Created in COMSOL Multiphysics 5.5

Small Concert Hall Acoustics

Introduction

Designing structures and open spaces with respect to sound quality is important for concert halls, outdoor environments, and even the rooms of a house. Simulating acoustics in the high-frequency limit, where the wavelength is smaller than the geometrical features, is best done with ray acoustics.

This tutorial model shows the basic steps and principles used when setting up a model using the Ray Acoustics physics interface. In the model, the acoustics of a small concert hall is analyzed. The model setup includes an omnidirectional sound source, wall boundary conditions for specular and diffuse scattering, sound pressure evaluation, a receiver data set used to reconstruct a temporal impulse response plot, and an energy response (reflectogram). The results are compared to simple reverberation time estimates.

Model Definition

In this model the acoustics of a generic "small concert hall" are analyzed. The model is that of a listening environment with a volume of $460\ \mathrm{m}^3$ and a total surface area of $390\ \mathrm{m}^2$. The listening environment is fitted with absorbers and diffusers; their location is not particularly optimized and does probably not follow design rules. The aim of this tutorial is to describe the important modeling steps to perform a room acoustics simulation using ray tracing. The geometry of the concert hall is depicted in Figure 1.

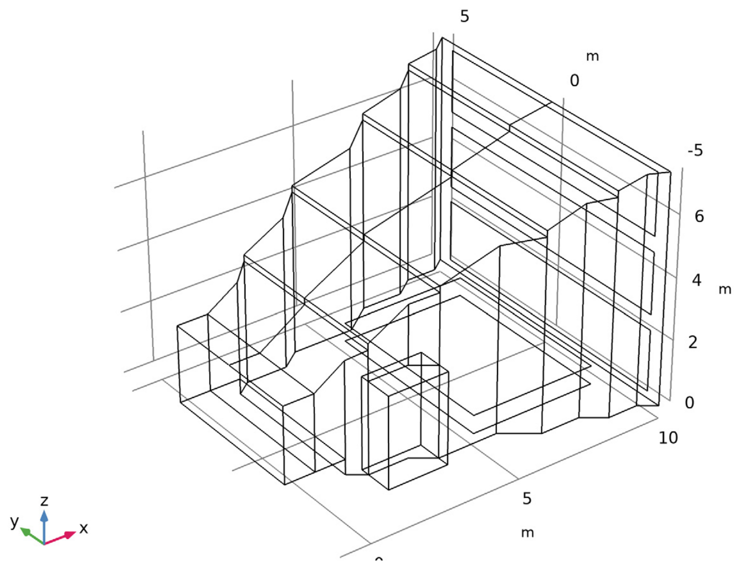


Figure 1: Geometry of the small concert hall.

An omni-directional source is located at the coordinates (x_s, y_s, z_s) near the stage. The receiver (microphone) is located at the coordinates (x_r, y_r, z_r). These are parameters found under **Global Definitions > Parameters 1**. The location of the receiver can be changed in postprocessing while the location of the source needs to be changed before running the model.

The absorption properties if the various surfaces (windows, seats, diffusers, floor, entrance, walls, and absorbers) are generic values taken from Ref. 1 and 2. The data is given in octave band values and imported from the file `small_concert_hall_absorption_parameters.txt` file into an interpolation function (lookup table). Moreover, the (intensity) attenuation of air is imported from the file `small_concert_hall_volume_absorption.txt`. Note that when the (volume) attenuation coefficient m (loaded from the file) is entered in the **Material Properties of Exterior and Unmeshed Domains** section it is the amplitude attenuation α_{ext} (= $0.5 \cdot m$) that should be used. The difference is a factor 0.5.

Results and Discussion

The local wavefront sound pressure level (SPL) is depicted in Figure 2 after 10 ms and in Figure 3 after 20 ms propagation. When the compute intensity option is selected in the Ray Acoustics interface, wavefront curvature, intensity, and SPL is calculated along each ray. They allow visualization of the local acoustic properties. However, it is the acoustic power transported by each ray that is important when calculating the impulse response (IR) and when visualizing the sound pressure level at surfaces. This means that the compute power option should always be selected for IR computation but the compute intensity can be turned off. Only selecting **Compute power** will also make the model run slightly faster and reduce the number of degrees of freedom (DOFs) solved for.

The temporal IR for the default source location configuration used in the model is depicted in Figure 4 and the frequency domain (FFT) of the IR is depicted in Figure 5. For most practical application and further postprocessing the impulse response should be exported under the **>Export** node by adding a **Plot** export. Per default the IR has a sampling frequency of 44100 Hz.

When an IR is reconstructed from a ray tracing simulation, information is inferred and put back into the time signal. The quality of the simulated IR increases with the number of rays (this model uses 10000) as well as the frequency resolution of the absorption, scattering, and source data (this data can be difficult to get from vendors but can often be simulated). In this model, octave band resolution is used. The Impulse Response plot also allows the use of 1/3 octave and 1/6 octave frequency resolution.

