Created in COMSOL Multiphysics 6.1

Vibrating Particle in Water

Introduction

This tutorial treats 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 asymptotic analytical (adiabatic) solution given in Ref. 1. For further comparison, the model is also set up using the Thermoviscous Boundary Layer Impedance (BLI) condition available in pressure acoustics.

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 the details of the thermal and viscous boundary layers. The Thermoviscous Boundary Layer Impedance condition used in Pressure Acoustics, Frequency Domain on the other hand treats what happens the boundary layers analytically. The condition is therefore not applicable in all situations. In this model, it applies well because the boundary layer is thin compared to the particle and the curvature of the particle surface.

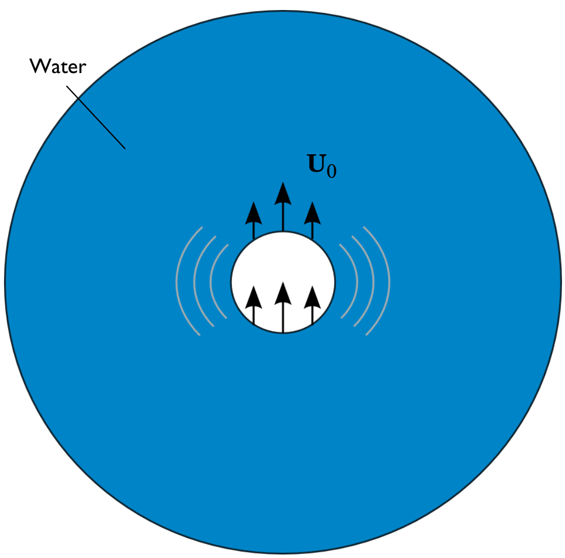

Figure 1: Sketch of the particle oscillating along the axis.

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 thermoviscous 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 represents the solution to the full equations of thermoviscous acoustics, the red line (on top of the blue) represent the pressure acoustics solution with the BLI condition, and the green line correspond to the analytical asymptotic solution, 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 and in the boundary layer in particular. The exact expressions for φ and $\mathbf{B}$ can be found in Ref. 1.

Finally, Figure 5 depicts the axial velocity near the particle surface extending 20 boundary layer thicknesses away. The figure shows how the full thermoviscous model matches the prescribed velocity U_0 (green line) by solving the details in the boundary layer. On the other hand, the details of the boundary layer are "lumped" with the Thermoviscous Boundary Layer Impedance condition used in pressure acoustics (red curve). The details have been treated analytically in the formulation of the condition.

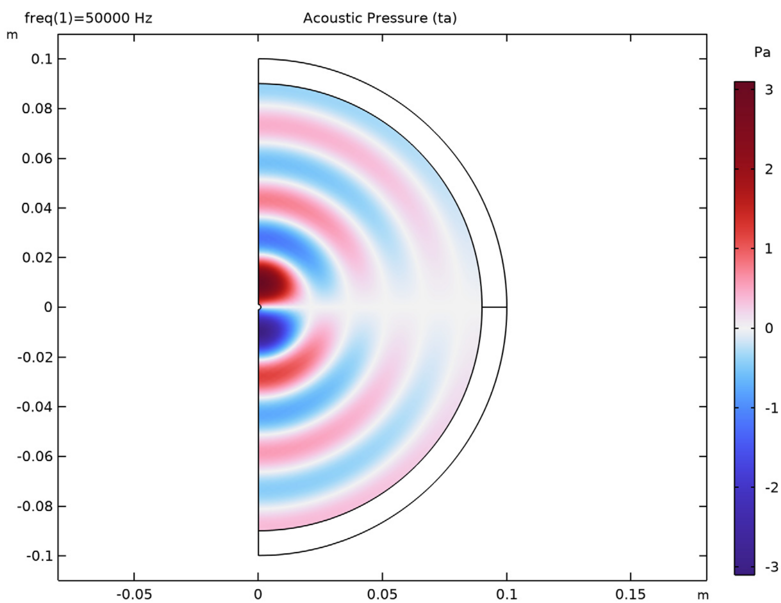

Figure 2: Pressure variations in the water outside the small vibrating particle.

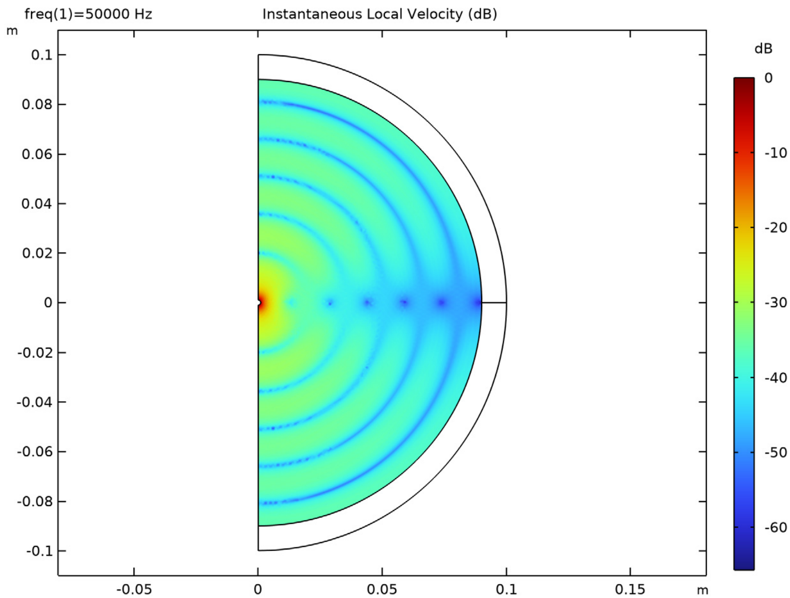

Figure 3: Instantaneous acoustic particle velocity in the water outside the small vibrating particle.

