* to 12 V. I set *HFE* to 100 and *Vbe* to 0.7 V. I pick an initial value of 4700 for *Rb2* knowing that (*Ic* / *HFE*) * *Rb2* is small.

To prevent saturation during worksheet operation, I set *RE* to 10000 ohms, a very high value, and *Rc* to 1 ohm, a very low value. I adjust *Rb1* for 2 V (actually 2.16 V) at the emitter and get *Rb1* = 15000 ohms.

I set *Rc* to the desired value of 2200 ohm then adjust *RE* for the mid-swing voltage at the collector (*Vc*) which is half way between 12 and 2 V ie 7 V. I get *Re* = 1000 ohms for a Vc of 7.45 V.

I split this *RE* into RE1 = 820 ohms and the 220 ohm unbypassed gain setting resistor. I enter *RE* as 820 + 220 = 1040 and read a *Vc* of 7.21 V - close enough to 7 V.

I have

- RB1 = 15000
- RB2 = 4700
- RE1 = 820

I vary *Vbe* from 0.5 to 0.8 volts and *HFE* from 50 to 200 and notice that Vc stays between 7 and 8 volts. My design will be stable over temperature and transistor variation.

Notes

A stable bias circuit is tolerant of transistor V_{BE} and H_{FE} values. If you vary V_{BE} and H_{FE} over the following ranges and your circuit currents and voltages remain OK then the exact transistor type is probably unimportant -

- Small signal transistor H_{FE} ranges from 50 to 500.
- Power transistor H_{FE} ranges from 10 to 100.
- V_{BE} ranges from 0.5 to 0.9 volts.

Summary of circuit types -

- *Standard* is stable and meets most needs.
- *CB* is low cost, less stable. Often used for low-level audio or RF stages where collector voltage swing is not critical. Provides a direct emitter ground.
- *CBE* and *CBE-Div* have poor stability.
- *CB-Div* is the “Vbe multiplier” circuit.
- *Vcc* and *Vbb-Div* are variations of *Standard.*

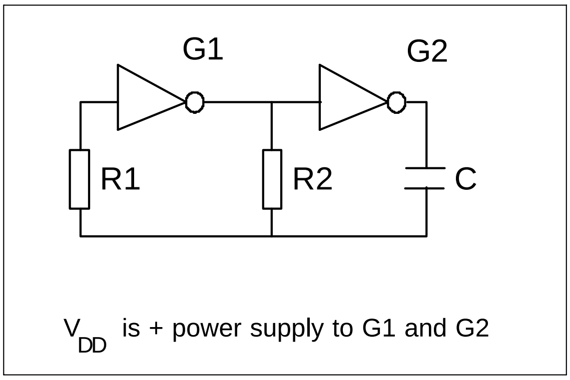
The Bias Worksheet only gives you resistor values - you may need coupling and bypass capacitors to complete the circuit.

<TOC 1 "CMOS Gate Oscillators">CMOS Gate Oscillators

Overview

The CMOS Gate Oscillator worksheet calculates component values for stable RC oscillators built from CMOS inverting logic gates.

Concept



[1]

G1 is an ordinary CMOS inverter with an input transition voltage of approximately $V_{DD}/2$, such as the 74HC04, 4049, 4069 or other gate operating as an inverter. In particular, G1 may not be a Schmitt input device such as the 74HC14, 40106, 4584 or a TTL input level device such as any of the 74HCT series.

The period of oscillation is given by

$$T = R_2C \ln\left[\frac{V_{DD}+V_{TR}}{V_{TR}}\right] + R_2C \ln\left[\frac{V_{DD}+V_{TR}}{V_{DD}-V_{TR}}\right] \qquad (1)$$

where

V_{DD} = supply voltage

V_{TR} = transition voltage of Inverter 1. Due to the high gain of the CMOS inverter, we can consider that the output voltage swings abruptly as the input voltage passes through the transition voltage.

The above assumes that no current flows through R_1. The input protection diodes of the CMOS gate mean that some current does flow through R_1 .The current can be neglected provided $R_1 \gg R_2$.

For a CMOS gate, $V_{TR} \approx V_{DD}/2$ giving

$T = 2\, R_2 C \ln[3$

or

$T = 2.2\, R_2 C$ (2)

In (1) small changes in V_{TR} tend to cancel, making the oscillator stable against V_{TR} shifts with temperature and against V_{TR} differences between individual CMOS gate samples.

CMOS datasheets show that V_{TR} in (1) remains at around half of V_{DD} as V_{DD} alters, making the oscillator stable against V_{DD} variation.

The worksheet evaluates the effect of parameters which make the circuit sensitive to change with temperature, CMOS manufacturing variations, circuit leakages and stray capacitance. The worksheet uses the following criteria to rate stability as "excellent" or "good". If neither set of conditions is met, stability is rated "poor".

