Pressure Reciprocity Calibration Coupler with Detailed Moist Air Material Properties

Introduction

When high-fidelity measurement microphones are calibrated, a pressure reciprocity calibration method is used. During calibration, two microphones are connected at each end of a closed cylindrical cavity. For the calibration procedure, it is important to understand the acoustic field inside such a cavity, including all the thermoviscous acoustic effects, for example, the acoustic boundary layers at higher frequencies and the transition to isothermal behavior at the lower frequencies.

This model sets up a simple calibration coupler model and discusses important considerations when performing a high-precision absolute-value simulation. The model results include the acoustic transfer impedance used for reciprocity calibration and the pressure in the coupler. The results are compared with analytical predictions.

The model also includes precise material property estimation using the Thermodynamics functionality in COMSOL Multiphysics. This allows setting up a moist air material that depends on the ambient pressure, temperature, and relative humidity. This functionality is available with the *Liquid & Gas Properties Module*. The model can also be set up using the default air material found in the *Material Library*, but this will represent dry air.

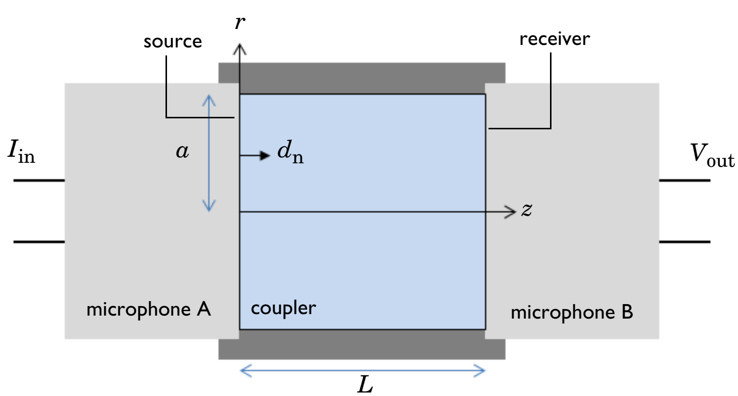


Figure 1: Sketch of a typical pressure reciprocity calibration coupler with two microphones. The cylindrical rz-coordinate system used in the model is also indicated, as well as the source and receiver boundaries.

Model Definition

In order to calibrate microphones a commonly used technique is the reciprocity calibration approach. The method relies on using microphones both as source and receivers (they are reciprocal) in a well described coupled setup (see Ref. 1 and 2). The setup consisting of two microphones and a coupler volume, is sketched in Figure 1. The coupler has a length L and a radius a. Both are parameters that can be changed in the COMSOL model to fit the dimensions of various standardized couplers. A 2D axisymmetric representation is used on the COMSOL model.

The ratio between the input current I_{in} and the output voltage V_{out} is the electric transfer impedance Z_{e} (this quantity is measured). The acoustic transfer impedance Z_{a} of the coupler is the ratio between the average pressure at the receiver boundary $p_{\mathrm{av,r}}$ and the volume velocity at the source boundary Q_{s}. The product of the microphone sensitivity M of microphone A and microphone B is given as

$$M_{\mathrm{A}}M_{\mathrm{B}} = \frac{Z_{\mathrm{e}}}{Z_a} \qquad Z_e = \frac{V_{\mathrm{out}}}{I_{\mathrm{in}}} \qquad Z_{\mathrm{a}} = \frac{p_{\mathrm{av,r}}}{Q_{\mathrm{s}}}$$

Using three microphones (A, B, and C), in all combinations of source and receiver, results in a system of three equations with the three sensitivities as unknowns. This system is solved to produce the calibrated microphone sensitivities M_{A}, M_{B}, and M_{C}. In this procedure, it is assumed that the acoustic transfer impedance Z_{a} of the coupler volume is known. To achieve high precision calibration the acoustic transfer impedance has to be well defined for a large frequency. Material properties of air also need to be well described.

The calibration procedure, acoustic transfer impedance, and material properties are described by the international standard, IEC 61094-2:2009, Ref. 3. As an alternative to the IEC standard, you can choose to model the acoustic transfer impedance in full detail using the **Thermoviscous Acoustics, Frequency Domain** physics interface. Detailed moist air material properties can be computed using the **Thermodynamics** feature. This will yield an acoustic transfer impedance that is correct at all frequencies. The simulation only assumes linear acoustics, while modeling all thermoviscous effects in details. That is, the dynamics of the thermal and viscous boundary layers, as well as the complex transition from adiabatic to isothermal behavior for decreasing frequencies. In the limit of large couplers and high frequencies correct wave propagation behavior is also captured.

The acoustic transfer impedance is computed in this model and compared to three different analytical models, as described below.

ISOTHERMAL LIMIT, VERY LOW FREQUENCY

At very low frequencies the system behaves isothermally (the thermal boundary layers fill the entire coupler volume). The pressure change dp, in the volume V, for a given volume change dV, follows from the definition of isothermal compressibility

$$\beta_T = \frac{1}{\rho}\frac{d\rho}{dp}\bigg|_T \qquad d\rho = -\rho\frac{dV}{V}$$

$$dp = -\frac{dV}{\beta_T V}$$

The variables for this expression are located under **Definitions > Variables - Isothermal Limit (very low frequency)**.

TRANSMISSION LINE, HIGH FREQUENCY

In the high frequency limit, disregarding boundary layer effects on the source and receiver boundaries, the coupler can be approximated with a transmission line defined by a transfer matrix $\mathbf{T}$. The model takes the boundary layer losses at the coupler walls into account through a complex-valued characteristic specific impedance and propagation constant. Details and an expression for the transfer matrix is described in the tutorial *Wax Guard Acoustics: Transfer Matrix Computation*. The acoustic transfer impedance (when the receiver is sound hard) is the simply defined as $Z_a = 1/T_{21}$. and the admittance is directly given by T_{21}.

