Sound Analysis Synthesis and Processing: SSSP Homework

May 22, 2025

The topic of this homework is Wave Digital Filter (WDF) modeling. Your task is to model the electro-mechano-acoustic behavior of a piezoelectric MEMS loudspeaker in the Wave Digital (WD) domain, starting from a linear lumped-element model (LEM) description.

Piezoelectric MEMS loudspeakers are attracting growing interest in the audio industry thanks to their compact form factors, made possible by the semiconductor manufacturing process. These miniaturized devices offer promising opportunities for integrating audio functionality into consumer electronics, paving the way for novel applications in embedded audio systems [1]. An example of such a device is shown in Fig. 1.

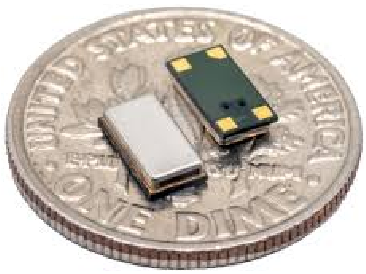

Figure 1: Piezoelectrically actuated MEMS loudspeakers.

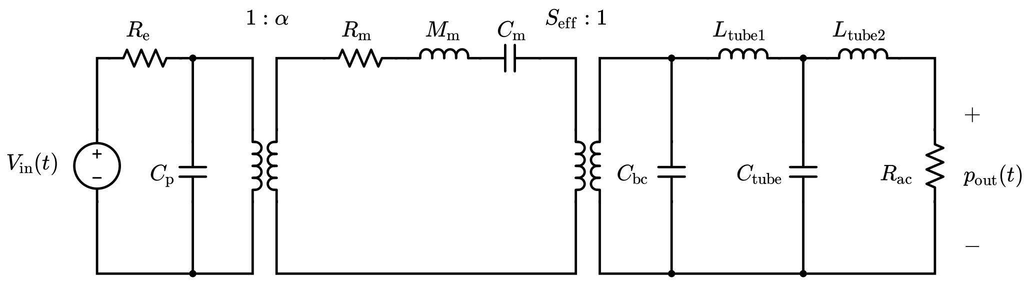

Figure 2: Linear lumped-element model of the target piezo-actuated MEMS loudspeaker for free-field applications.

The linear equivalent circuit model in Fig. 2 describes the multiphysics behavior of a possible device of the sort according to the impedance analogy, where mechanical velocity and acoustical volume velocity are represented by electrical currents, while mechanical force and acoustical pressure are represented by electrical voltages. In the electrical domain, it includes a voltage source V_{in}, an electrical resistance R_{e}, and a capacitor C_{p} representing the piezoelectric layer's static capacitance. Piezoelectric transduction is modeled by an ideal transformer with turns ratio $1 : \alpha$, where α is the electro-mechanical transduction coefficient.

In the mechanical domain, the vibrating diaphragm is modeled as a single-degree-of-freedom oscillator, with R_m, M_m, and C_m representing damping, mass and compliance of the moving components, respectively. It couples to the acoustic domain via another ideal transformer with a ratio determined by the effective area of the moving diaphragm S_eff.

In the acoustic domain, C_bc represents the acoustic compliance of the speaker enclosure's 100 mm^3 back chamber. The cylindrical short-tube filled with air in front of the loudspeaker, due to the loudspeaker packaging, is modeled using by inductors L_tube1, L_tube2, and capacitor, C_tube. Finally, the acoustic radiation impedance is represented by resistor R_ac.

The loudspeaker is driven by an input voltage signal $V_\mathrm{in}(t)$ and produces a corresponding radiated acoustic output pressure $p_\mathrm{out}(t)$, which can be interpreted in the electrical analogy as the voltage across resistor R_ac.

Parameter	**Value**	**Unit**
R_e	4	Ω
C_p	2.4×10^{-8}	F
α	3.7×10^{-4}	$\mathrm{N} \cdot \mathrm{V}^{-1}$
R_m	9.7×10^{-3}	$\mathrm{N} \cdot \mathrm{s} \cdot \mathrm{m}^{-1}$
M_m	1.0×10^{-6}	kg
C_m	2.2×10^{-3}	$\mathrm{m} \cdot \mathrm{N}^{-1}$
S_eff	2.0×10^{-5}	m^2
C_bc	3.6×10^{-13}	$\mathrm{Pa} \cdot \mathrm{m}^{-3}$
L_tube1	1.0×10^{2}	$\mathrm{Pa} \cdot \mathrm{s}^2 \cdot \mathrm{m}^{-3}$
L_tube2	1.0×10^{2}	$\mathrm{Pa} \cdot \mathrm{s}^2 \cdot \mathrm{m}^{-3}$
C_tube	6.5×10^{-13}	$\mathrm{Pa} \cdot \mathrm{m}^{-3}$
R_ac	5.0×10^{6}	$\mathrm{Pa} \cdot \mathrm{s} \cdot \mathrm{m}^{-3}$

Table 1: Values of the electro-mechano-acoustic circuital parameters in Fig. 2.