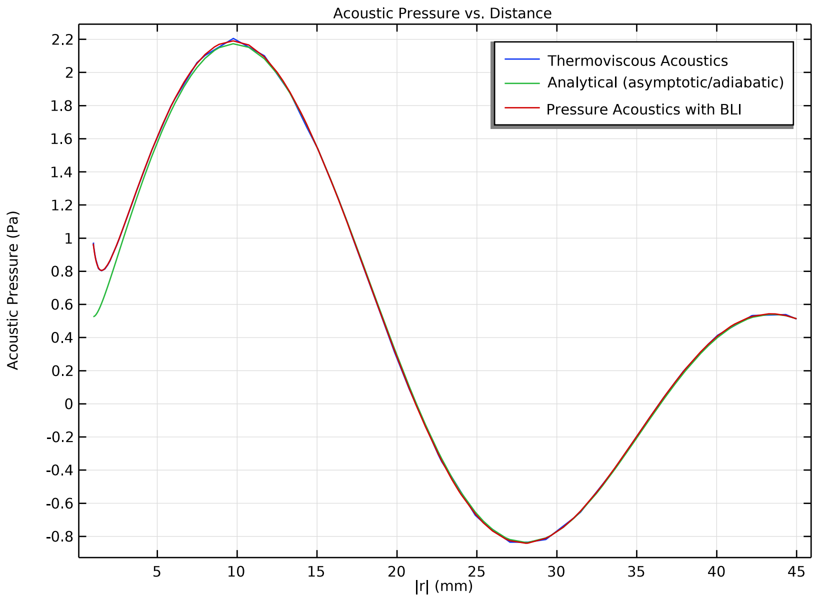

Figure 4: Pressure variation along the cut line: the numerical solutions and the analytical solutions.

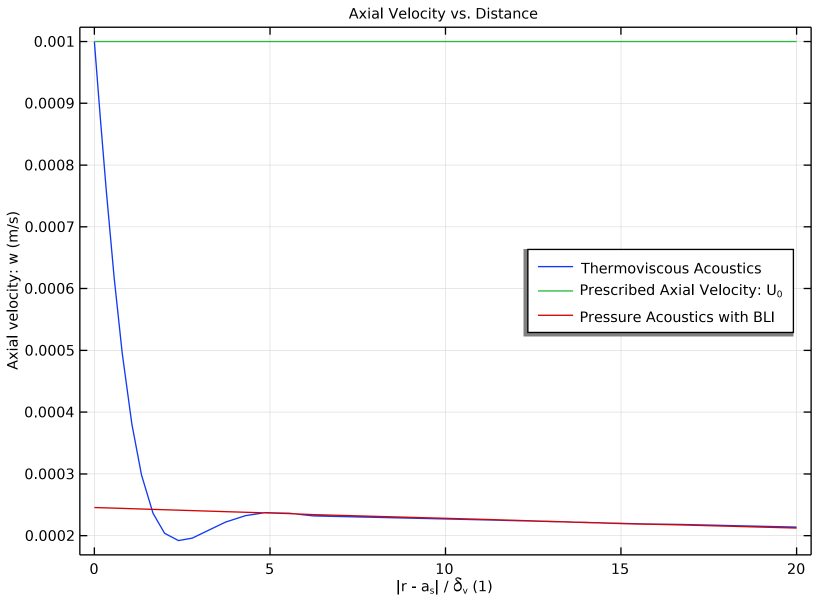

Figure 5: Comparison of the axial velocity close to the particle surface.

Reference

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. Click to 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)	m²/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 **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`.

Proceed and set up the model with Pressure Acoustics by using the **Thermoviscous Boundary Layer Impedance** boundary condition. Remember that the thermoviscous part is set up as adiabatic, so choose the **Viscous** fluid model (for the domain) and set the temperature condition to adiabatic. The setup is possible as the boundary layer thickness `dvisc` is several orders of magnitude smaller than the radius of curvature of the vibrating particle.

ADD PHYSICS

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

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

3 In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

4 Click **Add to Component 1** in the window toolbar.

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1. In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Sound Pressure Level Settings** section.
2. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

Pressure Acoustics 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 1**.
2. In the **Settings** window for **Pressure Acoustics**, locate the **Pressure Acoustics Model** section.
3. From the **Fluid model** list, choose **Viscous**.

