Ansys

2025 R1

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Mechanical Acoustic Analysis Guide

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

Published in the U.S.A.

Table of Contents

1. Acoustics Analysis Overview 1
2. Acoustics Analysis Types 3
2.1. Harmonic Acoustics Analysis 3
2.2. Harmonic Acoustics Analysis Using Prestressed Structural System 10
2.3. Harmonic Acoustics One-way Acoustic Coupling Analysis 11
2.4. Modal Acoustics Analysis 16
2.5. Static Acoustics Analysis 22
2.6. LS-DYNA Acoustics Analysis 27
3. Acoustics Loading Conditions 31
3.1. Mass Source 32
3.2. Surface Velocity 34
3.3. Diffuse Sound Field 37
3.4. Incident Wave Source 40
3.5. Port In Duct 43
3.6. Mass Source Rate 45
3.7. Surface Acceleration 48
3.8. Temperature 50
3.9. Impedance Sheet 53
3.10. Static Pressure 55
3.11. Pressure 57
3.12. Impedance Boundary 59
3.13. Absorption Surface 63
3.14. Radiation Boundary 65
3.15. Absorption Element 67
3.16. Free Surface 69
3.17. Thermo-Viscous BLI Boundary 71
3.18. Rigid Wall 73
3.19. Symmetry Plane 75
3.20. Port 77
3.21. Far-Field Radiation Surface 80
3.22. Fringe Plot 82
3.23. Transfer Admittance Matrix 83
3.24. Low Reduced Frequency Model 86
4. Acoustic Results 89
4.1. Acoustics Contour Results 89
4.2. Acoustic Far-field Results 90
4.3. Acoustic Frequency Response 93
4.4. Acoustic Power Loss Results 94
4.5. Acoustic Diffuse Sound Transmission Loss 95
4.6. Acoustic Waterfall Diagrams 95

Chapter 1: Acoustics Analysis Overview

In the Ansys Mechanical application™, acoustic analyses and simulations examine how acoustic waves are propagated in enclosed or open volumes. Acoustics is a special type of fluid analysis, one in which the fluid is essentially at rest (or in relatively restricted movement with no gross transport of the fluid, such as water sloshing in a tank). The variation of pressure throughout the acoustic medium is assumed to be small relative to the average pressure of the field.

Using acoustic simulations, you can explore various properties of an acoustic field, such as the pressure levels and how they vary throughout the field as a result of the geometry of the enclosure, the type of acoustic excitation present, the materials used in the space, and so on. You can also include the effects of how the acoustic waves interact with the solid structures that surround the space to predict sound transmission levels through walls, determine the sound levels produced by a vibrating structure, calculate the deformations and stresses in solids due to acoustic pressures, etc.

Acoustic simulations are valuable in a wide range of applications, including the design and analysis of hearing aids, vehicle interiors, acoustic sensors and actuators, sonar devices, wave guides, auditoriums, musical instruments, load speakers and microphones, acoustic test facilities, highway sound barriers, piping systems, environmental control systems, consumer devices of almost any type, noise mufflers, fire alarms, and on and on. Any application where sound levels are of concern is a candidate for acoustic analysis.

The basic workflow of an acoustic analysis is similar to the other workflows in Mechanical. Acoustic simulations do require some different material properties, such as the propagation of the speed of sound in the acoustic medium or the acoustic absorption characteristics of the materials in the space or at the boundaries. The types of boundary conditions are also different: you may have enclosing surfaces that bounce the sound energy back into the acoustic field, or you may have open boundaries that allow the energy to escape completely from the model. Loadings can include pressures, displacement constraints, or flexible surfaces that allow the sound energy to be transmitted into the surrounding structure. Each of these acoustic-specific modeling considerations are discussed in the documentation.

Chapter 2: Acoustics Analysis Types

Ansys Mechanical provides the following acoustic analyses. Within the scope of these analysis types, options are available to enable a wide variety of acoustic behaviors. The documentation sections listed below for each analysis type describe the specifics.

2.1. Harmonic Acoustics Analysis

2.2. Harmonic Acoustics Analysis Using Prestressed Structural System

2.3. Harmonic Acoustics One-way Acoustic Coupling Analysis

2.4. Modal Acoustics Analysis

2.5. Static Acoustics Analysis

2.6. LS-DYNA Acoustics Analysis

2.1. Harmonic Acoustics Analysis

Introduction

Harmonic Acoustics analyses are used to determine the steady-state response of a structure and the surrounding fluid medium to loads and excitations that vary sinusoidally (harmonically) with time. Examples of harmonic acoustics include Sonar (the acoustic counterpart of radar), the design of concert halls, the minimization of noise in a machine shop, noise cancellation in automobiles, audio speaker design, speaker housing design, acoustic filters, mufflers, and Geophysical exploration. Typical quantities of interest in the fluid and far-field location at different frequencies are pressure distribution, pressure gradient, sound power, and particle velocity of acoustic waves.

