Topology Optimization and Verification of an Acoustic Mode in a 2D Room

Introduction

This tutorial introduces the use of topology optimization in acoustics. The goal of the optimization is to find the optimal material distribution (solid or air) in a given design domain, here the ceiling of a 2D room, that minimizes the average sound pressure level in an objective region. The optimization is carried out for a single frequency. The topology optimized design is further transformed into a geometry and the results of the optimization are verified using sound hard boundaries. See the tutorial Optimizing the Shape of a Horn for instructions on how to perform the optimization for a frequency band.

The model was inspired by the work of M. B. Dühring, O. Sigmund, and J. Søndergaard Jensen (Ref. 1).

Note: This application requires the Acoustics Module and the Optimization Module.

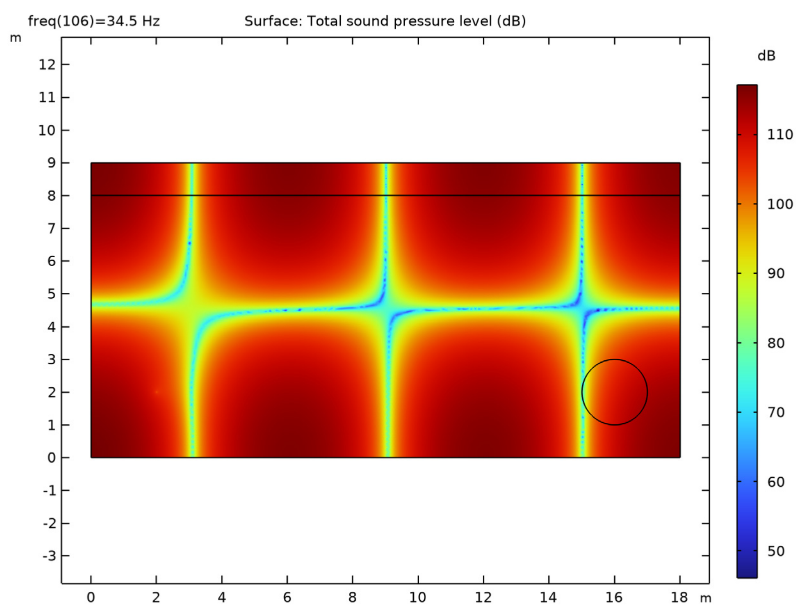

Figure 1: The initial design has a pressure node at the edge of the objective domain.

Model Definition

The room is exited with a monopole point source. The initial sound pressure level is shown in Figure 1. The objective is to minimize the average sound pressure level in the circular domain illustrated in the figure. The optimization is allowed to distribute material in the top of the room (a layer of the ceiling). The top domain is constrained to consists of at least 85% air, but there are no other constraints, so the material does not have to connect with the top boundary.

The gradient-based optimization solver, in the Optimization Module, by default evaluates derivatives of the objective function via the solution of an *adjoint equation*. This procedure requires that the symbolic derivative of any nonanalytic function is selected in a special way. The default behavior of the composite functions abs(z) and conj(z), which are most commonly used to obtain a real-valued objective function, is to return a derivative parallel to the real axis. However, this behavior is not appropriate for the adjoint method, where you instead need the definitions

$$\begin{aligned} \frac{d}{dz}|z| &= \frac{\bar{z}}{|z|} \\ \frac{d}{dz_1}(z_1\overline{z_2}) &= \overline{z_2} \\ \frac{d}{dz_2}(z_1\overline{z_2}) &= \overline{z_1} \end{aligned} \tag{1}$$

It is indeed possible to redefine the symbolic derivatives of built-in functions in COMSOL Multiphysics, but in this case it is more convenient to use the special function realdot(z_1,z_2), which evaluates as real(z_1·conj(z_2)) but differentiates according to Equation 1.

The model uses the **Density Model** feature to solve the topology optimization problem using the density method. This means that the geometry is defined implicitly using nonphysical material parameters. The local value of a given material parameter is determined by the local value of a control variable field, θ_c, which is bounded between zero and one, with zero corresponding to some solid material and one to air.

