Created in COMSOL Multiphysics 6.2

Thermoacoustic Engine and Heat Pump

Introduction

Thermoacoustic devices are interesting systems which can generate sound waves from temperature gradients or use sound waves to create temperature gradients. The devices have no moving parts; this feature not only makes the devices look mysterious but also allow the systems to keep working without frequent maintenance. Although a thermoacoustic engine has a long history from pioneering work in the 1970s (Ref. 1), the research field is still active and developing; see, for example, Ref. 2 and Ref. 3.

There are two types of thermoacoustic systems: those that use standing waves and those that use traveling waves. In this model, a standing wave-type thermoacoustic system is simulated. The thermoacoustic system in the model consists of both a thermoacoustic engine and a thermoacoustic heat pump, and the heat pump is driven by the power generated by the engine. The **Heat Transfer in Fluids** interface is used to simulate the steady temperature field around the engine, and then the **Thermoviscous Acoustics, Transient** interface is coupled with the **Heat Transfer in Solids** interface to reproduce the pressure amplification by the engine and the cooling effect by the heat pump. In the **Thermoviscous Acoustics, Transient** interface, nonlinear terms are taken into consideration by using the **Nonlinear Thermoviscous Acoustics Contributions** feature, which is necessary for simulating the heat pump effect.

Model Definition

The model is a 2D axisymmetric thermoacoustic device, as shown in Figure 1. The geometry dimensions are listed in Table 1. The device consists of an 8 m-long tube and two sets of thermal stacks made of acrylic. In the figure, it is also noted that one of the stacks is omitted from the simulation. This is because the temperature distribution in the stack which functions as an engine is determined by an external input and can be modeled simply by giving boundary conditions on the surface. The model uses the same geometry as the pipe and stack of the Simple Thermoacoustic Engine tutorial model available in the on-line COMSOL Application Gallery, which refers to the geometry in Ref. 4.

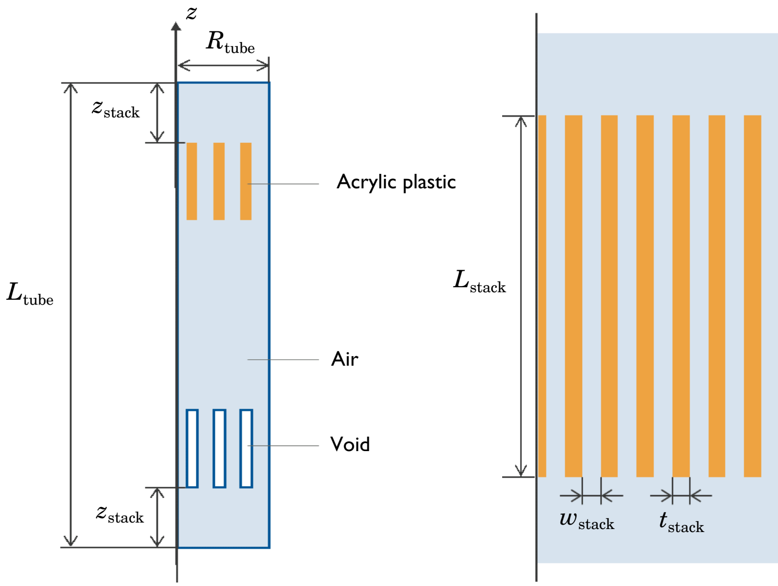


Figure 1: Sketch of the analysis domain (left) and the thermal stack (right). For sake of visibility the figures are not to scale, and some of the plates in the thermal stacks are omitted from the left figure.

The spacing between the plates of the stacks is 1.5 mm, which is roughly two and a half time the thickness of the thermal boundary layer

$$\delta_{th} = \sqrt{\frac{2\kappa}{\omega\rho C_p}} \approx 600\,\mu\text{m} \tag{1}$$

where κ, ρ, and C_p are the thermal conductivity, density, and isobaric heat capacity of the air, respectively, and ω is the angular frequency of the standing wave.

TABLE 1: GEOMETRY DIMENSIONS OF THE THERMOACOUSTIC SYSTEM.

Parameter	**Value**	**Description**
L_{tube}	8 m	Tube length
L_{stack}	140 mm	Stack length
R_{tube}	20.25 mm	Tube radius
t_{stack}	1.5 mm	Plate thickness

TABLE 1: GEOMETRY DIMENSIONS OF THE THERMOACOUSTIC SYSTEM.

Parameter	Value	Description
w_{stack}	1.5 mm	Plate spacing
z_{stack}	1.4 m	Stack placement

The relevant physical quantities are listed in Table 2. The boundary temperature of the engine stack varies by 270 K in the axial direction, which generates a temperature gradient large enough to amplify the oscillation. In the other stack, the temperature is fixed at a portion of the surface at 293.15 K, while the plates and the air are thermally coupled elsewhere. Around the heat pump stack, the **Nonlinear Thermoviscous Acoustics Contributions** node is enabled. It adds the nonlinear advection term to the governing equations, this term is important for modeling the thermoacoustic heat pump.

