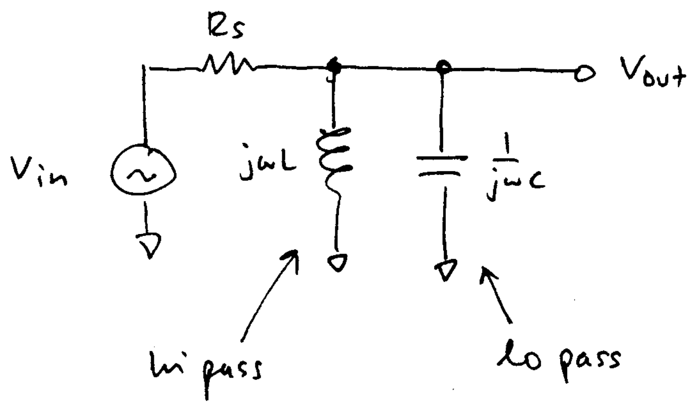
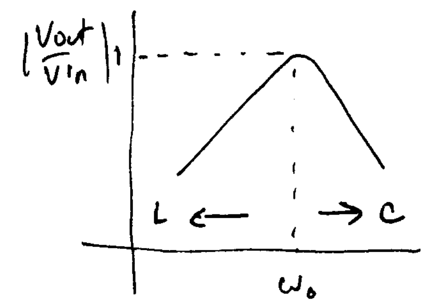
Resonant Circuits

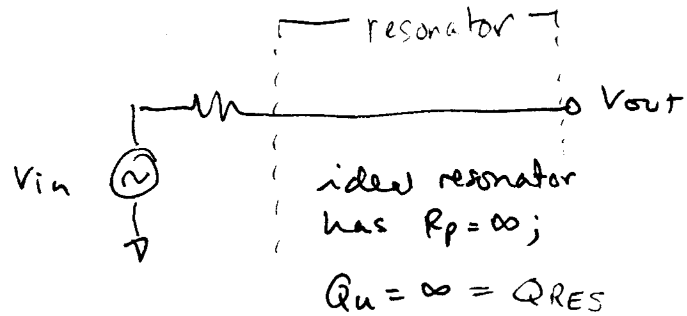
Widely used when bandpass characteristics are required for interstage matching or filtering.





at $\omega_0^2 = \frac{1}{LC}$ reactances equal and opposite

$j\omega L = -\frac{j}{\omega C}$; $V_0 = V_{in}$ in this highly ideal case w/o losses or load



If R_p, the parallel equivalent of component loss is finite, then $V_{out} < V_{in}$.

Quality factor, Q

Reactive components such as capacitors and inductors are often described with a figure of merit called Q. While it can be defined in many ways, it's most fundamental description is:

$$Q = \omega \frac{energy\ stored}{average\ power\ dissipated}$$

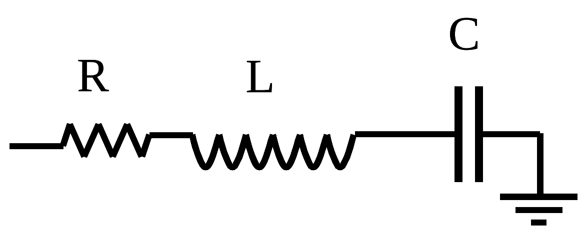
Thus, it is a measure of the ratio of stored vs. lost energy per unit time. Note that this definition does not specify what type of system is required. Thus, it is quite general. Recall that an ideal reactive component (capacitor or inductor) stores energy

$$E = \frac{1}{2} CV_{pk}^2 \quad or \quad \frac{1}{2} LI_{pk}^2$$

Since any real component also has loss due to the resistive component, the average power dissipated is

$$P_{avg} = \frac{1}{2} I_{pk}^2 R = \frac{V_{pk}^2}{2R}$$

If we consider an example of a series resonant circuit.



At resonance, the reactances cancel out leaving just a peak voltage, Vpk, across the loss resistance, R. Thus, Ipk = Vpk/R is the maximum current which passes through all elements. Then,

$$Q = \omega_O \frac{LI_{pk}^2 / 2}{I_{pk}^2 R / 2} = \frac{\omega_O L}{R} = \frac{1}{\omega_O RC}$$

In terms of the series equivalent network for a capacitor shown above, its Q is given by:

$$Q = \frac{1}{\omega RC} = \frac{X}{R}$$

where we pretend that the capacitor is resonated with an ideal inductor at frequency ω. X is the capacitive reactance, and R is the series resistance. Since this Q refers only to the capacitor itself, in isolation from the rest of the circuit, it is called unloaded Q or Q_U. The higher the unloaded Q, the lower the loss. Notice that the Q decreases with frequency.

The unloaded Q of an inductor is given by

$$Q_U = \frac{\omega_o L}{R}$$

where R is a series resistance as described above. Note that Q is proportional to frequency for an inductor. The Q of an inductor will depend upon the wire diameter, core material (air, powdered iron, ferrite) and whether or not it is in a shielded metal can.

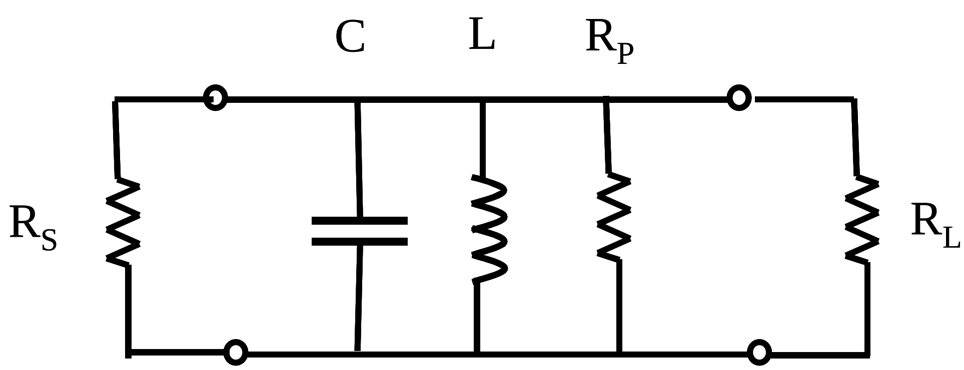
It is easy to show that for a parallel resonant circuit, the Q is given by susceptance/conductance:

$$Q = \frac{B}{G}$$

where B is the susceptance of the capacitor or inductor and G is the shunt conductance.

Loaded Q.

When a resonant circuit is connected to the outside world, its total losses (let's call them R_P or G_P) are combined with the source and load resistances, R_S and R_L. For example,



Here is a parallel resonant circuit (C,L and R_P)connected to the outside. The total Q of this circuit is called the loaded Q or Q_L and is given by

$$Q_L = \omega_o C (R_P \| R_S \| R_L)$$

or

$$Q_L = \frac{\omega_o C}{G_S + G_L + G_P} = \frac{\omega_o C}{G_{total}} = \frac{susceptance}{conductance}$$

The significance of this is that Q_L can be used to predict the bandwidth of a resonant circuit. We can see that higher Q_L leads to narrower bandwidth.

$$BW = \frac{\omega_o}{Q_L}$$

where

$$\omega_o = \frac{1}{\sqrt{LC}}$$

1. So, large C will increase the loaded Q at a given resonant frequency and reduce bandwidth.

2. Or, we could vary G_{total}. How?

A. Unloaded Q (Q_U) is an attribute of the passive L and C components and affects G_P. This will vary if we change L and C. And, $G_P = G_C + G_{ind}$,

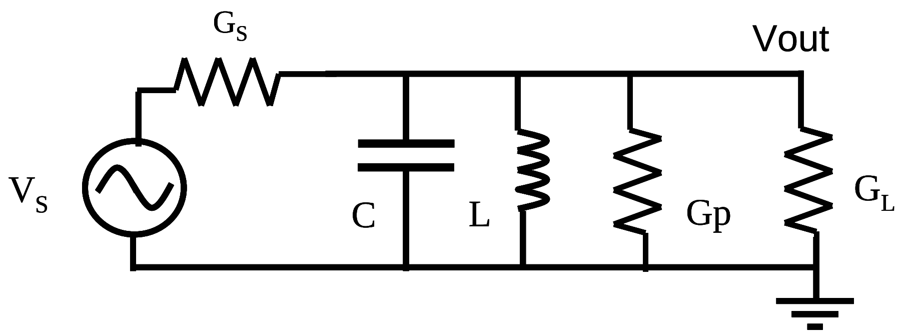
Capacitor: $$Q_U(capacitor) = \frac{\omega_o C}{G_C}$$

Inductor: $$Q_U(inductor) = \frac{1}{\omega_o L G_{ind}}$$

Using this to set the bandwidth of a resonator is not a good idea. If Q_U becomes comparable to Q_L, then the loss in the resonator becomes very high as we shall see.