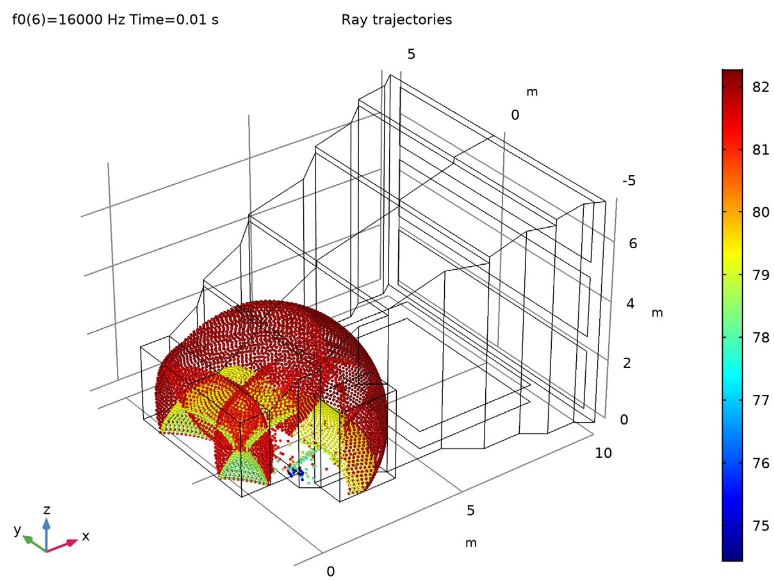


Figure 2: Ray location and SPL after 10 ms.

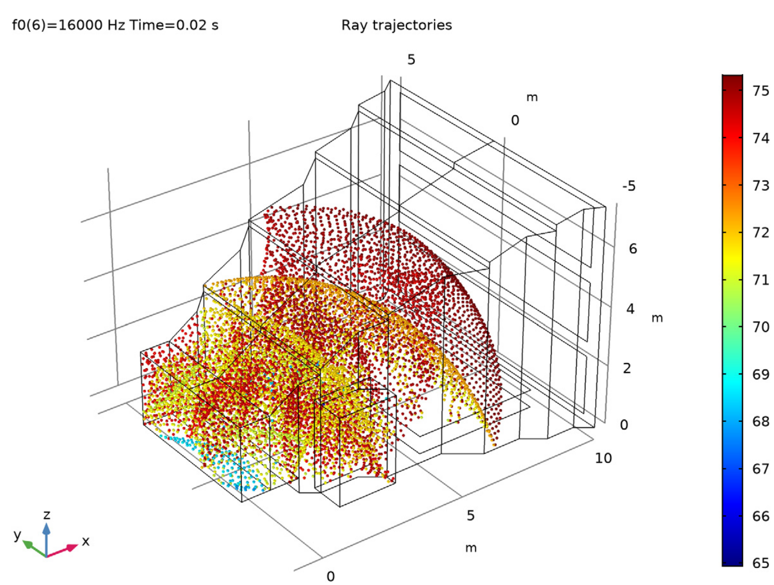


Figure 3: Ray location and field SPL after 20 ms.

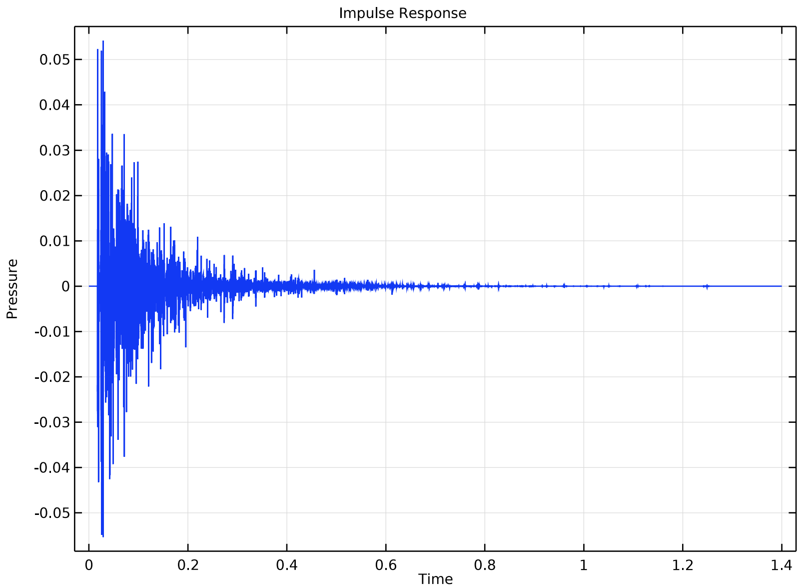


Figure 4: Temporal impulse response reconstructed from the ray data at the receiver location.

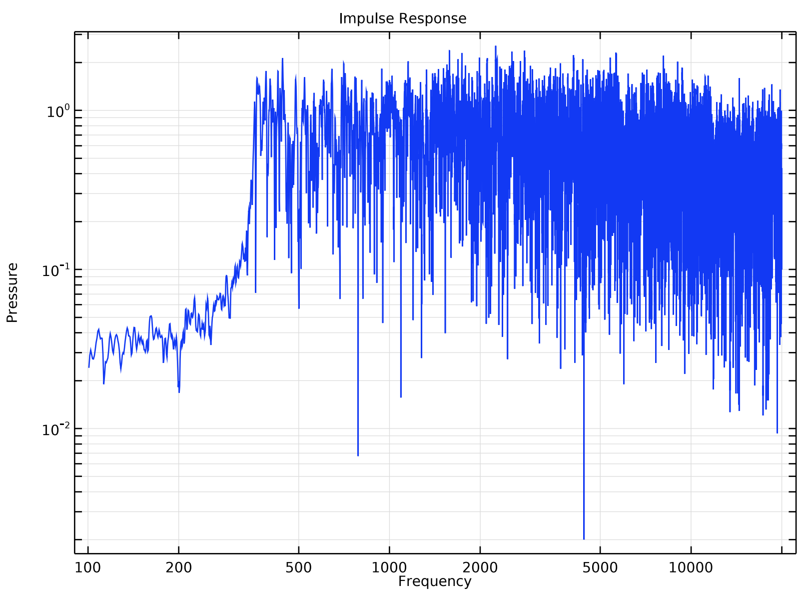


Figure 5: FFT of the impulse response (no smoothing/windowing is used).