TABLE 2: PHYSICAL QUANTITIES OF THE THERMOACOUSTIC SYSTEM.

Parameter	VAlue	Description
p_0	1 kPa	Initial pressure amplitude
T_0	293.15 K	Ambient Temperature
T_{engine}	270 K	Temperature difference

In actual applications, the oscillation would be initiated by an external device, such as a speaker. In this model, however, the initial pressure is given a sinusoidal distribution whose wave length is half the pipe length. The upper end of the pipe is set to the maximum pressure, 1 kPa, while the minimum pressure, –1 kPa, is set at the lower end. This condition enables the model to reach a steady acoustic pressure amplitude in a few oscillations.

Results and Discussion

Figure 2 shows the instantaneous distribution of the magnitude of the acoustic velocity around the heat pump stack. Because the Thermoviscous Acoustics interface solves the transportation equation of the velocity, the flow field induced by the oscillation is resolved. It is confirmed that the air flows through the stack due to the oscillation, and the magnitude of the velocity becomes large inside the stack passages. Capturing the behavior of the acoustic velocity is important for thermoacoustic simulations, because the advection due to the oscillatory flow plays a key role in transporting heat.

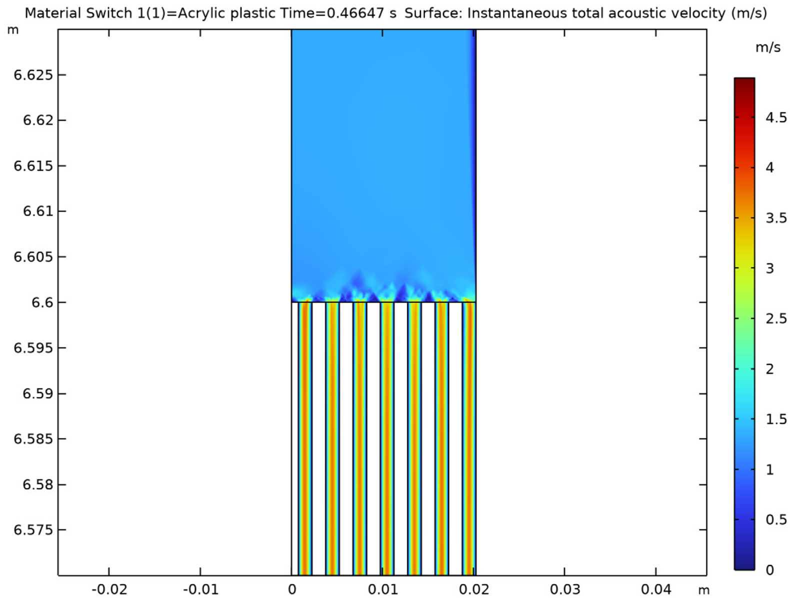


Figure 2: Instantaneous acoustic velocity around the heat pump stack which is induced by the standing wave.

In Figure 3, the pressure history at a point on the lower face of the heat pump stacks (z = 6.46 m) is plotted. The figure compares two different conditions: the blue line corresponds to the original setting, while the green line shows the result without any temperature gradient in the engine stack. It is confirmed that the temperature gradient in the engine stack is maintaining the pressure amplitude. Without the temperature gradient in the engine stack, the pressure oscillation is damped by viscous losses.

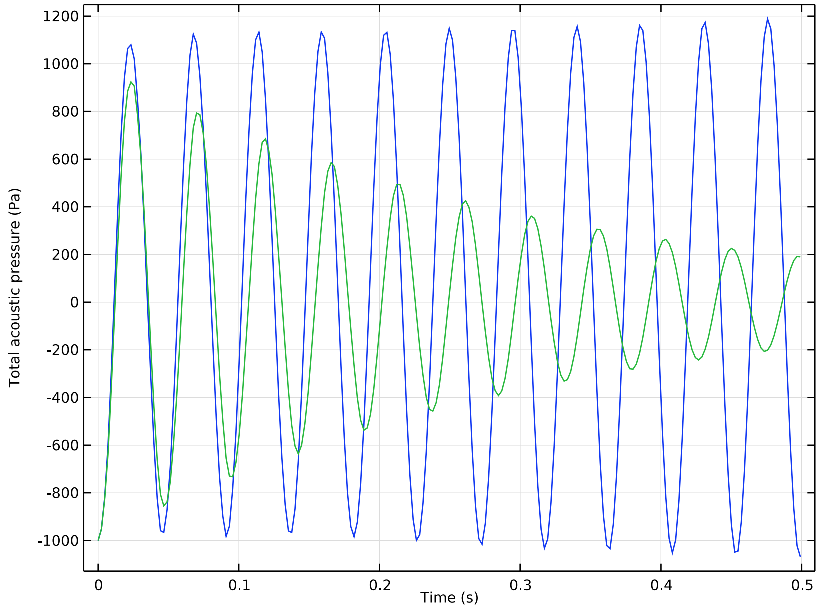


Figure 3: The pressure as a function of time at the heat pump stack for a temperature gradient in the engine stack (blue) and without the temperature gradient (green).