Excellent Stability	**Good Stability**	**Minimises effect of**
$R_1 C_{IN} < 100$ T	$R_1 C_{IN} < 10$ T	Input Capacitance of G1
$R_1 > 10$ R2	$R_1 > 2 R_2$	G1 protection diode loading
$R_1 < 500$ k ohm	$R_1 < 1$ M ohm	G1 input and PC board leakage
C > 500 pF	C > 150 pF	Stray capacitance
$R_2 > 100 R_{OUT}$	$R_2 > 10 R_{OUT}$	Loading of G1 & G2 outputs

where

C_{IN} = input capacitance of G1 (15 pF)

R_{OUT} = output resistance of G1 (500 ohm for 4000 and 74Cxx, 50 ohm for 74HCxx)

In auto-C mode, the worksheet sets $R_2 = 100 R_{OUT}$ and calculates C from (2). If C < 150 pF then sets $R_2 = 10 R_{OUT}$ and recalculates C.

Oscillators are surprisingly stable with supply voltage and temperature. The following shows stability with supply voltage for a test oscillator

R_1	R_2	**C**	**Shift in period for V_{DD} = 4 to 15 Volts**
430 k ohm	43 k ohm	1n0	1.4 %
100 k ohm	43 k ohm	1n0	2.4 %
43 k ohm	43 k ohm	1n0	3.7 %

References

Astable and Monostable Multivibrators, *RCA COS/MOS Integrated Circuits Manual*, RCA Solid State Division, pp 89-104, 1972.

Watts, Mike: "CMOS Oscillators", National Semiconductor Application Note AN-118, October 1974.

Notes

An oscillator rated “excellent” may vary 2 percent over a VDD range of 4 to 15 V, 1 percent over the temperature range 0 to 100 ºC and 3 percent between extreme samples of CMOS inverters.

At frequencies above 50 kHz, the 74HC family inverters provide better stability than the 4000 / 74C series due to lower output resistance and lower gate delay. There is little difference between logic families at the lower frequencies.

<TOC 1 "Curve Fit">Curve Fit

Overview

The Curve Fit worksheet produces the equation for a line passing through a set of user-specified points.

Concept

The worksheet uses the Least Squares criterion to find a polynomial which approximates a set of points. The total of the squares of the errors between the points and the curve is minimised.

Notes

The Curve Fit worksheet is a great way to get empirical data into a computer program. The polynomial from the Curve Fit worksheet can easily be evaluated in languages such as Basic, or in a spreadsheet. Data represented by graphs can be taken from engineering handbooks and the like and made available inside a program.

To fit a Nth order polynomial, at least N+1 points are required. If necessary the worksheet will reduce the polynomial order.

Fitting a curve requires some technique.

- The more points you enter, the better the fit.
- Points should be fairly evenly spaced along the X axis.
- You can add extra points near areas where the fit needs improving.
- Its not possible to get a good fit to a set of points with an abrupt step.
- If a curve has a sharp “bend”, break it into two or three sections and produce a polynomial for each section.
- Its easier to produce a couple of low order polynomials which cover a set of points, than it is to find a single high order polynomial.
- If the approximatin curve oscillates near its ends, add extra data points near the ends or invent extra data points beyond the curve ends to move the oscillation out of range.
- If points do not progress steadily along the X direction, it becomes impossible to fit a polynomial. You may need to swap the X and Y axis.

Suppose you produce a polynomial such as

$$Y = aX^3 + bX^2 + cX + d$$

and want to use it in a computer program. It is easy to write code which finds Y when you know X. However, to find X when you know Y, you will have to use a numerical method to “home in” on the value of X . The Newton Iteration is a popular method.

<TOC 1 "Inductor">Inductor

Overview

The Inductor worksheet designs and analyses single layer coils.

Concept

The accurate design of single layer solenoid inductors has been well understood since the 1940s. A group of formulas and graphs are used together to convert a set of coil dimensions into inductance, Q and self-resonant frequency values. The Inductance worksheet uses the identical method, except that the graphs are replaced by polynomial approximations.

The worksheet is based on an article by Geffe which gives polynomial approximations to graphed values and shows a method of adjusting the coil length to produce an integer number of turns.

Geffe, Philip R.: The Design of Single-Layer Solenoids for RF Filters, *Microwave Journal*, pp. 71-76, December, 1996.

The Geffe article contains a misprint on p 71. The corrected formula is

:$$s = 1 - \frac{2.length.(a+b)}{\pi.turndiam.n.K_n}$$

Notes

Use the *analysis* page to tell you the inductance, Q and self-resonant frequency of an existing coil or to play freely with coil dimensions.

Use the *design* page to quickly create a high Q inductor.

If you enter a coil length in the *design* page, the worksheet will adjust the length slightly to allow an integral number of turns while maintaining the target inductance value. For some applications, a whole number of turns is an advantage. In the *analysis* page, you can play with fractional turns.

In the *design* page, a space is inserted between turns in order to maximise coil Q. If you set the coil length, the worksheet chooses a wire thin enough to allow the correct spacing between turns. If you set the wire diameter, the worksheet chooses a coil length sufficient to allow the correct spacing between turns.

If you require a better coil Q, you can usually :

- Increase coil diameter.
- Optimise Q by making the coil length roughly equal to the diameter.

For any coil, Q peaks at a certain frequency and falls above and below that frequency.

Unlike simple formulas which work only for very restricted length to diameter ratios, the worksheet is accurate for length : diameter ratios between 10:1 and 1:10 .

<TOC 1 "Editor">Editor

Overview

The Editor worksheet stores notes and simple documents.