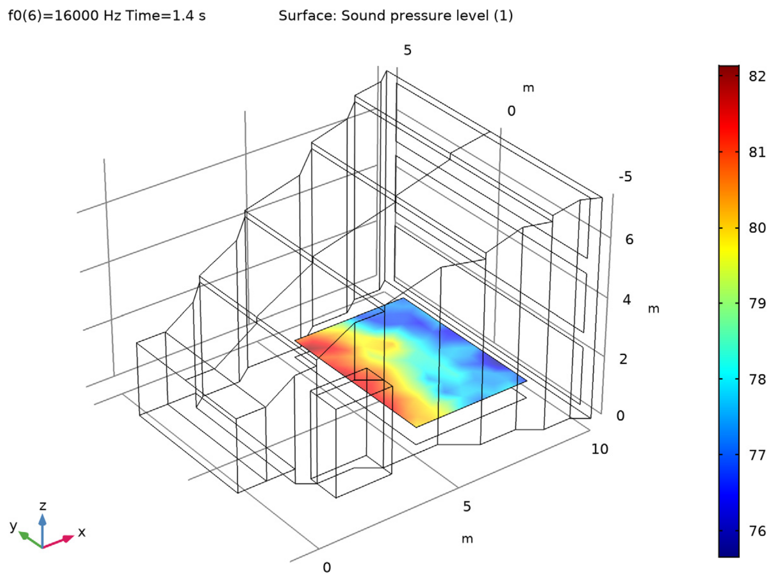


Figure 6: SPL in a cross section 60 cm above the ground at the location of the audience.

The sound pressure level in a cross section located above the seating section is depicted in Figure 6. It is calculated using the **Sound Pressure Level Calculation** feature, available as a sub-node to all wall conditions. In this case it is added to a transparent surface (**Pass through** used as **Wall condition**). The feature can be added to all other walls to postprocess the SPL distribution there if necessary.

In ray tracing methods, the intensity I and RMS pressure p_{rms} of the n'th ray detected by the receiver sphere is expressed as

$$I_{\mathrm{n}} = \frac{L_{\mathrm{r}}Q_{\mathrm{n}}}{V_{\mathrm{r}}} \qquad (p^2_{\mathrm{rms}})_{\mathrm{n}} = \rho c I_{\mathrm{n}}$$

where V_{r} is the receiver volume, L_{r} is the distance traveled by the ray inside the receiver, and Q_{n} is the power carried by the ray (see, for example, Ref. 3). The intensity is evaluated using the expression `re1dist*rac.Q/re1vol`. Plotting this information in a Ray plot as function of the arrival time yields the (discrete time) energy impulse response. It is also sometimes known as a reflectogram. It is plotted for the 500 Hz and the 16 kHz octave bands in Figure 7. The slope of the curves (point data) is related to the reverberation time of the room. The slope is usually assessed using, for example, Shroeder's backward integration or a moving integration/averaging. This is not done here but can be done in an external tool if the data is exported. In Figure 7 approximate trend lines have been added; their slope (60 dB down which is 6 decades for $\log_{10}(I_n)$) predict the T_{60} reverberation time. The values can be compared to the predicted values shown in Figure 8. The graph shows the values both with and without the effect of air attenuation. The values are calculated using the Sabine equation used in statistical room acoustics

$$T_{60} = 0.161\frac{V}{S\bar{\alpha} + 4mV}$$

where V is the room volume, S is the total surface area, m is the volume attenuation, and $\bar{\alpha}$ is the average wall absorption (see Ref. 2). For the two frequency bands the results from Figure 7 show good agreement with the results in Figure 8.

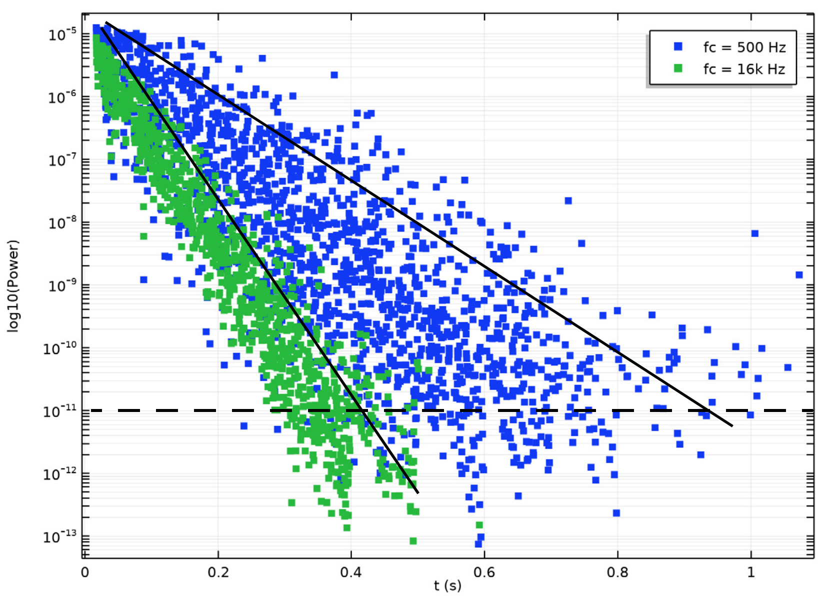


Figure 7: The raw data of the energetic impulse response or reflectogram. The slope represents the reverberation time in the given octave band.