Figure 4 shows the temperature at the lower end of the heat pump stack (where the temperature becomes lowest). The figure also shows how the temperature is affected by the nonlinear terms. It is shown that the nonlinear terms must be considered when simulating the cooling effect by the heat pump. The nonlinear terms include a convection term responsible for the heat transport due to the coupling between the acoustic velocity field and the acoustic temperature field. Therefore, the temperature will not be decreased without the nonlinear terms.

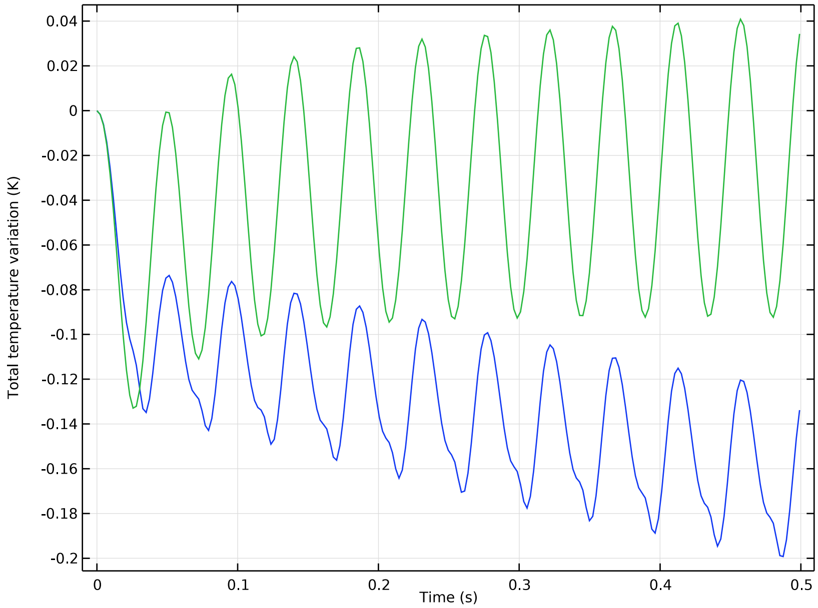


Figure 4: The temperature as a function of time at the heat pump stack with the nonlinear contributions included (blue) and without the nonlinear contribution (green).

In a thermoacoustic heat pump the material of the stack is important, especially the thermal conductivity. In Figure 5 the temperature at the cooling end of the heat stack is shown for two different stack materials, acrylic plastic and copper. The temperature decrease is largest for the acrylic plastic. If the model with acrylic plastic is run for 60 s the temperature at the heat pump is cooled down with approximately 2 K.

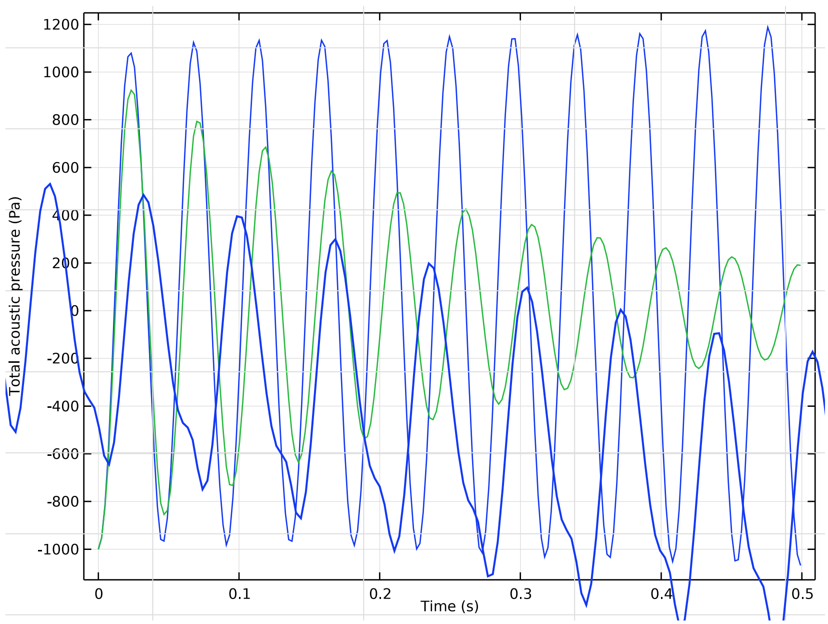


Figure 5: The temperature as a function of time at the heat pump stack for two different stack materials, copper (green) and acrylic plastic (blue).

References

1. P.H. Ceperley, "A pistonless Stirling engine — The traveling wave heat engine," *J. Acoust. Soc. Am.*, vol. 66, no. 5, pp. 1508–1513, 1979.