Insertion Loss

The resonator can be used as a bandpass filter or matching network. In these applications, insertion loss can be important.



Consider this resonant LCR circuit. When w = wo,

$$\frac{V_{out}(j\omega_o)}{V_S} = \frac{G_S}{G_S + G_L + G_P} < 1$$

Typically, $G_S = G_L = G$. We need to find S_{21} to determine insertion loss. Recall that $|S_{21}|^2$ = transducer gain = Power delivered to the load/available power from the source when the source and load are both Zo.

First note that

$$\frac{Q_L}{Q_U} = \frac{G_P}{G_P + 2G}$$

And,

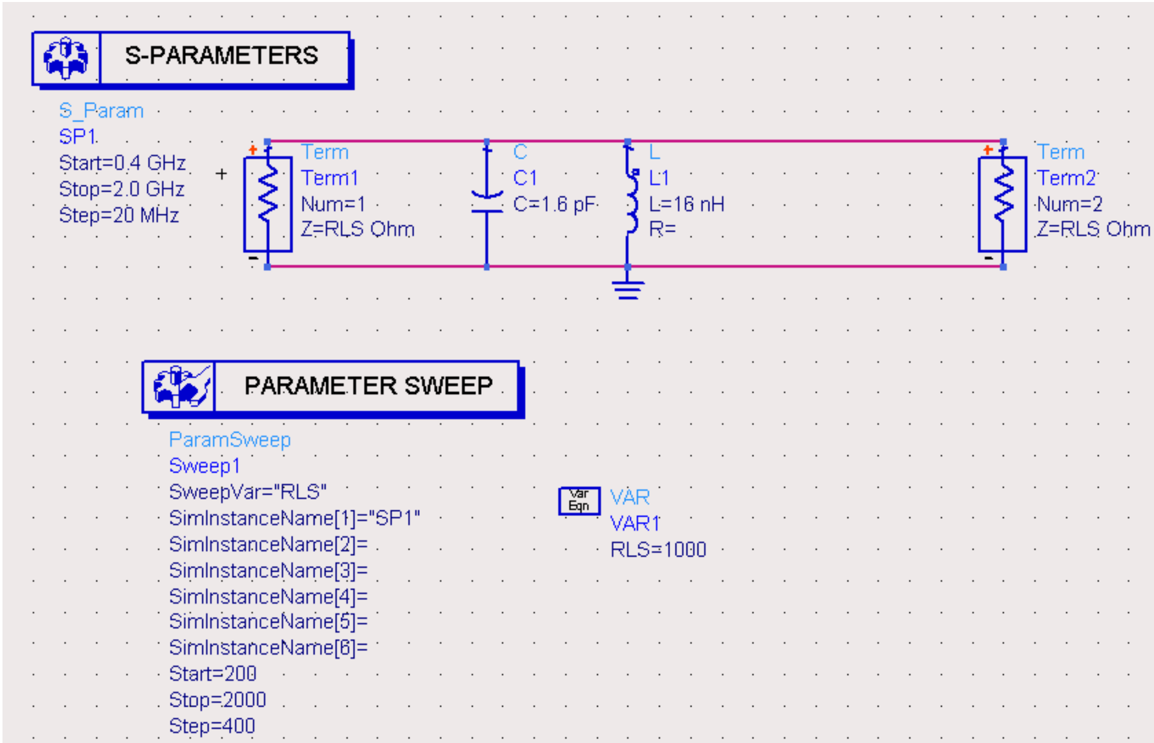
$$S_{21} = \frac{2V_{out}}{V_S} = 1 - \frac{Q_L}{Q_U} = \frac{2G}{G_P + 2G}$$

So, insertion loss (dB):

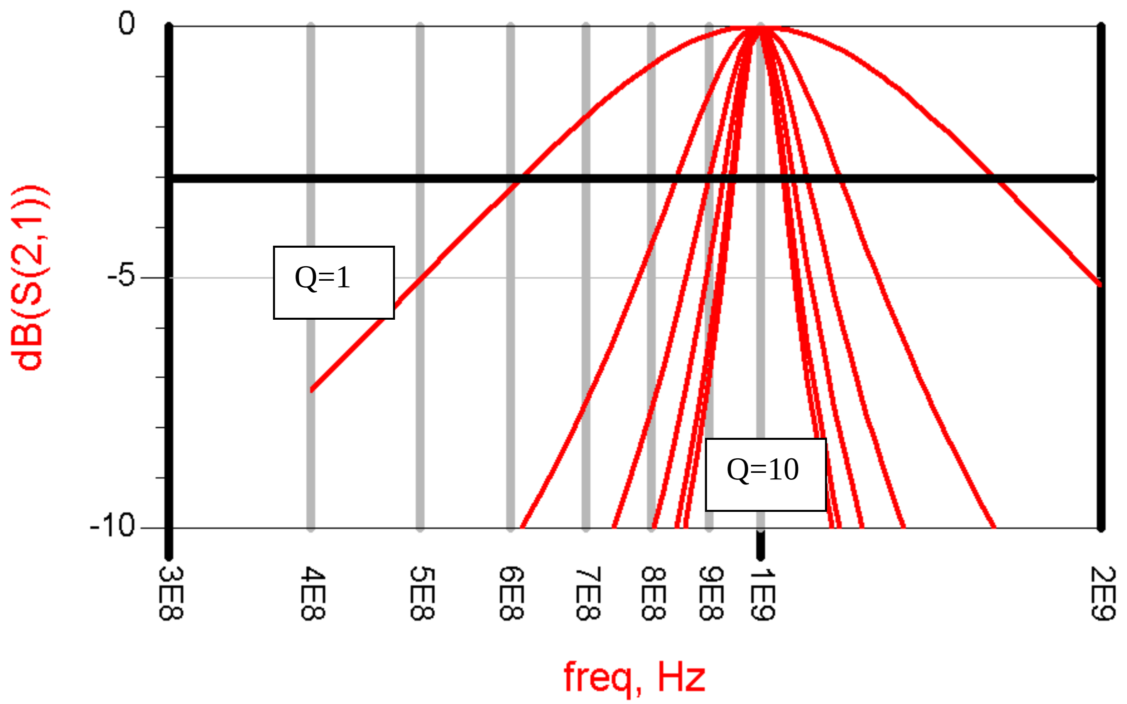
$$IL = 20\log\left(1 - \frac{Q_L}{Q_U}\right)$$

Thus, narrow bandwidth where Q_L and Q_U are similar can be very lossy.

