

Assignment 1: Mechanical-Acoustic coupling – Vibrating Disc

Figure 1 shows the axis-symmetric setup of a solid disc of height h_{disc} and radius r_{disc} which is embedded in a sound hard wall b_s . The disc is fixed at its outer edge b_f . The surface of the disc, denoted by the line b_e , is in contact with an unbounded fluid region. When the disc oscillates, e.g., due to mechanical forces, acoustic waves will be emitted into the neighboring fluid region. In this assignment we want to investigate on the following aspects:

- How does the frequency response of the vibrating disc look like?
- What is the influence of the material of the disc on the frequency response?
- In which way are mechanical vibrations damped by the presence of the neighboring fluid?

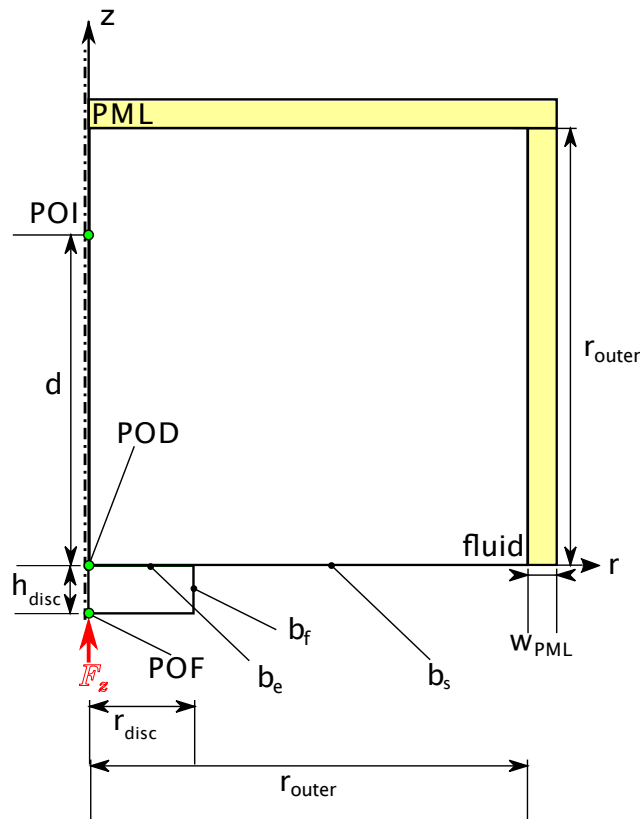


Figure 1: Mechanical-acoustic setup of a vibrating disc (solid) that radiates acoustic waves into an unbounded fluid domain. The disc is embedded into a sound hard wall (b_s).

General hints and remarks

- You should always correctly **label** the axes of the plots and add a **legend** if necessary. Format plots so that they can be **read easily**.
- **Save plots** as a `.png`-file before proceeding to the next task, as you will need to submit these.
- Write your **short answers** to all questions to a `Results.pdf` file (label each answer with its task number). Include your plots in this file.
- If you are unsure on how to perform the tasks, refer to
 - [COMSOL Tutorial](#)
 - [COMSOL Multiphysics Reference Manual](#)

1 Modeling

- 1.1. Use the model wizard to create a 2D-axisymmetric geometry and use the *Acoustics→Acoustic-Structure Interaction→Acoustic-Solid Interaction, Frequency Domain* physics. This Comsol template will add (1) a *Pressure Acoustics, Frequency Domain (acpr)* physics node, (2) a *Solid Mechanics (solid)* physics node and (3) an *Acoustic-Structure Boundary (asb1)* Multiphysics node. Multiphysics nodes are used in Comsol to model the coupling between two different physical domains. Instead of adding these three nodes with the *Acoustic-Solid Interaction, Frequency Domain* template, you could add the three nodes manually. In the last step of the model wizard, add a *Frequency Domain* study.
- 1.2. Model the geometry as depicted in Fig. 1. Thereby, define all required dimensions as **Parameters** and use the following values:
 - $h_{\text{disc}} = 5 \text{ mm}$
 - $r_{\text{disc}} = 40 \text{ mm}$
 - $r_{\text{outer}} = 200 \text{ mm}$
 - $w_{\text{PML}} = 5h_{\text{fluid}}$, where h_{fluid} is the element size in the fluid domain.
 - $d = 80 \text{ mm}$Make sure to also add a point for the POI. This ensures that the mesh will have a node at the POI.
- 1.3. Add *Air* as material for the fluid domain and *PET (polyethylene terephthalate or Mylar)* for the disc. Assign each of them to their corresponding domains. Comsol will show warnings because it thinks that you want to perform a mechanical simulation on the whole domain but air does not have mechanical material parameters. Ignore the warnings, as we will change this in a bit.
- 1.4. Assign the *Pressure Acoustics, Frequency Domain* physics to the fluid domains.
- 1.5. Assign the *Solid Mechanics* physics to the disc domain.
- 1.6. Go back to the two materials you added. Note that there is still a warning for the PET material. Indeed the Poisson's ratio is needed for the mechanical simulation but missing in the material definition. Set its value to 0.34.
- 1.7. Go to the *Acoustic-Structure Boundary 1* node and make sure that the correct *Acoustics* and *Structural* physics are being coupled and that the correct interface between the two physical domains is selected (the line b_e).
- 1.8. Disclose the *Equation* section and inspect the coupling equations. We have an exterior boundary (the fluid is outside the solid). In the equations, u_{tt} is the structural acceleration experienced by the fluid, \mathbf{n} is the surface normal, p_t is the total acoustic pressure and \mathbf{F}_A is the load (force per unit area) experienced by the structure due to the fluid.

