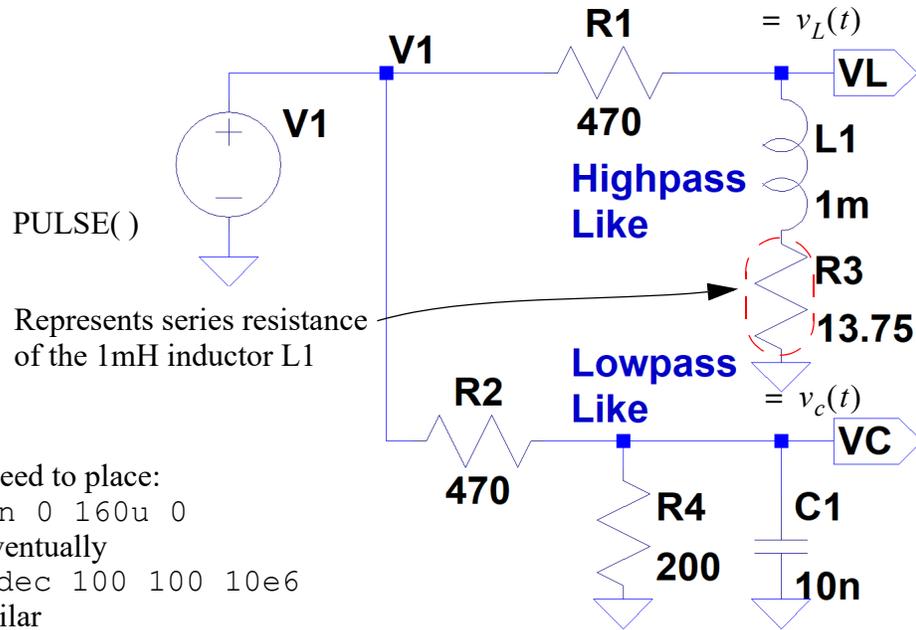


Lab 3: Circuits With R, L, and C

Due Friday October 25, 2019

Problems:

1. **RLC Circuit:** Consider the circuit shown below.



Will need to place:

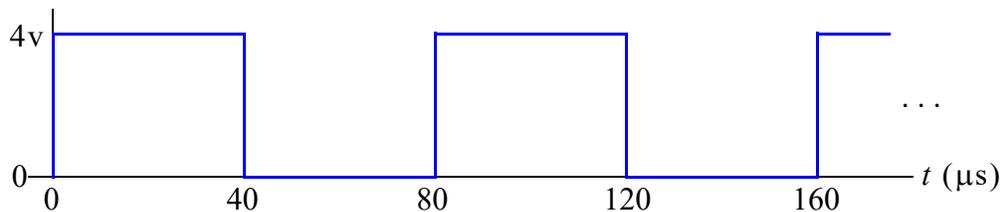
```
.tran 0 160u 0
```

 and eventually

```
.ac dec 100 100 10e6
```

 or similar

Configure the signal source V1 to produce a pulse train signal of the form:

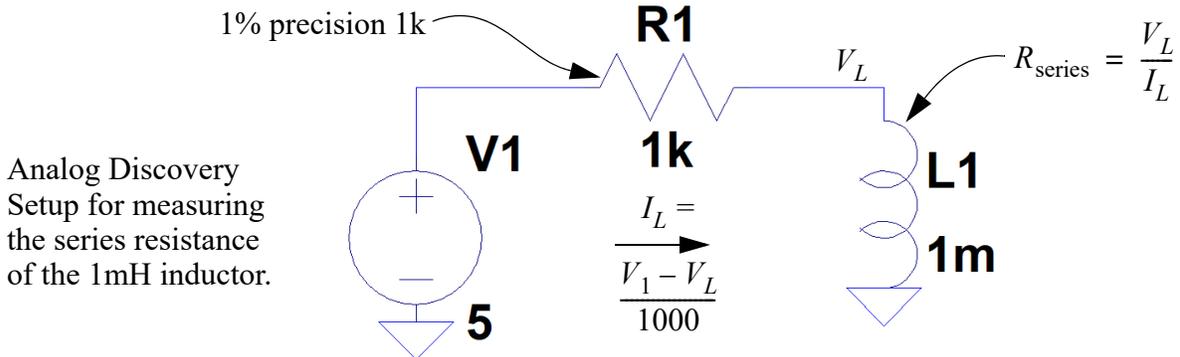


LTspice V1 generator settings for PULSE():