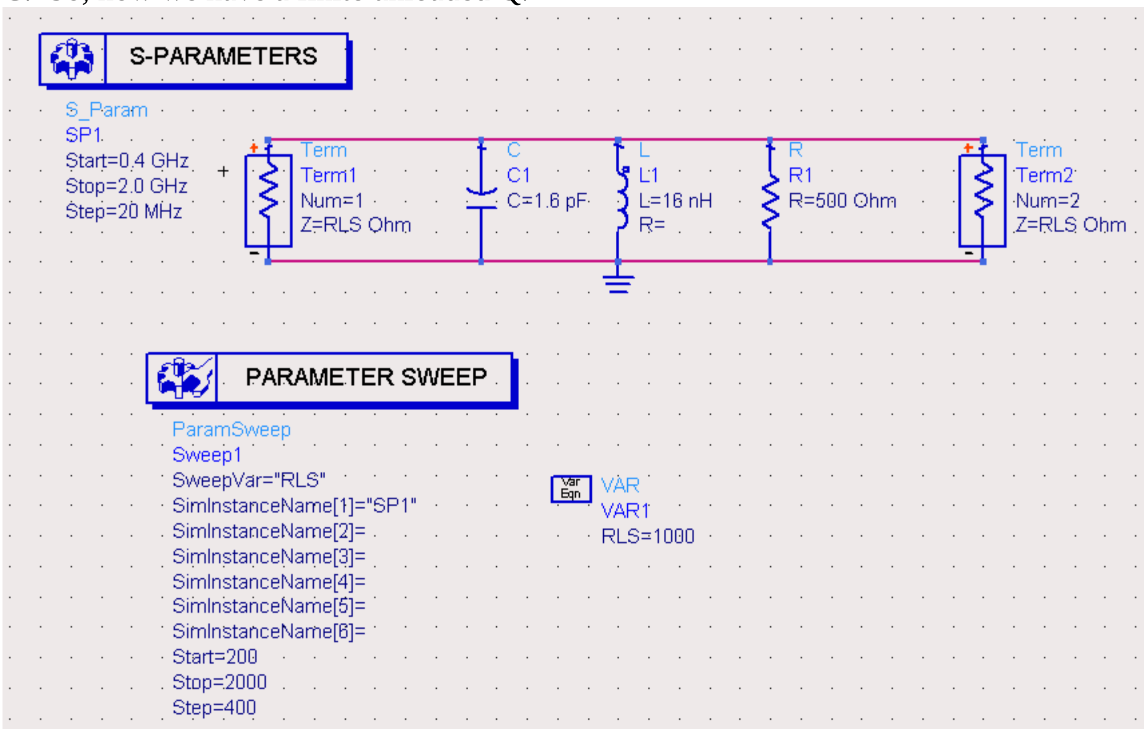
Here is an example of a simple resonant circuit. The unloaded Q is infinite, since no losses are included in the network. We see that there is no insertion loss in this case.



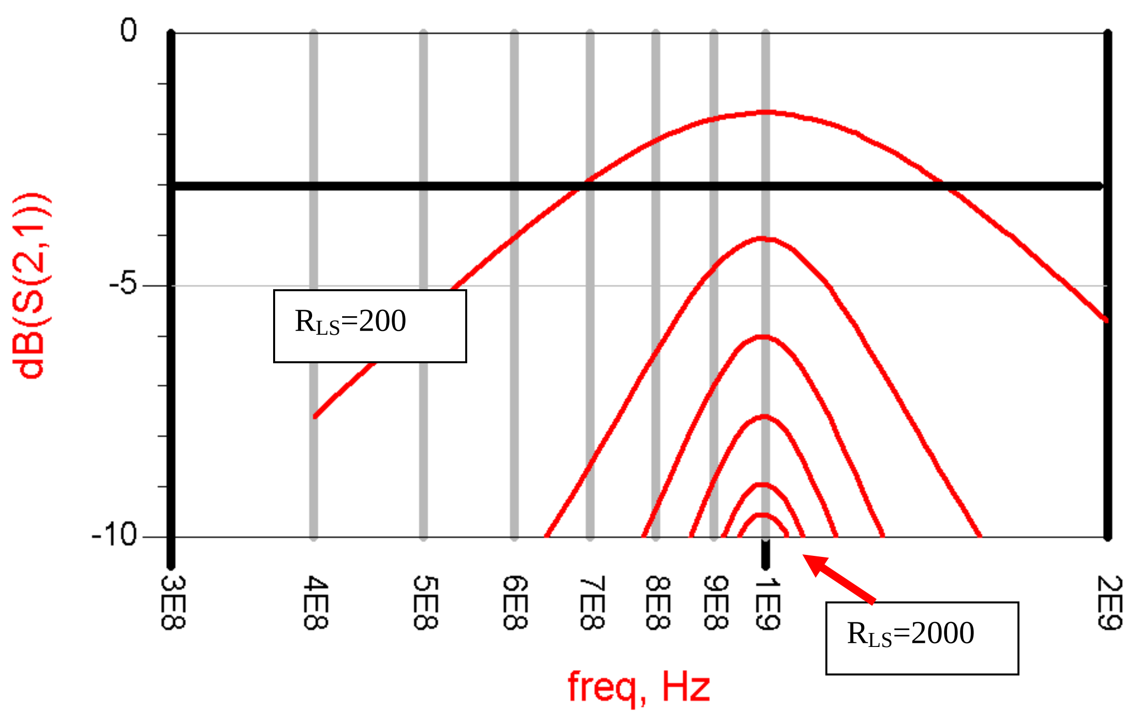
Loaded Q varies from 1 to 10 with the given parameter sweep.



Now, the circuit is modified to include a 500 ohm resistor (R_P) in parallel with the LC network. This resistance represents the parallel equivalent loss due to both the L and the C. So, now we have a finite unloaded Q.

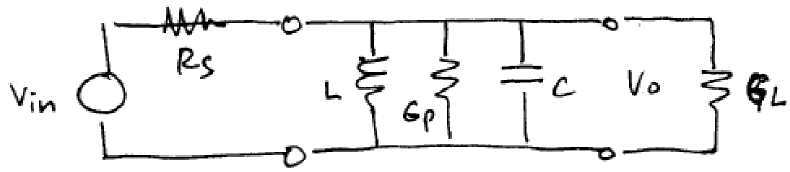


Note that the insertion loss increases as loaded Q, Q_L, approaches Q_U. Sweeping RLS, we see at resonance, the reactances cancel, and we are left with a resistive divider. Vout = Vin $[R_1/(2R_1+R_{LS})]$.



S21 (dB) = 20 log(2Vout/Vin)

2. B. Change external impedance level coupled to the intrinsic resonator.



$$\text{total } Q = \frac{\omega_o C}{G_S + G_p + G_L} = Q_L \text{ or } \underline{\text{loaded}}\ Q.$$

So, we can set the Q_L and bandwidth by adjusting the loading conductances/resistances.

$$BW = \frac{\omega_o}{Q_L}$$

But, we must make sure that $Q_U >> Q_L$ to avoid excessive losses.

So, if really narrow bandwidth is required, the solution generally requires multiple resonators or more complicated bandpass filter approaches or mechanical structures like quartz crystal filters.

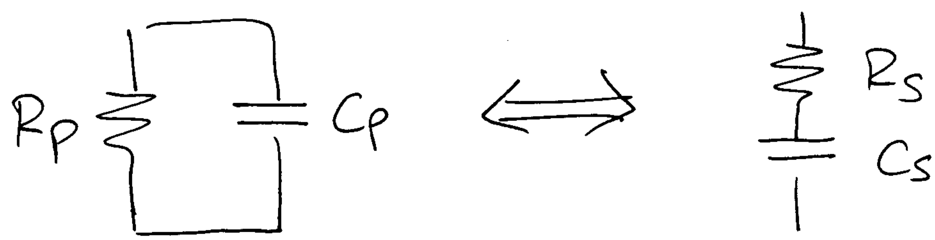
3. How can you set G_S and G_L? Aren't these dictated by the generator and load?

