Created in COMSOL Multiphysics 6.1

Noise Radiation by a Compound Gear Train

Introduction

This example illustrates the modeling of noise radiation from the housing of a gear train. First, a multibody analysis is performed in the time domain to compute the housing vibrations at the specified driver shaft speed. Then, an acoustic analysis is performed at a selected frequency to compute the sound pressure levels in the near, far, and exterior fields using the housing normal acceleration as a noise source.

Predicting the noise radiation from a dynamic system gives designers an insight into the behavior of moving mechanisms early in the design process. For example, consider a gearbox in which the change in the gear mesh stiffness causes sustained vibrations. These vibrations are transmitted to the gearbox housing through shafts and joints. The vibrating housing further transmits energy to the surrounding fluid, resulting in acoustic wave radiation.

Note: This model requires the Acoustics Module and the Multibody Dynamics Module.

Model Definition

The gear train enclosed in an acoustic domain is shown in Figure 1. The details of the compound gear train is shown in Figure 2.

This model is solved in two steps:

1. *Multibody analysis*: In the first part of the model, a transient analysis is performed to compute the dynamics of the gears as well as the acceleration levels on the gear train housing.
2. *Acoustic analysis*: In the second part of the model, a spherical domain enclosing the gear train housing is created. The computed accelerations on the housing are used as noise source for the acoustic domain. A frequency domain analysis is performed in order to compute the sound pressure levels outside the gear train.

Note: For the details of the multibody analysis of the gear train, see the model *Vibrations in a Compound Gear Train* in the Multibody Dynamics Module Application Library.

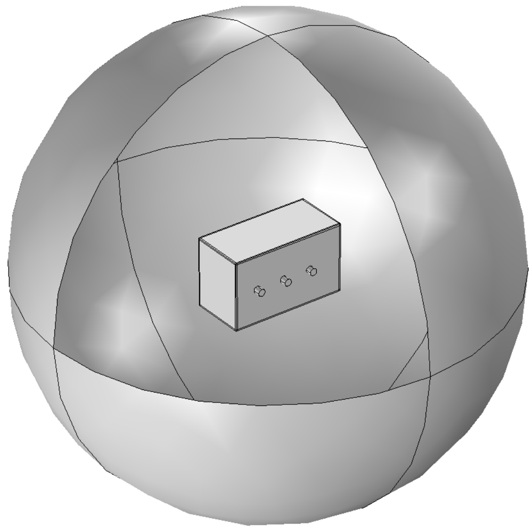

Figure 1: Model geometry of the gear train enclosed in a spherical acoustic domain.

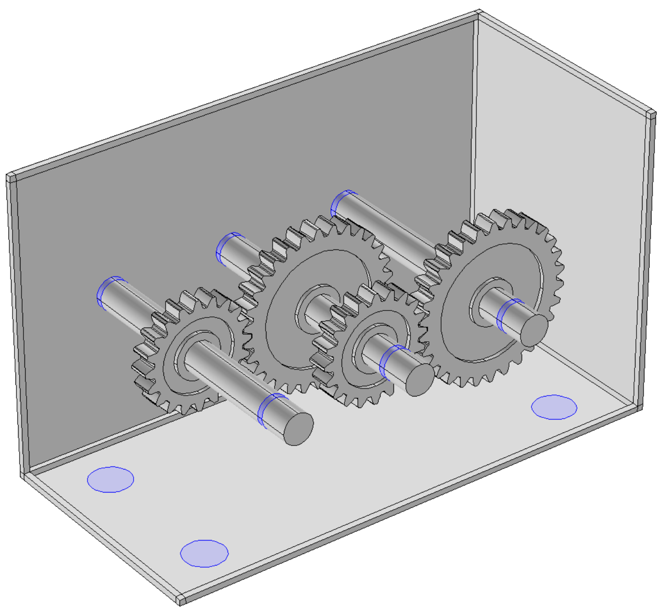

Figure 2: Details of the compound gear train.

Multibody-Acoustics Coupling

Unidirectional Coupling

When coupling multibody/structural and acoustics physics, unidirectional coupling can be assumed if the exterior fluid is air (or any other lighter fluid). This implies that the vibrations from the gear-train housing impacts the surrounding fluid, whereas the feedback from the acoustic waves to the structure is neglected. This model uses such an approach.

