Assignment 5: Acoustic Field – Vibrating Piston Transducer

In this assignment you shall investigate on the acoustic field that is produced by a piston type transducer vibrating with sinusoidal surface normal velocities of amplitude v_0 under different types of boundary conditions. Fig. 1 depicts the r-z-cross-section of a cylindrical fluid domain. This fluid domain of radius r_{outer} and height h is enclosed at its top and outer right hand side boundary by a perfectly matched layer (PML) of width w_{PML}. A piston type transducer of radius r_{inner} is located at the bottom of the fluid domain. Its center axis is aligned with the center axis of the cylindrical fluid domain. As the surface normal velocity v_0 at the interface b_{e} between piston and fluid domain shall be known, it is sufficient to model the piston type transducer in form of a boundary line.
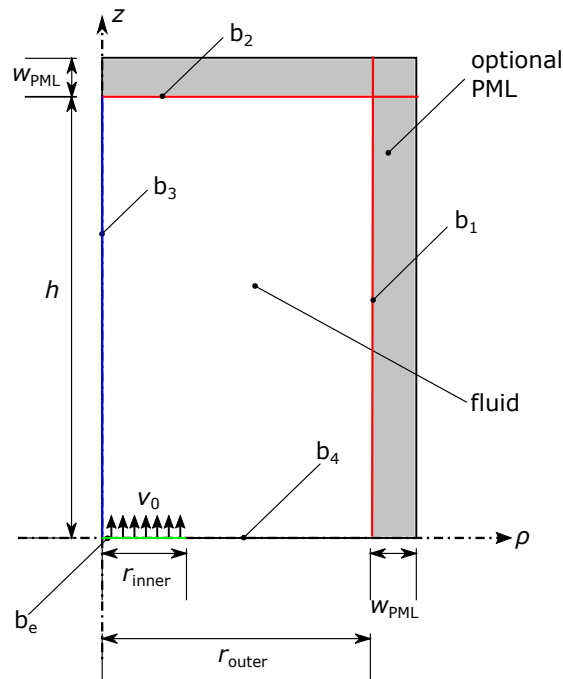

Figure 1: Acoustic setup consisting of a piston type transducer (green line) that is connected to a cylindrical fluid domain.

General hints and remarks

Please make sure to:

- <u>Save</u> all relevant plots you generate (e.g., as `png`-file) while working on the assignment.

- Always <u>label the axes</u> of the plots and <u>add a legend</u> if necessary.

- Format your plots so that they can be <u>read easily</u> (e.g., appropriate font size).

- Prepare a `Results.pdf` file with the plots and <u>short</u> answers to all questions (label each answer with its task number) for submission.

- If you are unsure on how to perform the tasks, refer to

1 Preparation

1.1. For the case of an ideal piston type transducer, i.e., each point on the surface oscillates with the same normal velocity v_0, the acoustic pressure $p(z)$ along the center line of the transducer can be computed via (see Lerch et al., "Technische Akustik", page 155)

$$p(z) = 2\mathrm{i}\rho c v_0 \sin\left[\frac{k}{2}\left(\sqrt{R^2 + z^2} - z\right)\right] \mathrm{e}^{-\frac{\mathrm{i}k}{2}\left(\sqrt{R^2+z^2}+z\right)} \ . \tag{1}$$

Thereby, the following variables are used

- ρ, c: density and speed of sound of the fluid medium (here water)
- $k = \frac{2\pi}{\lambda} = 2\pi f/c$: wave number, with wave length λ
- i: imaginary unit
- R: radius of the piston

What is the magnitude $|p(z)|$ of the complex valued function (1)?

2 Modeling

2.1. Create a new Comsol model using the *Pressure Acoustics, Frequency Domain (acpr)* interface. Add a *Frequency Domain* study to it.

2.2. Model the geometry as in Fig. 1. Thereby, define all required dimensions as `Parameters`. Do not model the PML region for now, as you will add it later on. Use the values $r_{\mathrm{inner}} = 8\,\mathrm{m}$, $r_{\mathrm{outer}} = 8\,r_{\mathrm{inner}}$ and $h = 10\,r_{\mathrm{inner}}$.

2.3. Add `Water, liquid` as a material and apply it to the whole simulation domain.

2.4. Select the `Pressure Acoustics, Frequency Domain (acpr)` node. Make sure to apply this physics to the entire domain.

- What PDE is solved by Comsol? Note: In the shown equations you can assume $\mathbf{q}_{\mathrm{d}} = \mathbf{0}$ and a material with homogeneous density $\rho_{\mathrm{c}} = \text{const.}$. What is the name of this well known equation?
- What is the default boundary condition (BC)?
- What is the default discretization order / element type used by Comsol?