Not necessarily! We can use tapped C matching circuits to transform source and load impedances to whatever we desire (assuming we don't increase loss too much by approaching Q_U).

But, first, we will introduce a convenient approximation which can be used to transform a parallel equivalent circuit into a series equivalent circuit and vice-versa.

Useful approximation for series-parallel conversion

(<u>at a single frequency</u>)



$$Y_p = \frac{1}{Z_s}$$

$$Re\left(\frac{1}{Y_p}\right) = Re(Z_s)$$

$$Im\left(\frac{1}{Y_p}\right) = Im(Z_s)$$

$$Q_{parallel} = Q_{series}$$

$$\frac{R_p}{X_p} = Q = \frac{X_s}{R_s}$$

$$\mathrm{Re}(Z_S) = R_S = \frac{1/R_P}{1/R_P^2 + \omega^2 C_P^2} = \mathrm{Re}\left(\frac{1}{Y_P}\right)$$

$$= \frac{1/R_P}{1/R_P^2 + Q^2 R_P^2} = \frac{R_P}{1+Q^2}$$

This leads to:

$$R_S = R_P\left(\frac{1}{Q^2+1}\right)$$

$$X_S = X_P\left(\frac{Q^2}{Q^2+1}\right)$$

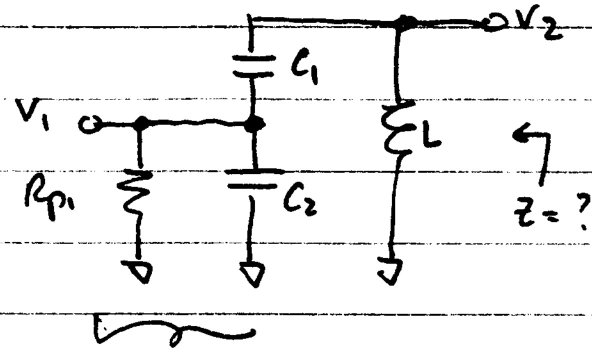
<u>if $Q > 10$</u>:

$$X_S \cong X_P = X$$

$$R_P \cong Q^2 R_S$$

$$R_P R_S = X^2$$

An example: Tapped capacitor network. This can serve as our impedance transform

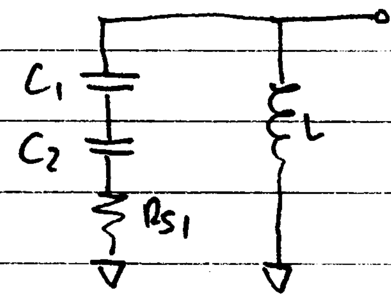


~~This network can be~~ used as an impedance transformer.

resonant circuit with L –

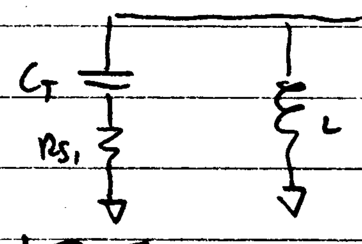
do series eq.

assume $Q_L > 10$



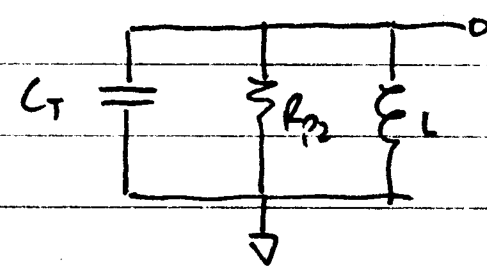
$$R_{S1} = \frac{X_1^2}{R_{p1}} = \frac{1}{R_{p1}}\left(\frac{1}{\omega C_2}\right)^2$$

~~combine C_1 and C_2~~



$$C_T = \frac{C_1 C_2}{C_1 + C_2} \quad \text{(series caps)}$$

series-parallel eq.



$$R_{p2} = \frac{X_2^2}{R_{S1}} = \frac{1}{R_{S1}}\left(\frac{1}{\omega C_T}\right)^2$$

$$= \frac{R_{p1} X_2^2}{X_1^2} = R_{p1}\left(\frac{C_1 + C_2}{C_1}\right)$$

New parallel equivalent resonator

- ~~C_T resonates L.~~

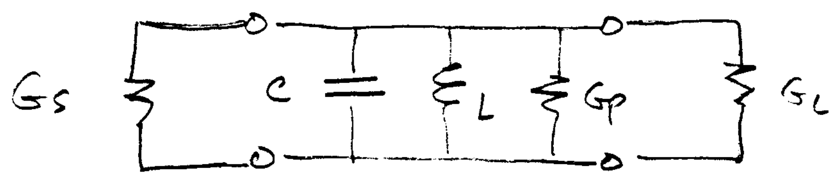
Or, if we define $n = \frac{C_1 + C_2}{C_1}$

we get $\boxed{R_{p2} = R_{p1} n^2}$

1. n is like the turns ratio on a transformer
2. but much easier to build than a transformer

we saw that you could transform impedances using a tapped capacitor network.

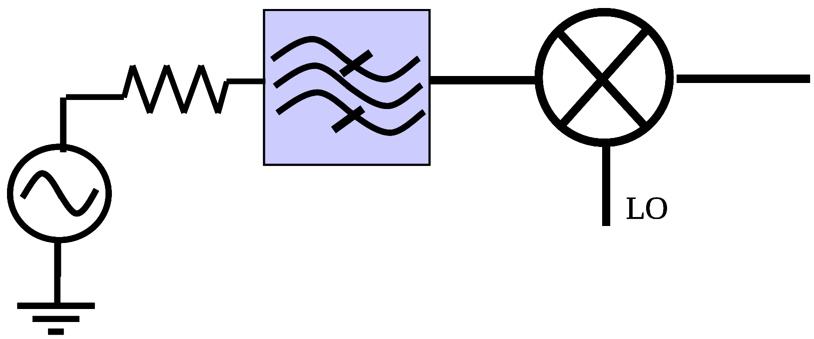
Let's see how we can use this network in conjunct with resonators to build bandpass filters with particular fo and BW.



$$f_0 = \frac{1}{2\pi\sqrt{LC}} \qquad BW = \frac{f_0}{Q_L}$$

$$Q_L = \frac{\omega_0 C}{G_S + G_L + G_P}$$

Example. Design bandpass matching network to transform 50 ohms to the input of an SA602 mixer.

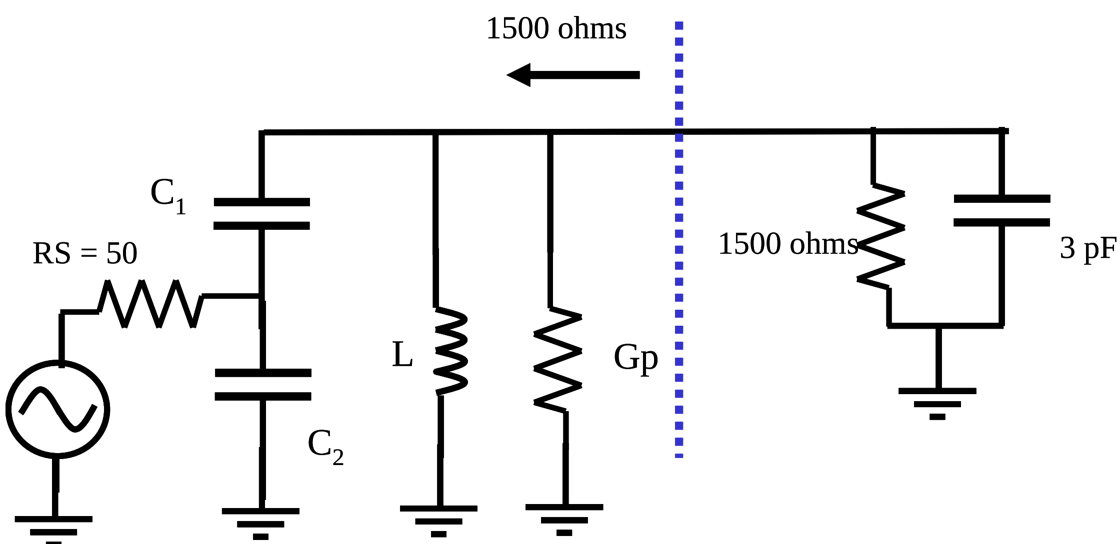


The mixer input impedance is given on the data sheet as

Z_{IN} = 1.5k || 3 pF for the SA602 IC.