FFT Solver

The multibody dynamics is solved in the time domain, whereas the acoustics is solved in the frequency domain. Therefore, the FFT solver is used to convert the housing accelerations from the time domain to the frequency domain.

Domain Equations

This model solves the problem in the frequency domain using the Pressure Acoustics, Frequency Domain interface. The model equation is a slightly modified version of the Helmholtz equation for the acoustic pressure p:

$$\nabla \cdot \left(-\frac{\nabla p}{\rho}\right) - \frac{\omega^2 p}{c^2 \rho} = 0$$

where ρ is the density, c is the speed of sound, and ω is the angular frequency.

Boundary Conditions

The following boundary conditions are applied in the acoustic domain:

- Normal acceleration of the gear train housing is applied on the interior boundaries of the acoustics domain.

  $$-\boldsymbol{n} \cdot \left\langle -\frac{\nabla p}{\rho} \right\rangle = a_n$$

  where a_n is the normal acceleration.
- A spherical wave radiation condition is applied on the exterior boundaries of the acoustic domain.
- An exterior field calculation is added on the exterior boundaries of the acoustic domain to compute the sound pressure levels in the exterior field.

MODEL PARAMETERS

The domain material is chosen as air. The following parameters are used in the model:

- The excitation frequency is set to 2000 Hz.
- The size of enclosing sphere is set to 0.5 m.
- Exterior field results are evaluated at a distance of 2 m from the center.

Results and Discussion

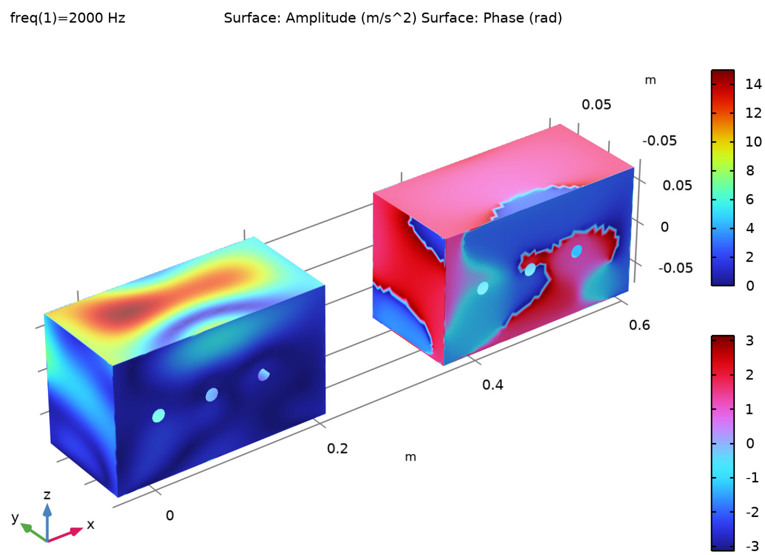

Figure 3: Amplitude and phase of normal acceleration of housing .

Figure 3 shows the amplitude (left) and phase (right) of the normal acceleration of the housing at 2000 Hz. The top part vibrates with higher velocity compared to the rest of the housing. This part also has constant phase as indicated by the color distribution in the second plot.

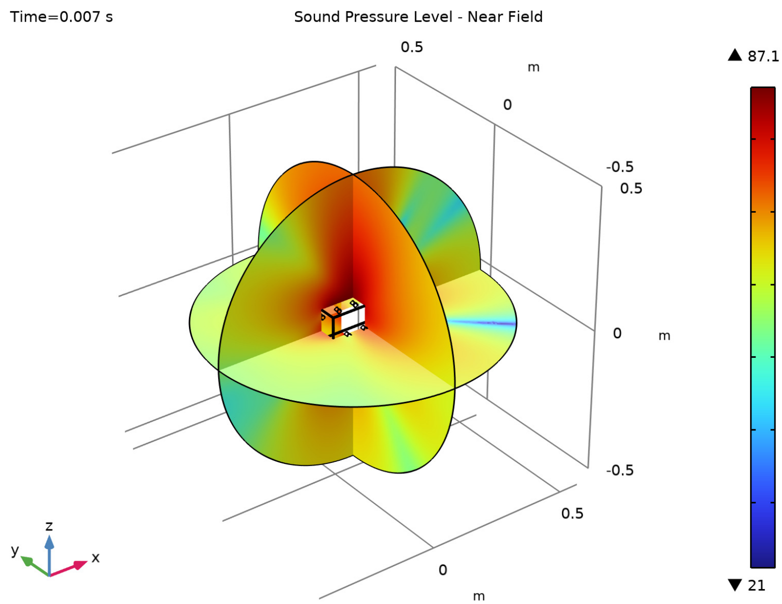

Figure 4: Sound Pressure Level (dB) in the near-field region at 2000 Hz.