Within the field of acoustic topology optimization it is common to interpolate the inverse density and bulk modulus linearly, that is,

$$\begin{aligned} \rho^{-1} &= \rho_2^{-1} + (\rho_1^{-1} - \rho_2^{-1})\theta \\ K^{-1} &= K_2^{-1} + (K_1^{-1} - K_2^{-1})\theta \end{aligned} \tag{2}$$

where ρ_1 and K_1 are the density and bulk modulus of aluminum, respectively, while ρ_2 and K_2 are the properties of air. Wherever θ is equal to one, the properties of air are thus used. θ is the material volume factor, which is related to the control variable field, θ_c, through a filtering and projection step; see the *Topology Optimization of an MBB Beam* model in the Optimization Module Application Library for details. The main difference is that this model uses milling constraints to reduce the design freedom. Acoustic topology optimization can be sensitive to small design changes, which can easily occur during postprocessing of the topology optimization results. To reduce these issues, a continuation strategy is applied in the projection slope β. The model thus solves a sequence of optimization problems starting from a low projection slope and initializing the optimization with higher values using the previous optimization results.

Results and Discussion

The topology optimization achieves a large reduction of the average sound pressure level in the objective domain by moving a pressure node into the domain. The topology optimization result is further transferred to a new component using a Filter dataset. In this step a solid geometry is generated from the optimized solution. This causes the node to shift a bit, but it still goes through the domain as shown in Figure 2.

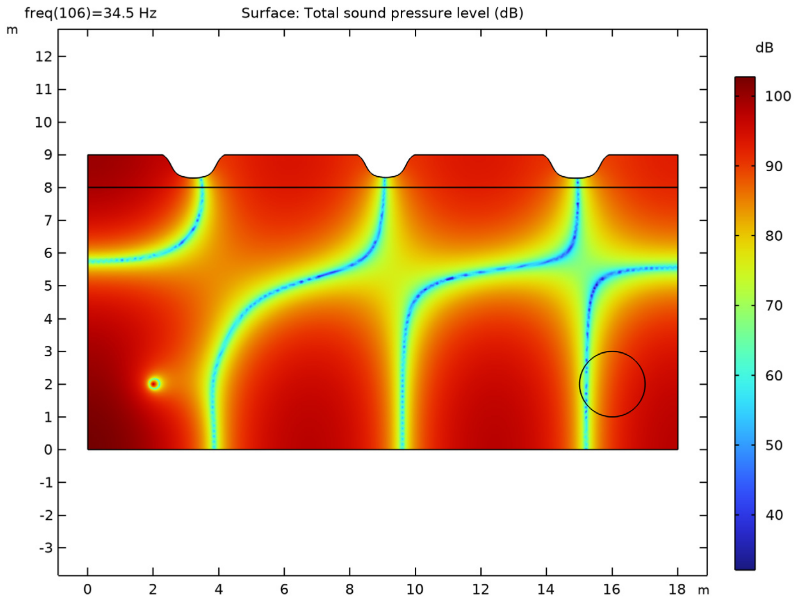

Figure 2: The sound pressure level is plotted after the optimization result has been transferred to a new component utilizing sound hard boundaries.

The value of the objective function is computed for a range of frequencies for the initial design as well as the topology optimized design (both before and after the verification step). The graph is plotted in Figure 3. It shows that the objective is only improved in a narrow frequency band around the optimization frequency. The verification step does not seem to cause the objective function to change noticeably.