2. M. McGaughy, C. Wang, E. Boessneck, T. Salem, and J. Wagner, "A Traveling Wave Thermoacoustic Engine — Design and Test," *ASME Lett. Dyn. Syst. Control*, vol. 31, no. 031006, 2021.

3. Z. Bouramdane, A. Bah, M. Alaoui, and N. Martaj, "Design optimization and CFD analysis of the dynamic behavior of a standing wave thermoacoustic engine with various geometry parameters and boundary conditions," *Int. J. Air-Cond. Refrig.*, vol. 31, no. 1, 2023.

4. K. Kuzuu and S. Hasegawa, "Effect of non-linear flow behavior on heat transfer in a thermoacoustic engine core," *Int. J. Heat Mass Transfer*, vol. 108, pp. 1591–1601, 2017.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/thermoacoustic_engine_heat_pump`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Transient (tatd)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.

10 Click **Done**.

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 `thermoacoustic_engine_heat_pump_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `tube_R`.

4 In the **Height** text field, type `tube_L*2`.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `stack_w`.

4 In the **Height** text field, type `stack_L`.

5 Locate the **Position** section. In the **r** text field, type `3/2*stack_w`.

6 In the **z** text field, type `stack_z`.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

An array is used to create two stacks, one for the thermoacoustic engine and one for heat pump.

2 Select the object **r2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **r size** text field, type `6`.

5 In the **z size** text field, type `2`.

6 Locate the **Displacement** section. In the **r** text field, type `2*stack_w`.

7 In the **z** text field, type `2*tube_L-2*stack_z-stack_L`.

Rectangle 3 (r3)

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

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

3 In the **Width** text field, type `stack_w*0.5`.

4 In the **Height** text field, type `stack_L`.

5 Locate the **Position** section. In the **z** text field, type `stack_z`.

Array 2 (arr2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r3** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **z size** text field, type `2`.

5 Locate the **Displacement** section. In the **z** text field, type `2*tube_L-2*stack_z-stack_L`.

Difference 1 (dif1)

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

Subtract the heat engine stack since it is not modeled.

2 Select the object **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **arr1(1,1)**, **arr1(2,1)**, **arr1(3,1)**, **arr1(4,1)**, **arr1(5,1)**, **arr1(6,1)**, and **arr2(1, 1)** only.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **z** text field, type `2*tube_L-stack_z-stack_end`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **r** text field, type `tube_R`.

7 In the **z** text field, type `2*tube_L-stack_z-stack_end`.

8 Right-click **Line Segment 1 (ls1)** and choose **Duplicate**.

Line Segment 2 (ls2)

1 In the **Model Builder** window, click **Line Segment 2 (ls2)**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **z** text field, type `stack_z`.

4 Locate the **Endpoint** section. In the **z** text field, type `stack_z`.

5 Right-click **Line Segment 2 (ls2)** and choose **Duplicate**.

Line Segment 3 (ls3)

1 In the **Model Builder** window, click **Line Segment 3 (ls3)**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **z** text field, type `stack_z+stack_L`.

4 Locate the **Endpoint** section. In the **z** text field, type `stack_z+stack_L`.

5 Right-click **Line Segment 3 (ls3)** and choose **Duplicate**.

Line Segment 4 (ls4)

1 In the **Model Builder** window, click **Line Segment 4 (ls4)**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **z** text field, type `2*tube_L-stack_z-stack_L`.

4 Locate the **Endpoint** section. In the **z** text field, type `2*tube_L-stack_z-stack_L`.

5 Right-click **Line Segment 4 (ls4)** and choose **Duplicate**.

Line Segment 5 (ls5)

1 In the **Model Builder** window, click **Line Segment 5 (ls5)**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **z** text field, type `2*tube_L-stack_z`.

4 Locate the **Endpoint** section. In the **z** text field, type `2*tube_L-stack_z`.

5 Right-click **Line Segment 5 (ls5)** and choose **Duplicate**.

Line Segment 6 (ls6)

1 In the **Model Builder** window, click **Line Segment 6 (ls6)**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **z** text field, type `2*tube_L-stack_z+stack_L`.

4 Locate the **Endpoint** section. In the **z** text field, type `2*tube_L-stack_z+stack_L`.

5 Right-click **Line Segment 6 (ls6)** and choose **Duplicate**.

Line Segment 7 (ls7)

1 In the **Model Builder** window, click **Line Segment 7 (ls7)**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **z** text field, type `2*tube_L-stack_z-2*stack_L`.

4 Locate the **Endpoint** section. In the **z** text field, type `2*tube_L-stack_z-2*stack_L`.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

The geometry is parameterized so that it can be changed by changing the parameters.

DEFINITIONS

Two selections are made for the air and stack domains.

Heat stack

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Heat stack` in the **Label** text field.

3 Select Domains 4, 5, 11, 12, 16, 17, 21, 22, 26, 27, 31, 32, 36, and 37 only.

Air

1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 In the **Label** text field, type `Air`.

