Created in COMSOL Multiphysics 5.5

Vibrating Particle in Water

Introduction

This is a model of a small particle oscillating in water, see Figure 1. It validates the numerical solution of the system of thermoviscous acoustics equations by comparison with an analytical solution given in Ref. 1. The Thermoviscous Acoustics, Frequency Domain interface is used for modeling the propagation of acoustic waves in small devices, where it is important to include losses in a detailed way. This is useful when modeling acoustics and vibrations in, for example, microphones, hearing aids, or MEMS devices. The interface provides a detailed way of solving the equations governing the propagation of acoustic waves in any fluid, including thermal conduction and viscous damping.

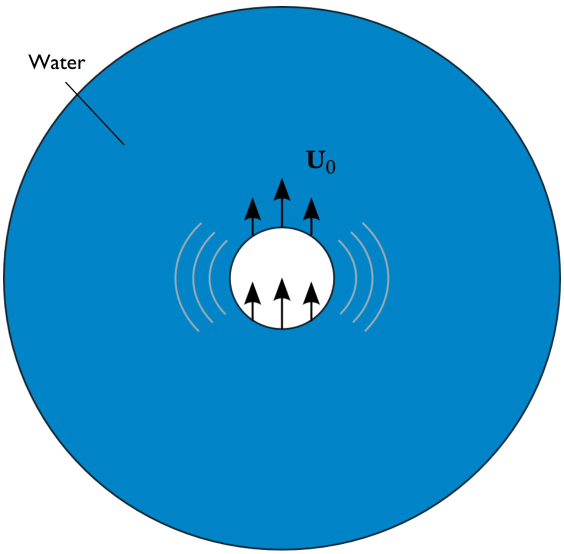


Figure 1: Sketch of the particle oscillating along the axis.

Acoustic waves are compressible waves and in the framework of the Thermoviscous Acoustics interface, a linearized equation of state is set up. This constitutive equation relates the acoustic variations in pressure p, density ρ, and temperature T through the expression

$$\rho = \rho_0(\beta_T p - \alpha_p T)$$

where ρ_0 is the equilibrium density, β_T is the isothermal compressibility, and α_p is the coefficient of thermal expansion. The model shows a small sphere of radius 1 mm which is oscillating along the polar axis at 50 kHz. The sphere is modeled in 2D axisymmetry.

Note: Details about the governing equations are found in the theory section of the thermoviscous acoustics physics interface documentation. See **File>Help>Documentation** and search or open the *Acoustics Module User's Guide* to the thermoviscous acoustics theory.

Model Definition

The model is set up in a 2D axisymmetric geometry; that is, the spatial coordinates are the radius, r, and the height, z. The spherical particle of the radius a_s vibrates along the z-axis with the velocity $\mathbf{U}_0 = U_0\mathbf{e}_z$.

The analytical solution is obtained from the Helmholtz decomposition of the acoustic particle velocity

$$\mathbf{u} = \nabla\varphi + \nabla\times\mathbf{B}$$

The velocity potential, φ , far from the sphere is defined as (Ref. 1)

$$\varphi(r,z) = U_0\left(\frac{a_s}{R}\right)^3\frac{ikR-1}{2-b^2-2ib}z\exp(ik(R-a)) \tag{1}$$

where k is the wave number, $R = (r + z)^{1/2}$, and $b = ka_s$. This yields the acoustic pressure

$$p(r,z) = i\omega\rho_0\varphi(r,z)$$

In this model, the adiabatic formulation of the system of thermoiviscous acoustics equations is solved. This formulation is appropriate because the thermal losses play a minor role in water compared to the viscous losses.

Since the acoustic waves radiated from the particle propagate in the free space, the computational domain used in the model should be truncated in a way that ensures wave propagation without reflections from the outer boundary. This is done in the model by surrounding the computational domain by a perfectly matched layer (PML).