Figure 4 shows the sound pressure level (SPL) in the acoustic domain. Different SPL values can be seen at different locations in space, with a maximum at the top of the gear train. The SPL field on the housing/casing surface is shown in Figure 5.

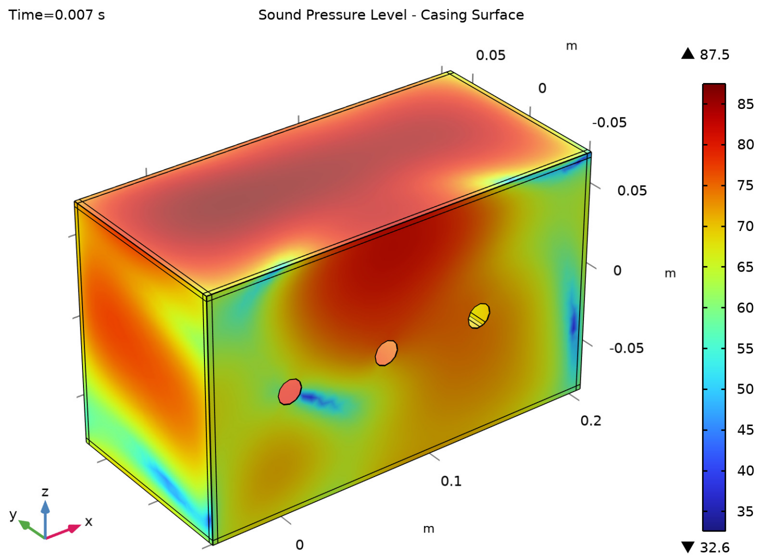

Figure 5: Sound Pressure Level (dB) on the casing surface at 2000 Hz.

The sound pressure levels in the far field at a distance of 2 m are shown in Figure 6, Figure 7, and Figure 8. All these plots show the sound pressure levels in different planes.

Figure 9 shows a three-dimensional plot of the sound pressure level, giving an idea of the dominant directions of noise radiation.

It is also possible to plot the acoustic wave propagation in a domain exterior to the modeled spherical domain. Figure 10 shows a plot of the acoustics waves propagating in the exterior domain.

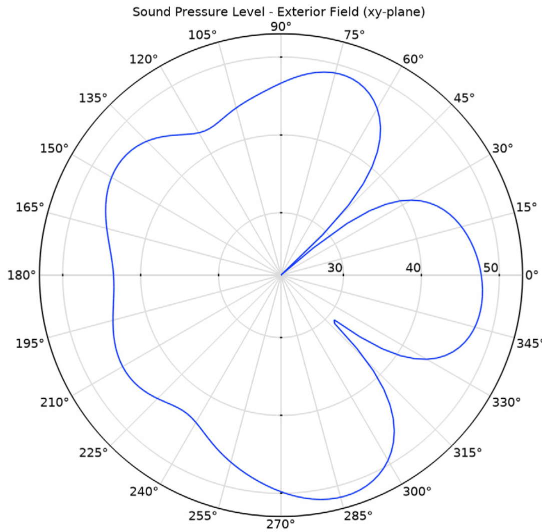

Figure 6: Exterior-field sound pressure level in the xy-plane at a distance of 2 m.

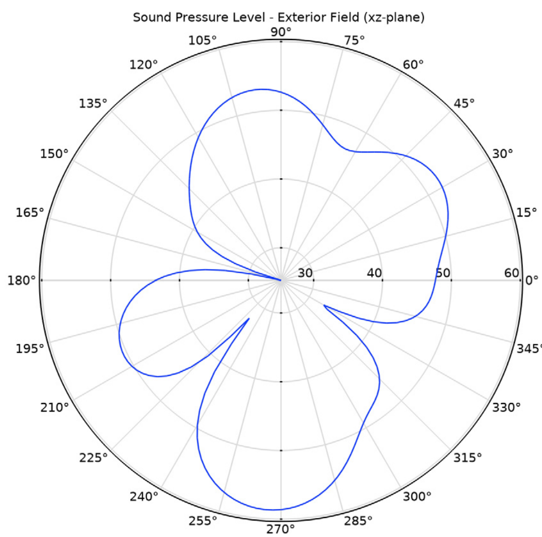

Figure 7: Exterior-field sound pressure level in the xz-plane at a distance of 2 m.