5 In the **Add** dialog box, select **Heat stack** in the **Selections to invert** list.

6 Click **OK**.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in>Air**.

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

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Air**.

Add a material switch to easy switch between acrylic plastic and copper for the heat pump stack.

Material Switch 1 (sw1)

In the **Model Builder** window, right-click **Materials** and choose **More Materials>Material Switch**.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in>Acrylic plastic** and **Built-in>Copper**.

3 Click the right end of the **Add to Component** split button in the window toolbar.

4 From the menu, choose **Add to Material Switch 1 (sw1)**.

MATERIALS

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material Switch 1 (sw1)**.

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

3 From the **Selection** list, choose **Heat stack**.

GLOBAL DEFINITIONS

Ramp 1 (rm1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Ramp**.

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

3 In the **Location** text field, type `stack_z+stack_end`.

4 In the **Slope** text field, type `-1/(stack_L-2*stack_end)`.

5 Select the **Cutoff** check box. In the associated text field, type `-1`.

6 Click to expand the **Smoothing** section.

7 Select the **Size of transition zone at start** check box. In the associated text field, type `0.02`.

8 Select the **Size of transition zone at cutoff** check box. In the associated text field, type `0.02`.

Tstack

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

The function describes the temperature distribution in the thermoacoustic engine.

2 In the **Settings** window for **Analytic**, type `Tstack` in the **Label** text field.

3 In the **Function name** text field, type `Tstack`.

4 Locate the **Definition** section. In the **Expression** text field, type `(1+rm1(z/1[m]))*DeltaT`.

5 In the **Arguments** text field, type `z`.

6 Locate the **Units** section. In the **Function** text field, type `K`.

7 In the table, enter the following settings:

Argument	**Unit**
z	m

p_initial

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

The function prescribes the initial condition of the pressure. A half wave resonance node in the length of the thermoacoustic system.

2 In the **Settings** window for **Analytic**, type `p_initial` in the **Label** text field.

3 In the **Function name** text field, type `p_initial`.

4 Locate the **Definition** section. In the **Expression** text field, type `sin((z-tube_L)/(2*tube_L)*pi)`.

5 In the **Arguments** text field, type `z`.

6 Locate the **Units** section. In the table, enter the following settings:

Argument	**Unit**
z	m

HEAT TRANSFER IN SOLIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Heat stack**.

Temperature 1

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

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

3 In the T_0 text field, type `T0`.

4 Select Boundaries 13, 22, 30, 32, 38, 46, 48, 54, 62, 64, 70, 78, 80, 86, 94, 96, 102, 110, 112, and 118 only.

HEAT TRANSFER IN FLUIDS 2 (HT2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids 2 (ht2)**.

2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

4 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids 2 (ht2)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the $T2$ text field, type `T0`.

Temperature - Engine stack

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

2 Select Boundaries 5, 17, 18, 25, 27, 33, 41–43, 49, 57–59, 65, 73–75, 81, 89–91, 97, 105–107, and 113 only.

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

4 In the T_0 text field, type `T0+Tstack(z)`.

5 In the **Label** text field, type `Temperature - Engine stack`.

Temperature - Outer and heat pump stack

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

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

3 In the T_0 text field, type `T0`.

4 Select Boundaries 2, 9, 13, 16, 20, 22, 28–30, 32, 36, 38, 44–46, 48, 52, 54, 60–62, 64, 68, 70, 76–78, 80, 84, 86, 92–94, 96, 100, 102, 108–110, 112, 116, 118, and 121–128 only.

5 In the **Label** text field, type `Temperature - Outer and heat pump stack`.

THERMOVISCOUS ACOUSTICS, TRANSIENT (TATD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Transient (tatd)**.

2 In the **Settings** window for **Thermoviscous Acoustics, Transient**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

Linear shape functions are used for modeling the acoustic fields.

4 Click to expand the **Discretization** section. From the **Element order for velocity** list, choose **Linear**.

5 From the **Element order for temperature** list, choose **Linear**.

6 Locate the **Transient Solver and Mesh Settings** section. In the f_{max} text field, type `f0`.

When using linear shape functions it is necessary to use stabilization. In this case the SUPG stabilization is used.

7 Click the **Show More Options** button in the **Model Builder** toolbar.

8 In the **Show More Options** dialog box, select **Physics>Stabilization** in the tree.

9 In the tree, select the check box for the node **Physics>Stabilization**.

10 Click **OK**.

11 In the **Settings** window for **Thermoviscous Acoustics, Transient**, click to expand the **Stabilization** section.

12 From the **Stabilization method** list, choose **Streamline upwind Petrov–Galerkin (SUPG) stabilization**.

Thermoviscous Acoustics Model 1

1 In the **Model Builder** window, expand the **Thermoviscous Acoustics, Transient (tatd)** node, then click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Model Input** section.