2.5. In the settings of the physics node there is a section where you can specify a *reference pressure*. What is this reference value and when does Comsol need to use it?

2.6. There is also a section where you can specify the *typical wave speed for PMLs*. When does Comsol need this value and how is it used? As you will be adding a PML later on, it is necessary to set this value to the sound speed of the medium that you are simulating. Do this with the three steps below:

2.6.1. First find the speed of sound of the material you added in task 2.3. by navigating to *Materials→Water, liquid (mat1)→Basic (def)→Interpolation (cs)*. Selecting the above node, you will find that the speed of sound c_s is a function of the temperature in degrees Kelvin. Plot this function by pressing the *Plot* button at the top of the settings window. Note that the wave speed of water has a strong dependence on the temperature. What is the wave speed at $20\,°\mathrm{C}$?

2.6.2. To find out the temperature at which you are running your simulation, go to the *Pressure Acoustics, Frequency Domain→Pressure Acoustics 1* node. Under the *Model Input* section you will find the temperature. What is the value? What is its value in degree Celsius?

2.6.3. Go back to the physics node and change the value of the *typical wave speed for PMLs* (c_{ref}) to `mat1.def.cs(293.15)`. This will evaluate the wave speed of the material at given temperature.

2.7. Add an excitation with a prescribed velocity of $v_0 = 1\,\text{mm/s}$ on the piston boundary, i.e., boundary b_e. You can do this using an appropriate BC.

2.8. Leave all other BCs as *sound hard* for now.

2.9. Mesh your geometry using a *Mapped* mesh. Choose an appropriate element size using the rule of thumb for quadratic elements you learned in class. Define `Parameters` to help you specify the maximum element size. Consider that the maximum and minimum frequencies to be simulated are $1000\,\text{Hz}$ and $100\,\text{Hz}$, respectively.

2.10. Under *Study 1$\rightarrow$Step 1*, specify three frequencies to solve for: $100\,\text{Hz}$, $316\,\text{Hz}$ and $1000\,\text{Hz}$.

3 Standing wave in a cylindrical closed tube

As a first task, consider a cylindrical tube. The model created in Section 2 represents a tube *closed* from all boundaries. It has sound hard walls, i.e., the walls are much stiffer than the fluid.

3.1. Run your simulation without modifications to your model of Section 2.

3.2. Comsol plots the *total acoustic pressure* `acpr.p_t`. In our case, this is the same as the *acoustic pressure* `p`. As we are performing frequency domain simulations, the resulting acoustic pressure is complex valued. Per default, Comsol plots the real part of complex variables and as you can see in the plot, the real part of `acpr.p_t` is zero.

3.2.1. Confirm the above by plotting `real(acpr.p_t)` for $f = 100\,\text{Hz}$. You should also get zero on the hole domain.

3.2.2. Now plot `imag(acpr.p_t)` for $f = 100\,\text{Hz}$. The result is apparently purely imaginary. Why is that? Give a motivation.

3.2.3. Plot `abs(acpr.p_t)` for all frequencies.

4 Optional: The influence of different boundary conditions

Note: You do not need to submit this task, as it is optional.

Next, you will study the influence of different BCs at the top boundary (b_2). As you will need your simulation results of the previous task later on, **save them** before overwriting them with the next simulation. You can do so by

4.3. Right clicking on the *Study 1* node and selecting *Create Solution Copy*.

4.4. Rename the newly created node *Study 1/Solution 1 - Copy 1* under the *Data Sets* node to *Study 1/Solution 1 - sound hard*.

Let's now investigate the behavior of an open tube, i.e., with a sound soft BC at the top. This represents a partially water filled pipe with air at the top.

4.5. Add a *Sound Soft Boundary*, apply it to the top line and perform the simulation as before.

4.6. Plot the resulting acoustic pressure **magnitude**, i.e., the absolute value, for all frequencies. What is the value of the acoustic pressure magnitude at the top boundary? Why is that?

4.7. Before continuing, duplicate the simulation results as in 4.3. and name it appropriately.

Investigate how a piston excites an "infinite" water filled tube, i.e., we do not consider reflections from the top of the pipe. For this you must prevent reflection of sound at the top boundary. As a first attempt:

4.8. Add a *Plane Wave Radiation*, apply it to the top line and perform the simulation as before.

4.9. Plot the resulting acoustic pressure **magnitude** for all frequencies.

4.10. Before continuing, duplicate the simulation results as in 4.3. and name it appropriately.

The *Plane Wave Radiation* is optimized for plane waves with near-normal incidence at the boundary, all other wave components are still (partially) reflected. To improve the model, use a PML at the top of the domain instead:

4.11. Add a PML domain to the **top** of your geometric model and set $w_{\mathrm{PML}} = 15\,\mathrm{m}$.

4.12. Right click on *Component 1→Definitions* and select *Perfectly Matched Layer*. Apply the PML only to the newly added domain region. Change the *Geometry Type* to *Cylindrical*. Leave the other parameters untouched.

