Created in COMSOL Multiphysics 5.6

Acoustics of a Living Room

Introduction

The sound we experience from our stereo or home theater systems in our living rooms is influenced not only by the quality of loudspeakers, but other factors, for example, the shape of the room, and the type and placement of the furniture. Reflections from walls and windows can both enhance and distort the sound that reaches our ears, and the low bass notes from the speaker woofer units can shake the windows and make the floor vibrate. This happens only for certain frequencies — the eigenfrequencies of the room. The simulation we set-up in this tutorial solves for the eigenfrequencies of a living room in the low frequency range, and analyzes the acoustic field in the room when the sound sources are the woofer units. The analysis is useful, for example, when optimizing for loudspeaker locations inside a living room.

Model Definition

The geometry for the living room used in this analysis is synchronized from Revit® through the LiveLink interface. The room is equipped with a flat-screen TV, a TV stand, a sideboard, a table, two speakers, a bookcase, and two couches. The Revit project file has been saved with the synchronization settings that generate and transfer the volume of the living room, the walls as solid objects, and the furniture. It is not necessary for the analysis to include fully detailed geometry for the furniture in the room. The bookcase is synchronized as a bounding box, while the other furniture items include the original detail level. Further simplifications are done to the synchronized furniture objects inside

COMSOL Multiphysics. Selections that are used for model settings are generated for all geometric objects during synchronization.

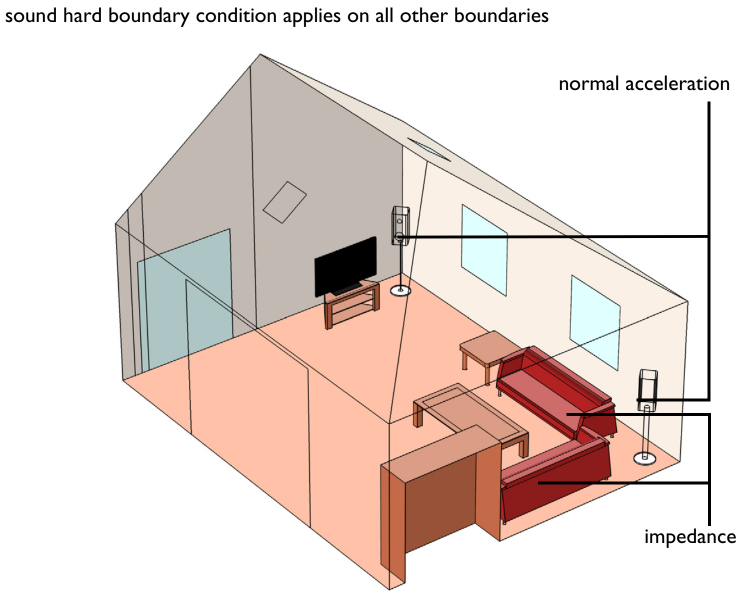


Figure 1: The room geometry and boundary conditions.

The simulation is set-up with the Pressure Acoustics, Frequency Domain interface to study the sound propagation in free air. Most boundaries in the model — walls, floor, ceiling, and furniture — are assumed to be perfectly rigid (sound hard boundaries), except for the soft surfaces of the two couches. On the couch surfaces an impedance value is set that represents the sound dampening and absorption properties of the material. A normal acceleration is applied to the boundaries that represent the woofer units in the model.

Two studies are added to the model. The eigenfrequency analysis computes the resonance frequencies of the room in the vicinity of 10 Hz together with the corresponding eigenmodes. The frequency domain analysis is set-up to run at 100 Hz.

Results and Discussion

The eigenmodes shows the pressure distribution at the resonance frequencies. Specifically they allow us to identify where there will be no sound (at the nodes) and where the sound will be amplified (at the antinodes). The absolute in an eigenfrequency study do not have any physical meaning.

The real part of the complex valued eigenfrequency represents the frequency at which the system is resonant. The imaginary part is related to the losses at the eigenfrequency an thus the Q-factor of the resonance.

All modes have local maxima in the corners of an empty room so speakers in the corners excite all eigenfrequencies. This simulation predicts eigenmodes that resemble those of the corresponding empty room. The higher the frequency, the more the placing of the furniture matters. The prediction that speakers placed in the corners of the room excite many eigenmodes and give a fuller and more neutral sound, however, holds for real-life rooms.