3 From the T_0 list, choose **Temperature (ht2)**.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `p0*p_initial(z)`.

Wall - Adiabatic

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Transient (tatd)** click **Wall 1**.

2 In the **Settings** window for **Wall**, type `Wall - Adiabatic` in the **Label** text field.

3 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

Nonlinear Thermoviscous Acoustics Contributions 1

1 In the **Physics** toolbar, click **Domains** and choose **Nonlinear Thermoviscous Acoustics Contributions**.

Include the nonlinear contributions near the thermoacoustic heat pump where they are important.

2 Select Domains 3, 6, 9, 10, 14, 15, 19, 20, 24, 25, 29, 30, 34, 35, 39, and 40 only.

3 In the **Settings** window for **Nonlinear Thermoviscous Acoustics Contributions**, locate the **Model Input** section.

4 From the T_0 list, choose **Temperature (ht2)**.

Wall - Isothermal

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

2 Select Boundaries 3, 5, 13, 17, 22, 25–27, 30, 32, 33, 38, 41–43, 46, 48, 49, 54, 57–59, 62, 64, 65, 70, 73–75, 78, 80, 81, 86, 89–91, 94, 96, 97, 102, 105–107, 110, 112, 113, and 118 only.

3 In the **Settings** window for **Wall**, type `Wall - Isothermal` in the **Label** text field.

No Slip 1

1 In the **Physics** toolbar, click **Boundaries** and choose **No Slip**.

2 Select Boundaries 9, 20, 28, 29, 36, 44, 45, 52, 60, 61, 68, 76, 77, 84, 92, 93, 100, 108, 109, and 116 only.

MULTIPHYSICS

Thermoviscous Acoustic–Thermal Perturbation Boundary 1 (tatpb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Thermoviscous Acoustic–Thermal Perturbation Boundary**.

2 Select Boundaries 9, 20, 28, 29, 36, 44, 45, 52, 60, 61, 68, 76, 77, 84, 92, 93, 100, 108, 109, and 116 only.

The coupling feature couples the temperature fields in the air and heat pump stack.

MESH 1

Mapped - Heat engine

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, type `Mapped - Heat engine` in the **Label** text field.

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

4 Select Domains 8, 13, 18, 23, 28, 33, and 38 only.

Distribution 1

1 Right-click **Mapped - Heat engine** and choose **Distribution**.

2 Select Boundaries 19, 35, 51, 67, 83, 99, and 115 only.

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

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped - Heat engine** and choose **Distribution**.
2 Select Boundaries 17, 25, 33, 41, 49, 57, 65, 73, 81, 89, 97, 105, 113, and 122 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `15`.

Mapped - Heat pump

1 In the **Mesh** toolbar, click **Mapped**.
2 In the **Settings** window for **Mapped**, type `Mapped - Heat pump` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 4, 5, 9–12, 14–17, 19–22, 24–27, 29–32, 34–37, 39, and 40 only.

Distribution 1

1 Right-click **Mapped - Heat pump** and choose **Distribution**.
2 Select Boundaries 24, 32, 40, 48, 56, 64, 72, 80, 88, 96, 104, 112, and 120 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `10`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped - Heat pump** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `4`.
4 Select Boundaries 22, 30, 38, 46, 54, 62, 70, 78, 86, 94, 102, 110, and 118 only.

Distribution 3

1 Right-click **Mapped - Heat pump** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `11`.
4 Select Boundaries 20, 28, 36, 44, 52, 60, 68, 76, 84, 92, 100, 108, 116, and 125 only.

Distribution 4

1 Right-click **Mapped - Heat pump** and choose **Distribution**.
2 Select Boundary 13 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `2`.

Boundary Layers 1

1. In the **Mesh** toolbar, click **Boundary Layers**.
2. In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.
3. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1. In the **Model Builder** window, click **Boundary Layer Properties**.
2. Select Boundaries 121, 123, 124, 127, and 128 only.
3. Select Boundaries 3, 5, 9, 13, 18, 19, 21, 24, 26, 27, 29, 32, 34, 35, 37, 40, 42, 43, 45, 48, 50, 51, 53, 56, 58, 59, 61, 64, 66, 67, 69, 72, 74, 75, 77, 80, 82, 83, 85, 88, 90, 91, 93, 96, 98, 99, 101, 104, 106, 107, 109, 112, 114, 115, 117, 120, 121, 123, 124, 127, and 128 only.
4. In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
5. In the **Number of layers** text field, type `3`.

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 `tube_R`.
5. Click **Build All**.

STUDY 1 - STEADY TEMPERATURE FIELD

In study 1 the stationary temperature field induced by the engine stack is modeled.

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1 - Steady Temperature Field` in the **Label** text field.

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1 - Steady Temperature Field** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for **Heat Transfer in Solids (ht)**.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

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

STUDY 2 - TIME DEPENDENT ACOUSTICS - STACK MATERIAL

In study 2 the acoustics is modeled for the two different stack materials.