The variables for this analytical model are located under **Definitions > Transmission Line (high frequency)**.

MODEL BY VINCENT ET AL., LOW FREQUENCY

More advanced analytical models exist for computing the transfer impedance of a cylinder, including all viscous and thermal effects. In a recent work by Vincent and others (see Ref. 5), the work of Gerber (see Ref. 4) is extended to be valid in the full low frequency range (from medium all the way to very low frequencies and the isothermal limit). The pressure in the coupler and the transfer impedance are given by equations 24, 26, 27, and 30 in Ref. 5. Note that the source and receiver admittances are 0 (perfect source and sound hard wall).

One of the analytical expressions include an infinite sum. In COMSOL, the sum is implemented using the `sum()` operator, with 10 terms in the sum. The analytical model also includes zeros of the Bessel function, the first 10 used are tabulated in the interpolation function `lam_n()` defined under **Definitions**.

The variables for this analytical model are located under **Definitions>Vincent et al. (low frequency)**.

Results and Discussion

The root mean square (RMS) velocity in the coupler and the temperature fluctuations, for four evaluation frequencies of 0.1 Hz, 1 Hz, 10 Hz, and 100 Hz, is depicted in Figure 2 and Figure 3, respectively. The dependency of the viscous and thermal boundary layers on frequency and the extent into the geometry, can be seen graphically. In the model, the coupler has a length L of 5 mm and a radius a of 4.5 mm.

The real and imaginary part of the pressure in the coupler is depicted in Figure 4 and Figure 5, respectively. The graphs show the expected correlation between the COMSOL model and the analytical models. The isothermal limit represents the asymptotic behavior for the frequency going to 0. The transmission line model shows the best agreement on the high frequency limit, without giving a perfect match. The model by Vincent and others agrees well with the COMSOL model (within 0.01 dB) on the real part below about 150 Hz and for the imaginary part below about 5 Hz.

The real and imaginary part of the acoustic transfer impedance Z_a, is plotted as function of frequency in Figure 6. The graph again shows good correlation between the model by Vincent *et al.* for frequencies below 100 Hz. Having consistent and correct values of the acoustic transfer impedance is essential for high fidelity calibration.

In this tutorial, the moist air material generated using the **Predefined System** option for **Moist air** of the **Thermodynamics** feature is used. This functionality requires the *Liquid & Gas Properties Module*. Once the mixture is set up using the steps of the wizard a moist air material is generated using the **Generate Material** option. Note that the content of the various species (mole fractions) that make up air, can be modified in the **Local properties** table inside the **Materials > Gas: Moist Air 1** material. This allows to model any air variant.

As an example, the dependency on the ambient temperature of the density, the dynamic viscosity, the thermal conductivity, and the speed of sound is depicted in Figure 7. The graphs shows the values for four values of the relative humidity. This dependency is automatically included in the COMSOL model results as **Relative humidity** ϕ_w, is a **Model Input** when the moist air material is used.

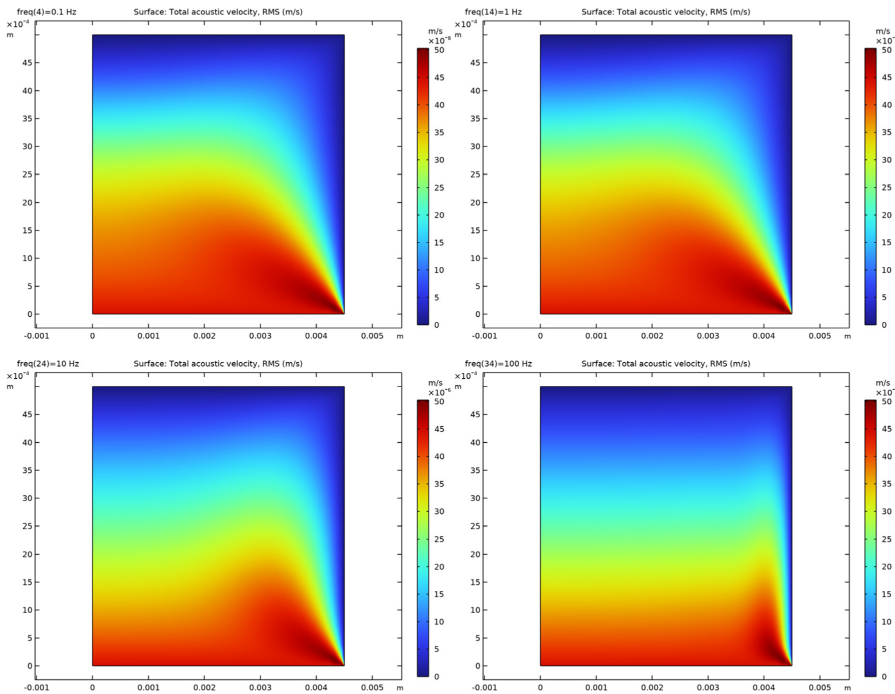


Figure 2: Acoustic velocity fluctuations (RMS) in the coupler volume at 0.1, 1, 10, and 100 Hz.