Thermoviscous Boundary Layer Impedance 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Thermoviscous Boundary Layer Impedance**.
2. Select Boundaries 8 and 9 only.
3. In the **Settings** window for **Thermoviscous Boundary Layer Impedance**, locate the **Mechanical Condition** section.
4. From the **Mechanical condition** list, choose **Velocity**.
5. Specify the $\mathbf{v}_0$ vector as

0	r
U0	z

6. Locate the **Thermal Condition** section. From the **Thermal condition** list, choose **Adiabatic**.
7. Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Water, liquid (mat1)**.

Proceed and generate the mesh based on the **Physics-controlled mesh** suggestion for Thermoviscous Acoustics. This is done by selecting Thermoviscous Acoustics as the only **Contributor**. The mesh needs to well resolve the physics near the particle so some manual changes to the mesh are necessary. Start by defining the frequency in the study.

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`.

MESH 1

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

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

3 In the table, clear the **Use** check box for **Pressure Acoustics, Frequency Domain (acpr)**.

4 Locate the **Thermoviscous Acoustics, Frequency Domain (ta)** section. From the **Number of mesh elements per wavelength** list, choose **User defined**.

5 In the text field, type `12`.

6 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Size 1

Now, modify the mesh resolution near the particle by explicitly setting the **Curvature factor**.

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

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

3 Select the **Curvature factor** check box. In the associated text field, type `0.03`.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

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

3 In the **Number of layers** text field, type `8`.

4 From the **Thickness specification** list, choose **All layers**.

5 In the **Total thickness** text field, type `2*pi*dvisc`.

This ensures that 8 mesh elements are used to resolve the wavelength of the exponentially decaying viscous wave.

6 Click **Build All**.

Proceed and solve the model.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

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

3 Clear the **Generate default plots** check box.

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

5 Click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Thermoviscous Acoustics, Frequency Domain>Acoustic Pressure (ta)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Acoustic Pressure (ta)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (ta)**.

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

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

4 Select Domain 2 only.

5 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

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

The figure should look like the one in Figure 2.

Instantaneous Local Velocity (dB)

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Instantaneous Local Velocity (dB)` in the **Label** text field.

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

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

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

Surface 1

1 Right-click **Instantaneous Local Velocity (dB)** and choose **Surface**.

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.

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]`.

4 In the **Instantaneous Local Velocity (dB)** toolbar, click **Plot**.

The figure should look like the one in Figure 3.

Next, create two **Cut Line 2D** datasets to compare the numerical thermoviscous results, the analytical (adiabatic/asymptotic), and the pressure acoustics solutions with the boundary layer impedance condition. The first spans half the air domain and the second extends 20 viscous boundary layers from the particle.

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 1**, set **R** to `a_s/sqrt(2)`.
4 In row **Point 1**, set **Z** to `a_s/sqrt(2)`.
5 In row **Point 2**, set **R** to `0.5*a_ta/sqrt(2)`.
6 In row **Point 2**, set **Z** to `0.5*a_ta/sqrt(2)`.

Cut Line 2D 2

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 1**, set **R** to `a_s/sqrt(2)`.
4 In row **Point 1**, set **Z** to `a_s/sqrt(2)`.
5 In row **Point 2**, set **R** to `(a_s+20*dvisc)/sqrt(2)`.
6 In row **Point 2**, set **Z** to `(a_s+20*dvisc)/sqrt(2)`.

Acoustic Pressure vs. Distance

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Acoustic Pressure vs. Distance` 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 **Label**.
5 Locate the **Plot Settings** section.
6 Select the **x-axis label** check box. In the associated text field, type `|r| (mm)`.
7 Select the **y-axis label** check box. In the associated text field, type `Acoustic Pressure (Pa)`.

Line Graph 1

1 Right-click **Acoustic Pressure vs. Distance** 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 From the **Unit** list, choose **mm**.

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

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

Legends
Analytical (asymptotic/adiabatic)

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

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

3 In the **Expression** text field, type acpr.p_t.

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

Legends
Pressure Acoustics with BLI

5 In the **Acoustic Pressure vs. Distance** toolbar, click **Plot**.

The figure should look like the one in Figure 4.

Axial Velocity vs. Distance

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type Axial Velocity vs. Distance in the **Label** text field.

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

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `|r - a<sub>s</sub> | / \delta<sub>v</sub> (1)`.
7 Select the **y-axis label** check box. In the associated text field, type `Axial velocity: w (m/s)`.
8 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Line Graph 1

1 Right-click **Axial Velocity vs. Distance** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3 In the **Expression** text field, type `w`.
4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
5 In the **Expression** text field, type `(R0-a_s)/dvisc`.
6 Locate 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

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 `U0`.
4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Prescribed Axial Velocity: U₀

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.
2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3 In the **Expression** text field, type `acpr.vz`.
4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Pressure Acoustics with BLI

5 In the **Axial Velocity vs. Distance** toolbar, click **Plot**.

The figure should look like the one below. It shows the radial velocity close to the particle surface. Notice how the results from pressure acoustics, that use the thermoviscous boundary layer impedance (BLI) condition, match the full thermoviscous model except in the boundary layer itself. The BLI condition treats what happens in the layer analytically.