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Time Dependent Acoustics - Stack Material` in the **Label** text field.

Material Sweep

1 In the **Study** toolbar, click **Material Sweep**.

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

3 Click **Add**.

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat Transfer in Fluids 2 (ht2)**.

4 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,t0/20, 0.5)`.

5 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 1 - Steady Temperature Field, Stationary**.

ADD STUDY

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

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

In study 3 we model the system with the thermoacoustic engine turned off.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>** **Time Dependent**.

4 Click **Add Study** in the window toolbar.

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

STUDY 3 - TIME DEPENDENT ACOUSTICS - ENGINE OFF

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - Time Dependent Acoustics - Engine off` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

1 In the **Model Builder** window, under **Study 3 - Time Dependent Acoustics - Engine off** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat Transfer in Fluids 2 (ht2)**.

4 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,t0/20, 0.5)`.

ADD STUDY

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

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

In study 4 we model the system without including the nonlinear contributions.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>** **Time Dependent**.

4 Click **Add Study** in the window toolbar.

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

STUDY 4 - TIME DEPENDENT ACOUSTICS - NO NONLINEAR CONTRIBUTIONS

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, type `Study 4 - Time Dependent Acoustics - No Nonlinear Contributions` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

1 In the **Model Builder** window, under **Study 4 - Time Dependent Acoustics - No Nonlinear Contributions** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat Transfer in Fluids 2 (ht2)**.

4 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,t0/20,0.5)`.

5 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 1 - Steady Temperature Field, Stationary**.

8 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

9 In the tree, select **Component 1 (comp1)>Thermoviscous Acoustics, Transient (tatd)>Nonlinear Thermoviscous Acoustics Contributions 1**.

10 Right-click and choose **Disable**.

STUDY 1 - STEADY TEMPERATURE FIELD

In the **Study** toolbar, click **Compute**.

STUDY 2 - TIME DEPENDENT ACOUSTICS - STACK MATERIAL

Click **Compute**.

STUDY 3 - TIME DEPENDENT ACOUSTICS - ENGINE OFF

Click **Compute**.

STUDY 4 - TIME DEPENDENT ACOUSTICS - NO NONLINEAR CONTRIBUTIONS

Click **Compute**.

RESULTS

Pressure - Engine on/off

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

2 In the **Settings** window for **1D Plot Group**, type `Pressure - Engine on/off` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Time Dependent Acoustics - Stack Material/Parametric Solutions 1 (sol3)**.

4 From the **Material Switch 1** list, choose **First**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Point Graph 1

1 Right-click **Pressure - Engine on/off** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `tatd.p_t`.

Point Graph 2

1 In the **Model Builder** window, right-click **Pressure - Engine on/off** and choose **Point Graph**.

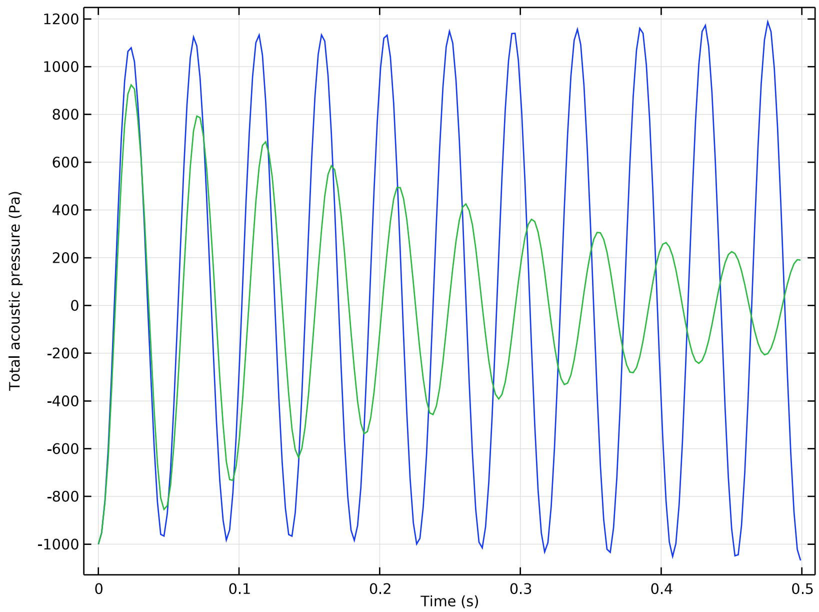
2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3 - Time Dependent Acoustics - Engine off/ Solution 6 (sol6)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `tatd.p_t`.

5 Select Point 1 only.

6 In the **Pressure - Engine on/off** toolbar, click **Plot**.



Temperature - Nonlinear Contributions On/Off

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

2 In the **Settings** window for **1D Plot Group**, type `Temperature - Nonlinear Contributions On/Off` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Time Dependent Acoustics - Stack Material/Parametric Solutions 1 (sol3)**.