Results and Discussion

The acoustic pressure variations and the instantaneous acoustic particle velocity in the physical domain are plotted in Figure 2 and in Figure 3. Figure 4 shows the pressure variations along the cut line directed from the particle top at the angle of 45° to the z-axis.

The blue solid line and the green dotted line correspond to the numerical and the analytical solutions, respectively. The results match well except for the area near the particle. This is explained by the fact that Equation 1 is an asymptotic expression that is invalid near the particle. The exact expressions for φ and $\mathbf{B}$ can be found in Ref. 1.

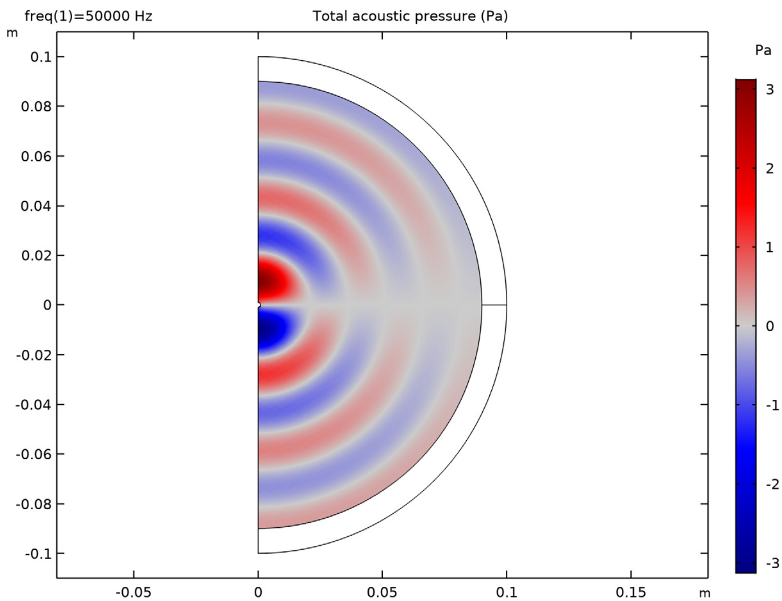


Figure 2: Pressure variations in the water outside the small vibrating particle.

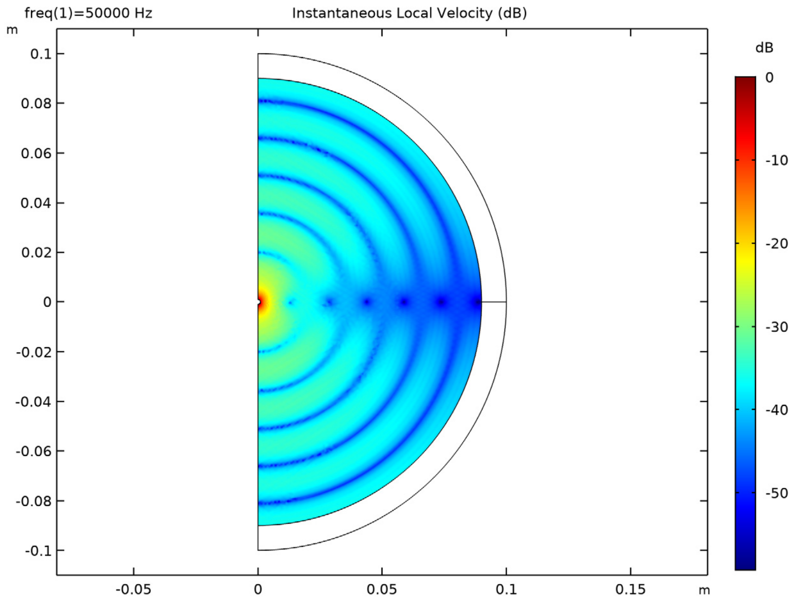


Figure 3: Instantaneous acoustic particle velocity in the water outside the small vibrating particle.