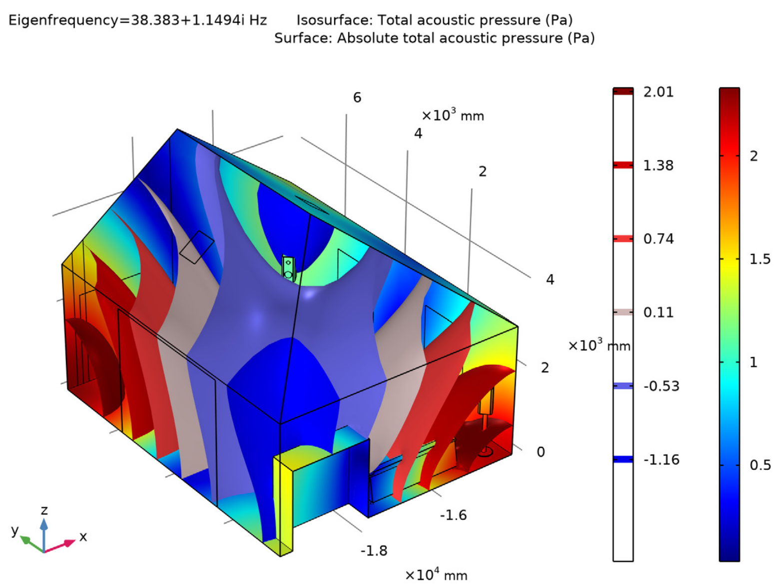


Figure 2: The sound pressure distribution for an eigenfrequency of 38.4 Hz. The real part of the pressure is visualized as an isosurface plot, and the absolute value of the pressure as a boundary plot.

The results of the frequency domain study allow us to study the acoustic field in the room for the chosen frequency, in this case 100 Hz. More specifically it can be used to extract a transfer function from the speaker to any listening location. This can, for example, be used to fine-tune the phase difference between the speakers to optimize the sound in a specific location of the room.

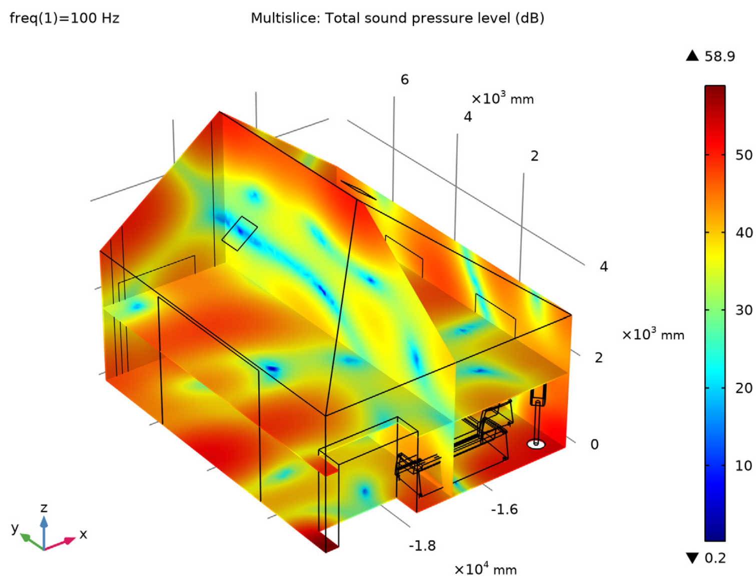


Figure 3: The sound pressure level produced from the speaker for a frequency of 100 Hz.

Intensity plots, such as the one displayed in Figure 4, allow a visual inspection of the energy flow from the loudspeakers. Typically near resonances the flow will be toward specific points in space.

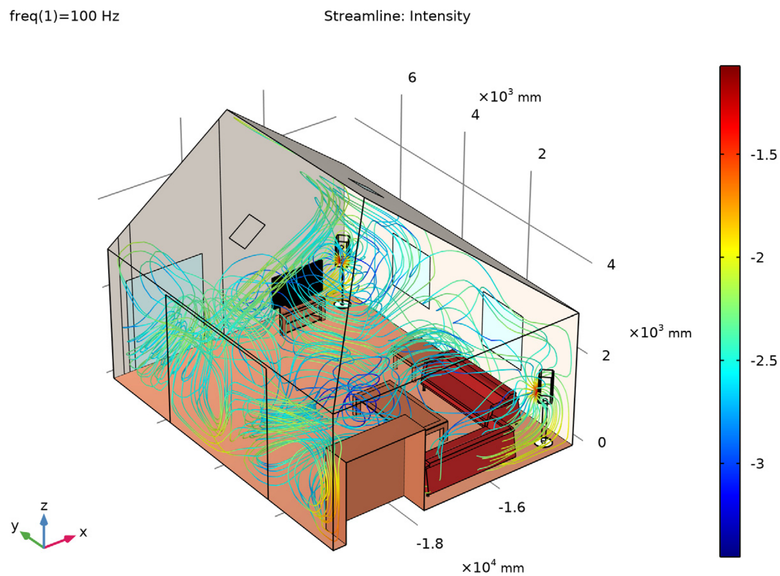


Figure 4: Streamline plot that visualizes the energy flow from the sound sources at 100 Hz.