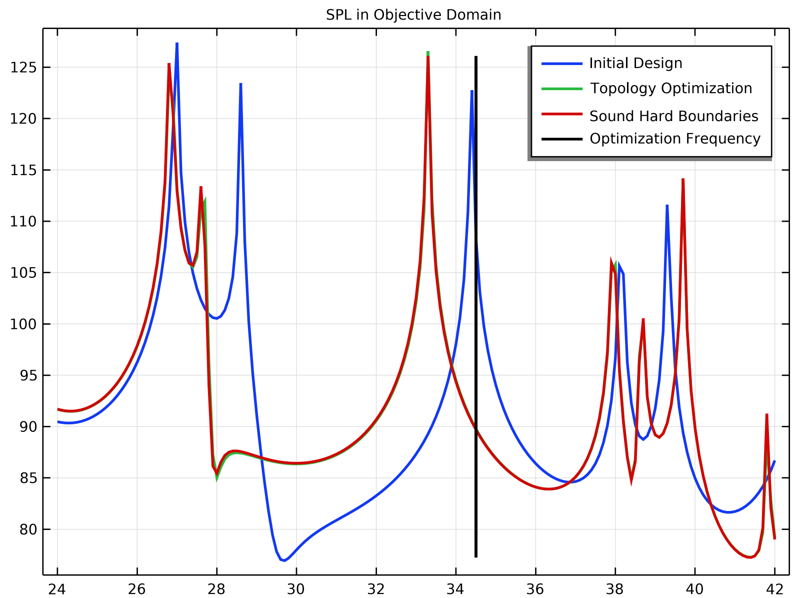

Figure 3: The objective function is plotted as a function of frequency for the initial and topology optimized designs.

Notes About the COMSOL Implementation

The model uses milling constraints to reduce the design freedom, but this also causes the objective function to be worse than it would be more design freedom.

Reference

1. M.B. Dühring, O. Sigmund, and J.S. Jensen, *Optimization of acoustic, optical and optoelastic devices*, Kgs. Lyngby, Denmark: Technical University of Denmark (DTU), DCAMM Special Report No. S109, 2009.

Application Library path: `Acoustics_Module/Optimization/topology_optimization_2d_room`

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**.
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**.

GLOBAL DEFINITIONS

Parameters 1

You can type in the parameters listed below or load them from `topology_optimization_2d_room_parameters.txt`.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
f0	34.5[Hz]	34.5 Hz	Frequency for optimization
W	18[m]	18 m	Room width
H	9[m]	9 m	Room height
dH	1[m]	1 m	Design domain height
Rob	1[m]	1 m	Objective domain radius
xob	16[m]	16 m	Objective domain x-coordinate
yob	2[m]	2 m	Objective domain y-coordinate
rho1	1.204[kg/m^3]	1.204 kg/m³	Air density

Name	Expression	Value	Description
K1	141.921e3[Pa]	1.4192E5 Pa	Air bulk modulus
rho2	2643.0[kg/m^3]	2643 kg/m³	Aluminum density
K2	68.7[GPa]	6.87E10 Pa	Aluminum bulk modulus
volfrac	0.85	0.85	Design domain volume fraction
alpha_K	0.001	0.001	Damping coefficient
hmax	0.3	0.3	Maximum element size
rmin	0.1*hmax	0.03	Minimum element size
beta	32	32	Projection slope

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 W.

4 In the **Height** text field, type H.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	dH

6 Clear the **Layers on bottom** check box.

7 Select the **Layers on top** check box.

Circle 1 (c1)

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

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

3 In the **Radius** text field, type Rob.

4 Locate the **Position** section. In the **x** text field, type xob.

5 In the **y** text field, type yob.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Point 1 (pt1)

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

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

3 In the **x** text field, type 2.

4 In the **y** text field, type 2.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Design Domain

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

2 In the **Settings** window for **Box Selection**, type `Design Domain` in the **Label** text field.

3 Locate the **Box Limits** section. In the **y minimum** text field, type `H-dH/2`.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

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

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

3 From the **Specify** list, choose **Bulk modulus and density**.

4 From the K list, choose **User defined**. In the associated text field, type `K1*(1+alpha_K*i)`.

5 From the ρ list, choose **User defined**. In the associated text field, type `rho1`.

Monopole Point Source 1

1 In the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.

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

3 From the **Selection** list, choose **Point 1**.

4 Locate the **Point Source** section. In the Q_S text field, type `0.02`.

MESH 1

Free Triangular 1

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

Size 1

1. 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 **Domain**.
4. From the **Selection** list, choose **Design Domain**.
5. Click to expand the **Element Size Parameters** section. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `hmax/2`.

Size 2

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 **Point**.
4. From the **Selection** list, choose **Point 1**.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `0.1*hmax`.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.
3. In the **Maximum element size** text field, type `hmax`.
4. Click **Build All**.

DEFINITIONS

Objective Function

1. In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.
2. In the **Settings** window for **Domain Probe**, type `Objective Function` in the **Label** text field.
3. In the **Variable name** text field, type `obj`.
4. Locate the **Source Selection** section. From the **Selection** list, choose **Circle 1**.

   The `realdot` operator is analytic, so if we use this to define the probe, it can be used as objective function without enabling splitting of complex variables.

