Introduction to Sinusoidal Steady-State Analysis and Frequency response

Learning outcomes: Student

+ is able to apply the Euler formula
+ knows how the time domain and frequency domain expressions are related to each other
+ know how the frequency response is defined and which information it contains

Sinusoidal Steady-State Analysis

From Wikipedia: The Euler's formula

$$e^{ix} = \cos x + i \sin x,$$

indicates that sinusoids can be represented mathematically as the sum of two complex-valued functions

$$A \cdot \cos(\omega t + \phi) = A \cdot \frac{e^{i(\omega t+\phi)} + e^{-i(\omega t+\phi)}}{2},$$

or as the real part of one of the functions

$$A \cdot \cos(\omega t + \phi) = \mathrm{Re}\{A \cdot e^{i(\omega t+\phi)}\} = \mathrm{Re}\{A \cdot e^{i\phi} e^{i(\omega t)}\}.$$

The quantity $Ae^{i\phi} \in \mathbb{C}(A, \phi \in \mathbb{R})$ is a complex number, the phasor representation of the signal. It can be expressed in the angle notation as

$$A\angle\phi.$$

So, in case of the cos –reference, transformation from time domain to phasors and back reads

$$A \cdot \cos(\omega t + \phi) \Leftrightarrow A\angle\phi.$$

Question: How do you determine ω from a phasor?

Question: Consider $y_1(t) = 20\cos(\omega t - 30°)$ and $y_2(t) = 40\cos(\omega t + 60°)$.

1. Find the phasors Y_1 and Y_2 that represent y_1 and y_2, respectively.
2. Find sum of the two functions, use phasors, i.e. find $Y = Y_1 + Y_2$.
3. Find the time domain signal $y(t) = y_1(t) + y_2(t)$ from Y.

Question: Let $v_{in}(t) = v_1(t) + v_2(t)$ where

$$\begin{aligned} v_1(t) &= 5\cos(\omega_1 t) \text{ and} \\ v_2(t) &= 0.5\cos(\omega_2 t) \end{aligned}$$

with $f_1 = 3$ kHz and $f_2 = 10$ kHz.

1. Find the phasors V_1 and V_2 that represent v_1 and v_2, respectively.
2. Can you add the two phasors V_1 and V_2?

   If **yes**, explain how to interpret the result. If **no**, what can you do?

Review of Transfer function Frequency response

Learning outcomes: Student

+ knows how the transfer function and frequency response are defined,

+ is able to determine the transfer function of a circuit,

+ is able to intepret the amplitude plot and the phase angle plot.

Transfer function & Frequency response

Transfer function $H(\omega)$ of a system is defined in the frequency domain. Let

$X(\omega)$ denote the input in the frequency domain, see Fig. 1,

$Y(\omega)$ denote the output in the frequency domain, see Fig. 1,

then

$$H(\omega) = \frac{Y(\omega)}{X(\omega)}.$$

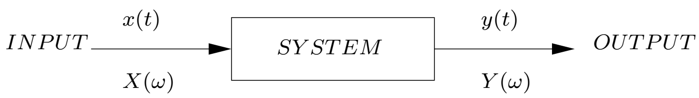


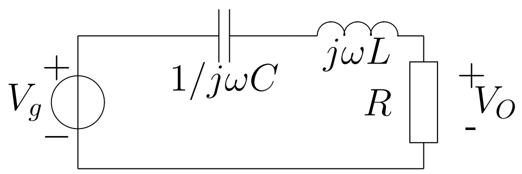
Figure 1: A general system with input and output in time and frequency domain.

Transfer function is commonly utilized, e.g, in

designing electronic filters, amplifiers,

signal processing, communication theory, ...

Example: Find transfer function, if $V_g(\omega)$ is the input and voltage $V_O(\omega)$ across the resistance is the output



Total *impedance* $Z(\omega) = R + j\omega L + \frac{1}{j\omega C}$, hence

$$V_g(\omega) = I(\omega) * Z(\omega) \Leftrightarrow I(\omega) = \frac{V_g}{Z} = \frac{V_g}{R + j\omega L + \frac{1}{j\omega C}}$$

and

$$V_O(\omega) = R * I = R * \frac{V_g}{R + j\omega L + \frac{1}{j\omega C}} \Leftrightarrow H(\omega) = \frac{V_O}{V_g} = \frac{\left(\frac{R}{L}\right) j\omega}{\left(\frac{R}{L}\right) j\omega + (j\omega)^2 + \frac{1}{LC}}$$

Once the transfer function of a system is known, also *frequency response* plots of the system are readily obtained

magnitude plot: $|H(\omega)|$

phase angle plot: angle$(H(\omega))$

Example continued: Let us return to the previous example, where the transfer function reads

$$H(\omega) = \frac{\left(\frac{R}{L}\right) * j\omega}{\left(\frac{R}{L}\right) * j\omega + (j\omega)^2 + \frac{1}{LC}},$$

and let us choose $R = 20\ \Omega$, $C = 1\ \mu$F, $L = 50$ mH. The frequency response with these values are shown in Figures 2 and 3.