Application Library path: `LiveLink_for_Revit/Tutorials,` `_LiveLink_Interface/living_room_acoustics_llrevit`

Modeling Instructions

1 In Revit open the file `house_living_room.rvt` located in the model's Application Library folder.

2 Switch to the COMSOL Desktop.

3 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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Frequency Domain**.
6 Click **Done**.

GEOMETRY 1

Make sure that the CAD Import Module kernel is used.

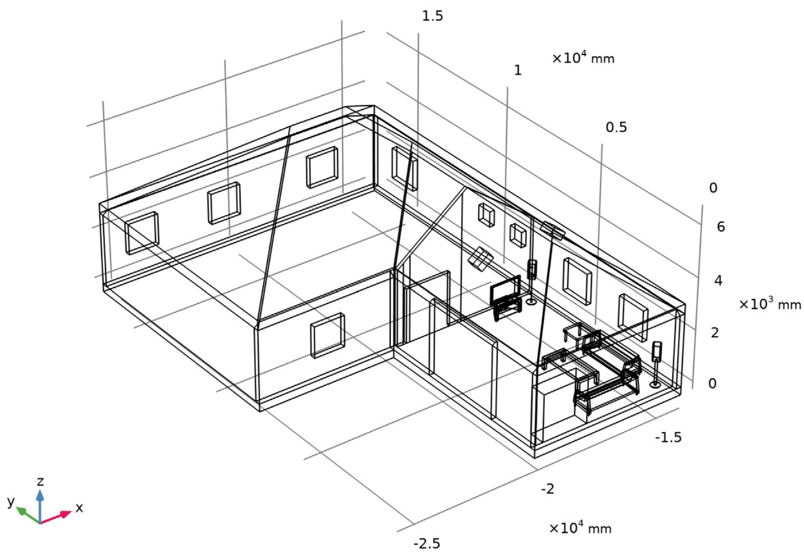
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
3 From the **Geometry representation** list, choose **CAD kernel**.

LiveLink for Revit 1 (cad1)

1 In the **Home** toolbar, click **LiveLink** and choose **LiveLink for Revit**.
2 In the **Settings** window for **LiveLink for Revit**, locate the **Synchronize** section.
3 Click **Synchronize**.

   For the synchronization to start a floor plan or section view needs to be active in the Revit software. In case the synchronization does not start switch to Revit, and activate the Level 1 floor plan view. During synchronization the geometry for the living room and the selected elements are generated and transferred to the COMSOL model.

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



5 Click to expand the **Object Selections** section. The synchronized objects are included in automatically generated selections that we will use for setting up the simulation. Clicking on a selection in the table highlights the corresponding objects in the **Graphics** window.

Room Bounding Solids

Start with creating union selections to be used as inputs to the geometry operations that will create the final geometry for the simulation.

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

2 In the **Settings** window for **Union Selection**, type `Room Bounding Solids` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Floors:Generic-12"**, **Roofs:Generic-12"**, **Walls:BasicWall:Generic-8"**, and **Walls:BasicWall:Interior-5"Partition(2-hr)**.To select several selections from the list hold down **Ctrl** while clicking the selections.

6 Click **OK**.

Room

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Air:Living room 5**, **Surface:Doors:M_Double-Glass2:1830x2134mm**, **Surface:Walls:CurtainWall:CurtainWall1**, **Surface:Windows:M_CasementDblwithTrim:1220x1220mm**, **Surface:Windows:M_Skylight:0610x0686mm**, and **Room Bounding Solids**.

6 Click **OK**.

Furniture

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Furniture:FloorstandingSpeaker:FloorstandingSpeaker**, **Furniture:M_Entertainmentcenter:1830x1830x0610mm**, **Furniture:M_Sofa-Pensi:2134mm**, **Furniture:M_Table-Coffee:0915x1830x0457mm**, **Furniture:M_Table-End:0762x0762mm**, **Furniture:M_TV-FlatScreen:1270mm**, and **Furniture:M_TVStand:M_TVStand**.

6 Click **OK**.

To be able to use the objects in the defined selections in the model set-up, they need to become part of the final geometry. To incorporate all objects into a single solid object use the Convert to Solid geometry operation.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 In the **Settings** window for **Convert to Solid**, locate the **Input** section.

3 From the **Input objects** list, choose **Room**.

4 From the **Repair tolerance** list, choose **Relative**.

5 Click **Build All Objects**.

Since only the room volume is needed at the end, delete the domains for the walls, roof and floor.

Delete Entities 1 (del1)

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

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 From the **Selection** list, choose **Room Bounding Solids**.

5 Click **Build All Objects**.

Difference 1 (dif1)

To obtaining the computational volume subtract the objects for the furniture from the object for the room.