- 1.8.1. Is this a strong or a weak coupling?
- 1.9. Go to *Component 1*→*Definitions* and add a *Perfectly Matched Layer*. Choose the correct domains for the PML and set the geometry type to *Cylindrical*.
- 1.10. Go back to *Component 1*→*Solid Mechanics* and add a *Fixed Constraint* on b_f .
- 1.11. Add a *Point Load (on Axis)* on the POF. Use a frequency-dependent force of $F_z = 5.0 \text{ N Hz}/f$. Comsol knows the variable `freq` when performing harmonic simulations.
- 1.12. Under *Study 1*→*Step1: Frequency Domain* set a logarithmic distribution of frequency values between 50 Hz and 50 kHz using 60 steps per decade.
- 1.13. Perform the harmonic simulation. This should take only a couple of minutes.
- 1.14. Inspect the resulting acoustic field for different frequencies.
- 1.15. Inspect the stress in the disc and its deformation for different frequencies. Does the disc vibrate uniformly? Will the radiation of this disc be similar to a piston transducer as in Assignment 5 of CAE of Sensors and Actuators?

2 Frequency response at POI

Investigate how the frequency-dependent pressure amplitude at the POI varies with different material combinations. Lets first plot the result for the previous simulation with *Mylar* for the disc and *Air* for the fluid:

- 2.1. Right click on *Solver Configurations* and select *Create Solution Copy*. A new Dataset will appear under *Results*. Rename the new dataset node to *Study 1/Mylar-Air*. This dataset will not be overwritten when performing a new simulation.
- 2.2. Create a *1D Plot Group* and add a *Point Graph* selecting the *Study 1/Mylar-Air* dataset as a source. Select the POI and plot the absolute acoustic pressure vs. the frequency. Make the graph double-logarithmic.
- 2.3. The frequency response exhibits maxima. Why is that? Motivate your answer.

Now compute the frequency response for *Water* instead of *Air* and plot the result into the same graph:

- 2.4. Add *Water, liquid* as a material and assign it to the fluid domains. Make sure to also change the *Typical Wave Speed for Perfectly Matched Layers* in the *Pressure Acoustics, Frequency Domain* node to an appropriate value. Otherwise the PML will not work correctly.
- 2.5. Perform the harmonic simulation. Repeat task 2.1. with an appropriate name for the node. Plot the resulting frequency response of the pressure at the POI into the same *1D Plot Group* as the response for *Air*. Thereby, make sure to use the newly created dataset as source for the plot. Both Graphs should now be displayed in the same axis.
- 2.6. Compare the two frequency responses. What do you observe? Which one radiates more into the fluid domain? Why?
- 2.7. The resonant frequencies are different. Why is that? Motivate why the resonant frequencies of the water loaded disc are lower than for the air loaded disc.

In the following you will compare a water loaded *Mylar* disc to a water loaded *Steel* disc:

- 2.8. Add *Structural Steel* as a material and assign it to the disc domain.
- 2.9. Perform the harmonic simulation and make a solution copy as before. Name the new dataset node appropriately and use it as a plotting source for the following task.

- 2.10. Create a new *1D Plot Group*. Plot again the frequency-dependent absolute pressure at the POI in a double-logarithmic scale. Plot both the water loaded Mylar disc as well as the corresponding steel disc.
- 2.11. How are the frequency responses different? Why is the response of the steel disc lower than of the Mylar disc?