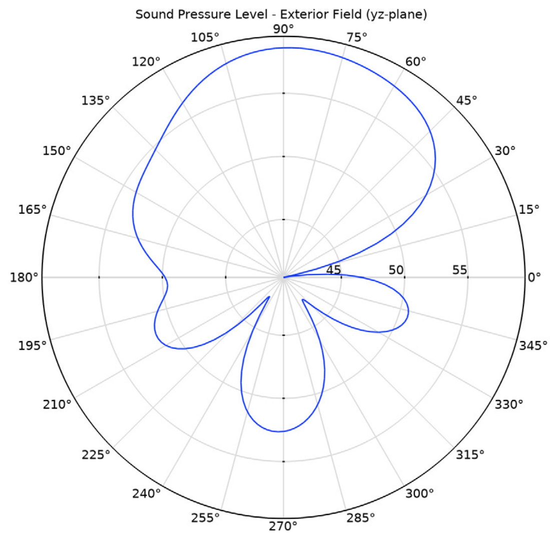

Figure 8: Exterior-field sound pressure level in the yz-plane at a distance of 2 m.

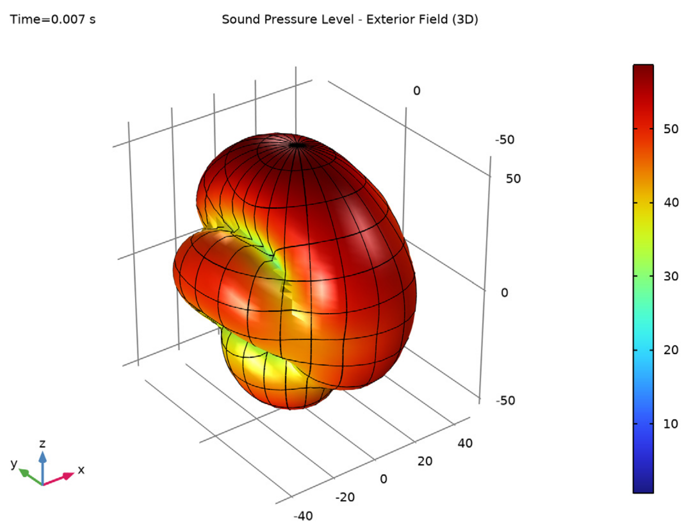

Figure 9: Exterior-field sound pressure level at a distance of 2 m.

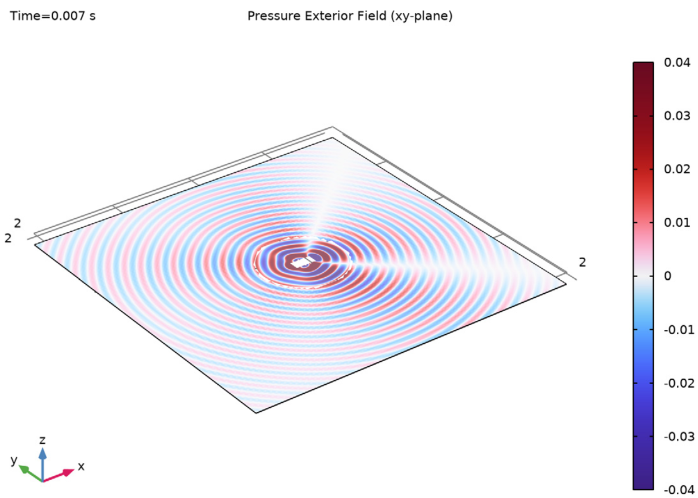

Figure 10: The acoustic wave propagation in the exterior domain.

Notes About the COMSOL Implementation

- The acoustics is set up in a separate component so that the multibody model setup is unaffected. A General Extrusion nonlocal coupling is used to map the acceleration values between the geometries of the two components.
- The normal acceleration of the housing is in the time domain. A Time to Frequency FFT study is used to convert it to the frequency domain.
- The presence of the Multibody Dynamics physics, by default, forces the solver to be nonlinear. Hence it is manually set to linear for the acoustic analysis.