Choose a convenient frequency for this exercise: 159 MHz = 1 x 10^9 rad/sec.

Use the tapped C procedure to transform 50 ohms to 1500 ohms. Include the 3 pF into the equivalent series capacitance of C1 and C2. The equivalent parallel resistance, R_P, due to the unloaded Q of the components should also be taken into account in calculating the loaded Q and the transformation ratio needed to match the source and load.



We need to specify a bandwidth to begin the process. Let's choose 10 MHz.

$$Q_L = \frac{159}{10} = 16$$

Q_U is generally limited by the inductor. The manufacturer's data generally specifies an unloaded Q at a certain frequency. For a T-30-12 powdered iron core material, Q_U of 120 is typical at this frequency. The external capacitors normally have a higher unloaded Q which can be neglected.

Our first design equations:

$$Q_U = \frac{1}{\omega_o L G_P} = 120$$

$$Q_L = \frac{B}{G_{total}} = \frac{1/\omega_o L}{G_S + G_L + G_P} = 16$$

Assume that $G_S = G_L = 1/1.5k = 6.7 \times 10^{-4}$. That is, design the tapped C network to provide a transformed impedance of 1.5k. This will get close to the solution needed.

So, now we have two equations with 2 unknowns. We can solve for L and for G_P.

$G_P = 2.04 \times 10^{-4}$S   or $R_P = 4900$ ohms.

$B = Q_U\, G_P = 2.44 \times 10^{-2}$ S

$$L = \frac{1}{\omega_o B} = 41 nH$$

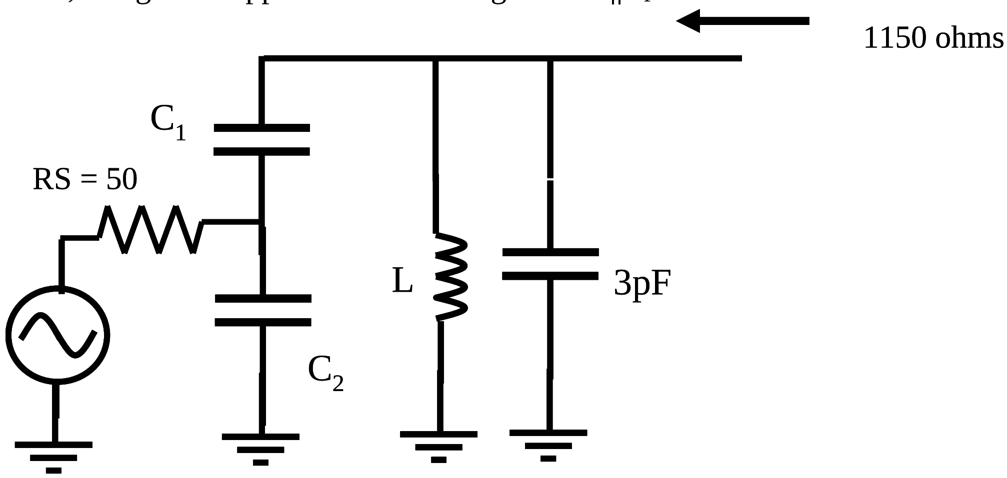
This is a rather small value and will require some care in layout to implement accurately on a PC board.

Check insertion loss:

$$IL = 20 \log\left(1 - \frac{Q_L}{Q_U}\right) = -1.2\, dB$$

If we required less loss, then a wider BW or higher Q_U is necessary.

Next, design the tapped C network to give 1.5k||R_P = 1150 ohms.



Because L is known, we can calculate the required C_{total} to resonate at 159 MHz.

$$C_{total} = \frac{1}{\omega_o^2 L} = 24\,pF$$

Now, deduct the 3 pF from C_{total} so that is is absorbed into the resonator. Design C1 and C2 for 21, not 24 pF.

The transformation ratio relates C_1 and C_2. It should transform 50 ohm source into the load impedance, 1500||RP = 1150 ohms.