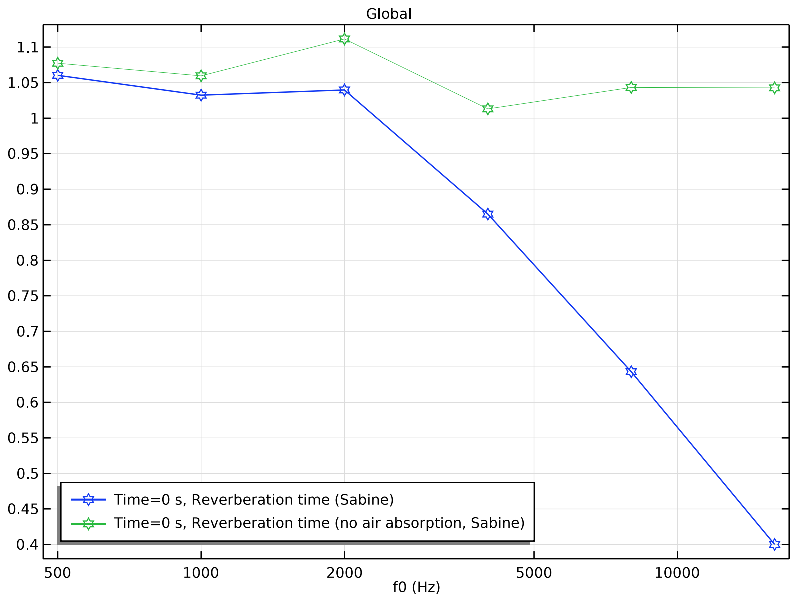


Figure 8: Reverberation time estimates based on the Sabine formula. The curve with and without the volume absorption of air. The importance of including air absorption is evident at the higher frequencies.

Notes About the COMSOL Implementation

RESULTS

There are several options that can be selected on the **>Results** node that are useful when working with ray acoustic models and especially when evaluating the impulse response.

- When first setting up plots, it is useful to select the **Only plot when requested** option as some plots take a long time to render. Another trick is to use only a few rays initially.
- Once the plots are set up, then before running the model (with a large number of rays), select the **Recompute all plot data after solving** option. Once the model has solved the plots will be rendered. This is very useful when running the model over lunch break or over night since rendering the IR plot often takes longer time than solving the model.
- Before saving the model, remember to set the **Save plot data** list to **On**. Then all plots do not need to be re-rendered once the model is opened again.

EXTRA TIME-STEPS CONVERGENCE ANALYSIS

When this tutorial model is solved the solution is only stored every 0.1 s. This reduces the file size when saving. For postprocessing it is typically necessary with a much finer time resolution. Ideally there should be one time before and after each ray-wall interaction and one time before and after a ray crosses the receiver. This is achieved using the **Extra Time Steps** in the Receiver data set. To find an adequate value a small convergence test can be run. This is done in the model (see the last plot also shown in Figure 9). For a single band, plot the power `rac.Q` detected at the receiver using a Ray plot; do this while increasing the number of extra time steps. In this model the **Proportionality factor** is increased. Once the solution does not change any more, enough extra times have been used. In Figure 9 you can see that the red dots are on top of all the cyan circles (except a few). This means that a proportionality factor of 30 is an adequate choice for all the postprocessing.

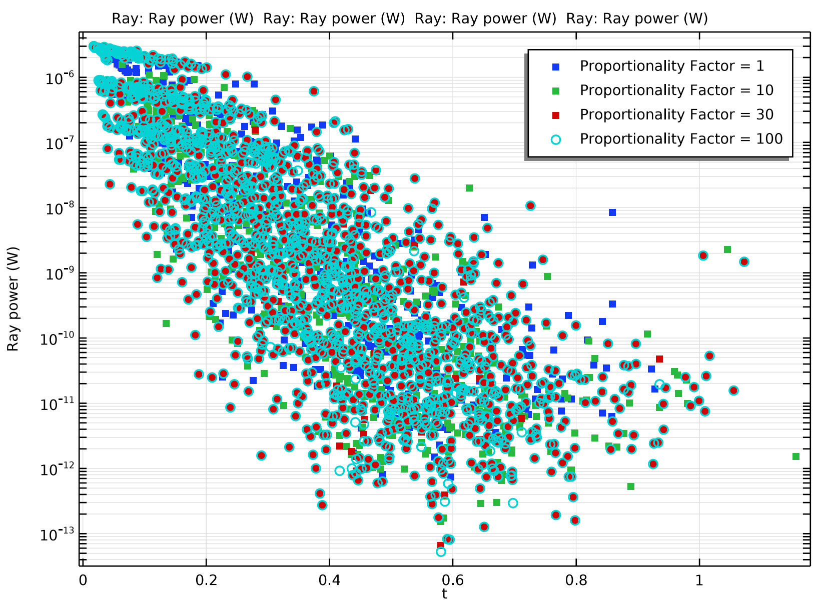


Figure 9: Graph used to evaluate how many extra time steps (time steps added in-between the stored times) that are needed when reconstructing the impulse response in order to get consistent results.

References

1. M. Vorländer, *Auralization, Fundamentals of Acoustics, Modeling, Simulation, Algorithms and Acoustic Virtual Reality*, Springer, 2008.

2. H. Kuttruff, *Room Acoustics*, CRC Press, 2009.

3. Z. Xiangyang, C. Ke'an, and S. Jincai, "On the accuracy of the ray-tracing algorithms based on various sound receiver models," *Appl. Acoust.*, vol. 64, pp. 433-441, 2003.

Application Library path: `Acoustics_Module/Building_and_Room_Acoustics/small_concert_hall`

Modeling Instructions

This section contains the modeling instructions for the Absorptive Muffler model. They are followed by the Geometry Sequence Instructions section.