Application Library path: `Acoustics_Module/Vibrations_and_FSI/gear_train_noise`

Modeling Instructions

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Multibody Dynamics Module>Tutorials, Transmission>gear_train** in the tree.

3 Click **Open**.

Enter the required set of parameters for the acoustic analysis.

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 In the table, enter the following settings:

Name	Expression	Value	Description
x0	0.07[m]	0.07 m	Sphere offset
R	0.5[m]	0.5 m	Sphere radius
f0	2000[Hz]	2000 Hz	Frequency of interest

COMPONENT 2 (COMP2)

Create a selection for the exterior boundaries of the gear train housing. The acceleration computed on these boundaries is used as a noise source for the acoustic analysis.

DEFINITIONS (COMP2)

Explicit 7

1 In the **Model Builder** window, expand the **Component 2 (comp2)** node.

2 Right-click **Component 2 (comp2)>Definitions** and choose **Selections>Explicit**.

3 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type 14,38,46,53,65,132,219,220,225,455, 456,461,919,920,925 in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

9 Select the **Group by continuous tangent** check box.

General Extrusion 1 (genext1)

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

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

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

4 From the **Selection** list, choose **Explicit 7**.

5 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

6 Locate the **Destination Map** section. In the **X-expression** text field, type `X`.

7 In the **Y-expression** text field, type `Y`.

8 In the **Z-expression** text field, type `Z`.

9 Click to expand the **Advanced** section. From the **Mesh search method** list, choose **Closest point**.

STUDY 1 - GEAR 2D

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

2 In the **Settings** window for **Study**, type `Study 1 - Gear 2D` in the **Label** text field.

STUDY 2 - GEAR TRAIN

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

2 In the **Settings** window for **Study**, type `Study 2 - Gear Train` in the **Label** text field.

ADD COMPONENT

Right-click **Study 2 - Gear Train** and choose **Add Component>3D**.

GEOMETRY 3

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 `gear_train_noise.mphbin`.

5 Click **Import**.

Create a domain surrounding the gear train assembly for the acoustic analysis. In order to do this, add a spherical domain of a finite size and remove the gear train assembly from it.

Convert to Solid 1 (csol1)

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

2 Click the **Transparency** button in the **Graphics** toolbar.

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

4 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type `R`.

4 Locate the **Position** section. In the **x** text field, type `x0`.

5 Click **Build Selected**.

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

Difference 1 (dif1)

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

2 Select the object **sph1** only.

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

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

5 Select the object **csol1** only.

6 Click **Build Selected**.

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 **Mathematics>ODE and DAE Interfaces>Boundary ODEs and DAEs (bode)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Study 1 - Gear 2D** and **Study 2 - Gear Train**.

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

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

7 In the table, clear the **Solve** check boxes for **Study 1 - Gear 2D** and **Study 2 - Gear Train**.

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

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

ADD MATERIAL

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

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

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

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

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

DEFINITIONS (COMP3)

Outer Boundaries

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

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

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

4 Select Boundary 1 only.

5 Select the **Group by continuous tangent** check box.

6 In the **Label** text field, type `Outer Boundaries`.

Casing

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

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

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

4 Locate the **Box Limits** section. In the **y minimum** text field, type `-0.06`.

5 In the **y maximum** text field, type `0.06`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

BOUNDARY ODES AND DAES (BODE)

1 In the **Model Builder** window, under **Component 3 (comp3)** click **Boundary ODEs and DAEs (bode)**.

2 In the **Settings** window for **Boundary ODEs and DAEs**, locate the **Boundary Selection** section.

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

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `acceleration` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Acceleration (m/s^2)**.

8 Click **OK**.

9 In the **Settings** window for **Boundary ODEs and DAEs**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `length` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Length (m)**.

14 Click **OK**.

15 In the **Settings** window for **Boundary ODEs and DAEs**, click to expand the **Discretization** section.

16 From the **Shape function type** list, choose **Lagrange**.

17 From the **Frame** list, choose **Material**.

18 Click to expand the **Dependent Variables** section. In the **Field name** text field, type `an`.

19 In the **Dependent variables** table, enter the following settings:

an

Distributed ODE 1

1 In the **Model Builder** window, under **Component 3 (comp3)> Boundary ODEs and DAEs (bode)** click **Distributed ODE 1**.