Vinitial[V]:	0
Von[V]:	4
Tdelay[s]:	0
Trise[s]:	100n
Tfall[s]:	100n
Ton[s]:	40u
Tperiod[s]:	80u
Ncycles:	

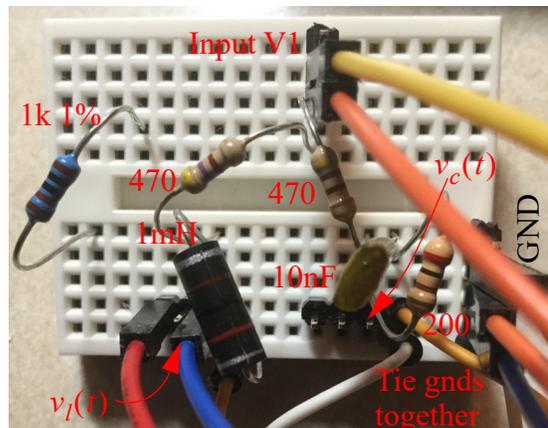
In LTspice set both the rise and fall time, which is the time it takes the waveform to change from 0 to 4 and 4 to 0V respectively, to 100 ns.

- (a) Construct an LTspice model of the circuit and plot the output waveforms $v_c(t)$ and $v_L(t)$. In your report show the LTspice model and a screen shot of the each waveform over at least $160 \mu\text{s}$ (two $80 \mu\text{s}$ periods). Notice that there is a small series resistance associated with the 1 mH inductor. This is due to the resistance (conductor loss) of the inductor windings over the magnetic core inside the package. For your LTspice model to match well with measurements in part (b), you will need to measure the resistance of *your* inductor using the Analog Discovery (AD) and the $1\text{k}\Omega$ 1% resistor in your parts kit. Recall Lab 3



where you measured current and voltage to form the ratio $R = V/I$.

- (b) Construct the circuit on a breadboard. Configure the AD waveform generator to produce the equivalent waveforms. Measure both the input waveform and output waveforms $v_c(t)$ and $v_L(t)$ using the AD scope channels. Comment on the agreement between the measured and simulated waveforms. I suggest using Excel to make an overlay plot as discussed in class. To generate the above waveform will require the use of the *Offset* adjustment on the AD AWG1 as shown below:



$$\text{Period is } \frac{1}{12.5 \times 10^3} = 80 \mu\text{s}$$

This is the peak value of the waveform; the peak-to-peak value is 4v

Combine with a positive offset of $\sim 2\text{v}$ creates the desired waveform

- (c) Obtain the theoretical expectation for both $v_c(t)$ and $v_L(t)$ over the time interval

$0 \leq t \leq 40\mu\text{s}$, using the inspection method described in notes Chapter 6. The assumption is that $+4\text{v}$ is applied to the circuit starting at $t \geq 0$ and there is no charge stored in the capacitor or magnetic flux stored in the inductor. Hence the voltage across the capacitor is initially zero. The capacitor voltage I am seeking is of the form:

$$v_c(t) = v_c(\infty) + [v_c(0) - v_c(\infty)]e^{-t/\tau}, 0 \leq t < 40\mu\text{s}$$

where in the *time constant* $\tau = R_{\text{parallel}}C$, R_{parallel} is the equivalent parallel resistance seen by the capacitor. Compare your theoretical results with the plots of (a) and (b) for just $0 \leq t \leq 40\mu\text{s}$ as an overlay of three plots. For the theory you will have to plot the expression, i.e., make a function plot like in Set #1. For the AD and LTspice you have to import the data into say Excel. I suggest using Excel, but Python and MATLAB both work very well. I want an overlay all three waveforms: theory, LTspice, and the AD in say Excel or Jupyter notebook. Exporting/importing comma separated value (CSV) files is best.