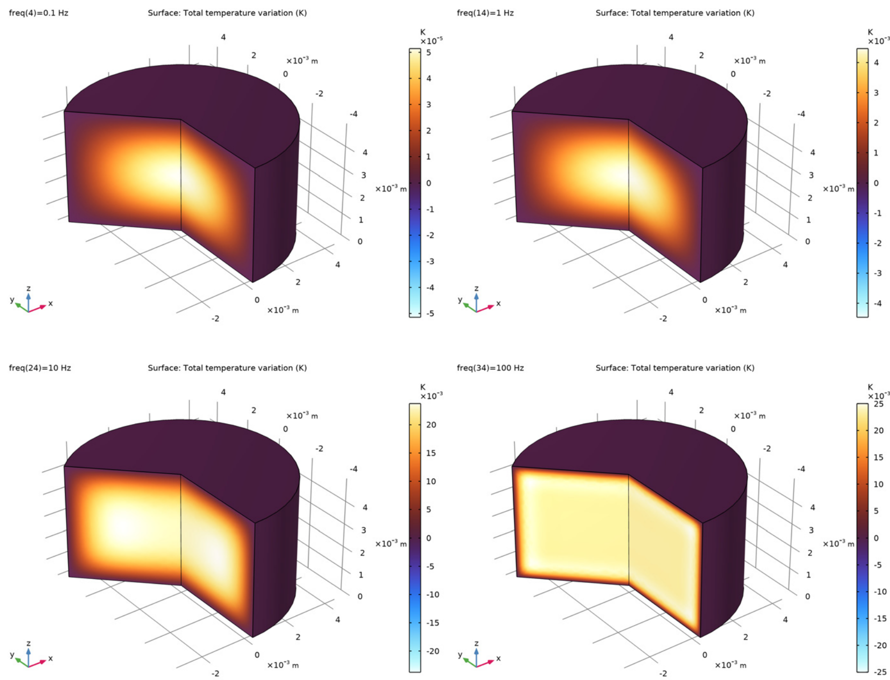


Figure 3: Acoustic temperature fluctuations in the coupler at 0.1, 1, 10, and 100 Hz.

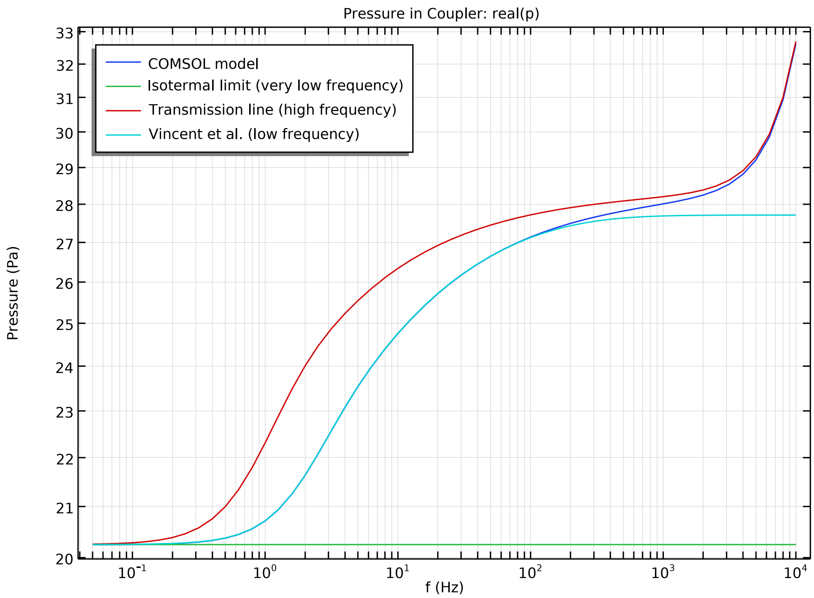


Figure 4: Real part of the pressure in the coupler evaluated using the COMSOL model and the three analytical approximations.

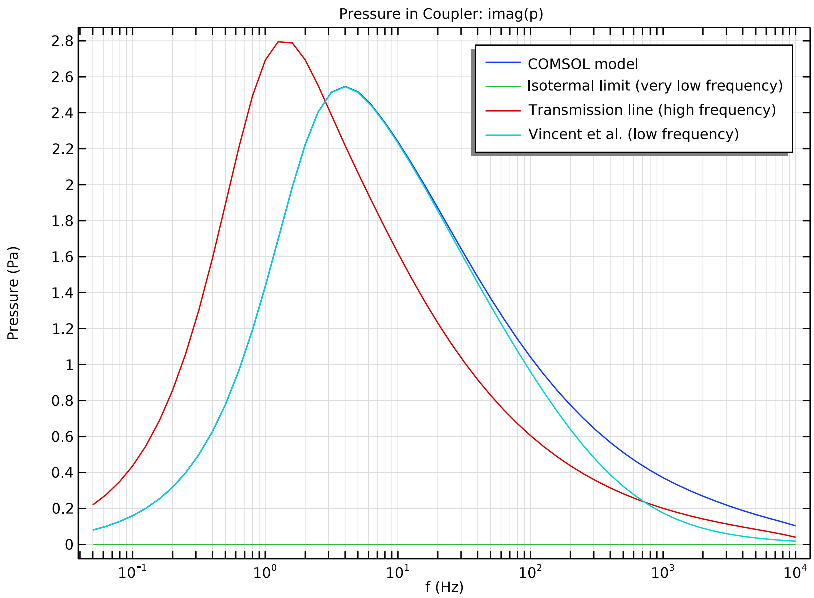


Figure 5: Imaginary part of the pressure in the coupler evaluated using the COMSOL model and the three analytical approximations.