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

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

3 From the **Objects to add** list, choose **Room**.

4 From the **Objects to subtract** list, choose **Furniture**.

5 Click **Build Selected**.

Now use the defeaturing tools to find and remove small details from the geometry of the furniture.

6 In the **Geometry** toolbar, click **Defeaturing and Repair** and choose **Delete Sliver Faces**.

7 In the **Tools** window for **Delete Sliver Faces**, locate the **Delete Sliver Faces** section.

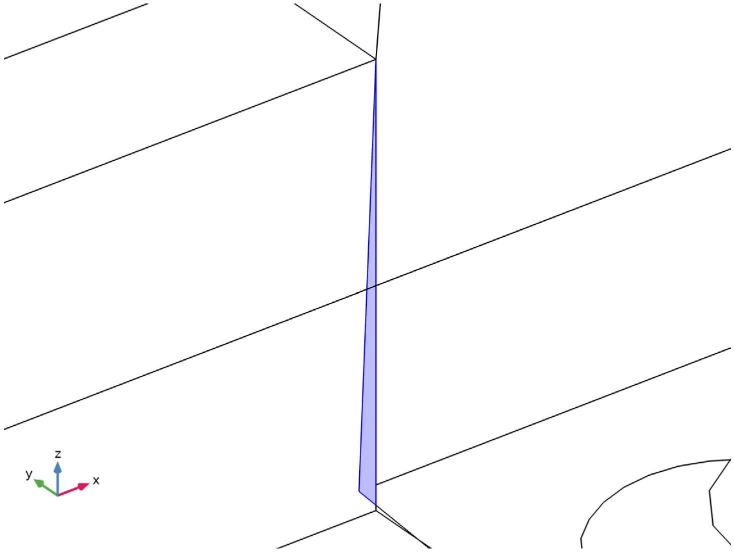
8 In the **Maximum face width** text field, type `5[mm]`.

9 Click **Find Sliver Faces**.

Sliver faces are faces with a very high aspect ratio.

10 In the list, select **Sliver face 8**.

11 Click the **Zoom to Selection** button in the **Graphics** toolbar.



12 Click **Delete All**.

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

14 Click **Delete Spikes**.

15 In the **Tools** window for **Delete Spikes**, locate the **Delete Spikes** section.

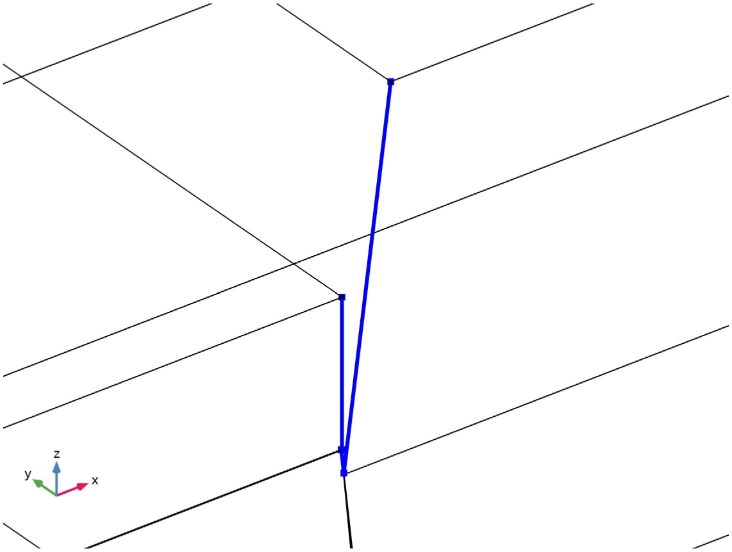
16 In the **Maximum spike width** text field, type `10[mm]`.

17 Click **Find Spikes**.

A spike is a long and narrow protrusion on an edge or corner of a face defined by two or three edges.