Obtaining the inductor voltage $v_l(t)$ is similar, except here there is initially no current flow through the inductor so there is no voltage drop across R_1 . The general form of the output voltage is:

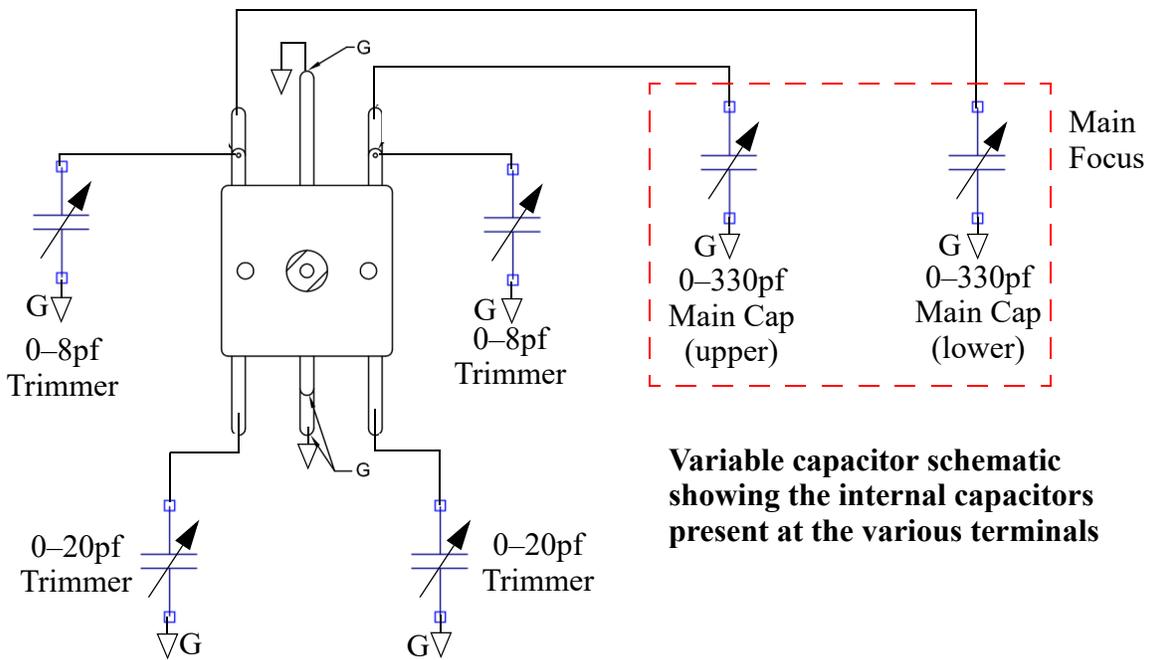
$$v_l(t) = v_l(\infty) + [v_l(0) - v_l(\infty)]e^{-t/\tau}, 0 \leq t < 40\mu\text{s}$$

where in the time constant $\tau = L/R_{\text{series}}$, R_{series} is the equivalent series resistance seen by the inductor.

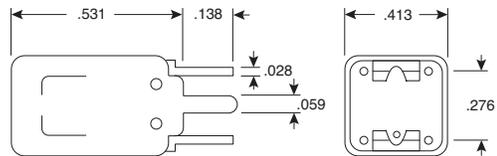
- (d) Obtain the frequency response of the circuit from the input to the outputs V_l and V_c by replacing the pulse waveform with an AC source in LTspice ($1 \angle 0$). The plot you obtain should be the default .ac magnitude in dB and phase in degrees versus log frequency plot. Set your frequency sweep to run from 1 kHz to 10 MHz. Thinking back to the ideal filters of notes Chapter 5, what type of filter is this (lowpass, highpass which is the complement of lowpass, or bandpass)? What is the 3 dB cutoff frequency (bandwidth) relative to the flat part of the filter magnitude response?
- (e) Measure the frequency response using the *network analyzer* capability of the AD for each signal, V1 to V_c and V1 to V_l . A classroom demo will help with the configuration of the AD. Overlay your measurements with LTspice in Excel (see notes p. 6–41 for an example). How well do the two results agree?
- (f) Obtain the theoretical frequency response using *s*-domain impedance modeling as described in notes Chapter 6. A plot of gain in dB is the end result. Use Excel and the complex math capabilities described in the notes and found in the spreadsheet `chapt6_freq_resp.xlsx`. This is challenging for sure, but study the example spreadsheet to see how the complex arithmetic is set up step-by-step.