4 From the **Material Switch 1** list, choose **First**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Point Graph 1

1 Right-click **Temperature - Nonlinear Contributions On/Off** and choose **Point Graph**.

2 Select Point 12 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `tatd.T_t`.

Point Graph 2

1 In the **Model Builder** window, right-click **Temperature - Nonlinear Contributions On/Off** and choose **Point Graph**.

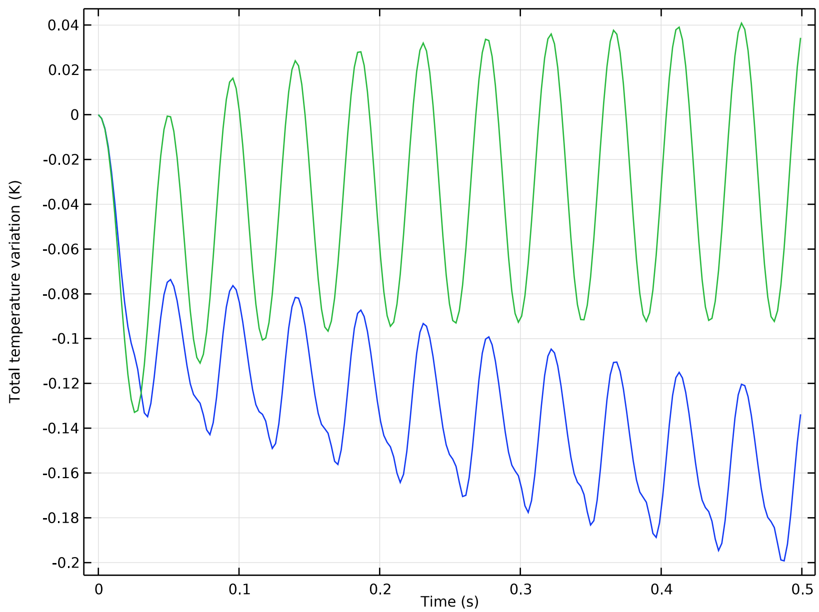
2 Select Point 12 only.

3 In the **Settings** window for **Point Graph**, locate the **Data** section.

4 From the **Dataset** list, choose **Study 4 - Time Dependent Acoustics - No Nonlinear Contributions/Solution 7 (sol7)**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `tatd.T_t`.

6 In the **Temperature - Nonlinear Contributions On/Off** toolbar, click **Plot**.



Temperature - Stack Material

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

2 In the **Settings** window for **1D Plot Group**, type `Temperature - Stack Material` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Time Dependent Acoustics - Stack Material/Parametric Solutions 1 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Point Graph 1

1 Right-click **Temperature - Stack Material** and choose **Point Graph**.

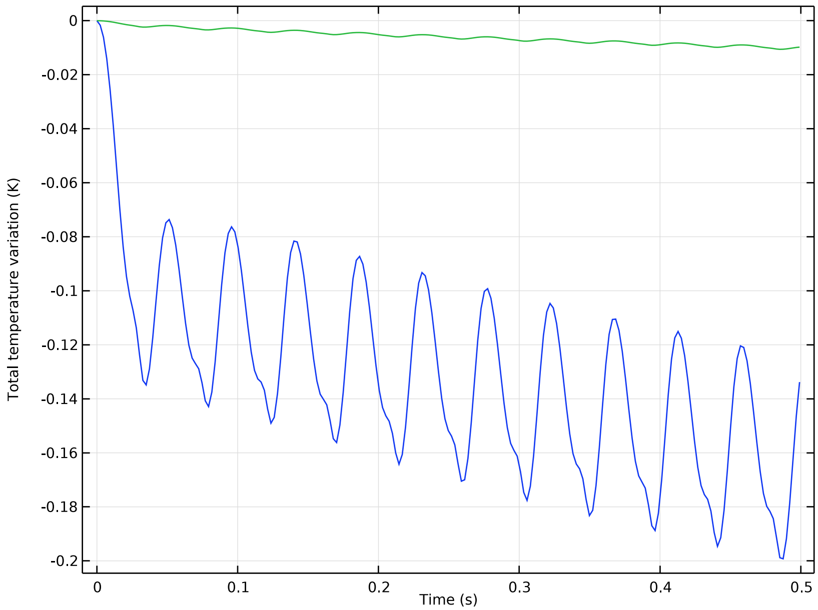
2 Select Point 12 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `tatd.T_t`.

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Time**.

6 In the **Temperature - Stack Material** toolbar, click **Plot**.



Acoustic Velocity (tatd)

1 In the **Model Builder** window, under **Results** click **Acoustic Velocity (tatd)**.

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

3 From the **Material Switch 1** list, choose **Acrylic plastic**.

4 In the **Acoustic Velocity (tatd)** toolbar, click **Plot**.

5 From the **Time (s)** list, choose **0.46647**.

6 In the **Acoustic Velocity (tatd)** toolbar, click **Plot**.