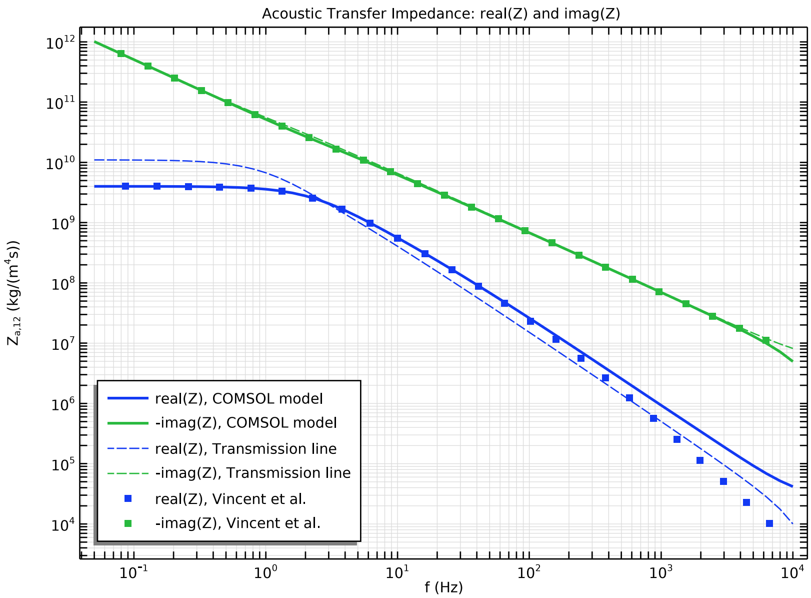


Figure 6: Acoustic transfer impedance for the coupler volume evaluated using the COMSOL model as well as the low and high frequency models.

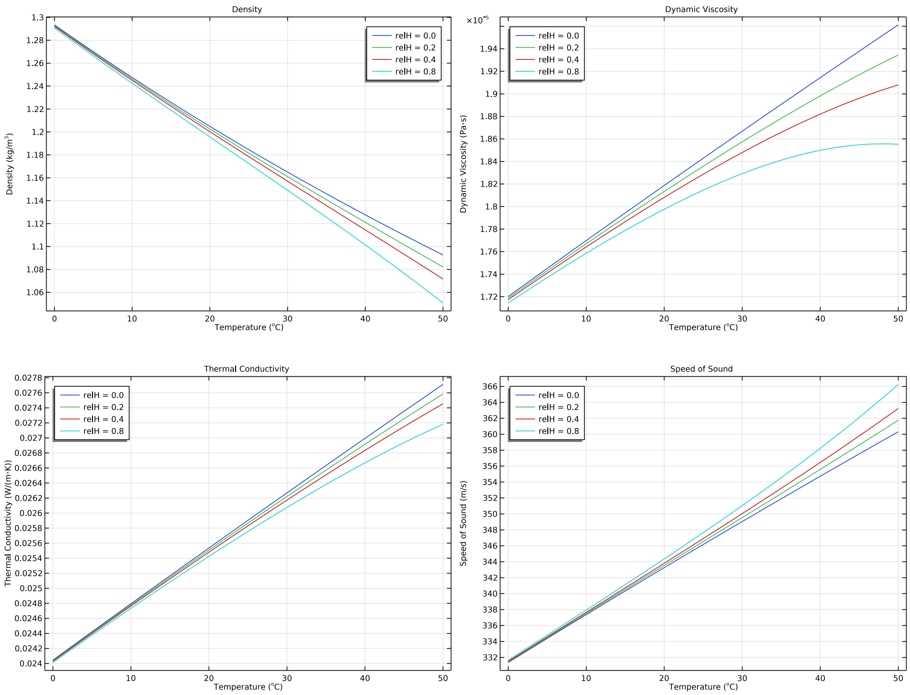


Figure 7: Various material properties of moist air (density, dynamic viscosity, thermal conductivity, and speed of sound) evaluated as function of temperature (from 0 to 50 deg C) for four different values of the relative humidity (0.0, 0.2, 0.4, and 0.8).

Notes About the COMSOL Implementation

BULK LOSSES

When modeling fluids with the thermoviscous acoustics interface, the damping and dissipation due to the thermal and viscous boundary layers is included in full detail. In the bulk/volume of the fluid (away from the boundaries) the model captures the attenuation that corresponds to the classical thermoviscous attenuation α_{tv}. This is not the correct bulk attenuation as it does not include losses due to relaxation. These losses that are, for example, captured when using the **Atmosphere** attenuation model in pressure acoustics. In most microacoustic applications the boundary layer losses are orders of magnitude higher than the bulk losses, so it is not important. In the present coupler application, the bulk losses may start to play a small role at very high frequencies. In this case, the true bulk losses may be included by defining a frequency dependent bulk viscosity $\mu_B = \mu_B(f)$, according to

$$\mu_B(f) = \frac{2\rho c^3}{\omega^2}\alpha(T_0, p_0, \phi_w) - \mu\left(\frac{4}{3} + \frac{\gamma - 1}{\Pr}\right)$$

where $\omega = 2\pi f$ and α is the true atmosphere attenuation. This will introduce consistent bulk/volume losses that also include relaxation processes and other mechanisms included in the expression for α.

MEMBRANE DEFORMATION

In this model, plane wave propagation is assumed. For certain larger couplers, the effects of the true membrane deformation can start to play a role a high frequencies. In this case, the membrane can be included in the model. See for example *The Brüel & Kjær 4134 Condenser Microphone* tutorial.

MESH

The mesh used in this model uses two boundary layer meshes. The first resolves the physics by resolving the thermal and viscous boundary layer thickness (also known as the penetration depth). The second adds a small single layer that is used to resolve the numerical singularities at the corners of the geometry.

References

1. Danish Primary Laboratory of Acoustic Microphone Reciprocity Calibration Calculation Program for Reciprocity Calibration, Technical Review No. 1-1998, Brüel & Kjær, Denmark.

2. E. Frederiksen, “Acoustic metrology - an overview of calibration methods and their uncertainty,” *Int. J. Metrol. Qual. Eng.*, vol. 4, pp. 97–107, 2013.