Notes

Use the Editor for design task notes. Usually a single Editor worksheet is all you will need for a project. If you like, you can use formatting such as bold, underline and italics.

Simply type into the Editor text area - you cannot “Save” or “Open” a document since the document is saved into the Project file.

Click the floppy disk icon when you want to export the text into a separate text file. You could then load the file into a wordprocessor.

Click the open folder icon when you want to import a text file into the Editor.

You have a choice of two import and export file formats : RTF (Rich Text Format) and Text. RTF preserves formatting such as typefaces, font size, bold and underline. Text is a universal format which omits all formatting. When you click the import or export icon, a file dialog appears. Look for the dropbox labelled “Files of Type” and select “Rich Text Files (*.RTF)” or “Text Files (*.TXT)”.

<TOC 1 "Match">Match

Overview

The Match worksheet calculates component values for resistance transforming networks. The networks work over a narrow band of frequencies determined by the network Q. A resistance connected to either end of the network is transformed to the design resistance at the other end.

Concept

Apart from the simple transformer, all the networks are based on the impedance transforming property of an LC tuned circuit. Most of the networks use *three* reactances rather than two. The third reactance gives freedom to choose a Q above the basic value.

The networks are analysed by writing an expression for the input impedance at one end when the other end is terminated in a resistance. To make the input impedance resistive the imaginary part of the expression is set to zero, which gives a condition which must be met. Putting this condition into the expression for the real part of the input impedance leaves only one remaining degree of freedom - the circuit Q.

Example

I have a 7 MHz, 10 Watt radio transmitter with a single ended output stage. I need a matching network to connect the output transistor to a 50 ohm coax . My transmitter is powered from 12 Volts.

The output stage for an output stage of this type requires a load of approximately

$$R_L = \frac{(V_{CC})^2}{(2 \times P)}$$ where Vcc = supply voltage, P = Power output.

ie $\frac{(12)^2}{(2 \times 10)}$ = 7 ohms.

In the Match worksheet I enter *R1* = 50 and *R2* = 7 . I enter *Q* = 12 because this is a commonly used value for transmitters - too high a Q means high matching circuit losses and narrow bandwidth, while too low a Q means poor harmonic suppression. I then press the *Enter* key to recalculate.

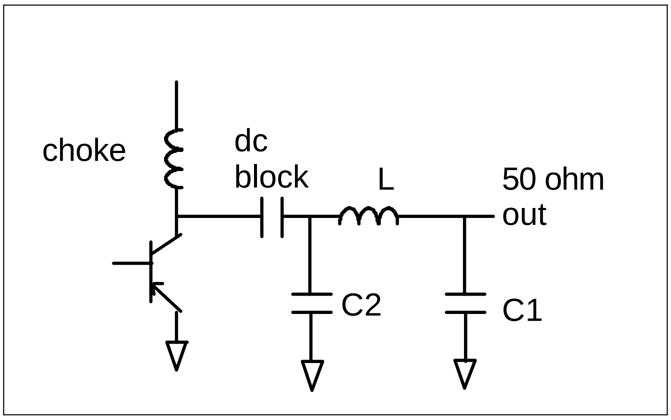
I click the *Pi* Network tab because the Pi network is a good choice, being a low pass network with good harmonic suppression. The capacitive input can also be a useful aid to output stage stability.

I read

- *C2* = 14.269 nF

- *L* = 17.204 uH
- *C1* = 5.4567 nF

To tune my output stage, I will need to make L variable or perhaps part of C1 can be provided by a variable capacitor. C2 should include any output capacitance of the output transistor - probably insignificant at these frequencies. A DC feed choke and a DC blocking capacitor complete the network.



The large Pi capacitor values are not easily made tunable - the LCC 1 network is an interesting alternative.

Single and Double Termination

The network Q is valid for a terminating resistance connected at one end only. If you need a matching network terminated at both ends, design with a Q *twice* the desired value.

The following are examples of singly terminated use -

- A matching network connects the antenna to the base of a dual gate mosfet in a receiver front end. At frequencies up to 100 MHz or so, the dual gate mosfet does not resistively load the matching network. The only load is the antenna resistance.
- A matching network connects the output stage of a transmitter to an antenna or other resistive load. Most output stages force current pulses into the matching network and do not resistively load the network.

A doubly terminated matching network occurs when a resistive source, like a correctly tuned antenna or a signal generator connects through a matching network to a resistive load such as an attenuator or bipolar amplifier

Notes

Apart from the transformer circuit, all the networks have similar properties for frequencies close to the design frequency, provided they are designed with the same Q value.

Away from the design frequency, the various networks show different behaviours -

- The Pi, Tee and L networks pass low frequencies unattenuated, while the others block low frequencies.
- The Tapped C and Top C networks degenerate into a shunt and series capacitors pair at high frequencies where they offer limited attenuation.

The Top C network is interesting because it demonstrates that a single capacitor can provide matching to the hot end of a tuned circuit.

<TOC 1 "Power & Noise">Power & Noise

Overview

- Power is produced by voltage across a resistance.
- Power is produced by thermal noise in a resistance.
- The Power & Noise worksheet handles the relationships between power, voltage, resistance, temperature and bandwidth.