$$n = \frac{C_1 + C_2}{C_1} = \sqrt{\frac{1150}{50}} = 4.8$$

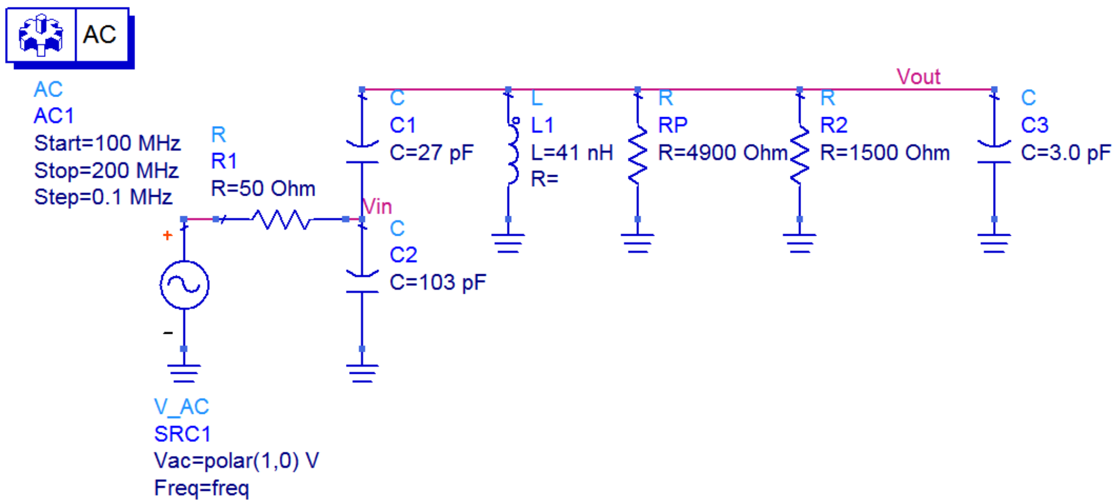
The series combination of C_1 and C_2

$$\frac{C_1 C_2}{C_1 + C_2} = 24 - 3 = 21\,pF$$

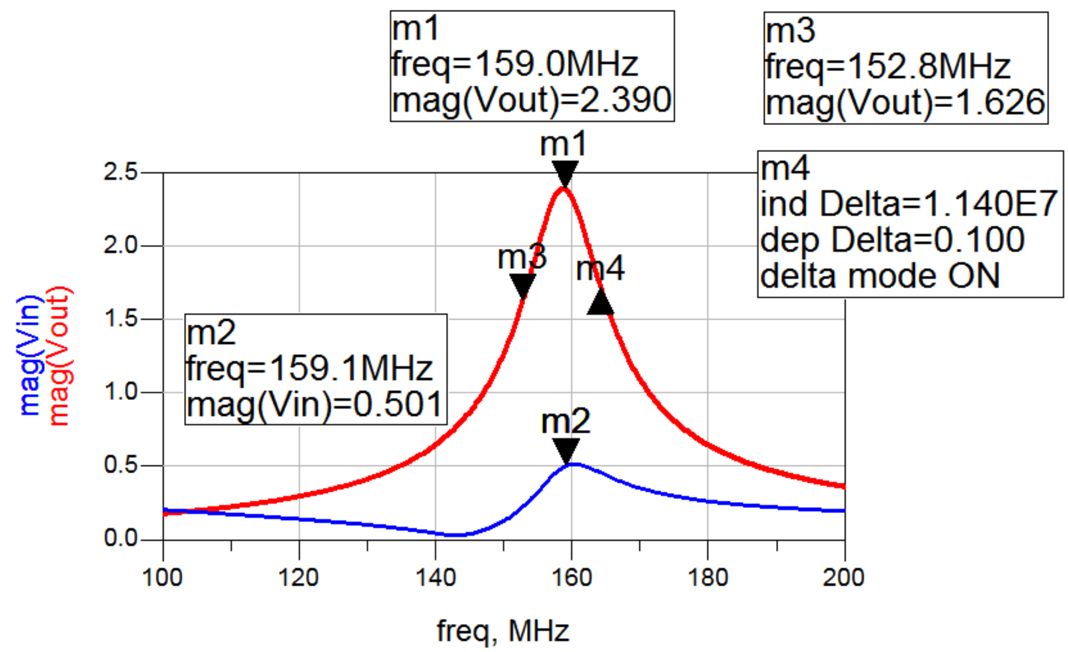
2 equations; 2 unknowns. Solve for C_1 and C_2.

C_1= 27 pF; C_2 = 103 pF.

Check the result with ADS:



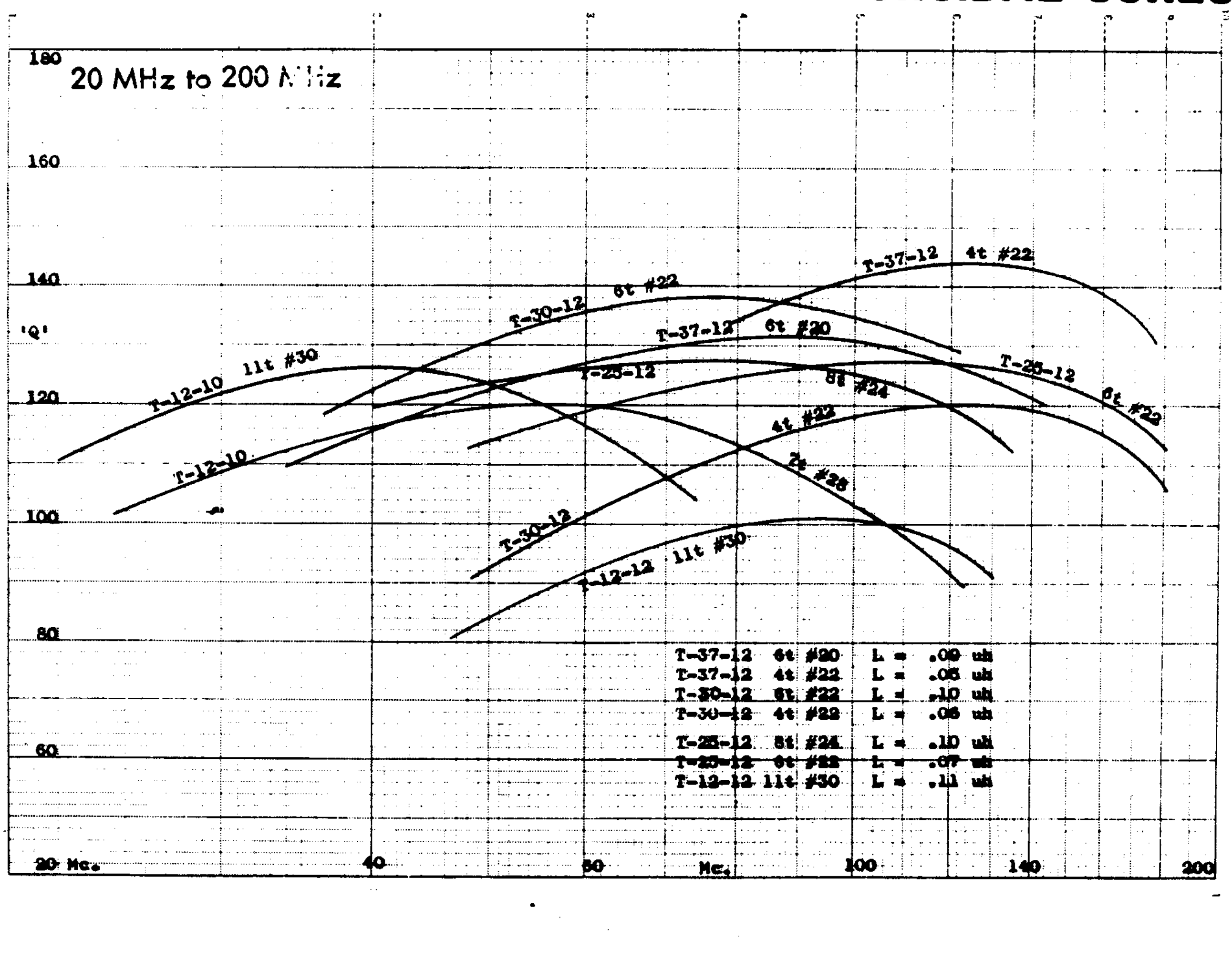
A small signal AC simulation is performed which includes the RP and excess C of the load. If the load is matched correctly to the source, we should see half of the source voltage at node Vin, 0.5V (available power). Vout should be 4.8 x 0.5 = 2.4V.



Everything looks as expected except for the bandwidth, 11.4 MHz rather than 10 MHz. This is due to the effect of R_P. If it mattered, we could go back through again and include this effect and get a 10 MHz BW, but most applications as this one are not so critical. The important thing is to get the correct impedance transformation.

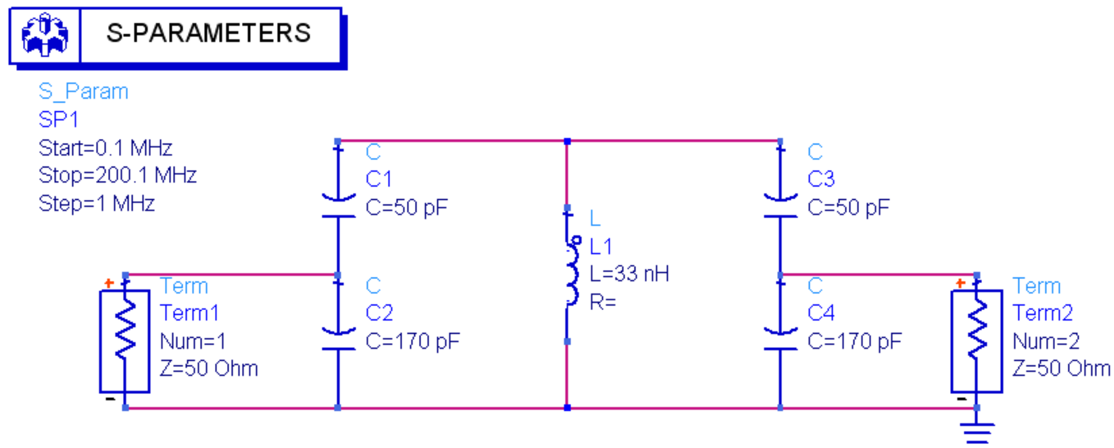
Of course, a L network, PI or T network could also have been used here with somewhat less flexibility in choosing loaded Q.