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

NEW

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

MODEL WIZARD

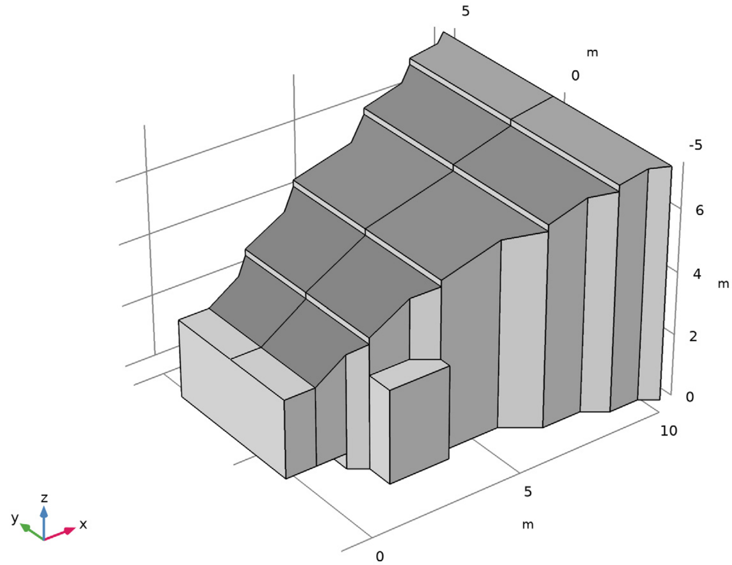
1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Acoustics>Geometrical Acoustics>Ray Acoustics (rac)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Ray Tracing**.
6. Click **Done**.

The geometry is set up by importing a geometry sequence. The sequence imports the small concert hall geometry and sets up several selections. The predefined selections simplify the rest of the model setup.

GEOMETRY 1

1. In the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `small_concert_hall_geom_sequence.mph`.
3. In the **Geometry** toolbar, click **Build All**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.



Import the model parameters form a file. The parameters include the band center frequency `f0`, the location of the source and receiver, as well as the room volume.

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 `small_concert_hall_parameters.txt`.

Proceed and set up interpolation functions for the absorption coefficients of the different surfaces in the concert hall. The data is easily stored in one `.txt` file. Also define an interpolation function for the intensity attenuation of air (given at 50 % relative humidity and 20 deg. C).

Interpolation 1 (int1)

1 In the **Home** toolbar, click **Functions** and choose **Global>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 `small_concert_hall_absorption_parameters.txt`.

6 In the **Number of arguments** text field, type `1`.

7 Click **Import**.

8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
a_walls	1
a_entrance	2
a_windows	3
a_floor	4
a_diffuser	5
a_seats	6
a_absorbers	7

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

10 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

11 In the **Function** text field, type `1`.

Interpolation 2 (int2)

1 In the **Home** toolbar, click **Functions** and choose **Global>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 `small_concert_hall_volume_absorption.txt`.

6 Click **Import**.

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

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

9 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

10 In the **Function** text field, type `1/m`.

Now import the variables that define the reverberation time (T60) based on the simple Sabine equations. This also requires setting up integration operators for all the surfaces.

DEFINITIONS

Variables 1

1. In the **Model Builder** window, expand the **Definitions** node.
2. Right-click **Component 1 (comp1)>Definitions** and choose **Variables**.
3. In the **Settings** window for **Variables**, type `Variables: Reverberation Time Estimates` 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 `small_concert_hall_variables.txt`.

Integration 1 (intop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_windows` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Windows**.

Integration 2 (intop2)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_seats` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Seats**.

Integration 3 (intop3)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_diffusers` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Diffusers**.

Integration 4 (intop4)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_floor` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Floor**.

Integration 5 (intop5)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_entrance` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Entrance**.

Integration 6 (intop6)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_walls` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Walls**.

Integration 7 (intop7)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `intop_absorbers` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Absorbers**.

Now it is time to set up and define the physics and boundary conditions of the model. To compute the impulse response, it is necessary to model the intensity and power along the rays. The model only uses a surface mesh. Propagation in the unmeshed domains requires the definition of material properties at the interface level (in the section **Material Properties**

of Exterior and Unmeshed Domains). Also set up boundary conditions for the different walls (boundaries).

RAY ACOUSTICS (RAC)

1 In the **Model Builder** window, click **Ray Acoustics (rac)**.
2 In the **Settings** window for **Ray Acoustics**, locate the **Intensity Computation** section.
3 From the **Intensity computation** list, choose **Compute intensity and power**.
4 Locate the **Material Properties of Exterior and Unmeshed Domains** section. In the c_{ext} text field, type `c0`.
5 In the ρ_{ext} text field, type `rho0`.
6 In the α_{ext} text field, type `0.5*m_air(f0)`.

   Multiplication by 0.5 is necessary as the input in COMSOL is defined for the amplitude attenuation and not the intensity attenuation (as given in the interpolation data).

Ray Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Ray Acoustics (rac)** click **Ray Properties 1**.
2 In the **Settings** window for **Ray Properties**, locate the **Ray Properties** section.
3 In the f text field, type `f0`.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 In the **Settings** window for **Wall**, type `Wall: Walls` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Walls**.
4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Specular reflection**.
5 Locate the **Reflection Coefficient Model** section. In the α text field, type `a_walls(f0)`.

Wall 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 In the **Settings** window for **Wall**, type `Wall: Entrance` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Entrance**.
4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Specular reflection**.
5 Locate the **Reflection Coefficient Model** section. In the α text field, type `a_entrance(f0)`.

Wall 4

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

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

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Windows**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Specular reflection**.

5 Locate the **Reflection Coefficient Model** section. In the α text field, type `a_windows(f0)`.

Wall 5

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

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

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Floor**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Specular reflection**.