18 In the list, select **Spike 8**.

19 Click the **Zoom to Selection** button in the **Graphics** toolbar.

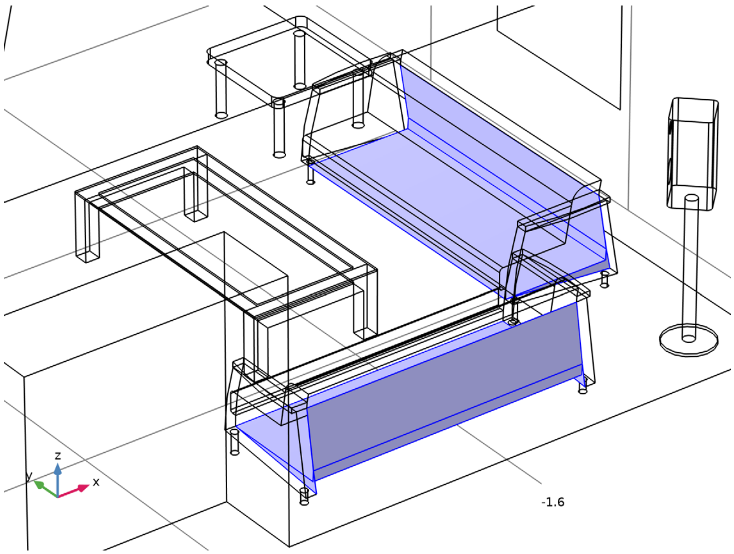


20 Click **Delete All**.

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

22 Click **Delete Faces**.

23 On the object **dsp1**, select Boundaries 30–33, 156, 189, 195, 196, 265, and 280 only. These are faces on the back and bottom of the couches.



24 In the **Model Builder** window, click **Geometry 1**.

25 In the **Tools** window for **Delete Faces**, locate the **Delete Faces** section.

26 Click **Delete Selected**.

The geometry is now ready for the setting up the simulation. Before defining the physics create selections to make the set-up easier.

Adjacent Selection 1 (adjsel1)

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

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

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

4 Click **Add**.

5 In the **Add** dialog box, select **Floors:Generic-12"** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Resulting Selection** section.

8 From the **Show in physics** list, choose **Off**.

Couches

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Furniture:M_Sofa-Pensi:2134mm** in the **Selections to add** list.

6 Click **OK**.

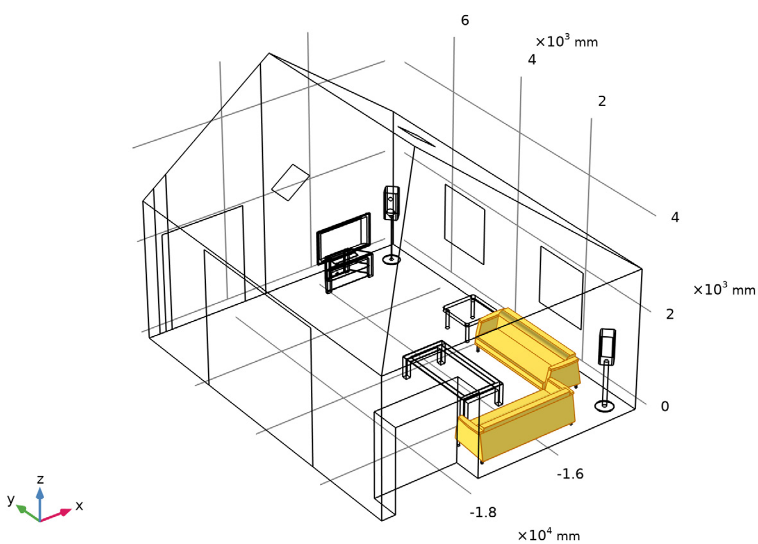
7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog box, select **Adjacent Selection 1** in the **Selections to subtract** list.

10 Click **OK**.

The `Couches` selection contains only the soft parts of the couches, the legs are excluded.



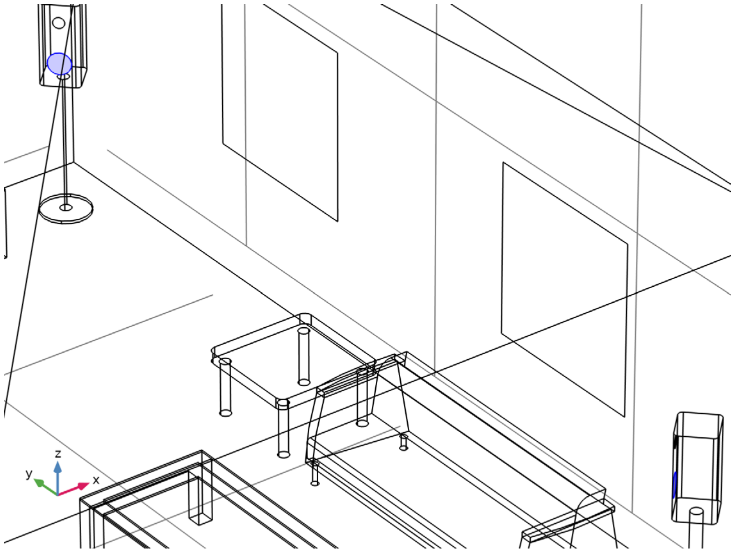
Woofer Units

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

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

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

4 On the object **dfa1**, select Boundaries 237 and 239 only. These are the boundaries that represent the woofer units of the loudspeakers.



Roof and Walls

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Roofs:Generic-12"**, **Walls:BasicWall:Generic-8"**, and **Walls:BasicWall:Interior-5"Partition(2-hr)**.