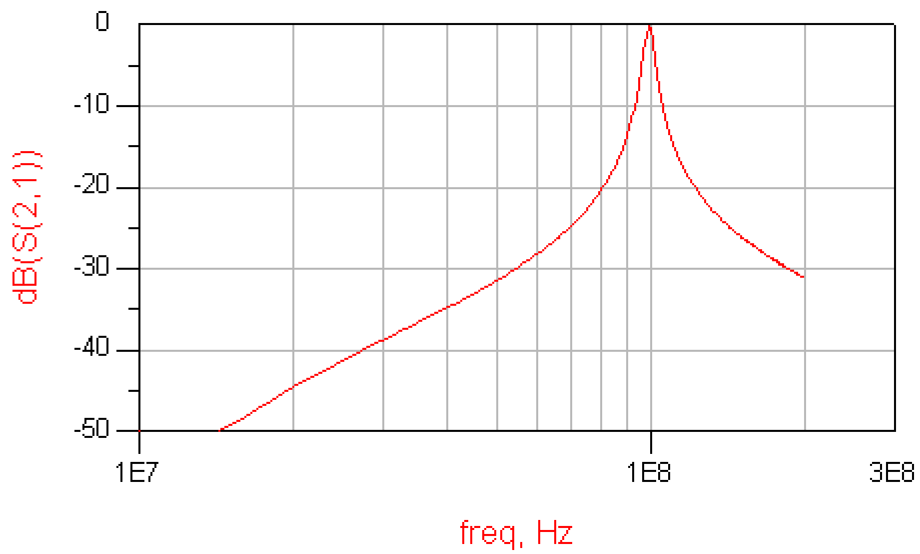
Q—CURVES IRON-POWDER TOROIDAL CORES



Example of resonator used as bandpass filter: 50 ohms to 50 ohms. 100 MHz. The tapped C transforms up to 1000 ohms for this example. You can work through the design – similar to the earlier example.

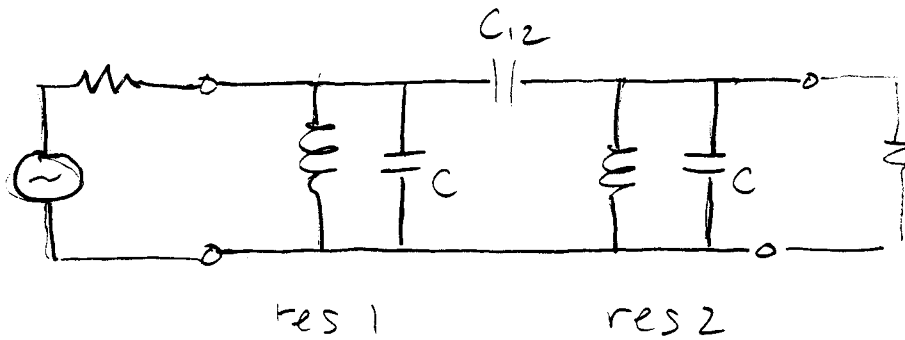
Tapped Capacitor Resonator





Coupled Resonators

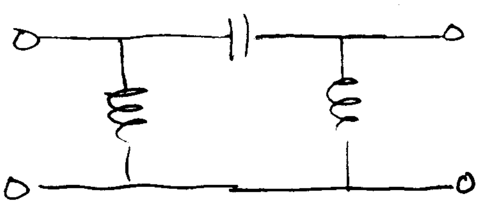
To get better out-of-band attenuation, two resonators can be coupled:



$$C_{12} = \frac{C}{Q}$$ $Q =$ Q_L of single resonator

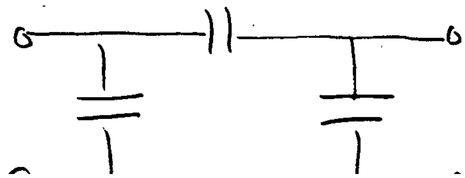
$$Q_L \cong \frac{Q}{\sqrt{2}}$$

Below resonance:



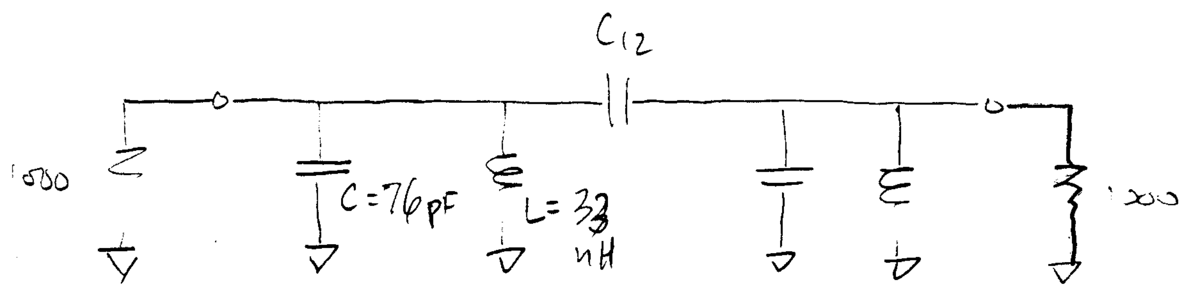
18 dB/octave rolloff

above:



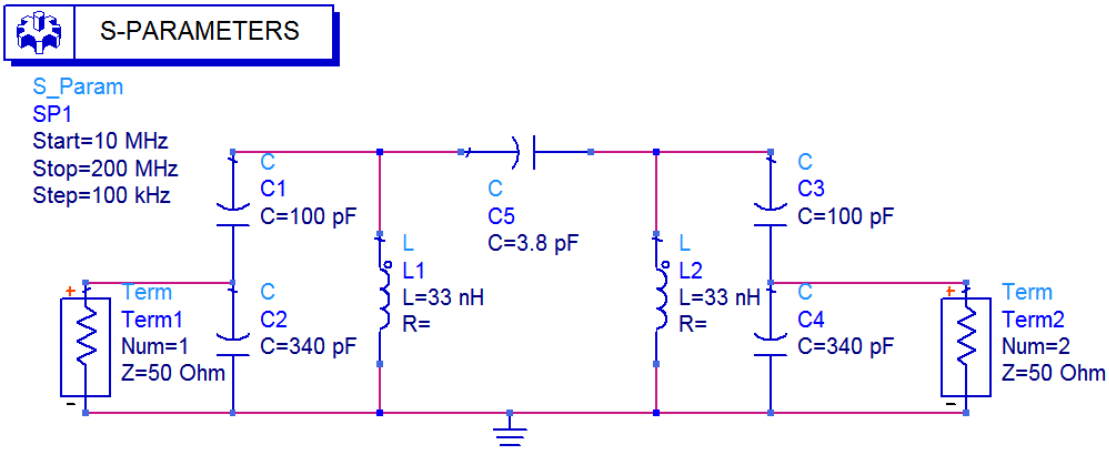
only 6 dB/octave

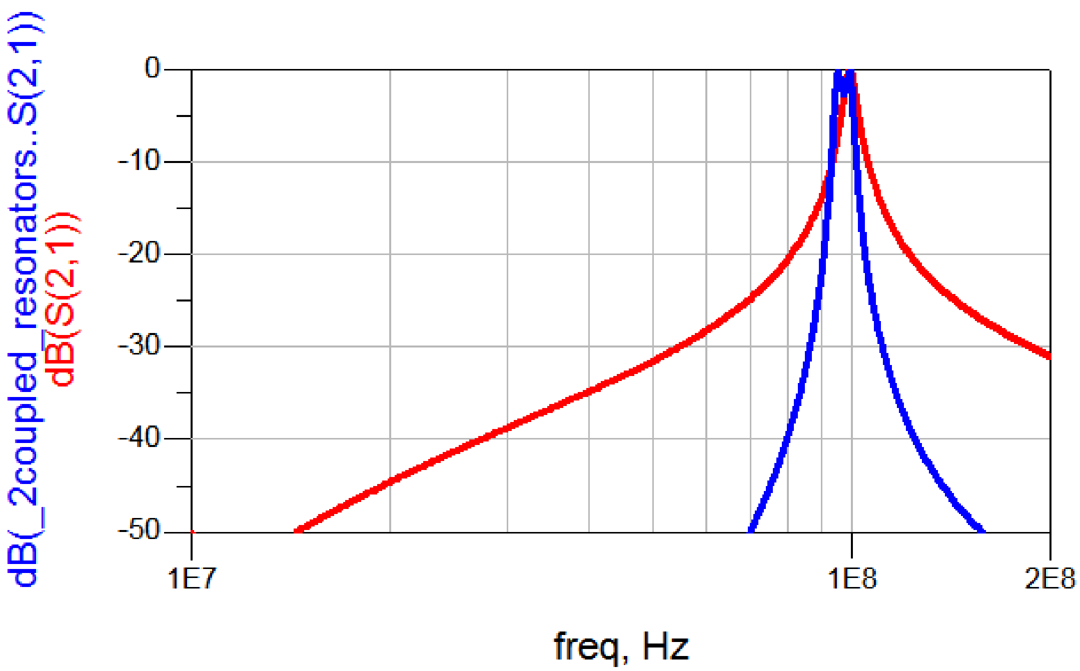
Look at 1st example. use for coupled resonator:



$$C_{12} = \frac{76\text{ pF}}{20} = 3.8\text{ pF}$$

$$Q_L = \frac{20}{\sqrt{2}} = 14$$





Note the improvement in the stopband rejection provided by the coupled resonator compared with the single resonant tapped C circuit.

Temperature Compensation of Resonant Circuits

Oscillators are frequently used to set the transmit or receive frequency in a communication system. While many applications use a phase locked loop technique to correct for frequency drift, it is good practice to build oscillators with some attempt to minimize such drift by selecting appropriate components.

Inductors and capacitors often drift in value with temperature. Permeability of core materials or thermal expansion of wire causes inductance drift. Variations in dielectric constant with temperature in capacitors is the main source of drift for these components.

Temperature drift is expressed as a temperature coefficient in ppm/^{o}C or %/temp range.

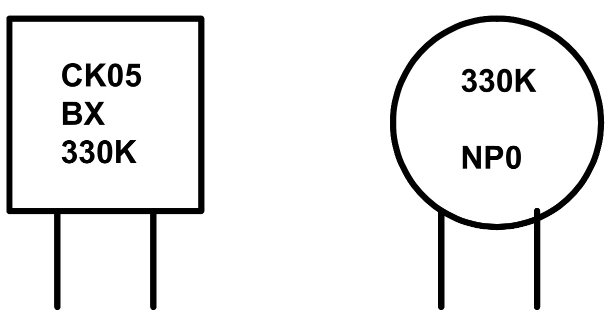
Capacitors

The 3 most common types of dielectrics for RF capacitors are:

Dielectric type	Temp coefficient (TC)	Temp range
C0G (or NP0)	+/- 30 ppm/^{o}C	-55 to +125C
X7R (BX)	- 1667 (+15% to -15%)	-55 to +125
Z5U	- 10^4 (+22% to -56%)	10 to 85

Clearly, the Z5U is not much good for a tuned circuit and should be used for bypass and AC coupling (DC block) applications where the value is not extremely critical. At lower radio frequencies, polystyrene capacitors can be used. These have a – 150 ppm/C TC.

The C0G and X7R can be used in tuned circuits if their values are selected to compensate for the inductor drift.



The two leaded capacitors above illustrate the labels found on typical capacitors of the X7R and NP0 types. The value is given by the numerals: 330. In this example, this is 33 pF. It goes 1st significant digit (3), 2nd significant digit (3), and multiplier (10^0). The letter K is the tolerance, which is +/- 10%.

As always, the parasitic inductance and self resonance of any capacitor must be considered for RF applications.

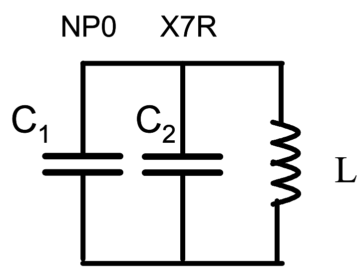
Inductors

There are many types of inductor core materials which are intended for different frequency ranges, permeability, and TC. Powdered Iron and Ferrites are the two categories of these materials.

For example, the material you will have available for the VCO lab is powdered iron, Type 12 (green/white). This is useful from 50 to 200 MHz and gives Qu in the 100 – 150 range. $\mu/\mu_0 = 4$. Manufacturer's data sheets can be found on the web that specify TCs for the many materials. This one has a weird TC vs temperature behavior, but we are mainly interested in the 25 to 50C range for this project.

Temperature range	TC
25 – 50C	+50 ppm/C
50 – 75	- 50
75 – 125	+ 150

So, how can you compensate for component drift?



The equation below shows how the TCs of individual components combine[1]. Suppose that the inductor was resonated with a drift free capacitor (NP0). The frequency drift will be – 25 ppm/C. If the design frequency is 100 MHz, this corresponds to a drift of 2.5 kHz/C. But, the equation shows that you can set the total frequency TC (TCF) of a circuit to zero by combining capacitors with different TCs.

$$TCF = \frac{\Delta f}{f_0} = -\frac{1}{2}\left(TC_L + TC_{C1}\frac{C1}{C_{TOTAL}} + TC_{C2}\frac{C2}{C_{TOTAL}} \right)$$

Thus, if the inductor has a positive TC, you can correct for temperature drift with the right combination of non drift and drifty capacitors. In this case, we want the total capacitance of C1 and C2 to have a net TC of – 50 ppm/C. The best oscillators will be designed with components with low intrinsic TCs so that you do not have to compensate them with different components having large and possibly unreliable TCs.

---

[1] W. Hayward, R. Campbell, and B. Larkin, Experimental Methods and RF Design, ARRL Press, 2003.

如 何 学 习 天 线 设 计

天线设计理论晦涩高深，让许多工程师望而却步，然而实际工程或实际工作中在设计天线时却很少用到这些高深晦涩的理论。实际上，我们只需要懂得最基本的天线和射频基础知识，借助于 HFSS、CST 软件或者测试仪器就可以设计出工作性能良好的各类天线。

易迪拓培训(www.edatop.com)专注于微波射频和天线设计人才的培养，推出了一系列天线设计培训视频课程。我们的视频培训课程，化繁为简，直观易学，可以帮助您快速学习掌握天线设计的真谛，让天线设计不再难…

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助你快速学习掌握如何使用 HFSS 软件进行天线设计，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

CST 天线设计视频培训课程套装

套装包含 5 门视频培训课程，由经验丰富的专家授课，旨在帮助您从零开始，全面系统地学习掌握 CST 微波工作室的功能应用和使用 CST 微波工作室进行天线设计实际过程和具体操作。视频课程，边操作边讲解，直观易学；购买套装同时赠送 3 个月在线答疑，帮您解答学习中遇到的问题，让您学习无忧。

详情浏览：http://www.edatop.com/peixun/cst/127.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

关于易迪拓培训：

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，一直致力和专注于微波、射频、天线设计研发人才的培养；后于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验

※ 一直专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 视频课程、既能达到了现场培训的效果，又能免除您舟车劳顿的辛苦，学习工作两不误

※ 经验丰富的一线资深工程师主讲，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com