2 In the **Settings** window for **Distributed ODE**, locate the **Source Term** section.

3 In the f text field, type `0`.

4 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `0`.

Initial Values 1

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

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

3 In the an text field, type `comp2.genext1(comp2.mbd2.an)`.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

In the **Model Builder** window, under **Component 3 (comp3)** click **Pressure Acoustics, Frequency Domain (acpr)**.

Spherical Wave Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spherical Wave Radiation**.
2 In the **Settings** window for **Spherical Wave Radiation**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Outer Boundaries**.
4 Locate the **Spherical Wave Radiation** section. Specify the $\mathbf{r}_0$ vector as

x0	x
0	y
0	z

The normal acceleration computed in **Study 2 - Gear Train** is used as an acceleration source for the acoustic analysis.

Normal Acceleration 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.
2 In the **Settings** window for **Normal Acceleration**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Casing**.
4 Locate the **Normal Acceleration** section. In the a_n text field, type `comp3.an`.

Exterior Field Calculation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.
2 In the **Settings** window for **Exterior Field Calculation**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Outer Boundaries**.
4 Click the **Show More Options** button in the **Model Builder** toolbar.
5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
6 Click **OK**.
7 In the **Settings** window for **Exterior Field Calculation**, click to expand the **Advanced Settings** section.
8 Clear the **Use polynomial-preserving recovery for the normal gradient** check box.

MESH 3

The mesh size is chosen such that there are 5 elements per wavelength at the frequency of interest.

Free Tetrahedral 1

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `343[m/s]/f0/5`.

5 In the **Model Builder** window, right-click **Mesh 3** and choose **Build All**.

Boundary Layers 1

In the **Mesh** toolbar, click **Boundary Layers**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Outer Boundaries**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `1`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `343[m/s]/f0/5/10`.

7 In the **Model Builder** window, right-click **Mesh 3** and choose **Build All**.

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 **Empty Study**.

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

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

STUDY 2 - GEAR TRAIN

In the **Study** toolbar, click **Update Solution**.

STUDY 3 - ACOUSTICS

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

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

Transform the time-dependent acceleration data to the frequency domain using a **Time to Frequency FFT** study step, then use this data as input for the **Frequency Domain** study step.

Time to Frequency FFT

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Time to Frequency FFT**.

2 In the **Settings** window for **Time to Frequency FFT**, locate the **Study Settings** section.

3 From the **Prescribed by** list, choose **Initial expression**.

4 From the **Input study** list, choose **Study 2 - Gear Train, Time Dependent**.

5 In the **End time** text field, type `7e-3`.

6 In the **Maximum output frequency** text field, type `1/(7e-5)`.

7 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Multibody Dynamics (mbd)**, **Multibody Dynamics 2 (mbd2)**, and **Pressure Acoustics, Frequency Domain (acpr)**.

Step 2: Frequency Domain

1 In the **Model Builder** window, right-click **Study 3 - Acoustics** and choose **Study Steps>Frequency Domain>Frequency Domain**.

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

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

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Multibody Dynamics (mbd)**, **Multibody Dynamics 2 (mbd2)**, and **Boundary ODEs and DAEs (bode)**.

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

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

7 From the **Study** list, choose **Study 3 - Acoustics, Time to Frequency FFT**.

8 From the **Selection** list, choose **Automatic (all solutions)**.

Set the solver to linear as by default Multibody Dynamics sets it to nonlinear.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 From the **Linearity** list, choose **Linear**.

5 In the **Model Builder** window, click **Study 3 - Acoustics**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

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

RESULTS

Study 3 - Acoustics/Solution 3 (9) (sol3)

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

2 Right-click **Results>Datasets>Study 3 - Acoustics/Solution 3 (8) (sol3)** and choose **Duplicate**.

Selection

1 In the **Model Builder** window, right-click **Study 3 - Acoustics/Solution 3 (9) (sol3)** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Casing**.

Study 3 - Acoustics/Solution 3 (10) (sol3)

In the **Model Builder** window, under **Results>Datasets** right-click **Study 3 - Acoustics/Solution 3 (9) (sol3)** and choose **Duplicate**.