1 Reference Circuit

The equivalent circuit model shown in Fig. 2 includes two ideal transformers, which are linear multi-port elements commonly used to model domain coupling. While such elements can be implemented in the WD domain framework – for instance, by embedding them within junctions as described in [2] – an alternative and often more convenient approach is to eliminate the transformers altogether.

To this end, we reformulate the circuit so that all three subcircuits (electrical, mechanical, and acoustic) are mapped into the mechanical domain. This transformation avoids the explicit presence of transformers, simplifying the circuit analysis. The resulting electrical circuit, which we adopt as reference for designing the WD structure, is shown in Fig. 3. To account for the transformers' constitutive equations, the input ideal voltage source is redefined as $F_\mathrm{in}(t) = \alpha V_\mathrm{in}(t)$, and all component values are transformed accordingly, as summarized in Table 1.

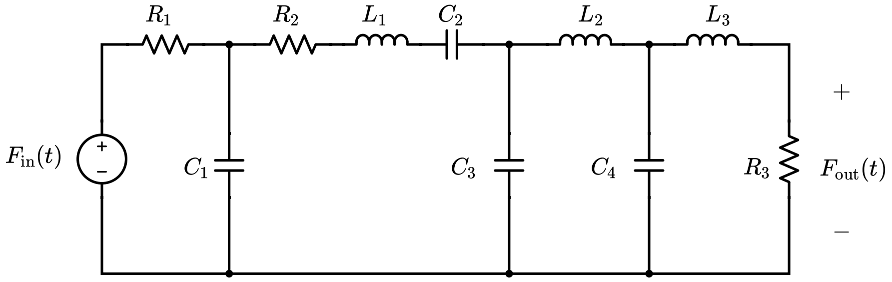

Figure 3: Reference circuit: MEMS loudspeaker equivalent circuit model into the mechanical domain.

Resistances	Inductances	Capacitances
$R_1 = R_\mathrm{e} \cdot \alpha^2$	$L_1 = M_\mathrm{m}$	$C_1 = C_\mathrm{p}/\alpha^2$
$R_2 = R_\mathrm{m}$	$L_2 = L_\mathrm{tube1} \cdot S_\mathrm{eff}^2$	$C_2 = C_\mathrm{m}$
$R_3 = R_\mathrm{ac} \cdot S_\mathrm{eff}^2$	$L_3 = L_\mathrm{tube2} \cdot S_\mathrm{eff}^2$	$C_3 = C_\mathrm{bc}/S_\mathrm{eff}^2$
		$C_4 = C_\mathrm{tube}/S_\mathrm{eff}^2$

Table 2: Definitions of the circuit parameters values for the reference circuit in Fig. 3.

1.1 Simscape Implementation of the Reference Circuit (optional)

You are not expected or required to download Simscape and to install it in order to complete this homework and achieve the maximum grade. The output signal of the reference circuit will be provided to you such that you can use it as a ground truth when you implement the required WDF. However, Simscape could help you to analyze the behavior of the reference circuit and search for a smart solution in designing the corresponding WDF structure. A file containing the Simscape implementation of the reference circuit (`mems_spk_SSC.slx`) is provided, along with its accompanying MATLAB script (`ssc_script.m`) that allows to define the simulation input signal. You can download Simscape for free, using your student license, at the following link: `https://it.mathworks.com/products/simscape.html`.

1.2 Input Signal

The model of the ideal voltage source element is not characterized by any internal series resistance, hence its continuous-time constitutive equation is given by

$$v(t) = F_\mathrm{in}(t) \ , \tag{1}$$

where $v(t)$ is the port voltage of the ideal voltage source element and $F_\mathrm{in}(t) = \alpha V_\mathrm{in}(t)$ is the input signal applied to the circuit.

The input signal $V_\mathrm{in}(t)$ is defined as an exponential swept sine (log chirp) with amplitude A_in, starting from frequency f_1, ending at frequency f_2, and lasting T_end seconds. Its mathematical expression is [3]

$$V_\mathrm{in}(t) = A_\mathrm{in} \sin\left(\frac{2\pi f_1 T_\mathrm{end}}{\ln(f_2/f_1)}\left(\exp\left(\frac{t}{T_\mathrm{end}}\ln(f_2/f_1)\right) - 1\right)\right) \ . \tag{2}$$

This signal is synthesized using the built-in MATLAB function `sweeptone`. In particular, the following command is used to generate the desired waveform

```
Vin = A * sweeptone(1.99, 0.01, fs, 'SweepFrequencyRange', [500 20000]);     (3)
```

This generates an exponential sweep signal `Vin` with an amplitude of $A_\mathrm{in} = 10$ V, starting at $f_1 = 500$ Hz and ending at $f_2 = 20$ kHz, with a duration of $T_\mathrm{end} = 1.99$ seconds, followed by 0.01 seconds of silence. The variable `fs` denotes the sampling frequency, which is set for this application to 192 kHz.