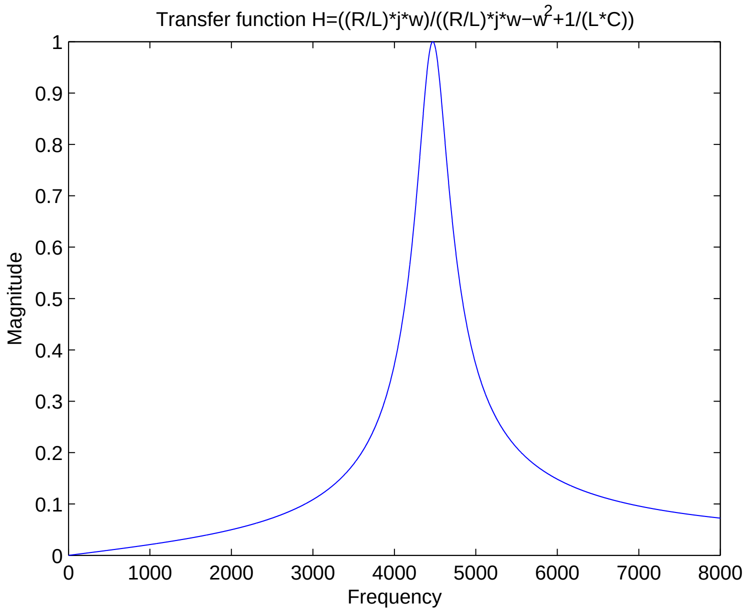


Figure 2: Magnitude plot of the transfer function.

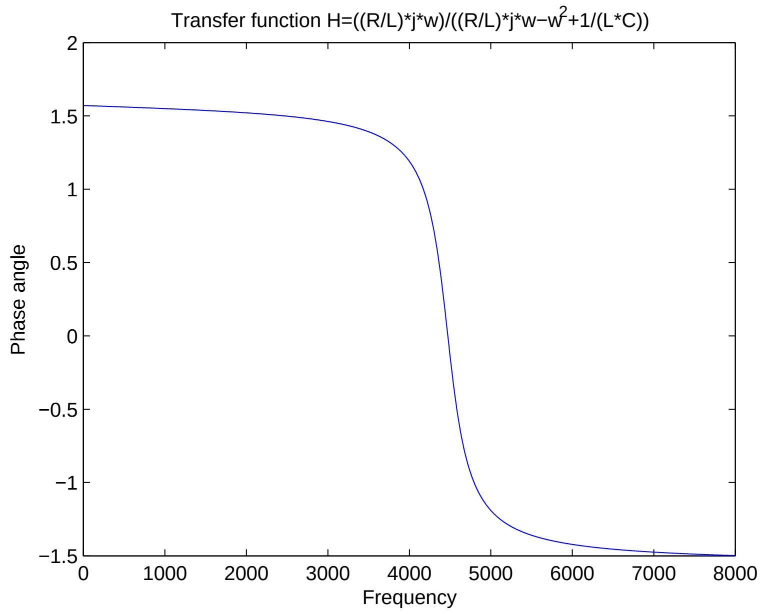


Figure 3: Phase angle plot of the transfer function.

The above is an example of a *bandpass filter.* To characterize response of such a system, we can use:

Cutoff Frequency: frequency/frequencies where magnitude of a transfer function is decreased by factor $\frac{1}{\sqrt{2}}$ from its maximum value, i.e,

$$|H(\omega_C)| = \frac{1}{\sqrt{2}} H_{max}.$$

We obtain two frequencies ω_{c1}, ω_{c2} (where $\omega_{c1} < \omega_{c2}$) and find values for three parameters:

center frequency $\omega_0 = \sqrt{\omega_{c1} * \omega_{c2}}$,

bandwidth $BW = \omega_{c2} - \omega_{c1}$

quality factor $Q \approx \frac{\omega_0}{BW}$

In our example $\omega_1 = 4276.6$, $\omega_2 = 4676.6$,:

$$\omega_0 = 4472,\ BW = 400,\ Q \approx 11.183.$$

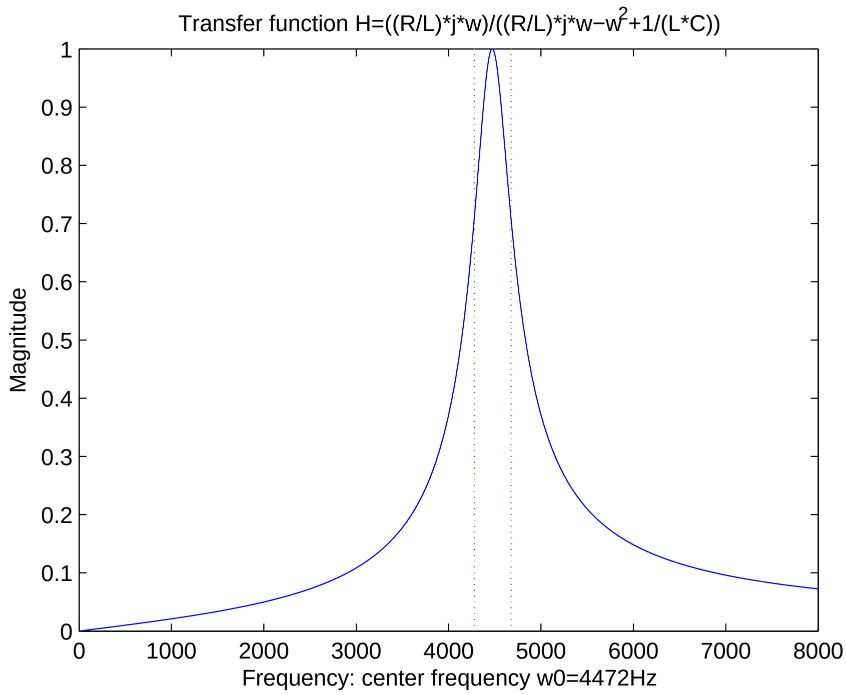


Figure 4: Magnitude plot of the transfer function. Half power cutoff frequencies marked on the plot.

Use of the Transfer function: Some theory

It can be shown (see, for example, Nilsson & Riedel "*Electric Circuits*" 7th edition, chapter 13.7) that if a steady-state sinusoidal input

$$x(t) = A\cos(\omega t + \phi) \qquad \Leftrightarrow \qquad A\angle\phi$$

is applied to a circuit whose transfer function

$$H(\omega) = |H(\omega)|e^{j\theta(\omega)} = |H|e^{j\theta} \in \mathbb{C}$$

is known, the output is

$$y_{ss}(t) = A|H|\cos(\omega t + \phi + \theta). \tag{1}$$

<u>Question:</u>

1. How to interpret equation (1)?
2. Express $y_{ss}(t)$ as a phasor.
3. Could you rewrite the phasor such that it includes a term related to the transfer function and a term related to the input?

Notes on some more involved theory

If we analyze in mathematical rigour, we should apply the Laplace transform.

From Wikipedia: The Laplace transform of a function $f(t)$, defined for all real numbers $t \geq 0$, is the function $F(s)$, which is defined by

$$F(s) = \int_0^\infty f(t)e^{-st}\,dt,$$

where s is a complex number

$$s = \sigma + i\omega \text{ with } \sigma, \omega \in \mathbb{R}.$$

Remarks

Equation (1) is derived using the Laplace transform.

In case of sinusoidal steady–state s is replaced by $j\omega$.

For example, after the Laplace transform “impedance of an inductor” is expressed as sL and in the sinusoidal steady–state case as $j\omega L$.

TASKS

1. (a) Design a first-order passive low-pass filter, cutoff-frequency 340 Hz. Use 470 nF capacitor and choose suitable resistor from the list attached (Table 1).
   (b) Determine transfer function of the circuit by paper and pen. From it check
      i. magnitude and phase at the cutoff-frequency.
      ii. magnitude and phase at low and at high frequencies.

2. Lets use Multisim and myDAQ to work with the filter, in particular, use so called *Bode Analyzer*. We are going to (a) simulate and (b) measure the circuit. You use both Multisim and myDAQ such that their results can be easily compared.

   For that you use *NI myDAQ design* in Multisim, you find guidelines in a video:

   `https://www.youtube.com/watch?v=ZpmAEj_biJ0`

   (a) Simulate the circuit in Multisim.
   (b) Construct and measure the circuit using breadboard and myDAQ.

   Checklist:

   AO-0 is used as the input source.

   Analogue ground (AGND) is attached to the ground reference point of your circuit i.e. the bottom leg of the Capacitor.