5 Locate the **Reflection Coefficient Model** section. In the α text field, type `a_floor(f0)`.

Wall 6

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

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

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Diffusers**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Mixed diffuse and specular reflection**.

5 In the γ_s text field, type `1-s_diffuser`.

6 Locate the **Reflection Coefficients Model** section. In the α_s text field, type `a_diffuser(f0)`.

7 In the α_d text field, type `a_diffuser(f0)`.

In this model the scattering coefficient `s_diffuser` is constant across the frequency bands. It can of course also be defined as an interpolation function.

Wall 7

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

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

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Seats**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Specular reflection**.

5 Locate the **Reflection Coefficient Model** section. In the α text field, type `a_seats(f0)`.

Wall 8

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

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

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Absorbers**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Specular reflection**.

5 Locate the **Reflection Coefficient Model** section. In the α text field, type `a_absorbers(f0)`.

Wall 9

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

2 In the **Settings** window for **Wall**, type `Wall: SPL cross section` in the **Label** text field.

3 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Pass through**.

The **Pass through** option is used here as this surface only is meant for visualizing the SPL in a cross section. Use **Wireframe** rendering to select the surface that is located inside the concert hall.

4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

5 Select Boundary 40 only.

Sound Pressure Level Calculation 1

In the **Physics** toolbar, click **Attributes** and choose **Sound Pressure Level Calculation**.

Release from Grid 1

1 In the **Physics** toolbar, click **Global** and choose **Release from Grid**.

2 In the **Settings** window for **Release from Grid**, locate the **Initial Coordinates** section.

3 In the $q_{x,\,0}$ text field, type `x_s`.

4 In the $q_{y,\,0}$ text field, type `y_s`.

5 In the $q_{z,\,0}$ text field, type `z_s`.

6 Locate the **Ray Direction Vector** section. From the **Ray direction vector** list, choose **Spherical**.

7 In the N_{w} text field, type `Nrays`.

8 Locate the **Total Source Power** section. In the P_{src} text field, type `P0`.

Ray Termination 1

1 In the **Physics** toolbar, click **Global** and choose **Ray Termination**.

2 In the **Settings** window for **Ray Termination**, locate the **Termination Criteria** section.

3 From the **Spatial extents of ray propagation** list, choose **Bounding box, from geometry**.

4 From the **Additional termination criteria** list, choose **Intensity**.

5 In the I_{th} text field, type `1e-13[W/m^2]`.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

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

3 From the **Selection** list, choose **All boundaries**.

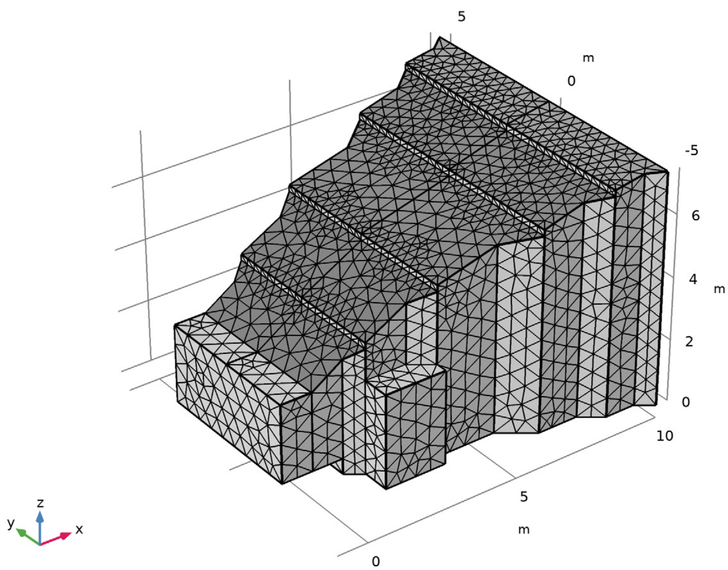
Size

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

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

3 From the **Predefined** list, choose **Finer**.

4 Click **Build All**.



Proceed and solve the model by adding a parametric sweep over the center frequency variable `f0`. This represents the center frequency of the octave bands analyzed in this model, in order to get a broadband response. The first time you set up and solve the model it can be useful to reduce the number of rays by changing the value of the parameter `Nrays` to, for example, 1000. This will make postprocessing faster. Remember that the quality of the results in acoustic ray tracing increase for an increasing number of rays and more narrow frequency bands (you need to have wall absorption data with the desired resolution). In the Ray Acoustics interface the impulse response plot can handle octave, 1/3 octave, and 1/6 octave data.

STUDY 1

Step 1: Ray Tracing

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Ray Tracing**.

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

3 From the **Time unit** list, choose **s**.

4 In the **Times** text field, type `0 0.01 0.02 range(0.1,0.1,1.4)`.

The times entered here represent instances where the solution is stored (the model size on disc depends in part on this). Much smaller time steps are used internally by the solver. In postprocessing when reconstructing the impulse response additional time steps are rendered and used by the data sets.

Parametric Sweep

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

Using the parametric sweep is important as this gives the frequency resolution (here in full octaves). The ray propagation model is solved once per frequency band. The data is collected in postprocessing, by the receiver data set and the impulse response plot, to generate the broadband response.

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

3 Click **Add**.

4 Click **Range**.

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

6 In the **Start frequency** text field, type `500`.

7 In the **Stop frequency** text field, type `20000`.

8 Click **Replace**.

Solving the model takes a couple of minutes and uses less than 2 GB of RAM (depending on your hardware). This will increase for an increasing number of rays.

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

RESULTS

Ray Trajectories (rac)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Time (s)** list, choose **0.01**.