5 Locate the **Expression** section. In the **Expression** text field, type `0.5*realdot(p,p)/acpr.pref_SPL^2`.

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, type `24` in the **Start** text field.

5 In the **Step** text field, type `0.1`.

6 In the **Stop** text field, type `42`.

7 Click **Replace**.

8 In the **Settings** window for **Frequency Domain**, click to expand the **Results While Solving** section.

9 From the **Probes** list, choose **None**.

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

11 In the **Settings** window for **Study**, type `Study 1 - Initial Design` in the **Label** text field.

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

RESULTS

Acoustic Pressure (acpr)

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

2 From the **Parameter value (freq (Hz))** list, choose **34.5**.

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

Sound Pressure Level (acpr)

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

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

3 From the **Parameter value (freq (Hz))** list, choose **34.5**.

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

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

Acoustic Pressure (acpr), Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr)** and **Sound Pressure Level (acpr)**.

2 Right-click and choose **Group**.

Study 1 - Initial Design

In the **Settings** window for **Group**, type `Study 1 - Initial Design` in the **Label** text field.

COMPONENT 1 (COMP1)

Set up the topology optimization by adding a **Density Model** and interpolate the inverse density and inverse bulk modulus linearly.

Density Model 1 (dtopo1)

1 In the **Definitions** toolbar, click **Optimization** and choose **Topology Optimization> Density Model**.

2 In the **Settings** window for **Density Model**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Design Domain**.

4 Locate the **Filtering** section. From the $R_{\min}$ list, choose **User defined**.

5 In the text field, type `hmax`.

Use milling constraints to restrict the design freedom.

6 Click to expand the **Milling** section. From the **Milling constraints** list, choose **Enabled**.

7 In the table, enter the following settings:

X	Y
0	-1

8 Locate the **Projection** section. From the **Projection type** list, choose **Hyperbolic tangent projection**.

9 In the β text field, type `beta`.

10 Locate the **Interpolation** section. From the **Interpolation type** list, choose **Linear**.

11 Locate the **Control Variable Discretization** section. From the **Element order** list, choose **Constant**.

12 Locate the **Control Variable Initial Value** section. In the θ_0 text field, type `1`.

Prescribed Material Boundary 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Topology Optimization> Prescribed Material Boundary**.

2 Select Boundary 4 only.

DEFINITIONS

Design Domain Variables

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Design Domain Variables` in the **Label** text field.

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

4 From the **Selection** list, choose **Design Domain**.

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

Name	Expression	Unit	Description
rhod_inv	1/(1/rho2+ dtopo1.theta_p*(1/ rho1-1/rho2))	kg/m³	Design domain density
Kd_inv	1/(1/K2+ dtopo1.theta_p*(1/K1- 1/K2))	Pa	Design domain bulk modulus

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics (Design Domain)

1 In the **Physics** toolbar, click **Domains** and choose **Pressure Acoustics**.

2 In the **Settings** window for **Pressure Acoustics**, type `Pressure Acoustics (Design Domain)` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Design Domain**.

4 Locate the **Pressure Acoustics Model** section. From the **Specify** list, choose **Bulk modulus and density**.

5 From the K list, choose **User defined**. In the associated text field, type `Kd_inv*(1+alpha_K*i)`.

6 From the ρ list, choose **User defined**. In the associated text field, type `rhod_inv`.

Disable the feature in the 1st study.

STUDY 1 - INITIAL DESIGN

Step 1: Frequency Domain

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

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)> Pressure Acoustics (Design Domain)**.

5 Click **Disable**.

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> Frequency Domain**.

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: Frequency Domain

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

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

Topology Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Topology Optimization**.

2 In the **Settings** window for **Topology Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type `50`.

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>comp1.obj - Objective Function**.

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

Expression	Description
`log10(comp1.obj)`	`Objective Domain SPL`

6 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)>Definitions>Density Model 1>Global>comp1.dtopo1.theta_avg - Average material volume factor**.

7 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound
`comp1.dtopo1.theta_avg`	`volfrac`	

8 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

Initialize the study to generate a plot for use while solving.

9 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Acoustic Pressure (acpr) 1, Sound Pressure Level (acpr) 1

In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr) 1** and **Sound Pressure Level (acpr) 1**.