Concept

temperature and noise to power

A resistor delivers a noise power which depends only on its temperature and the bandwidth of interest.

$$P = KTB$$

P = power delivered into a noiseless resistance connected to the noisy resistor terminals and equal in value to the noisy resistor.

K = Boltzman’s Constant (1.3806 x 10^{-23})
B = Bandwidth Hz
T = temperature in degrees Kelvin.

power to dbm

The dbm is a measure of power. 0 dbm is 1 milliwatt.

$$dbm = 10\log(P \times 1000)$$

P = Power in watts.

Voltage to Power Relationship

The power delivered into a resistance depends on the voltage across the resistance terminals.

$$P = \frac{V^2}{RP}$$

P = Power, watts.
V = RMS voltage across the resistor.

R = value of resistance, ohms.

Examples

example 1

My scanner claims to have a sensitivity of 1 microvolt for an SSB signal to noise ratio of 10 db. How good is it ?

In the Power and Noise worksheet I enter

Resistance = 50 (my scanner antenna socket is marked 50 ohm)
Bandwidth = 6000 (my scanner handbook mentions a 6 KHz bandwidth)
Temperature = Room Temp (we're operating on planet earth)

I read a calculated voltage of 0.035 microvolts. This is the noise produced by a 50 ohm resistor at room temperature or by an antenna immersed in an environment at room temperature. Our scanner needs 20 log(1.0 uV / 0.035 uV) = 29 db more signal than the noise. A perfect receiver would need 10 db more for a 10 db signal to noise ratio, so we are 19 db short. There's room for improvement!

example 2

My digital multimeter tells me that my audio signal generator is putting out 2V RMS. How many dbm is that in a 600 ohm system ?

In the Power & Noise worksheet I enter

Resistance = 600
voltage = 2

I readout a calculated value of 8.24 dbm.

example 3

My exciter is rated at +25 dbm output. How many watts is that ?

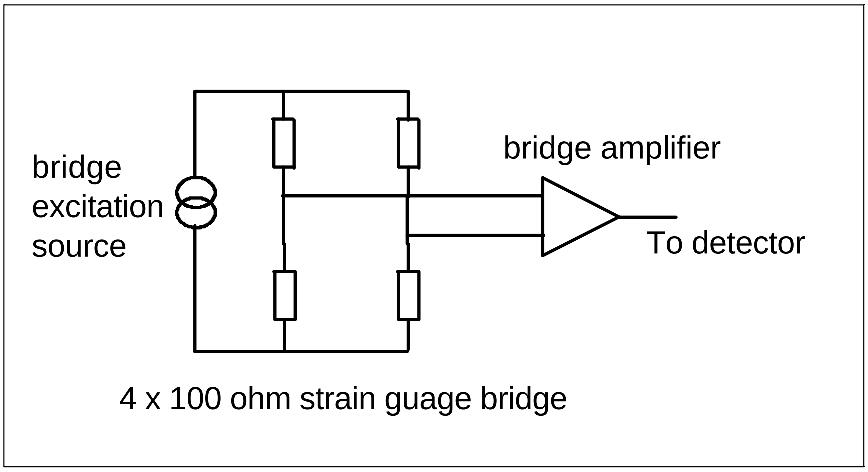
In the Power & Noise worksheet I enter

dbm = 25

I readout a calculated power of 0.316 watts.

Example 4

I need to measure a 20 microvolt full scale signal output from a strain gauge bridge made up of four 100 ohm resistance elements. How narrow a bandwidth should my bridge amplifier have in order to keep the strain gauge signal well above the noise?



In the Power & Noise worksheet I enter

Resistance = 100 (resistance looking into 2 terminals of the bridge)
Voltage = 20e-6

I readout a bandwidth of 972 MHz . This is the bandwidth of an amplifier which lets through 20 microvolts of noise from the bridge. For a 0.01 percent noise power contribution, I should narrow the bandwidth to 0.01 percent ie 97.2 KHz (0.01 percent noise power is 1 percent noise voltage). If need be, I can further reduce the noise contribution by reducing the bandwidth even further.

<TOC 1 "Rectifier">Rectifier

Overview

The Rectifier worksheet predicts the performance and component stresses of a capacitor input power supply.

Concept

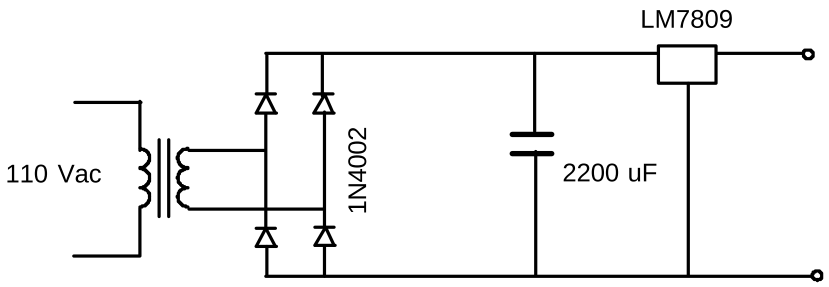
One or more diodes pump current pulses into each filter capacitor. The current pulse waveform is determined by the transformer winding resistance, transformer leakage inductance, diode voltage drop, surge resistor, capacitor equivalent series resistance and load current.