3 In the **Ray Trajectories (rac)** toolbar, click **Plot**.

Ray Trajectories 1

1 In the **Model Builder** window, expand the **Ray Trajectories (rac)** node, then click **Ray Trajectories 1**.

2 In the **Settings** window for **Ray Trajectories**, locate the **Coloring and Style** section.

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

4 Find the **Point style** subsection. From the **Type** list, choose **Point**.

Color Expression 1

1 In the **Model Builder** window, expand the **Ray Trajectories 1** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `rac.Lp`.

4 In the **Ray Trajectories (rac)** toolbar, click **Plot**.

This should reproduce the image in Figure 2.

Ray Trajectories (rac)

1 In the **Model Builder** window, click **Ray Trajectories (rac)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **0.02**.

4 In the **Ray Trajectories (rac)** toolbar, click **Plot**.

This should reproduce the image in Figure 3.

On the **Results** node you have several options that are useful when postprocessing ray tracing simulations, where rendering plots can be time consuming. This is especially true for the impulse response plots. While setting up plots, it is useful to select **Only plot when requested** such that the plots are not generated every time you change an option.

It is also good practice to save the plots in the model such that they are already rendered when you open your model at a later stage. Finally, once you have set up all the plots and you are ready to run the model again, it can be useful to enable **Recompute all plot data after solving**. All plots will then be recomputed after the model is solved, for example, running over night.

5 In the **Model Builder** window, click **Results**.

6 In the **Settings** window for **Results**, locate the **Update of Results** section.

7 Select the **Only plot when requested** check box.

8 Locate the **Save Data in the Model** section. From the **Save plot data** list, choose **On**.

Now set up all the data sets you will need for postprocessing the results. First, set up the receiver data set that includes all the solved frequency bands (for a broadband analysis). Then, set up a receiver for the first and for the last frequency band. Finally, set up a series of data sets that will be used in a convergence analysis performed later on. The latter is optional, but it shows how the **Proportionality factor** value is chosen and set equal to 30 in all the receiver data sets used.

Receiver 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Receiver 3D**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - All Bands` in the **Label** text field.

3 Locate the **Receiver** section. Find the **Center** subsection. In the **x** text field, type `x_r`.

4 In the **y** text field, type `y_r`.

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

6 Find the **Radius** subsection. In the **Number of rays** text field, type `Nrays`.

7 In the **Room volume** text field, type `Vol`.

8 In the **Source-receiver distance** text field, type `dsr`.

9 Locate the **Extra Time Steps** section. From the **Maximum number of extra time steps rendered** list, choose **Proportional to number of solution times**.

10 In the **Proportionality factor** text field, type `30`.

Receiver 3D - All Bands 1

1 Right-click **Receiver 3D - All Bands** and choose **Duplicate**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - 500 Hz Band` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (f0)** list, choose **From list**.

4 In the **Parameter values (f0 (Hz))** list, select **500**.

Receiver 3D - 500 Hz Band 1

1 Right-click **Receiver 3D - 500 Hz Band** and choose **Duplicate**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - 16k Hz Band` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (f0 (Hz))** list, select **16000**.

Group 1

1 In the **Model Builder** window, right-click **Datasets** and choose **Node Group**.

2 In the **Settings** window for **Group**, type `Group: Convergence Analysis Data Sets` in the **Label** text field.

Receiver 3D 4

1 In the **Results** toolbar, click **More Datasets** and choose **Receiver 3D**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - Single Band (prop = 1)` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (f0)** list, choose **First**.

4 Locate the **Receiver** section. Find the **Center** subsection. In the **x** text field, type `x_r`.

5 In the **y** text field, type `y_r`.

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

7 Find the **Radius** subsection. In the **Number of rays** text field, type `Nrays`.

8 In the **Room volume** text field, type `Vol`.

9 In the **Source-receiver distance** text field, type `dsr`.

10 Locate the **Extra Time Steps** section. From the **Maximum number of extra time steps rendered** list, choose **Proportional to number of solution times**.

Receiver 3D - Single Band (prop = 1) 1

1 Right-click **Receiver 3D - Single Band (prop = 1)** and choose **Duplicate**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - Single Band (prop = 10)` in the **Label** text field.

3 Locate the **Extra Time Steps** section. In the **Proportionality factor** text field, type `10`.

Receiver 3D - Single Band (prop = 10) 1

1 Right-click **Receiver 3D - Single Band (prop = 10)** and choose **Duplicate**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - Single Band (prop = 30)` in the **Label** text field.

3 Locate the **Extra Time Steps** section. In the **Proportionality factor** text field, type `30`.

Receiver 3D - Single Band (prop = 30) 1

1 Right-click **Receiver 3D - Single Band (prop = 30)** and choose **Duplicate**.

2 In the **Settings** window for **Receiver 3D**, type `Receiver 3D - Single Band (prop = 100)` in the **Label** text field.

3 Locate the **Extra Time Steps** section. In the **Proportionality factor** text field, type `100`.

1D Plot Group 2

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Receiver 3D - All Bands**.

Impulse Response 1

1 In the **Impulse Response** toolbar, click **More Plots** and choose **Impulse Response**.

Rendering the impulse response will typically take longer time than solving the model (up to 20-30 min for this plot alone). The plot is not just a representation of computed data but consists of a reconstruction (computation) of a signal based on the ray data picked up at the receiver.

2 Click **Plot**.

This should reproduce the impulse response shown in Figure 4. The impulse response is the most important result of this model. The signal can be exported under the **Export** node and used for further analysis in an external signal processing tool. The response is reconstructed from the ray data detected by the **Receiver** data set (arrival time, power, and band center frequency). It has a default sampling frequency of 44100 Hz. This can be changed under the **Advanced** section in the plot settings window.