1.3 Output Signal

The output signal $F_\mathrm{out}(t)$ of the reference circuit is the voltage measured at resistor R_3. This signal can be converted into the corresponding acoustic pressure via the relation $p_\mathrm{out}(t) = F_\mathrm{out}(t)/S_\mathrm{eff}$.

The ground truth output acoustic pressure signal obtained by the Simscape simulation is provided and can be imported in MATLAB using the command (already implemented in the script)

```
load('output_ssc.mat');
```

In this way, you can compare the output signal of the implemented WDF with a ground-truth. The comparison is going to be performed both in the time-domain and in the frequency domain, as described later.

2 Port-wise Definition of Wave Variables

The port-wise definition of wave variables considered in this homework is the same definition of voltage waves we have already seen in class. Therefore, Kirchhoff variables in the discrete-time domain at one port of an element are expressed in terms of wave variables as

$$v[k] = \frac{a[k] + b[k]}{2} \quad , \quad i[k] = \frac{a[k] - b[k]}{2Z[k]} \; , \tag{4}$$

where $v[k]$ is the port voltage, $i[k]$ is the port current, $a[k]$ is the wave incident to the element, $b[k]$ is the wave reflected by the element and $Z[k]$ is a scalar free parameter different from zero.

3 WD Model of the Ideal Voltage Source

The WD scattering relation of an ideal voltage source is obtained by substituting the definition of port voltage (4) in terms of wave variables into the constitutive equation (1) and then solving for $b[k]$. Hence the discrete-time WD model of the ideal voltage source is the following

$$b[k] = 2F_{\text{in}}[k] - a[k] \; . \tag{5}$$

It is important to notice that an ideal voltage source with scattering relation (5) cannot be adapted; this means that there is no value of the free parameter $Z[k]$ that allows us to eliminate the instantaneous dependence between $b[k]$ and $a[k]$.

4 WDF Design

There are many possible ways of representing the reference circuit of Fig. 3 as a WDF structure, all leading to equally accurate results.

You are asked to choose one valid WDF representation of the reference circuit and draw the corresponding WDF scheme. For your convenience, you are encouraged to choose the WDF representation that, according to you, minimizes computational complexity. However, the only constraint on the required WD structure is that it must be computable in a fully explicit fashion; this means that no iterative solvers can be used to implement it in the WD domain.

You have to propose a WDF based on one connection tree characterized by:

- a root (hint - the root should be an element that cannot be adapted);
- one or more nodes (WD topological junctions called adaptors);
- many leaves (linear one-port elements that can be adapted).

As far as nodes of connection trees are concerned, you can use

- N-port series adaptors with $N \geq 3$.
  Remember that one port of a series adaptor, e.g. port 1, can be made reflection-free by setting

$$Z_1 = \sum_{n=2}^{N} Z_n$$

- N-port parallel adaptors with $N \geq 3$.
  Remember that one port of a parallel adaptor, e.g. port 1, can be made reflection-free by setting

$$Z_1 = \frac{1}{\sum_{n=2}^{N} Z_n^{-1}}$$

- N-port arbitrary topological junctions (adaptors) with $N \geq 3$.
  In this general case, the adaptation condition for making, e.g., port 1 reflection-free can be found

using the symbolic environment in MATLAB. As an example, here follows a MATLAB-like pseudocode for finding the value of Z_1 that makes a generic 5-port topological junction reflection free at port 1

```
syms Z1 Z2 Z3 Z4 Z5 real

Z=diag([Z1,Z2,Z3,Z4,Z5]);

...

S = eye(5) - 2*Z*B'*inv(B*Z*B')*B;

Z1 = solve(S(1,1)==0,Z1)
```

where `S` is the scattering matrix of the topological junction and `B` is the corresponding fundamental-loop matrix that maps the subset of independent port currents to all port currents. Matrix `B` depends on the specific topological junction you are considering, as we have seen in class. Additional details on how to compute the fundamental loop matrix **B** or the fundamental cut-set matrix **Q** of a given directed graph corresponding to a connection network can be found at the following link: `https://www.tutorialspoint.com/network_theory/network_theory_topology_matrices.htm`.

In summary, you are asked to:

1. Draw the proposed WDF scheme including T-shaped stubs at all ports of WD elements and WD junctions that are adapted (reflection-free ports). Please provide a figure created using a drawing software of your choice or a photo of a clear and legible drawing you made on paper. (Note that if we cannot understand something in your drawing, it will be counted as an error).