Acoustic Pressure (acpr) 1

Drag and drop on **Topology Optimization**.

Study 2 - Topology Optimization

1 In the **Model Builder** window, under **Results** click **Topology Optimization**.

2 In the **Settings** window for **Group**, type `Study 2 - Topology Optimization` in the **Label** text field.

Surface 1

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

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.Lp_t - Total sound pressure level - dB**.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Color table**.

STUDY 2

Topology Optimization

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

2 In the **Settings** window for **Topology Optimization**, locate the **Output While Solving** section.

3. Select the **Plot** check box.
4. From the **Plot group** list, choose **Threshold**.

Use a **Segregated** solver to reduce the computational time.

Solver Configurations

In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

Solution 2 (sol2)

1. In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Optimization Solver 1** node.
2. Right-click **Stationary 1** and choose **Segregated**.
3. In the **Settings** window for **Segregated**, locate the **General** section.
4. From the **Termination technique** list, choose **Iterations**.
5. Right-click **Segregated 1** and choose **Segregated Step**twice.
6. In the **Settings** window for **Segregated Step**, type `Optimization` in the **Label** text field.
7. Locate the **General** section. In the **Variables** list, choose **Milling material volume factor (comp1.dtopo1.theta_m1)** and **Pressure (comp1.p)**.
8. Under **Variables**, click **Delete**.
9. In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)> Optimization Solver 1>Stationary 1>Segregated 1** click **Segregated Step 1**.
10. In the **Settings** window for **Segregated Step**, type `Milling` in the **Label** text field.
11. Locate the **General** section. Under **Variables**, click **Add**.
12. In the **Add** dialog box, select **Milling material volume factor (comp1.dtopo1.theta_m1)** in the **Variables** list.
13. Click **OK**.
14. In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)> Optimization Solver 1>Stationary 1>Segregated 1** click **Segregated Step 2**.
15. In the **Settings** window for **Segregated Step**, type `Acoustics` in the **Label** text field.
16. Locate the **General** section. Under **Variables**, click **Add**.
17. In the **Add** dialog box, select **Pressure (comp1.p)** in the **Variables** list.
18. Click **OK**.
19. In the **Model Builder** window, click **Study 2**.
20. In the **Settings** window for **Study**, type `Study 2 - Topology Optimization` in the **Label** text field.

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

Use a **Parametric Sweep** to perform continuation in the projection slope parameter `beta`.

Parametric Sweep

1. In the **Study** toolbar, click **Parametric Sweep**.
2. In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3. Click **Add**.
4. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
beta (Projection slope)	4 8 16 32	

5. Locate the **Output While Solving** section. From the **Probes** list, choose **None**.
6. Click to expand the **Advanced Settings** section. Select the **Reuse solution from previous step** check box.
7. In the **Study** toolbar, click **Compute**.

ROOT

Add a study to generate the spectrum for the topology optimized design.

ADD STUDY

1. In the **Study** 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> Frequency Domain**.
4. Click **Add Study** in the window toolbar.
5. In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain

1. In the **Settings** window for **Frequency Domain**, locate the **Results While Solving** section.
2. From the **Probes** list, choose **None**.
3. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Topology Optimization (Component 1)**.

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

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

6 From the **Study** list, choose **Study 2 - Topology Optimization, Frequency Domain**.

7 Locate the **Study Settings** section. Click **Range**.

8 In the **Range** dialog box, type `24` in the **Start** text field.

9 In the **Step** text field, type `0.1`.

10 In the **Stop** text field, type `42`.

11 Click **Replace**.

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

13 In the **Settings** window for **Study**, type `Study 3 - Optimized Spectrum` in the **Label** text field.

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

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

RESULTS

Add a **1D Plot Group** to visualize the objective function as a function of the frequency.

Response Comparison

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

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

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

4 In the **Title** text area, type `SPL in Objective Domain`.

Global 1

1 Right-click **Response Comparison** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>obj - Objective Function**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`10*log10(obj)`		`Initial Design`

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

Global 2

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

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

3 From the **Dataset** list, choose **Study 3 - Optimized Spectrum/Solution 8 (sol8)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`10*log10(obj)`		`Topology Optimization`

Add a 2nd component with sound hard boundaries to verify that the optimization result can be trusted.