4.13. Deactivate all BCs we had previously set up for the line b_2, as we will no longer need them.

4.14. Re-compute *Study 1*.

4.15. Plot again the pressure magnitude for all frequencies. Notice how the PML damps the pressure inside the PML region. Does the PML perform equally well for all frequencies? Why is that and how could you improve damping for high frequencies?

4.16. Before continuing, duplicate the simulation results as in 4.3. and name it appropriately.

To better assess the results, plot the pressure magnitude along the center line for all the above cases:

4.17. Add a *1D Plot Group* to plot the results of the different simulations at $100\,\mathrm{Hz}$. Add 4 *Line Graphs* to it for sound hard, sound soft, plane wave radiation and PML, respectively. Each of the line graphs, shall plot the corresponding pressure magnitude. Make sure to add a meaningful legend to the plot. Note: you will need to select the lines on which to plot for each of the *Line Graphs* separately, include the PML region for the PML case.

4.18. Do the same as above for $316\,\mathrm{Hz}$.

4.19. Note that the pressure magnitude is exponentially damped in the PML region. It is not damped to zero, however. This could be improved by increasing the PML size in expense of computational time.

4.20. Compare the plotted results and add your observations to `Results.pdf`. Go into detail about the following points

   4.20.1. Depending on the BC, you observe very different values for the steady-state pressure magnitude. What is the reason for this?

   4.20.2. Is there a visible difference between the plane wave radiation conditions and the PML regarding the pressure along the centerline? Is there an influence of the frequency or do both methods perform equally well for all three frequencies? You should also take a look at the pressure distribution in the whole fluid domain. This will give you additional information (take a look at the acoustic field near the right hand side boundary, too!).

   4.20.3. Compare the pressure values resulting from a sound soft and a sound hard boundary at the top of the tube. What pressure value is obtained at the top boundary? Does the pressure go to zero or to a maximum/minimum there?

5   Free field wave propagation

After investigating on the wave propagation inside a tube, you shall now simulate the free field behavior of the piston transducer, i.e., without reflecting walls.

5.1. Similar to task 4.11., model the PML domain including both the right hand side as well as the top of the simulation domain, as depicted in Fig. 1.
Important: Make sure to model the PML geometry with 3 different regions – top, corner and right part. Comsol requires this because it needs to treat each of these regions differently.

5.2. Specify the PML as in task 4.12. and make sure that all 3 regions are selected in the PML node this time.

For the following tasks, perform only **one** harmonic simulation at $f = 500\,\text{Hz}$.

5.3. Perform a parametric sweep for $r_{\text{outer}} = [2, 5, 8] \cdot r_{\text{inner}}$.

5.4. Plot the acoustic pressure magnitude, i.e., `abs(p)`, on the whole domain for $r_{\text{outer}} = 8 r_{\text{inner}}$ as a surface plot.

5.5. Plot the acoustic pressure magnitude on the center line (b_3) for each of the values of r_{outer} into one single graph. Additionally, plot the analytic result of equation (1) into the same graph. You can do this by

    5.5.1. Adding an *Analytic Function* to *Component 1→Definitions*. Comsol does not have full support for complex functions, it is therefore important that you return real values. As *Expression* enter the **magnitude** of the analytic solution of equation (1). If required, you can define and use additional `Parameters` for this function.

    5.5.2. Go to *Study 1* and click *Update Solution*. This will compute the values of the newly added function on your mesh.

    5.5.3. Now you can plot those values along the center line by adding a *Line Graph* to your *1D Plot Group* and evaluating `an1(z)`. Make sure to plot the function for only one value of your parametric sweep as the function is independent of the outer radius.

5.6. Should the distance to the right hand side PML have any effect on the pressure along the center line? According to your figure of task 5.5., do you see any such influence?

5.7. After certain distance – known as the *near field distance* z_{nf} – the pressure magnitude radiated by a piston transducer falls monotonically. The region $z < z_{\text{nf}}$ is called the *near field*, while the zone with $z > z_{\text{nf}}$ is referred to as the *far field* of the transducer. What is the value of z_{nf} for the present case?

Submitting your assignment

Hand in your group's

- COMSOL `mph`-file (clear all meshes and results beforehand, see the COMSOL Tutorial), and

- `Results.pdf` file with the plots and <u>short</u> answers to all questions (label each answer with its task number),

before the due date by uploading to the corresponding StudOn section.

When handing in for the first time you need to create a group. Please add all members when doing so! Once the group has been create, you can and should <u>re-use the group</u> for the subsequent submissions.