Selection

1 In the **Model Builder** window, expand the **Study 3 - Acoustics/Solution 3 (10) (sol3)** node, then click **Selection**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5-7,10-14,16-27,29-39,43-45,48-49,53-55,57,59,67-69,71,73-96` in the **Selection** text field.

6 Click **OK**.

Grid 3D 1

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

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

3 From the **Dataset** list, choose **Study 3 - Acoustics/Solution 3 (8) (sol3)**.

4 Locate the **Parameter Bounds** section. Find the **First parameter** subsection. In the **Minimum** text field, type `-2`.

5 In the **Maximum** text field, type `2`.

6 Find the **Second parameter** subsection. In the **Minimum** text field, type `-2`.

7 In the **Maximum** text field, type `2`.

8 Find the **Third parameter** subsection. In the **Maximum** text field, type `0`.

9 Click to expand the **Grid** section. In the **x resolution** text field, type `100`.

10 In the **y resolution** text field, type `100`.

11 In the **z resolution** text field, type `2`.

Use the following instructions to plot the amplitude and phase of the normal acceleration of the gear train housing as shown in Figure 3.

Housing Normal Acceleration: Amplitude and Phase

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 From the **Dataset** list, choose **Study 3 - Acoustics/Solution 3 (9) (sol3)**.

4 In the **Label** text field, type `Housing Normal Acceleration: Amplitude and Phase`.

Surface 1

1 Right-click **Housing Normal Acceleration: Amplitude and Phase** and choose **Surface**.

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

3 In the **Expression** text field, type `abs(acpr.nacc)`.

Surface 2

1 In the **Model Builder** window, right-click **Housing Normal Acceleration: Amplitude and Phase** and choose **Surface**.

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

3 In the **Expression** text field, type `arg(acpr.nacc)`.

Translation 1

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

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

3 In the **x** text field, type `0.4`.

Surface 2

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

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

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Wave>Disco** in the tree.

5 Click **OK**.

Housing Normal Acceleration: Amplitude and Phase

1 In the **Model Builder** window, click **Housing Normal Acceleration: Amplitude and Phase**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Surface: Amplitude (m/s^2) Surface: Phase (rad)`.

5 Locate the **Color Legend** section. From the **Position** list, choose **Right double**.

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

7 In the **Housing Normal Acceleration: Amplitude and Phase** toolbar, click **Plot**.

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

9 Click the **Transparency** button in the **Graphics** toolbar.

Use the following instructions to plot the spatial distribution of the sound pressure level in the spherical domain and on the surface of gear train housing as shown in Figure 4 and Figure 5, respectively.

SPL Near Field

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Acoustics/ Solution 3 (8) (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Near Field`.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Slice 1

1 Right-click **SPL Near Field** and choose **Slice**.

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

3 In the **Expression** text field, type `acpr.Lp_t`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

Slice 2

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

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

3 From the **Plane** list, choose **XY-planes**.

4 In the **Planes** text field, type `1`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Slice 3

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

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

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

5 In the **SPL Near Field** toolbar, click **Plot**.

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

SPL Casing Surface

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Acoustics/ Solution 3 (9) (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Casing Surface`.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **SPL Casing Surface** and choose **Surface**.

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

3 In the **Expression** text field, type `acpr.Lp_t`.

4 In the **SPL Casing Surface** toolbar, click **Plot**.

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

Follow the instructions below to plot the spatial distribution of the sound pressure level in the exterior-field region as shown in Figure 6, Figure 7, Figure 8, and Figure 9, respectively.

Polar SPL xy-plane

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

2 In the **Settings** window for **Polar Plot Group**, type `Polar SPL xy-plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Acoustics/ Solution 3 (8) (sol3)**.

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

5 In the **Title** text area, type `Sound Pressure Level - Exterior Field (xy-plane)`.

Radiation Pattern 1

1 In the **Polar SPL xy-plane** toolbar, click **More Plots** and choose **Radiation Pattern**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

4 Find the **Center** subsection. In the **x** text field, type `x0`.

5 Find the **Evaluation distance** subsection. In the **Radius** text field, type `4*R`.

6 In the **Polar SPL xy-plane** toolbar, click **Plot**.

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

Polar SPL xz-plane

1 In the **Model Builder** window, right-click **Polar SPL xy-plane** and choose **Duplicate**.

2 In the **Model Builder** window, click **Polar SPL xy-plane 1**.

3 In the **Settings** window for **Polar Plot Group**, type `Polar SPL xz-plane` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Sound Pressure Level - Exterior Field (xz-plane)`.