3. International Standard, *Electroacoustics - Measurement microphones - Part 2: Primary method for pressure calibration of laboratory standard microphones by the reciprocity technique*, IEC 61094-2:2009.

4. H. Gerber, “Acoustic Properties of Flui.Filled Chambers at Infrasonic Frequencies in the Absence of Convection,” *J. Acoust. Soc. Am.*, vol. 36, p. 1427, 1964.

5. P. Vincent, D. Rodrigues, F. Larsonnier, C. Guianvarc’h, and S. Durand, “Acoustic transfer admittance of cylindrical cavities in infrasonic frequency range,” *Meterologica*, vol. 56, p. 015003, 2019.

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

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**.

GLOBAL DEFINITIONS

Parameters - Physics

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_parameters_physics.txt`.

Parameters - Geometry

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

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

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_parameters_geometry.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 `a`.

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

5 Click **Build All Objects**.

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>Air**.

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

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

MATERIALS

Air (dry)

1 In the **Settings** window for **Material**, type `Air (dry)` in the **Label** text field.

You just added the default air material from the **Material Library**. This is a model of dry air where certain properties like density and speed of sound are based on the ideal gas law. A detailed moist air material can be conveniently set up using a **Predefined System** defined by the **Thermodynamics** feature.

Setting up the moist air material requires the Liquid and Gas Properties Module. If you do not have that product, you can skip the next steps and use the **Air (dry)** material instead.

GLOBAL DEFINITIONS

In the **Physics** toolbar, click **Thermodynamics** and choose **Predefined System**.

SELECT SYSTEM

1 Go to the **Select System** window.

2 From the **Predefined thermodynamic system** list, choose **Moist air**.

3 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Next** in the window toolbar.

SELECT THERMODYNAMIC MODEL

1 Go to the **Select Thermodynamic Model** window.

2 From the **Gas phase model** list, choose **Peng-Robinson**.

The choice of **Gas phase model** depends on the exterior conditions like ambient pressure and temperature. The different models are described in the Liquid and Gas Properties Module User's Guide in the Thermodynamics Models section.

3 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Moist Air 1 (pp1)

In the **Model Builder** window, under **Global Definitions>Thermodynamics** right-click **Moist Air 1 (pp1)** and choose **Generate Material**.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Next** in the window toolbar.

SELECT PROPERTIES

1 Go to the **Select Properties** window.

2 Click **Next** in the window toolbar.

DEFINE MATERIAL

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

2 Click **Finish** in the window toolbar.

Note that the bulk viscosity is defined in terms of the dynamic viscosity. The relation is the same as in the default (dry) air material. Values of the bulk viscosity are experimentally obtained using high-frequency absorption techniques. More details can be found in the Acoustic Properties of Fluids chapter of the Acoustics Module User's Guide.

Use the moist air material for the model.

MATERIALS

Gas: Moist Air 1 (pp1mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Gas: Moist Air 1 (pp1mat1)**.

2 Select Domain 1 only.

DEFINITIONS

Variables - Material Properties

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, type `Variables - Material Properties` in the **Label** text field.

4 Locate the **Variables** section. Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_variables_material.txt`.

Variables - Isotermal Limit (very low frequency)

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables - Isotermal Limit (very low frequency)` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_variables_isothermal.txt`.

Variables - Transmission Line (high frequency)

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables - Transmission Line (high frequency)` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_variables_transmission.txt`.

Variables - Vincent et al. (low frequency)

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables - Vincent et al. (low frequency)` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_variables_vincent.txt`.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_s` in the **Operator name** text field.

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

4 Select Boundary 2 only.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_r` in the **Operator name** text field.

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

4 Select Boundary 3 only.

Integration 3 (intop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_pnt` in the **Operator name** text field.

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

4 Select Point 4 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Interpolation 1 (int1)

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `pressure_reciprocity_calibration_coupler_bessel_zeros.txt`.

6 Click **Import**.

7 In the **Function name** text field, type `lam_n`.

8 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Nearest neighbor**.

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

Argument	Unit
t	1

10 In the **Function** table, enter the following settings:

Function	Unit
lam_n	1

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

12 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

13 Click **OK**.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics Model 1

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

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

3 In the ϕ_w text field, type `relH`.

Because the relative humidity is set up as a **Model Input** for the moist air material, it automatically appears as an input in the physics.

Velocity 1

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

2 Select Boundary 2 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 `ta.iomega*dn`.

7 Click to expand the **Excluded Points** section. Select Point 3 only.

Isothermal 1

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

2 Select Boundary 2 only.

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh component.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, 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 `a/10`.
5 In the **Minimum element size** text field, type `dvisc0`.

Size 1

1 In the **Model Builder** window, 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 2–4 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7 In the associated text field, type `dvisc0`.

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 Boundary 4 only.
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4 From the **Thickness specification** list, choose **First layer**.
5 In the **Thickness** text field, type `0.2*dvisc`.

Boundary Layers 2

1 In the **Mesh** toolbar, click **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, locate 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 2 and 3 only.
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4 From the **Thickness specification** list, choose **First layer**.
5 In the **Thickness** text field, type `0.2*dvisc`.

Boundary Layers 3

1 In the **Mesh** toolbar, click **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, locate 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 Boundary 4 only.
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4 In the **Number of layers** text field, type `1`.
5 From the **Thickness specification** list, choose **First layer**.
6 In the **Thickness** text field, type `2e-6`.

Boundary Layers 4

1 In the **Mesh** toolbar, click **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, locate 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 2 and 3 only.
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4 In the **Number of layers** text field, type `1`.
5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `2e-6`.