Filter

The exported geometry becomes smoother, if the milling material volume factor is used instead of the (projected) material volume factor.

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Filter**.

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

3 In the **Expression** text field, type `if(isdefined(dtopo1.theta_m), dtopo1.theta_m, 1)`.

4 Locate the **Evaluation** section. From the **Smoothing** list, choose **Inside material domains**.

5 Select the **Use derivatives** check box.

6 Click **Plot**.

7 Right-click **Results>Datasets>Filter** and choose **Create Mesh Part**.

MESH PART 1

Import 1

1 In the **Settings** window for **Import**, locate the **Import** section.

2 Click **Import**.

3 Click **Build All**.

4 In the **Model Builder** window, right-click **Mesh Part 1** and choose **Create Geometry**.

ADD PHYSICS

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

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

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

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Study 1 - Initial Design**, **Study 2 - Topology Optimization**, and **Study 3 - Optimized Spectrum**.

5 Click **Add to Component 2** in the window toolbar.

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 2 (ACPR2)

Pressure Acoustics 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Pressure Acoustics, Frequency Domain 2 (acpr2)** click **Pressure Acoustics 1**.

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

3 From the **Specify** list, choose **Bulk modulus and density**.

4 From the K list, choose **User defined**. In the associated text field, type `K1*(1+alpha_K*i)`.

5 From the ρ list, choose **User defined**. In the associated text field, type `rho1`.

Monopole Point Source 1

1 In the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.

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

3 From the **Selection** list, choose **Point 1 (Import 1)**.

4 Locate the **Point Source** section. In the Q_S text field, type `0.02`.

MESH 2

Free Triangular 1

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

Size 1

1 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 **Point**.

4 From the **Selection** list, choose **Point 1 (Import 1)**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.1*hmax`.

Size

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Size**.

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

3 In the **Maximum element size** text field, type `hmax`.

4 Click **Build All**.

DEFINITIONS (COMP2)

Objective Function

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `Objective Function` in the **Label** text field.

3 In the **Variable name** text field, type `obj`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Circle 1 (Import 1)**.

5 Locate the **Expression** section. In the **Expression** text field, type `0.5*realdot(p2,p2)/acpr2.pref_SPL^2`.

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>Frequency Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Pressure Acoustics, Frequency Domain (acpr)**.

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

6 In the **Model Builder** window, click the root node.

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

STUDY 4

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Results While Solving** section.

2 From the **Probes** list, choose **None**.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Topology Optimization (Component 1)**.

4 Locate the **Study Settings** section. Click **Range**.

5 In the **Range** dialog box, type `24` in the **Start** text field.

6 In the **Step** text field, type `0.1`.

7 In the **Stop** text field, type `42`.

8 Click **Replace**.

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

10 In the **Settings** window for **Study**, type `Study 4 - Verification` in the **Label** text field.

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

RESULTS

Topology Optimization

In the **Model Builder** window, under **Results** right-click **Topology Optimization** and choose **Delete**.

Acoustic Pressure (acpr2), Sound Pressure Level (acpr2)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr2)** and **Sound Pressure Level (acpr2)**.

2 Right-click and choose **Group**.

Study 4 - Verification

In the **Settings** window for **Group**, type `Study 4 - Verification` in the **Label** text field.

Acoustic Pressure (acpr2)

1 In the **Model Builder** window, click **Acoustic Pressure (acpr2)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **34.5**.

4 In the **Acoustic Pressure (acpr2)** toolbar, click **Plot**.

Sound Pressure Level (acpr2)

1 In the **Model Builder** window, click **Sound Pressure Level (acpr2)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **34.5**.

4 In the **Sound Pressure Level (acpr2)** toolbar, click **Plot**.

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

Global 3

1 In the **Model Builder** window, under **Results>Response Comparison** right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 4 - Verification/Solution 9 (7) (sol9)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`10*log10(obj)`		`Sound Hard Boundaries`

The transition to sound hard boundaries causes the spectrum to shift slightly. Plot the position of the optimization frequency to see that this results in a slightly higher objective function.

Global 4

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`10*log10(obj)`		`Optimization Frequency`

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `f0`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

7 In the **Response Comparison** toolbar, click **Plot**.

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