   AI-0+ is used to measure the input voltage

   AI-1+ is used to measure the output voltage across the Capacitor

   All voltages are measured across two points, in this case with respect to ground -> connect AI-0- and AI-1- to analogue ground (AGND)

3. Let transfer function of a circuit be $H(\omega) = \frac{1}{1+j\omega*4*10^{-5}}$, see Fig. 5.
   Define output of the circuit, if $v_{in}(t) = 5\cos(\omega_1 t) + 0.5\cos(\omega_2 t)$ where $f_1 = 3$ kHz and $f_2 = 10$ kHz.

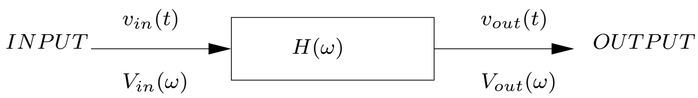


Figure 5: A circuit whose transfer function $H(\omega)$ is known.

4. **OPTIONAL Extra task**, if time allows
   (a) First, paper-and-pen:
      i. Design a first-order passive high-pass filter (HP), cutoff-frequency 300 Hz. Use a 100 nF capacitor and select resistor from the list attached (Table 1).

ii. Design a first order passive low-pass filter (LP), cutoff-frequency 3000 Hz. Use a 470 pF capacitor and select resistor from the list attached (Table 1).

(b) Second, simulations (see the guidelines below about simulations):

i. Simulate the high-pass filter using Multisim.

ii. Simulate the low-pass filter using Multisim.

(c) Then connect the two filters in cascade in Multisim to have a band-pass filter.

i. Connect first HP->LP.

ii. Connect then LP->HP.

Do you notice any differences between the outputs in these two cases? If so, could you think of a reason for that.

HINTS: It is likely easier to use the AC Analysis in Multisim:

Open a new "Blank" design, place the components by selecting Place -> Component.

Place the voltage source by selecting

- Group: Sources,
- Family: POWER_SOURCES, Component: AC_POWER (default values will be ok for now, since we aim to sweep over a range of frequencies).

Label parts appropriately by double–clicking a wire to refer to them when setting up a simulation (see the Fig. 6 where labels IN1 and OUT1 have been added). You may need to tick 'Show net name' to get the label visible.

Once the circuit is ready, select Simulate -> Analysis -> AC Analysis. For settings use:

- Set "Frequency parameters" appropriately (e.g. 10 Hz - 100 kHz)
- Under Output select Add Expression and set expression where you use labels to define ratio between input and output voltages e.g. "V(OUT1)/V(IN1)" -> you get frequency response of the filter(s).

Item	Type	Needed
Capacitor	100nF	
	47nF	
	470pF	
	220pF	
Resistor	360k	
	330k	
	240k	
	180k	
	110k	
	100k	
	91k	
	82k	
	75k	
	68k	
	56k	
	16k	
	11k	
	8.2k	
	6.8k	
	5.6k	
	4.7k	
	3.6k	
	2.7k	
	1.0k	
	820	
	680	

Table 1: Filter part list

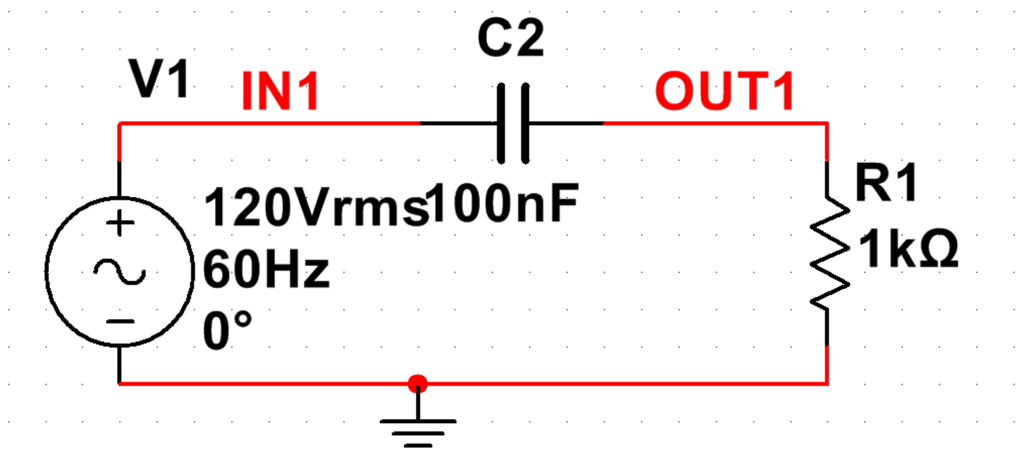


Figure 6: Example of labeling parts of a circuit.