6 Click **OK**.

Doors and Windows

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

2 In the **Settings** window for **Union Selection**, type `Doors and Windows` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Surface:Doors:M_Double-Glass2:1830x2134mm**, **Surface:Walls:CurtainWall:CurtainWall1**,

Surface:Windows:M_CasementDblwithTrim:1220x1220mm, and **Surface:Windows:M_Skylight:0610x0686mm**.

6 Click **OK**.

Tables and Stands

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

2 In the **Settings** window for **Union Selection**, type `Tables and Stands` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Furniture:M_Entertainmentcenter:1830x1830x0610mm**, **Furniture:M_Table-Coffee:0915x1830x0457mm**, **Furniture:M_Table-End:0762x0762mm**, and **Furniture:M_TVStand:M_TVStand**.

6 Click **OK**.

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

GLOBAL DEFINITIONS

Continue with loading the parameters used for the physics and mesh 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 `living_room_acoustics_parameters.txt`.

ADD MATERIAL

Add air as the material for the room volume.

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.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Normal Acceleration 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Pressure Acoustics, Frequency Domain (acpr)** and choose **Normal Acceleration**.

2 In the **Settings** window for **Normal Acceleration**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Woofer Units**.

4 Locate the **Normal Acceleration** section. In the a_n text field, type `Nacc`.

Impedance 1

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

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

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

4 Locate the **Impedance** section. In the Z_i text field, type `Zicouch`.

MESH 1

For acoustics simulations it is important that the mesh resolves the acoustic wavelength, which translates into at least 5 second order elements per wavelength. Specify the maximum allowed mesh size in the domain to comply with this requirement.

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

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

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda0/6`.

5 In the **Minimum element size** text field, type `100[mm]`.

6 In the **Resolution of narrow regions** text field, type `0.8`.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 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 From the **Selection** list, choose **Furniture**.

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 `100[mm]`.

8 Select the **Minimum element size** check box.

9 In the associated text field, type `20[mm]`.

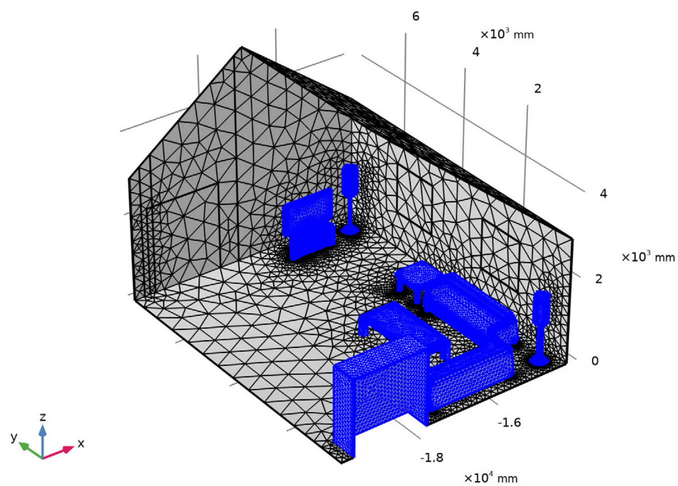
10 Click **Build All**.

To view the mesh on the interior boundaries hide some of the wall boundaries.

11 Click the **Click and Hide** button in the **Graphics** toolbar.

12 In the **Graphics** window, click on boundaries 1, 2, 4, 5, 6, 15, 43, and 117 to get a view similar to the figure below.

13 Click the **Click and Hide** button in the **Graphics** toolbar.



STUDY 1 FREQUENCY DOMAIN

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

2 In the **Settings** window for **Study**, type `Study 1 Frequency Domain` in the **Label** text field.

Step 1: Frequency Domain

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

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

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

For large acoustic models it is efficient to use an iterative solver. Switch from the default solver to a recommended iterative solver.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1 Frequency Domain>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (GMRES with GMG) (acpr)** and choose **Enable**.

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

RESULTS

Plots of the acoustic pressure and the sound pressure level are generated by default. Add a multislice plot for a better visualization of the sound pressure level in the room.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

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

3 Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Sound Pressure Level (acpr)** node.

2 Right-click **Surface 1** and choose **Delete**.

Sound Pressure Level (acpr)

In the **Model Builder** window, click **Sound Pressure Level (acpr)**.

Multislice 1

1 In the **Sound Pressure Level (acpr)** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp_t - Total sound pressure level - dB**.

3 Locate the **Multiplane Data** section. Find the **X-planes** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `-14181 -17000`.