The non-linear diode makes the circuit difficult to solve analytically. The Rectifier worksheet uses a time stepped simulation where circuit charges are tracked at small increments of time. The non-linear diode is handled at each time step by iterating to find a circuit current that simultaneously meets the diode V vs I characteristic. The diode model uses 3 parameters to describe any diode type.

The model ignores iron losses. Iron losses can be modelled as an additional secondary winding connected to a resistive load. The additional load reduces the output resistance of the transformer, improving its regulation. Lossy transformers rated at just a couple of watts, tend to have high primary resistance and high core losses, giving a rectified output voltage above the value predicted by the simulator.

The worksheet is designed for supply frequencies of 25 to 500 Hz and neglects all frequency-dependent diode behaviour.

Example



I have a 110 volt to 12 volt at 700 mA transformer feeding a full wave rectifier consisting of four 1N4002 diodes which charge a 2200 uF capacitor. My multimeter measures a primary resistance of 40 ohms and a secondary resistance of 0.5 ohms. With no load on the transformer I measure 14 Vac output at the secondary.

I want to connect a LM7809 regulator to the capacitor to supply 0.5 amp of regulated DC at 9 volts. Will my circuit work? What are the ratings for each component?

In the Rectifier worksheet

I select the *Schematic* tab then click *full-wave*.

I select the *Values* tab then enter
Primary Volts = 110
Primary Ohms = 40
Secondary Volts = 14 (no-load volts - not the 12V nominal loaded voltage)
Secondary Ohms = 0.5
Leakage Inductance = 0 (I don't know the actual value)
Capacitor = 2200
ESR = 0 (I don't know the actual value)
Diode = 1N4001 thru 1N4007
Load Amps = 0.5
Rs = 0 (I don't use Rs)
Supply Frequency = 60

I click the *Results* tab and observe the filter cap voltage. The design looks OK because the voltage stays up between current pulses.

I click the *Report* tab and observe -

The Transformer Secondary RMS Current is 914 mA . My 700 mA transformer is going to overheat so I'll have to choose a bigger transformer!

The Capacitor ripple current is 766 mA. I have to be sure my cap is specified for at least this ripple current!

Ripple Minimum (at the ripple troughs) is 14.4 Volt. This is 5.4 volts above the regulator output. The National Semiconductor datasheet for the LM7809 tells me that I need at least 1.7 volts across the regulator at 0.5 amp. I've got 5.4 - 1.7 = 3.7 volts to spare, so my circuit will function with a generous margin of safety.

Diode Dissipation is 276 mW per diode. My 1N4002 datasheet quotes a thermal resistance of 45 ^{o}C / watt, giving a temperature rise of a mere 0.276 x 45 = 12.4 ^{o}C above ambient. The circuit will handle ambient temperatures of 100 ^{o}C or more and still have a diode junction temperature well below the datasheet 175 ^{o}C limit.

The transformer secondary current graph shows a switch-on half cycle surge of 8 A peak, well below the datasheet 30 amp surge limit.

The typical tolerance extremes for an electrolytic might be -20% and +100% (1760 and 4400 uF). I click the *Values* tab and change the capacitor to the tolerance extremes, checking the *Report* each time. I note that the output voltage holds up well and the transformer and capacitor ratings don't change much.

Notes

You can measure most transformer parameters with a digital multimeter. Connect the primary to the mains and measure mains (primary) voltage and secondary no load voltage. Disconnect from the mains and measure the DC primary and secondary resistances.

Transformer leakage inductance has the effect of broadening the current pulses. However, for typical transformers the effect on rectified voltage is small, so its hardly worth bothering to measure the leakage inductance when a value of zero is close enough.

The Rectifier worksheet does not calculate diode Peak Inverse Voltage (PIV). For a secondary voltage of Vrms, use PIV = 1.4 x Vrms for the half wave and bridge circuits. For the other circuits, use PIV = 2.8 x Vac. Note that Vrms is half the total secondary voltage for the centre-tapped circuit.

Capacitor ESR can be zero for design purposes since it usually drops much less voltage than the transformer resistances and the diodes.

The Diode Dissipation value given in the *Report* is useful for diode selection. Because diode dissipation is hard to calculate without a simulator like the Rectifier Worksheet, manufacturer's datasheets rate diodes by a roundabout method using average forward current. However, a diode dissipation figure can often be extracted from the datasheet. Sometimes you can use figures for thermal resistance and maximum junction temperature to calculate a maximum allowed dissipation.

The Rectifier worksheet offers a selection of diode types. If you have diode that is not offered in the diode selection, you can get good results by selecting a listed diode of similar current rating and same type (shottky vs silicon).

The effect of variation of diode forward voltage with temperature has a small effect on circuit performance and can be neglected.

Don't try for very low ripple voltage unless load current is low. The big capacitor needed causes high diode dissipation and high transformer RMS current. If you really need a couple of volts more output at the ripple troughs, try a schottky rectifier or a rectifier of much higher rating - like a 25 A bridge in place of a 5 A bridge.

If you have an extremely lightly loaded rectifier, the simulator will produce an obviously incorrect "serrated" current waveform. Such a circuit is so lightly loaded that it doesn't need designing anyway!

The simulation breaks down if the leakage inductance value is so high that current flow does not fall to zero between current pulses. Such leakage inductance values are unrealistic anyway.