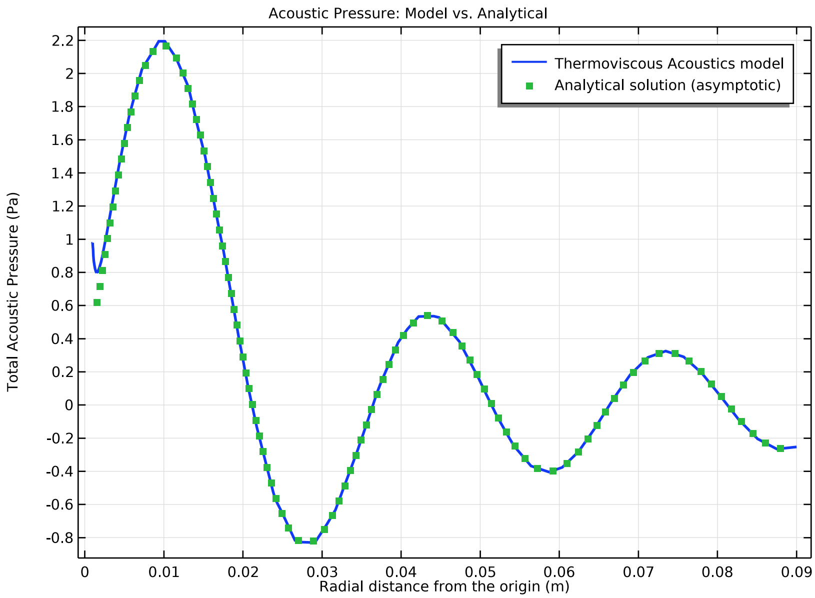


Figure 4: Pressure variation along the cut line: numerical and analytical solutions.

References

1. S. Temkin, *Elements of Acoustics*, Acoustical Society of America, 2001.

Application Library path: `Acoustics_Module/Tutorials,` `_Thermoviscous_Acoustics/vibrating_particle_water`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Frequency Domain**.
6. Click **Done**.

Load the parameters from the file `vibrating_particle_water_parameters.txt`.

GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `vibrating_particle_water_parameters.txt`.

GEOMETRY 1

Circle 1 (c1)

1. In the **Geometry** toolbar, click **Circle**.

   The thermal losses only play a minor role in water and can therefore be neglected by solving the system of thermoviscous equations in adiabatic formulation.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `a_s`.
4. Click **Build Selected**.

Circle 2 (c2)

1 Right-click **Circle 1 (c1)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type a_tot.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	a_tot - a_ta

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c2** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate selection** toggle button.

5 Select the object **c1** only.

6 Click **Build All Objects**.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
b	a_s*k0		Help variable
R0	sqrt(r^2 + z^2)	m	Radial distance from the origin
phi_an	U0*a_s^3/R0^3*z*exp(i*k0*(R0 - a_s))*(i*k0*R0 - 1)/(2 - b^2 -2*i*b)	m^2/s	Velocity potential (asymptotic)
p_an	i*omega0*rho0*phi_an	Pa	Acoustic pressure (asymptotic)

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 3 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Frequency Domain (ta)**.

2 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Sound Pressure Level Settings** section.

3 From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

4 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `c0`.

5 Locate the **Thermoviscous Acoustics Equation Settings** section. Select the **Adiabatic formulation** check box.

Velocity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Velocity**.

2 Select Boundaries 8 and 9 only.

3 In the **Settings** window for **Velocity**, locate the **Velocity** section.

4 Select the **Prescribed in r direction** check box.

5 Select the **Prescribed in z direction** check box.

6 In the u_{0z} text field, type `U0`.

Now, proceed and set up the mesh.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lam0/10`.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8 and 9 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Curvature factor** check box.

7 In the associated text field, type `0.1`.

Add **Boundary Layers** around the oscillating sphere to resolve the viscous penetration depth `dvisc`.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundaries 8 and 9 only.