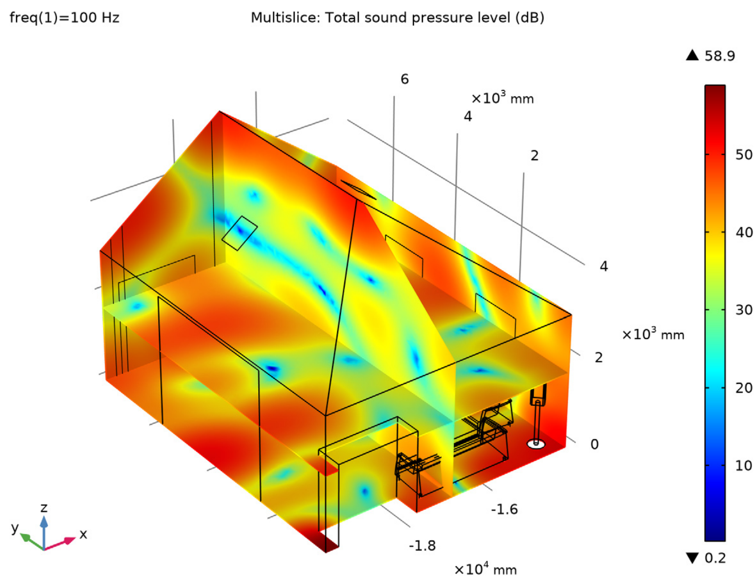
5 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `7542`.

7 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

8 In the **Coordinates** text field, type `0 1700`.

9 In the **Sound Pressure Level (acpr)** toolbar, click **Plot**.



Intensity

Now add a streamline plot of the sound intensity to visualize the flow of energy from the sound sources.

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Intensity` in the **Label** text field.

Streamline 1

1 In the **Intensity** toolbar, click **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Pressure Acoustics, Frequency Domain>Intensity>acpr.Ix,acpr.Iy,acpr.Iz - Intensity**.

3 Locate the **Streamline Positioning** section. In the **Number** text field, type `80`.

4 Locate the **Selection** section. From the **Selection** list, choose **Woofer Units**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

6 In the **Tube radius expression** text field, type `10`.

7 Select the **Radius scale factor** check box.

Color Expression 1

1 In the **Intensity** toolbar, click **Color Expression**.

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

3 In the **Expression** text field, type `log10(acpr.p_t)`.

4 In the **Intensity** toolbar, click **Plot**.

Intensity

In the **Model Builder** window, click **Intensity**.

Surface 1

1 In the **Intensity** toolbar, click **Surface**.

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

3 In the **Expression** text field, type `1`.

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

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Custom**.

7 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

8 Click **Define custom colors**.

9 Set the RGB values to 250, 240, and 230, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 In the **Intensity** toolbar, click **Selection**.

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

3 From the **Selection** list, choose **Roof and Walls**.

Surface 2

1 In the **Model Builder** window, under **Results>Intensity** right-click **Surface 1** and choose **Duplicate**.

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

3 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

4 Click **Define custom colors**.

5 Set the RGB values to 224, 255, and 255, respectively.

6 Click **Add to custom colors**.

7 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Doors and Windows**.

Surface 3

1 In the **Model Builder** window, under **Results>Intensity** right-click **Surface 2** and choose **Duplicate**.

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

3 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

4 Click **Define custom colors**.

5 Set the RGB values to 255, 160, and 122, respectively.

6 Click **Add to custom colors**.

7 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Floors:Generic-12"**.

Surface 4

1 In the **Model Builder** window, under **Results>Intensity** right-click **Surface 3** and choose **Duplicate**.

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

3 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

4 Click **Define custom colors**.

5 Set the RGB values to 196, 106, and 72, respectively.

6 Click **Add to custom colors**.

7 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 In the **Model Builder** window, expand the **Surface 4** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Tables and Stands**.

Surface 5

1 In the **Model Builder** window, under **Results>Intensity** right-click **Surface 4** and choose **Duplicate**.

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

3 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

4 Click **Define custom colors**.

5 Set the RGB values to 178, 34, and 34, respectively.

6 Click **Add to custom colors**.

7 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 In the **Model Builder** window, expand the **Surface 5** node, then click **Selection 1**.