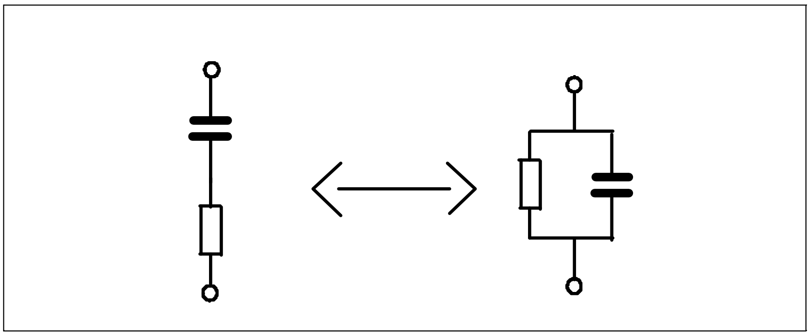
<TOC 1 "Serial - Parallel Converter">Serial - Parallel Converter

Overview

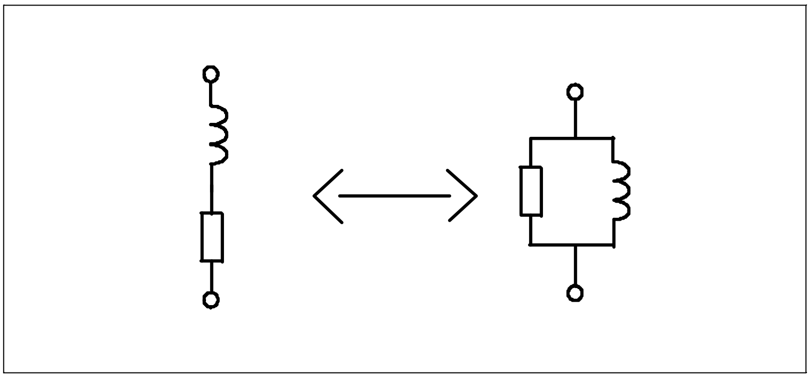
The Serial - Parallel worksheet converts a parallel resistor and reactance into an equivalent series resistor and reactance. This is useful for network analysis and impedance matching.

Concept

For every series RC network, an equivalent parallel RC network can be found which presents the identical impedance. The networks are equivalent only at a single frequency.



In the same way, RL series and parallel equivalents exist.



Derivation of the RC Parallel to Series formula -:

Impedance of a parallel network R_P and C_P :

$$Zp = \frac{1}{\frac{1}{R_P} + \frac{1}{jwC_P}}$$

rearranging gives -

$$Z_P = \frac{R_P\left(wC_p\right)^2 + jwC_P(R_P)^2}{(R_P)^2 + (wC_P)^2} \qquad (1)$$

Impedance of a series network R_S and C_S :

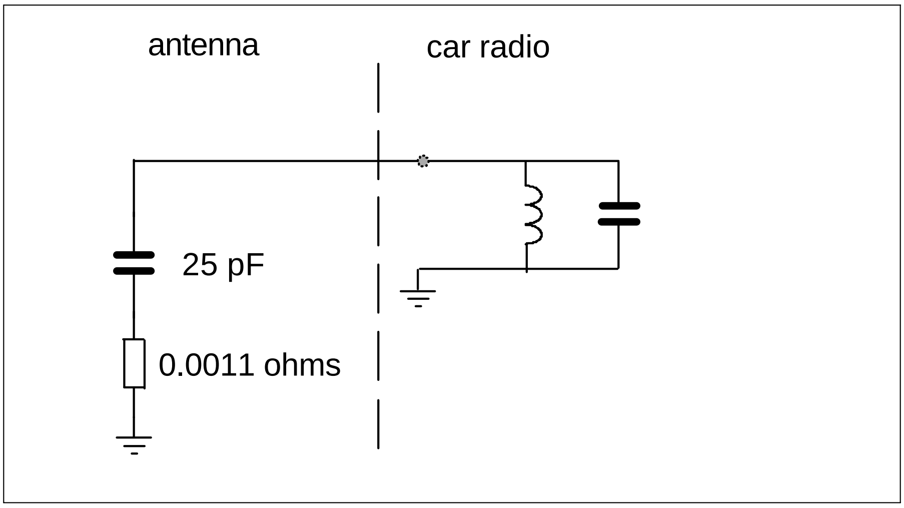
$$Z_s = R_S + jwC_S \qquad (2)$$

for $Z_S = Z_P$ we set $Re(Z_S) = Re(Z_P)$ and $Im(Z_S) = Im(Z_P)$ which gives the parallel to series formulas -

$$R_S = \frac{R_P(wC_P)^2}{(R_P)^2+(wC_P)^2} \text{ and } \qquad Cs = \frac{C_P(R_P)^2}{(R_P)^2+(wC_P)^2}$$