Impulse Response 1

1 In the **Model Builder** window, right-click **Impulse Response** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Impulse Response FFT` in the **Label** text field.

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

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

Impulse Response 1

1 In the **Model Builder** window, expand the **Results>Impulse Response FFT** node, then click **Impulse Response 1**.
2 In the **Settings** window for **Impulse Response**, locate the **x-Axis Data** section.
3 From the **Transformation** list, choose **Frequency spectrum**.
4 Select the **Frequency range** check box.
5 In the **Minimum** text field, type `100`.
6 In the **Maximum** text field, type `20000`.
7 In the **Impulse Response FFT** toolbar, click **Plot**.

This should reproduce the image in Figure 5.

3D Plot Group 4

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Cross Section SPL` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

Surface 1

1 Right-click **Cross Section SPL** and choose **Surface**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Ray Acoustics> Accumulated variables>Wall intensity comp1.rac.wall9.spl1.Iw>rac.wall9.spl1.Lp - Sound pressure level - dB**.
3 In the **Cross Section SPL** toolbar, click **Plot**.

This should reproduce the image in Figure 6. Choose the 16k data set to see the SPL at this frequency.

1D Plot Group 5

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Energetic Impulse Response (Reflectogram)` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
5 In the associated text field, type `t (s)`.
6 Select the **y-axis label** check box.

7 In the associated text field, type `log10(Power)`.

8 Locate the **Data** section. From the **Dataset** list, choose **None**.

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

Ray 1

1 In the **Energetic Impulse Response (Reflectogram)** toolbar, click **More Plots** and choose **Ray**.

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

3 From the **Dataset** list, choose **Receiver 3D - 500 Hz Band**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `re1dist*rac.Q/re1vol`.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

7 From the **Positioning** list, choose **In data points**.

8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

9 Select the **Show legends** check box.

10 In the table, enter the following settings:

Legends
`fc = 500 Hz`

Ray 2

1 Right-click **Ray 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Receiver 3D - 16k Hz Band**.

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

Legends
`fc = 16k Hz`

5 In the **Energetic Impulse Response (Reflectogram)** toolbar, click **Plot**.

This should reproduce the image in Figure 7.

1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, type `Reverberation Time Estimates` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Time selection** list, choose **First**.

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

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

Global 1

1 Right-click **Reverberation Time Estimates** 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**
`T60`	`m`	`Reverberation time (Sabine)`
`T60_na`	`m`	`Reverberation time (no air absorption, Sabine)`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **f0**.

5 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Star**.

6 From the **Positioning** list, choose **In data points**.

7 In the **Reverberation Time Estimates** toolbar, click **Plot**.

This should reproduce the image in Figure 8.

1D Plot Group 7

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

2 In the **Settings** window for **1D Plot Group**, type `Extra Time-Steps Convergence Analysis` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Locate the **Axis** section. Select the **y-axis log scale** check box.

Ray 1

1 In the **Extra Time-Steps Convergence Analysis** toolbar, click **More Plots** and choose **Ray**.

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

3 From the **Dataset** list, choose **Receiver 3D - Single Band (prop = 1)**.

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

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

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

7 From the **Positioning** list, choose **In data points**.

8 Locate the **Legends** section. Select the **Show legends** check box.

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

10 In the table, enter the following settings:

Legends
`Proportionality Factor = 1`

Ray 2

1 Right-click **Ray 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Receiver 3D - Single Band (prop = 10)**.

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

Legends
`Proportionality Factor = 10`

Ray 3

1 Right-click **Ray 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Receiver 3D - Single Band (prop = 30)**.

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

Legends
`Proportionality Factor = 30`

Ray 4

1 Right-click **Ray 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Receiver 3D - Single Band (prop = 100)**.

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

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

Legends
`Proportionality Factor = 100`

6 In the **Extra Time-Steps Convergence Analysis** toolbar, click **Plot**.

This should reproduce the image in Figure 9.

Finally, evaluate the arrival time of the first ray using the built-in data set variable `re1first`. This time can, for example, be used when analyzing the temporal alignment of speakers.

Evaluation Group 1

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group: Arrival Time of First Ray` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Receiver 3D - 500 Hz Band**.

Global Evaluation 1

1 Right-click **Evaluation Group: Arrival Time of First Ray** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`re1first`	`s`	

4 In the **Evaluation Group: Arrival Time of First Ray** toolbar, click **Evaluate**.

Geometry Sequence Instructions

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 Click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `small_concert_hall.mphbin`.
5 Click **Import**.
6 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Explicit Selection 1 (sel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `Windows` in the **Label** text field.
3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
4 On the object **imp1**, select Boundaries 60–62 only.

Explicit Selection 2 (sel2)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `Seats` in the **Label** text field.
3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
4 On the object **imp1**, select Boundary 39 only.

Explicit Selection 3 (sel3)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `Diffusers` in the **Label** text field.
3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
4 On the object **imp1**, select Boundaries 13, 15, 29, 30, 41, 42, 49, and 50 only.

Explicit Selection 4 (sel4)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `Floor` in the **Label** text field.
3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundaries 3, 8, 12, 14, 18, and 21 only.

Explicit Selection 5 (sel5)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundaries 16, 19, 20, 23, 31, and 32 only.

Explicit Selection 6 (sel6)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundaries 7, 9–11, 17, 22, 24–28, 34–37, 43–48, and 51–59 only.

Explicit Selection 7 (sel7)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundaries 1, 2, 4–6, 33, and 38 only.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 Click **Build Selected**.