2. **LC Tank Circuit with variable Capacitor:** In this experiment you will be studying the frequency response of an LC tank (parallel L and C) circuit, which is at the heart of a radio receiver. The LC tank acts as a bandpass filter for received RF signals, signals being processed at the IF frequency, and for building a variable frequency LO in a superheterodyne receiver. The capacitor is a multi-section (*ganged*) variable unit having two sections that are adjustable via the main shaft and four small trimer capacitors. The trimmers can be adjusted independently via a small (preferably plastic) screwdriver. The inductor, 42IF100, is also variable and is a transformer too, since it has primary and secondary windings.

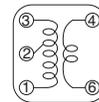
Details on the L and C components used in this part of the lab are given below in the following figure. Note that the variable capacitor is shown with the various elements outside the package. The real values sit inside the package.



42IF100 variable inductor schematic and specifications



- Frequency: 796kHz
- Inductance 1-3: $360\mu\text{H}$ @ 455kHz
- Unloaded Q 1-3: 80 min.
- Turns: 1-2: 104, 2-3: 3, 1-3: 107, 4-6: 8
- Dielectric strength: 100VDC for 1 minute
- Insulating resistance: $100\text{M}\Omega$ @ 100VDC
- RoHS Compliant

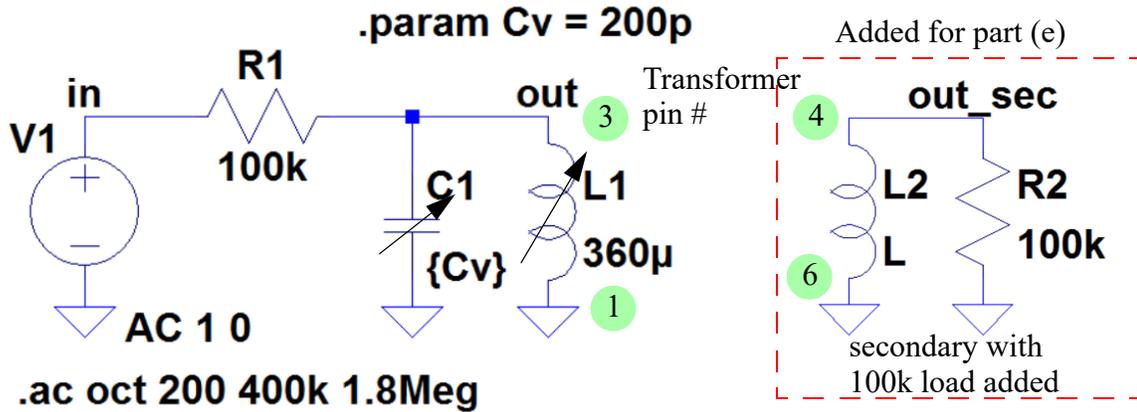


Schematic

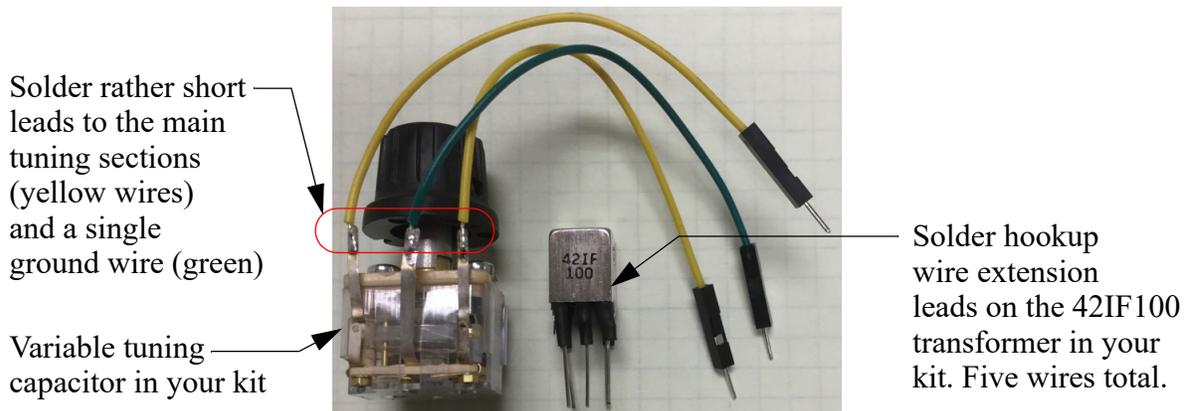


Dimensions (in.)

- (a) Construct the following circuit in LTSpice and plot the frequency response in dB as C_v is stepped from 10 to 330 pf using the `.step` LTSpice directive. Step over 5 values or so.



- (b) Repeat part (a) with R_1 changed to just 10k. Note this will change the loading on the tank circuit and lower the circuit Q, which should widen the bandpass response. Do you agree?
- (c) Now its time to take some real measurements using the AD. Configure the components on a solderless breadboard according to the schematic (return to using 100k for R_1). The first step is to prepare the C and L components as shown below:



To see that things are working OK apply a 1v peak 1 MHz sinusoid from AWG1. Measure the input and output waveforms using the two scope channels. Verify that with an input at 1 MHz there is a value of the variable tuning capacitor that *peaks* the output waveform amplitude. When this occurs the tank circuit is at its resonant frequency, that is