3 Transient simulation

In the following you will perform a transient analysis of the vibrations of the water-loaded Mylar disc.

- 3.1. Change the material of the disc back to *Mylar*.
- 3.2. The physics node for acoustics is only appropriate for harmonic simulations. Add a *Pressure Acoustics, Transient* physics to your model. Assign it to the appropriate domains.
- 3.3. Add a *Spherical Wave Radiation* condition to the physics and apply it to the outer domain boundaries (but b_s is still sound hard).
- 3.4. The new physics is not yet coupled to the mechanics. Right click on *Multiphysics* and add a *Acoustic-Structure Boundary* node. Couple the *Pressure Acoustics, Transient* physics to the *Solid Mechanics*.
- 3.5. You will need a time-dependent exciting force at the POF.
 - 3.5.1. Define the excitation signal under *Component 1* → *Definitions* as an *Analytic* function. You might also want to add a *Rectangle* function to be used in the *Analytic* function. Define any Parameters you need. Use the following excitation signal:
 - Sinus-burst with 4 periods.
 - center frequency of $f_c = 1550$ Hz
 - amplitude of 0.025 N
 - 3.5.2. Add an extra *Point Load (on Axis)* node to the *Solid Mechanics* physics. Use the time-dependent force you just defined to load the disc at the POF.
- 3.6. The newly created nodes should not be used in the harmonic simulation. Go to *Study 1* → *Step 1: Frequency Domain* and under *Physics and Variable Selection* check *Modify model configuration for study step*, then disable the following nodes:
 - 3.6.1. *Point Load (on Axis) 2*
 - 3.6.2. *Pressure Acoustics, Transient*
 - 3.6.3. *Acoustic-Structure Boundary 2*
- 3.7. Coarsen the mesh as far as possible while still ensuring correct simulation results (one third of the minimum wavelength contained in your excitation signal should be enough). The coarser the mesh, the faster the simulation.
- 3.8. Add a new *Time-Dependent* study. Only include the *Pressure Acoustics, Transient* and the *Solid Mechanics* physics as well as the newly created *Acoustic-Structure Boundary 2* coupling in the study.
- 3.9. Simulate a total time of three times the excitation signal. Choose the time step as $f_c/20$. Set corresponding time points in the Study settings.
- 3.10. Select the *Modify model configuration for study step* option.
 - 3.10.1. Disable the PML node. Why should you deactivate PMLs for transient simulations? → We added the Spherical Wave Radiation condition instead.
 - 3.10.2. Disable the *Pressure Acoustics, Frequency Domain* physics (if it is marked orange, it is already disabled).

- 3.10.3. Disable the *Point Load (on Axis) 1* node (it depends on the `freq` variable, which does not exist for the transient simulation).
- 3.10.4. Disable the *Acoustic-Structure Boundary 1* node (if it is marked orange, it is already disabled).
- 3.11. Right click on *Solver Configurations* and select *Show Default Solver*.
- 3.12. Under *Solution $x \rightarrow$ Time-Dependent Solver 1* go to *Time Stepping* and change *Steps taken by solver* to *Strict*. This makes sure that the solver actually evaluates within the time steps you specified in 3.9.
- 3.13. Perform the transient simulation. This may take 10 min or so.

4 Analyzing the disc's vibrations

Plot the mechanical displacements in z -direction at the POD:

- 4.1. Create a new *1D Plot group* and add a *Point Graph* to it.
- 4.2. Plot the mechanical displacements in z -direction at the POD over time.
- 4.3. What do you observe?
 - 4.3.1. In especial, after the excitation, why does the amplitude decrease over time? Remember that we have not modeled damping in the material.
- 4.4. Create an animation showing the vibration of the entire disc.

Submitting your assignment

Hand in your

- Comsol `.mph` file and
- a `Results.pdf` file with short answers to all questions within this assignment sheet (label each answer with its task number). All plots you are asked to create should also be contained in this file at the position of the corresponding task. Animations should be reduced to just one frame at a “representative” time. You can, for example, use Word or LaTeX to produce the PDF file.

Upload the above files to the corresponding Assignment section on StudOn:
<https://www.studon.fau.de/exc2992793.html>.

Important: Make sure to delete all results and meshes from the Comsol file before handing in. This reduces the file size. Refer to the Comsol Tutorial on how to do this.