7 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 Click **Range**.

4 In the **Range** dialog box, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type `0.05`.

6 In the **Stop frequency** text field, type `10000`.

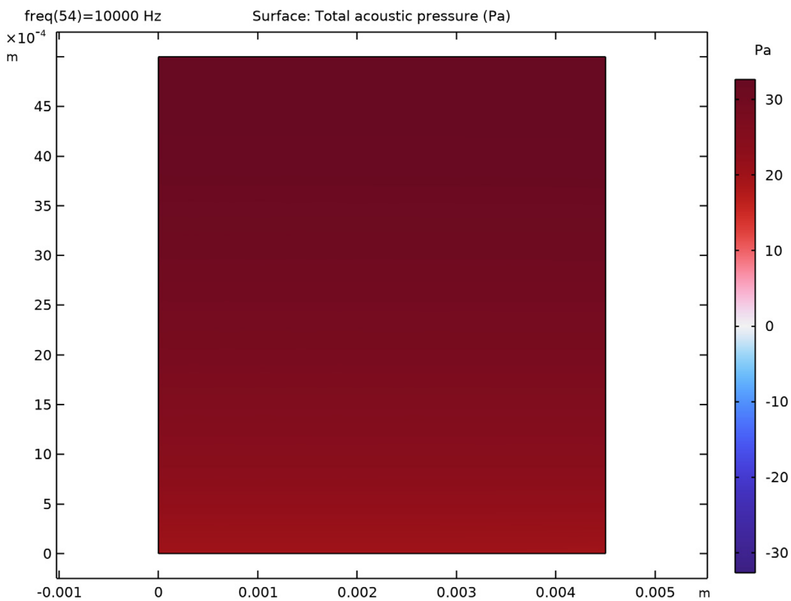
7 From the **Interval** list, choose **1/3 octave**.

8 Click **Replace**.

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

RESULTS

Acoustic Pressure (ta)



RMS Acoustic Velocity (ta)

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

2 In the **Settings** window for **2D Plot Group**, type `RMS Acoustic Velocity (ta)` in the **Label** text field.

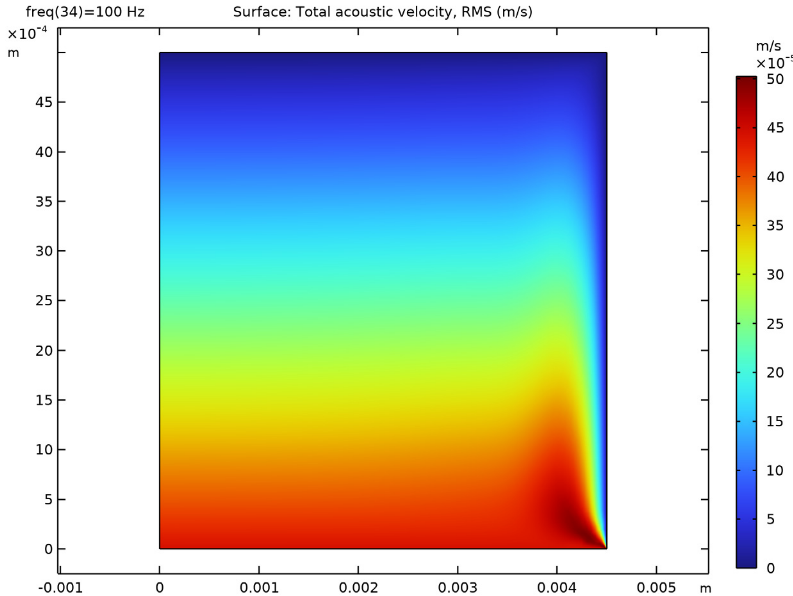
Surface

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

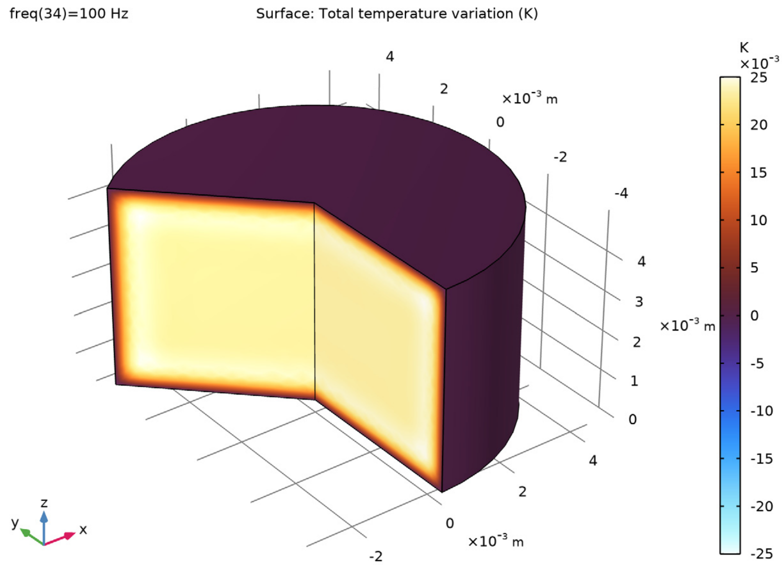
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.v_rms`.

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



Temperature Variation (ta)



Pressure in Coupler: real(p)

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 in Coupler: real(p)` in the **Label** text field.

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

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `f (Hz)`.

6 Select the **y-axis label** check box.

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

8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Pressure in Coupler: real(p)** and choose **Point Graph**.