Example

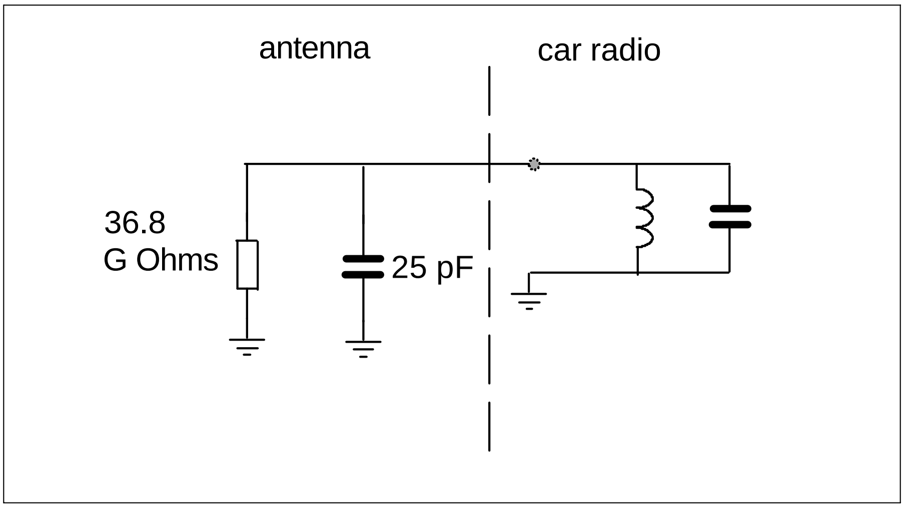
An antenna handbook gives figures for a car radio AM whip antenna at 1 MHz. The book quotes a radiation resistance of 0.0011 ohms and a capacitive reactance equivalent to 25 pF. The antenna is connected directly to the "hot" end of a parallel tuned circuit inside the car radio. What resistance does the radiation resistance present across the tuned circuit ?



In the Series - Parallel worksheet, I enter -

frequ, Hz	=	1MHz
Rs	=	0.0011 Ohms
Cs	=	25pF

I hit the enter key and read
Rp : 36.8 G Ohms
Cp : 25 pF



This very high resistance has negligible effect on the tuned circuit, which demonstrates just how ineffecient AM antennas can be! Only the antenna capacitance is significant.

Notes

Most RLC networks can be analysed with nothing more than the series to parallel converter.

A series - parallel equivalent circuit can be considered valid over a small bandwidth such as a the bandwidth of a radio channel. Q and impedance calcuation are performed as though the transformed circuit is the actual circuit in use.

<TOC 1 "Tuned Circuit Calculator">Tuned Circuit Calculator

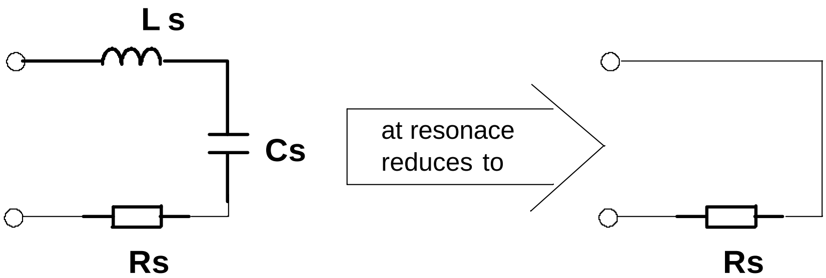
Overview

The tuned circuit calculator handles the inductance, capacitance, Q and series or parallel resistance of a tuned circuit.

Concept

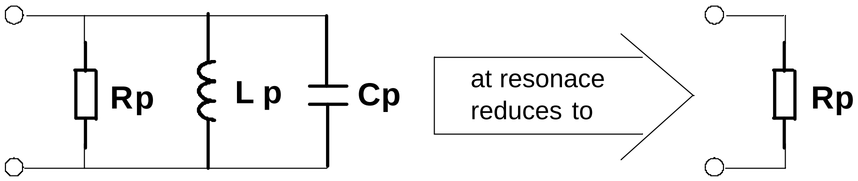
Resonance

The series resonant circuit consists of R, C, L in series. At resonance L and C “cancel out” leaving just the resistance.



.

The parallel resonant circuit consists of R, C, L in parallel. . At resonance L and C “cancel out” leaving just the resistance.



fo, the frequency for resonance is the same for parallel and series circuits :

$$f_O = \frac{1}{2.\pi.\sqrt{L.C}} \text{ (Hz)} \quad (1)$$

Reactance

The reactance (X) of a circuit element which is a capacitor or inductor is a kind of “ac resistance” and at any particular frequency is defined as :

$$X = \frac{\text{ac voltage across element}}{\text{ac current through element}}$$

The reactance of a capacitor of value C (Farads) at frequency f (Hz) is given by

$X_C = \frac{1}{2.\pi.f.C}$ ohms. (2)

The reactance of an inductor of value L (Henries) at frequency f (Hz) is given by

$X_L = 2.\pi.f.L$ ohms. (3)

At resonance, the reactances of the inductor and capacitor are equal, ie

$X_C = X_L$. (4)

Q

The Q of a tuned circuit tells us the ratio of power exchanged between inductor and capacitor per cycle and the power lost in the resistance R_S or R_P per cycle. Thus a high Q circuit has low losses.

For the series resonant circuit : $Q = \frac{X}{R_S}$ (5)

where X is the reactance of C_S or L_S.

For the parallel resonant circuit : $Q = \frac{R_P}{X}$ (6)

where X is the reactance of C_P or L_P.