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

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

Surface 6

1 In the **Model Builder** window, under **Results>Intensity** right-click **Surface 5** and choose **Duplicate**.

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

3 From the **Color** list, choose **Black**.

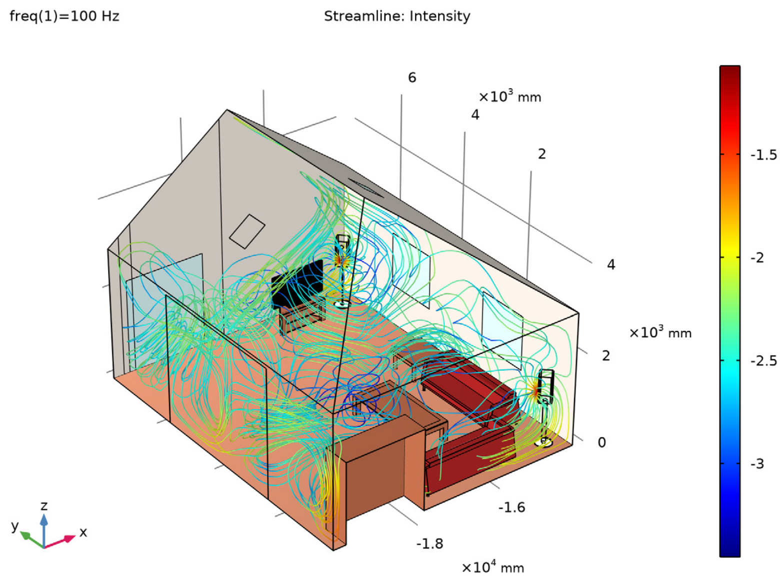
Selection 1

1 In the **Model Builder** window, expand the **Surface 6** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Furniture:M_TV-FlatScreen:1270mm**.

Intensity



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> Eigenfrequency**.

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

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

STUDY 2

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

2 In the **Search for eigenfrequencies around** text field, type `10[Hz]`.

3 From the **Eigenfrequency search method around shift** list, choose **Larger real part**.

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

5 In the **Settings** window for **Study**, type `Study 2 Eigenfrequency` in the **Label** text field.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2 Eigenfrequency>Solver Configurations> Solution 2 (sol2)>Eigenvalue Solver 1** node.

4 Right-click **Suggested Iterative Solver (GMRES with GMG) (acpr)** and choose **Enable**.

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

RESULTS

Sound Pressure Level (acpr) 1

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr) 1**.

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

3 From the **Eigenfrequency (Hz)** list, choose **49.511+0.89681i**.

Isosurface 1

1 In the **Sound Pressure Level (acpr) 1** toolbar, click **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp_t - Total sound pressure level - dB**.

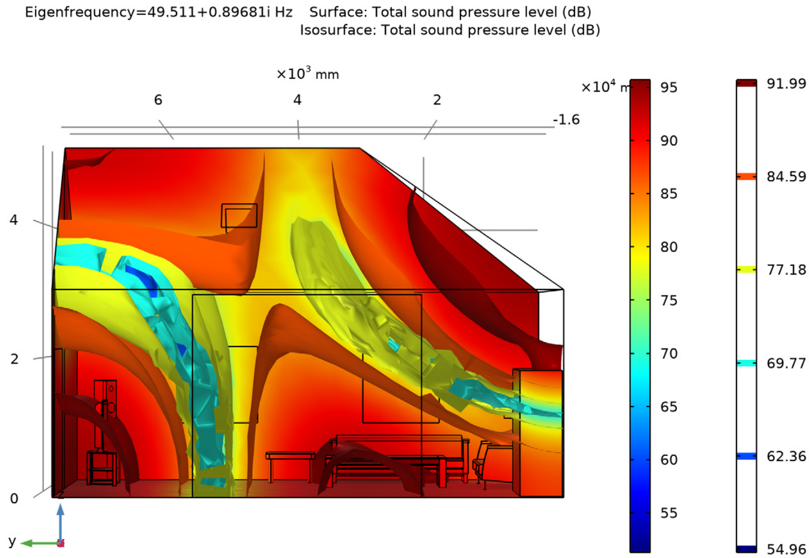
3 Locate the **Levels** section. In the **Total levels** text field, type `6`.

4 In the **Sound Pressure Level (acpr) 1** toolbar, click **Plot**.

5 Click the **Go to YZ View** button in the **Graphics** toolbar.

6 Click the **Go to YZ View** button in the **Graphics** toolbar.

7 Click the **Go to YZ View** button in the **Graphics** toolbar.



Acoustic Pressure, Isosurfaces (acpr) 1

1 In the **Model Builder** window, click **Acoustic Pressure, Isosurfaces (acpr) 1**.

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

3 From the **Eigenfrequency (Hz)** list, choose **38.383+1.1494i**.

Isosurface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure, Isosurfaces (acpr) 1** node, then click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 In the **Total levels** text field, type 6.

Acoustic Pressure, Isosurfaces (acpr) 1

In the **Model Builder** window, click **Acoustic Pressure, Isosurfaces (acpr) 1**.

Surface 1

1 In the **Acoustic Pressure, Isosurfaces (acpr) 1** toolbar, click **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 (comp1)>**

Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp_t - Absolute total acoustic pressure - Pa.

3 In the **Acoustic Pressure, Isosurfaces (acpr) 1** toolbar, click **Plot**.

4 Click the **Go to Default View** button in the **Graphics** toolbar.