2 Select Point 2 only.

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

4 In the **Expression** text field, type `real(ta.p_t)`.

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

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

7 In the table, enter the following settings:

Legends
`COMSOL model`

Global 1

1 In the **Model Builder** window, right-click **Pressure in Coupler: real(p)** and choose **Global**.

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

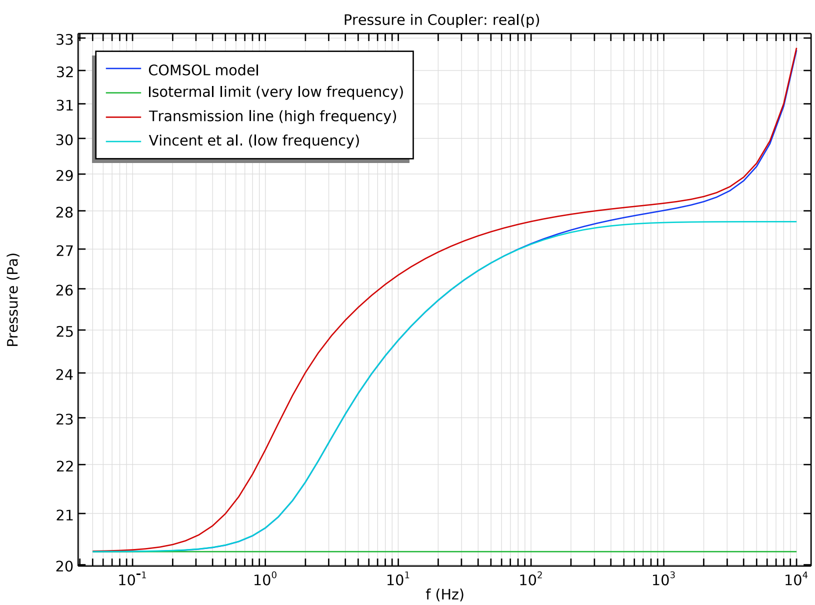
3 In the table, enter the following settings:

Expression	Unit	Description
`real(dpT)`	`Pa`	`Isotermal limit (very low frequency)`
`real(i*omega*dV/Y_tl)`	`Pa`	`Transmission line (high frequency)`
`real(p_galf)`	`Pa`	`Vincent et al. (low frequency)`

4 In the **Pressure in Coupler: real(p)** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

6 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.



Pressure in Coupler: imag(p)

1 Right-click **Pressure in Coupler: real(p)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure in Coupler: imag(p)` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Point Graph 1

1 In the **Model Builder** window, expand the **Pressure in Coupler: imag(p)** node, then click **Point Graph 1**.

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

3 In the **Expression** text field, type `imag(ta.p_t)`.

Global 1

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

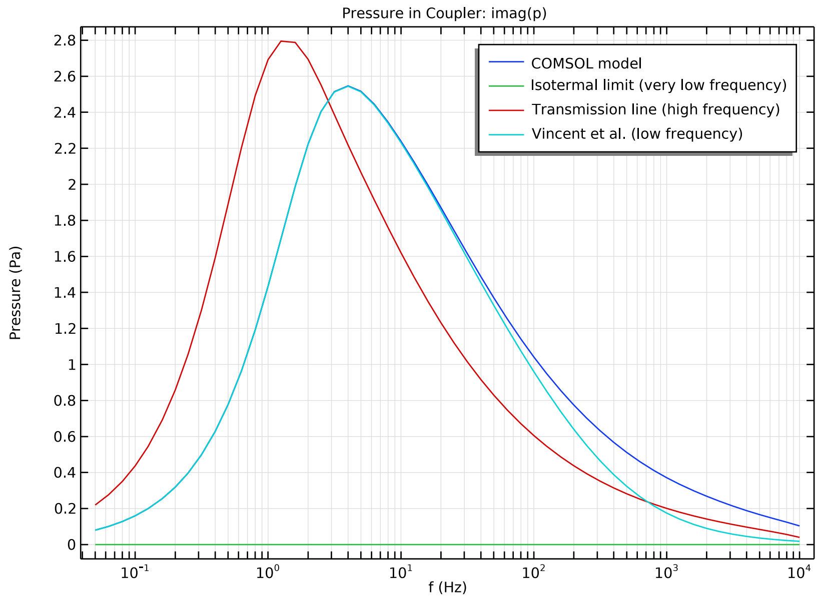
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
imag(dpT)	Pa	Isotermal limit (very low frequency)
imag(i*omega*dV/Y_tl)	Pa	Transmission line (high frequency)
imag(p_galf)	Pa	Vincent et al. (low frequency)

4 In the **Pressure in Coupler: imag(p)** toolbar, click **Plot**.

5 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.



Acoustic Transfer Impedance: real(Z) and imag(Z)

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

2 In the **Settings** window for **1D Plot Group**, type `Acoustic Transfer Impedance: real(Z) and imag(Z)` in the **Label** text field.

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

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `f (Hz)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type Z_{a,12} (kg/(m⁴s)).