Radiation Pattern 1

1 In the **Model Builder** window, click **Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Normal vector** subsection. In the **y** text field, type `1`.

4 In the **z** text field, type `0`.

5 In the **Polar SPL xz-plane** toolbar, click **Plot**.

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

Polar SPL yz-plane

1 In the **Model Builder** window, right-click **Polar SPL xz-plane** and choose **Duplicate**.

2 In the **Model Builder** window, click **Polar SPL xz-plane 1**.

3 In the **Settings** window for **Polar Plot Group**, type `Polar SPL yz-plane` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Sound Pressure Level - Exterior Field (yz-plane)`.

Radiation Pattern 1

1 In the **Model Builder** window, click **Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Normal vector** subsection. In the **y** text field, type `0`.

4 In the **x** text field, type `1`.

5 In the **Polar SPL yz-plane** toolbar, click **Plot**.

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

3D Polar SPL

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Acoustics/ Solution 3 (8) (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Exterior Field (3D)`.

Radiation Pattern 1

1 In the **3D Polar SPL** toolbar, click **More Plots** and choose **Radiation Pattern**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of elevation angles** text field, type `50`.

4 In the **Number of azimuth angles** text field, type `50`.

5 Find the **Sphere** subsection. From the **Sphere** list, choose **Manual**.

6 In the **X** text field, type `x0`.

7 In the **Radius** text field, type `4*R`.

8 Locate the **Coloring and Style** section. From the **Grid** list, choose **Fine**.

9 In the **3D Polar SPL** toolbar, click **Plot**.

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

Finally, plot the spatial distribution of the sound pressure level in the exterior field as shown in Figure 10.

Pressure Exterior Field

1. In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Pressure Exterior Field` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Grid 3D 1**.
4. Locate the **Title** section. From the **Title type** list, choose **Manual**.
5. In the **Title** text area, type `Pressure Exterior Field (xy-plane)`.

Surface 1

1. Right-click **Pressure Exterior Field** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `if(sqrt((x-x0)^2+y^2+z^2)>R,pext(x,y,z), NaN)`.
4. Click to expand the **Range** section. Select the **Manual color range** check box.
5. In the **Minimum** text field, type `-.04`.
6. In the **Maximum** text field, type `.04`.
7. Locate the **Coloring and Style** section. Click **Change Color Table**.
8. In the **Color Table** dialog box, select **Wave>Wave** in the tree.
9. Click **OK**.
10. In the **Pressure Exterior Field** toolbar, click **Plot**.

Slice 1

1. In the **Model Builder** window, right-click **Pressure Exterior Field** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 3 - Acoustics/Solution 3 (9) (sol3)**.
4. Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.
5. In the **Planes** text field, type `1`.
6. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
7. In the **Pressure Exterior Field** toolbar, click **Plot**.
8. Click the **Zoom Extents** button in the **Graphics** toolbar.

SPL Casing Surface

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

Thumbnail

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Thumbnail` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Acoustics/ Solution 3 (10) (sol3)**.
4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **Thumbnail** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 Click **Change Color Table**.
4 In the **Color Table** dialog box, select **Wave>Wave** in the tree.
5 Click **OK**.
6 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
7 From the **Scale** list, choose **Linear symmetric**.

Surface 2

1 In the **Model Builder** window, right-click **Thumbnail** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2 - Gear Train/Solution 2 (4) (sol2)**.
4 From the **Time (s)** list, choose **0.00301**.
5 Locate the **Expression** section. In the **Expression** text field, type `mbd2.disp`.
6 Locate the **Coloring and Style** section. Click **Change Color Table**.
7 In the **Color Table** dialog box, select **Aurora>AuroraAustralis** in the tree.
8 Click **OK**.

Deformation 1

1 Right-click **Surface 2** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** check box. In the associated text field, type `1`.

Isosurface 1

1 In the **Model Builder** window, right-click **Thumbnail** and choose **Isosurface**.

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

3 In the **Total levels** text field, type `11`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Filter 1

1 Right-click **Isosurface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `y>0`.

Thumbnail

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

2 In the **Thumbnail** toolbar, click  **Plot**.