$$f_o = \frac{1}{2\pi\sqrt{LC}}$$

For your report capture a screen shot of the scope viewed waveforms at the peak.

- (d) Continuing from part (c), now configure the AD to make a network analyzer measurement of the frequency response from input to output. This will obtain a measurement of $H(j2\pi f) = V_o(f)/V_i(f)$ for a particular capacitor setting. Proceed to take five network analyzer sweeps of the circuit as C is varied its shaft rotation range of 0%, 25%, 50%, 75%, and 100% of maximum capacitance. Note at 100% capacitance the resonant frequency will be at its lowest value. Do you agree? Overlay all five magnitude frequency

response plots in dB using Excel. Make note of the resonant frequencies observed in each of the five plots. This plot should be similar to the LTspice simulation of part (a). DO NOT attempt overlay the LTspice measurements. Why?

- (e) Repeat part (d) by taking a single sweep with the shaft at 50%, but measuring the output from the secondary of the 42IF100 (terminals 4 and 6). Note how many dB down the frequency response peak is relative to the same measurement of part (e).
- (f) Pull the variable capacitor out of the circuit and sweep as in part (d) making note of the resonant frequency. Assuming the inductor is $360\mu\text{H}$, what shunt capacitance must be present in the breadboard and wiring? **HINT:** You need to use the resonant frequency formula to solve for C given you know L and f_o . Are you surprised to find this much *parasitic capacitance* in your wiring?

Notes: 1 mH Inductor

Specifications	Features	Documents (1)	My Notes
Manufacturer:	Taiyo Yuden		<input checked="" type="checkbox"/>
Product Category:	Fixed Inductors		<input checked="" type="checkbox"/>
RoHS:	 Details		
Inductance:	1 mH		<input type="checkbox"/>
Tolerance:	10 %		<input type="checkbox"/>
Maximum DC Current:	190 mA		<input type="checkbox"/>
Maximum DC Resistance:	13.2 Ohms		<input type="checkbox"/>
Operating Temperature Range:	- 25 C to + 105 C		<input type="checkbox"/>
Self Resonant Frequency:	2.5 MHz		<input type="checkbox"/>
Q Minimum:	80		<input type="checkbox"/>
Termination Style:	Axial		<input type="checkbox"/>
Brand:	Taiyo Yuden		
Diameter:	4.4 mm		
Length:	8 mm		

- Of significance is the series resistance of 13.2 Ohms
- Can measure with an Ohm meter