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

9 Select the **y-axis log scale** check box.

10 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Acoustic Transfer Impedance: real(Z) and imag(Z)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`real((intop_s(ta.p_t)/S)/(ta.iomega*dn*S))`	`kg/(m^4*s)`	`real(Z), COMSOL model`
`-imag((intop_s(ta.p_t)/S)/(ta.iomega*dn*S))`	`kg/(m^4*s)`	`-imag(Z), COMSOL model`

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

Global 2

1 In the **Model Builder** window, right-click **Acoustic Transfer Impedance: real(Z) and imag(Z)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`real(1/Y_tl)`	`kg/(m^4*s)`	`real(Z), Transmission line`
`-imag(1/Y_tl)`	`kg/(m^4*s)`	`-imag(Z), Transmission line`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

Global 3

1 Right-click **Acoustic Transfer Impedance: real(Z) and imag(Z)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`real(1/Ya)`	`kg/(m^4*s)`	`real(Z), Vincent et al.`
`-imag(1/Ya)`	`kg/(m^4*s)`	`-imag(Z), Vincent et al.`

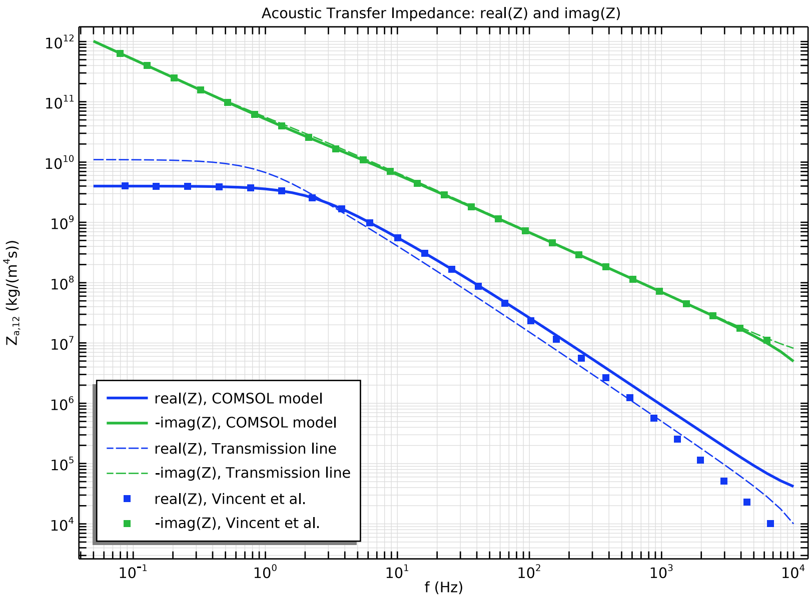
4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

7 In the **Number** text field, type `25`.

8 In the **Acoustic Transfer Impedance: real(Z) and imag(Z)** toolbar, click **Plot**.



Next create a grid dataset that will be used to plot the material properties as function of temperature for several values of the relative humidity.

Grid 1D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

2 In the **Settings** window for **Grid 1D**, locate the **Parameter Bounds** section.

3 In the **Name** text field, type Tg.

4 In the **Minimum** text field, type 273.15.

5 In the **Maximum** text field, type 323.15.

Density

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

2 In the **Settings** window for **1D Plot Group**, type Density in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Grid 1D 1**.

4 From the **Parameter selection (freq)** list, choose **First**.

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

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

7 In the associated text field, type Temperature (^oC).

8 Select the **y-axis label** check box.

9 In the associated text field, type Density (kg/m³).

Line Graph 1

1 Right-click **Density** 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 subst(pp1mat1.def.rho,minput.T,Tg[K/m], minput.pA,p0,minput.phi,0).

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

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

6 In the table, enter the following settings:

Legends
relH = 0.0

7 In the **Density** toolbar, click **Plot**.

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 subst(pp1mat1.def.rho,minput.T,Tg[K/m], minput.pA,p0,minput.phi,0.2).

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

Legends
relH = 0.2

5 In the **Density** toolbar, click **Plot**.

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 subst(pp1mat1.def.rho,minput.T,Tg[K/m], minput.pA,p0,minput.phi,0.4).

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

Legends
relH = 0.4

5 In the **Density** toolbar, click **Plot**.

Line Graph 4

1 Right-click **Line Graph 3** and choose **Duplicate**.

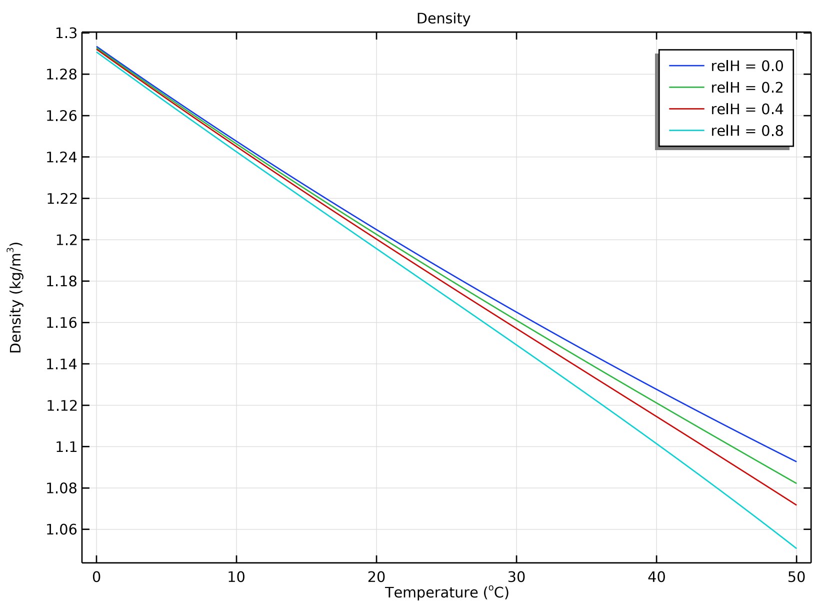
2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `subst(pp1mat1.def.rho,minput.T,Tg[K/m], minput.pA,p0,minput.phi,0.8)`.

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

Legends
relH = 0.8

5 In the **Density** toolbar, click **Plot**.



Duplicate the density plot in order to create plots of the dynamic viscosity (`pp1mat1.def.mu`), thermal conductivity (`pp1mat1.def.k_iso`), and speed of sound (`pp1mat1.def.c`). Finally, group the plots in a **Node Group**. All the plots are depicted in the Results and Discussion section of the documentation.