Real World Resonant Circuits

In practical situations we use RS or RP to represent all tuned circuit circuit losses. These losses are due to 3 causes :

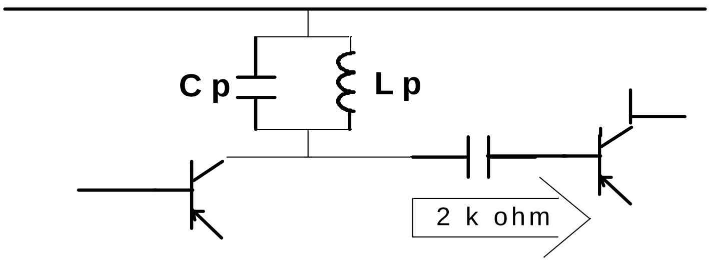
- Capacitor losses
- Inductor losses
- Losses in other components such as transistors or antennas which are connected to the resonant circuit.

Lumping all losses into R_S or R_P, provided Q is reasonably high, say 5 or greater.

The Tuned Circuit Worksheet simply manages equations (1) to (6) above. However, in real world situations there are complications due to the fact that R_S and R_P are not produced by physical resistors.

Example 1

I need to place a tuned circuit in the collector of a transistor, in order to reject out of band noise. I want a Q of 30. The input resistance of the following stage at 2 k ohms dominates other circuit resistances. The circuit operates at 1.5 MHz. What values of C_P and L_P should I use ?



In the Tuned Circuit Worksheet I enter :

Frequency =	1.5 MHz	
Rp	=	2 k
Q	=	30

I hit the ENTER key to recalculate and read :

L	: 7.07 uH
C	: 1.59 nF

We have assumed that the inductor has a much higher Q than 30 so that the 2K input resistance dominates over the R_P due to inductor losses. This is quite practical with inductors wound on a pot core or ferrite ring. Mosts capacitor types will have insignificant losses.

Example 2

I want to clip my oscilliscope probe onto the “hot” end of a tuned circuit in an oscillator in order to check signal amplitude. The oscillator uses a 350pF capacitor. The oscillator runs at 7 MHz. At 7 MHz my oscilliscope probe presents an input resistance of 4 K ohms and a 12 pF capacitance. Will the measurement be of useful accuracy ?

In the Tuned Circuit Worksheet I enter :

Frequency	= 7 MHz
C	= 350 pF
Rp	= 4 K ohms

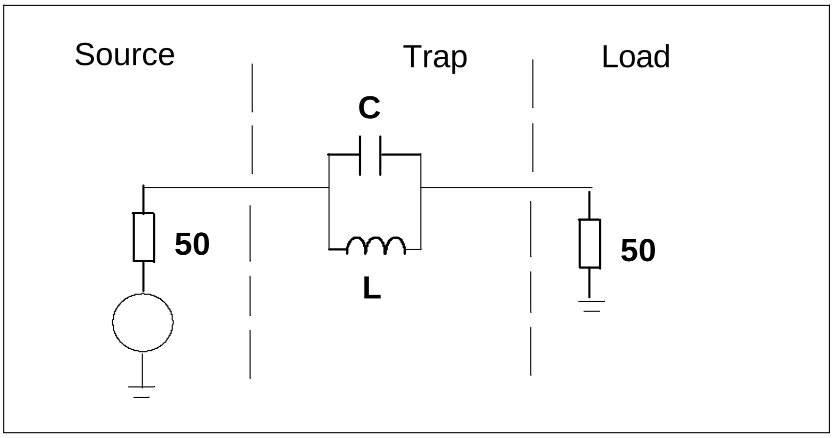
I hit the Enter key to recalculate and read :

Q	: 61.5

This is quite a low Q for a tuned circuit used in a stable oscillator. From experience, I expect coil Q values of 120 upwards to define tuned circuit Q. Unless the oscillator is clipping or has some form of amplitude control, my proble input resistance will load the oscillator heavily and my oscilliscope wil display an unreasonably low amplitude signal.

Example 3

I want to insert a notch filter or “trap” between a 50 ohm source and a 50 ohm receiver to remove an unwanted frequency. The trap is to reject a signal at 5 MHz. At resonance, I want the trap to have 40 db rejection. I estimate that I can wind a coil with a Q of 100. What are the values of L, C and what is the insertion loss of my trap ?



The parallel tuned circuit “sees” a resistance of 50 + 50 = 100 ohmns. 40 db represents a voltage ratio of 100, so for 40 db rejection, R_P due to the Q of LP must be 99 times 100 ohms, ie 9900 ohms.

In the Tuned Circuit Worksheet I enter :.

Frequency	= 5 MHz
Q	= 100
RP	= 9900 ohms

I hit the Enter key to recalculate and read

C	= 321 pF
L	= 3.15 uH
RS	== 0.99 ohms

C and L are as read. Insertion loss *well above* the trap frequency depends on the series resistance of C which is very low, giving a low insertion loss. Insertion loss *well below* the trap frequency depends on the series resistance of L, which depends on the Q of L. We can estimate the series resistance as R_S and calculate the insertion loss as :

$$I_L = -20 \log\left(\frac{50}{R_S+50}\right) \text{ db}$$

= 0.17 db