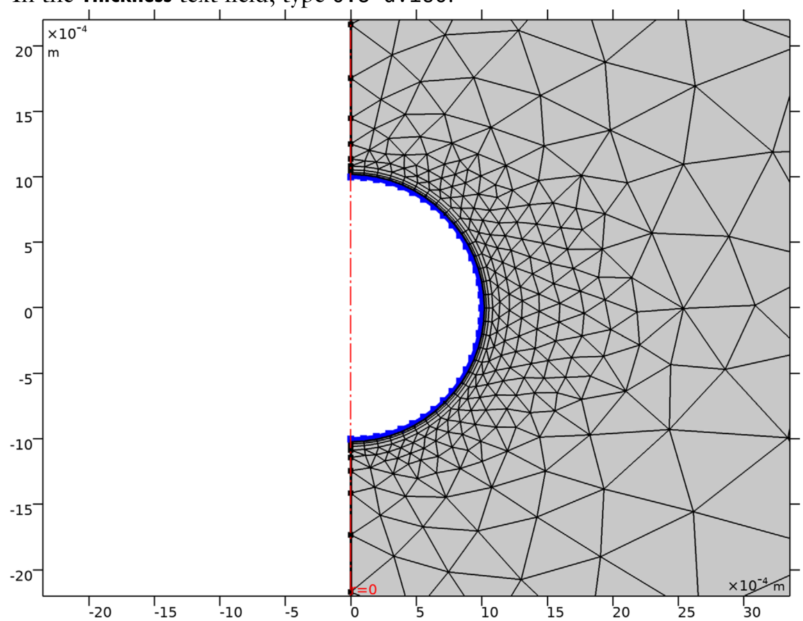
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** text field, type `10`.

5 In the **Boundary layer stretching factor** text field, type `1.5`.

6 From the **Thickness of first layer** list, choose **Manual**.

7 In the **Thickness** text field, type `0.3*dvisc`.



Mapped 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `8`.

5 Click **Build All**.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `f0`.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (ta)

1 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Total acoustic pressure (Pa)`.

4 Locate the **Color Legend** section. Select the **Show units** check box.

Selection 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (ta)** node.

2 Right-click **Surface** and choose **Selection**.

3 Select Domain 2 only.

4 In the **Acoustic Pressure (ta)** toolbar, click **Plot**.

The figure should look like the one in Figure 2.

Acoustic Velocity (ta)

1 In the **Model Builder** window, click **Acoustic Velocity (ta)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Instantaneous Local Velocity (dB)`.

5 Locate the **Color Legend** section. Select the **Show units** check box.

The acoustic velocity quickly fades away as the distance from the particle grows. Use the logarithmic scale with the reference value `U0` for better visualization of the velocity profile.

Surface

1 In the **Model Builder** window, expand the **Acoustic Velocity (ta)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `10*log10(abs(ta.v_inst/U0))[dB]`.

Selection 1

1 Right-click **Surface** and choose **Selection**.

2 Select Domain 2 only.

3 In the **Acoustic Velocity (ta)** toolbar, click **Plot**.

The figure should look like the one in Figure 3.

Create a **Cut Line 2D** data set to compare the numerical and the analytical solutions.

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 2**, set **R** to `a_ta/sqrt(2)`.

4 In row **Point 2**, set **Z** to `a_ta/sqrt(2)`.

1D Plot Group 4

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Total Acoustic Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Acoustic Pressure: Model vs. Analytical`.

6 Locate the **Plot Settings** section. Select the **y-axis label** check box.

7 In the associated text field, type `Total Acoustic Pressure (Pa)`.

Line Graph 1

1 Right-click **Total Acoustic Pressure** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `R0`.

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Thermoviscous Acoustics model`

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `p_an`.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

5 Find the **Line style** subsection. From the **Line** list, choose **None**.

6 In the **Width** text field, type `1`.

7 Find the **Line markers** subsection. In the **Number** text field, type `100`.

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Analytical solution (asymptotic)`

9 In the **Total Acoustic Pressure** toolbar, click **Plot**.

The figure should look like the one in Figure 4.