2. Assign numbers or names to all ports of topological junctions (adaptors).

3. Write down in symbolic form how to set each free parameter of the WDF.
   As an example, if an adapted resistor with resistance R_1 is connected to a port whose free parameter is called Z_4, you will write $Z_4 = R_1$.
   As another example, if a 3-port series adaptor is characterized by free parameters Z_1, Z_2, Z_3 and port 1 is made reflection-free, you will write $Z_1 = Z_2 + Z_3$.

5 WDF Implementation

You are asked to implement the WDF you designed according to the instructions presented in the previous section. You will do this by completing the MATLAB script `mems_spk_WD.m` that we are providing.

The WD models that you can use for implementing the one-port elements of the reference circuit (apart from the already discussed ideal voltage source) are summarized in Table 5 for your convenience.

Constitutive Eq.	**Wave Mapping**	**Adaptation Condition**
$v(t) = V_g(t) + R_g i(t)$	$b[k] = V_g[k]$	$Z[k] = R_g$
$v(t) = Ri(t)$	$b[k] = 0$	$Z[k] = R$
$i(t) = C\frac{dv(t)}{dt}$	$b[k] = a[k-1]$	$Z[k] = \frac{T_s}{2C}$
$v(t) = L\frac{di(t)}{dt}$	$b[k] = -a[k-1]$	$Z[k] = \frac{2L}{T_s}$

Table 3: Wave mappings of common WD linear one-port elements.

To verify the correctness of your implementation, you are invited to compare the output signal of the WDF structure (i.e., the voltage across R_3 converted into acoustic pressure) with the ground-truth pressure signal provided in `output_ssc.mat`, as described in Subsection 1.3. The comparison is carried out both in the time domain, by directly evaluating the waveform correspondence, and in the frequency domain, by comparing the corresponding Sound Pressure Level (SPL) curves. Additionally, you are required to plot the error signal defined as the difference between the ground-truth signal and the output signal of the WDF.

In the script `mems_spk_WD.m`, the input signal F_{in} is already defined, the ground-truth output pressure signal computed from the Simscape model is already imported. Additionally, the conversion from F_{out} to pressure signal is pre-defined, along with all circuit parameters. The script also includes the necessary plotting routines for visualizing both the output signals and the error signal. The only remaining task is the actual implementation of the WDF.

Files to be delivered

You are required to deliver the following files:

1. A short **report** (max. 2 pages) in PDF format including
   - the picture of the WDF scheme following the recommendations discussed in Section 4;
   - a description of the WDF scheme in words, in case you believe that your scheme requires some further explanation;
   - the setting of free parameters due to adaptation conditions (please define the free parameters at each port coherently to the WDF scheme and specify how to set each free parameter using symbolic expressions as indicated in Section 4);
   - the plots of the output signal and of the error signal described in Section 5;
   - the value of the Mean Squared Error (MSE) between the output signal of the WDF structure and the ground-truth.
2. The folder containing the completed MATLAB script named `mems_spk_WD.m` along with all the other files already provided (namely, the ground truth signal `output_ssc.mat` and the provided MATLAB functions), such that we can directly execute the script in that folder. Please, **deliver just one MATLAB script and avoid producing auxiliary files such as MATLAB functions**.

The homework can be completed individually or in groups of up to two students. Remember to write your names both in the report and at the beginning of the MATLAB script as a comment. Put both the report and the folder containing the MATLAB script in another folder called 'SSSP_HW2_Surname' (where 'Surname' is your surname) in case you are doing the homework individually, or called 'SSSP_HW2_Surname1_Surname2' (where your surnames 'Surname1' and 'Surname2' are in alphabetical order) in case you are doing the homework in groups of 2. Finally, compress the folder in a zip file and upload it using the WeBeep platform in the delivery folder. One student will submit a zip file for the entire group; **do not upload the same HW twice**.

References

[1] H. Wang, Y. Ma, Q. Zheng, K. Cao, Y. Lu, and H. Xie, "Review of recent development of MEMS speakers," *Micromachines*, vol. 12, no. 10, 2021. [Online]. Available: https://www.mdpi.com/2072-666X/12/10/1257

[2] K. J. Werner, A. Bernardini, J. O. Smith, and A. Sarti, "Modeling circuits with arbitrary topologies and active linear multiports using wave digital filters," *IEEE Transactions on Circuits and Systems I: Regular Papers*, vol. 65, no. 12, pp. 4233–4246, 2018.

[3] A. Farina, "Advancements in impulse response measurements by sine sweeps," in *Proc. 122nd Audio Engineering Society (AES) Convention*, Vienna, Austria, 5 2007.