In Harmonic Response analyses, the following equation is resolved for pure acoustic problems:

$$(\omega^2[M_a] + j\omega[C_a] + [K_a])\{p\} = \{f_F\}$$

For fluid structure interaction problems, the acoustic and the structural matrices are coupled using the following equation:

$$\left(-\omega^2\begin{bmatrix} M_s & 0 \\ \rho_0 R^T & M_f \end{bmatrix} + j\omega\begin{bmatrix} C_s & 0 \\ 0 & C_f \end{bmatrix} + \begin{matrix} K_s & -R \\ 0 & K_f \end{matrix}\right)\begin{Bmatrix} u \\ p \end{Bmatrix} = \begin{Bmatrix} f_s \\ f_F \end{Bmatrix}$$

Mesh Adaptation

During a purely harmonic acoustic analysis (all bodies in the Physics Region object set to **Acoustic**), use the Morphing Region object to perform mesh adaptation during the solution process.

This feature models an enclosure that includes an interior space. The surface (face) around this space defines a boundary. You also specify an external boundary surrounding the enclosure. The area between the two boundaries acts as an infinite acoustic space in the form of a mesh that will be adapted, per frequencies, during the solution process.

During the solution, the solver identifies if the mesh must be updated based on how you specify the object's properties. Base mesh and morphing parameters include **Base Frequency**, **Morphing Region Thickness**, **Minimum**/**Maximum Frequency**, **Morphing Intervals**.

Mesh adaptations are based on an applied offset in the normal direction of the faces selected as the Moving Boundaries and as needed, the mesh is morphed/adapted. The offset is calculated as follows:

Offset = Morphing Region Thickness * ((Base Frequency/current frequency) – 1)

Points to Remember

Note that:

- This analysis supports 3D geometries only.
- If possible, model your fluid region as a single solid multibody part.
- This analysis requires that the air surrounding the physical geometry be modeled as part of the overall geometry. The air domain can be easily modeled in DesignModeler using the Enclosure feature.
- The Physics Region object(s) need to identify all of the active bodies that may belong to the acoustic and structural (if FSI) physics types. For your convenience, when you open a Modal Acoustics or Harmonic Acoustics system, the application automatically inserts a **Physics Region** object and scopes it to all bodies. You need to specify the physics selection.
- Use the Morphing Region object to specify a domain in which the mesh is changed, or "morphed," based on node locations and coordinates, during the solution process to adapt the mesh to the current frequency

Automatic Boundary Condition Detection

The **Harmonic Acoustics** Environment object provides the following context menu (right-click) options:

- **Create Automatic** > **FSI**: This selection creates a **Fluid Solid Interface** object with all possible Fluid Solid Interface face selections based on the physics region definitions.
- **Create Automatic** > **Far-field Radiation Surface**: This selection automatically creates an **Far-field Radiation Surface** object that includes all possible Far-field Radiation Surfaces available in the analysis. Mechanical identifies the following faces as Far-field Radiation Surfaces:
  - Interface between the normal acoustic element and PML acoustic element (Interface between Normal Acoustic and PML Acoustic Region)
  - Face selections of Radiation Boundary (faces of elements flagged with **SF,,INF**)
  - Face selections of Impedance Boundary (faces of element flagged with **SF,,IMPD**)
  - Face selection of Absorption Element (faces of elements of type FLUID130)

– Face selection of Absorption Surface (faces of element flagged with **SF,,ATTN**)

- **Create Automatic** > **FSI and Far-field Radiation Surface**: This selection performs both of the above object generation options.

Preparing the Analysis

If you have not already created a **Harmonic Acoustics** system in the **Project Schematic**, see the Harmonic Acoustics section in the *Workbench User's Guide* for the steps to create this system.

Define Material Data

All of your acoustic bodies must be assigned a material that contains the properties **Density** and **Speed of Sound**.

Important:

The Fluid Materials library in the Engineering Data workspace includes the fluid materials Air and Water Liquid. Each of these materials includes the property Speed of Sound. Any other material to be used in the Acoustics Region requires you to specify the properties Density and Speed of Sound in Engineering Data workspace (**Toolbox** > **Physical Properties**).

Define Part Behavior

You **cannot** apply an acoustics-based **Physics Region** object to a body or part that has the **Stiffness Behavior** property set to **Rigid**.

Caution:

If you scope a structural-based **Physics Region** to a body or part that has the **Stiffness Behavior** property set to **Rigid**, this body/part is in contact with an acoustic-based **Physics Region**, then fluid-structure interaction may not behave as expected.

Define Connections

To define contact between two acoustic bodies or an acoustic and a structural body (FSI contact) which have non-conforming meshes, you must set the:

- **Type** property to **Bonded**.
- **Formulation** property must be set to **MPC**.

For FSI contacts:

- The Contact side must be on the acoustic body and the Target must be on the structural body.
- The Bonded contact type setting and the Pure Penalty formulation is supported in addition to MPC formulation.
- Pure Penalty formulation is not supported for contact conditions between two acoustic bodies.

- The Nodal-Dual Shape Function Projection (keyo,cid,4,4) option, of the Detection Method property, is used by default for FSI contact defined using the Pure Penalty formulation.
- The Combined option (keyo,cid,4,5), of the Detection Method property, is not supported for the MPC formulation type

---

Note:

Contact settings other than Bonded using MPC or Pure Penalty formulation (keyo,cid,2,1) are ignored and are overwritten with the following preferred key options of Bonded and MPC contact:

- For fluid-fluid contact: keyo,cid,1,10 ! select only PRES dof
- For FSI contact:
  - keyo,cid,8,2 ! auto create asymmetric contact
  - keyo,tid,5,2 ! For case of solid-shell body contact
  - keyo,tid,5,1 ! For case of solid-solid body contact
- Bonded Always: keyo,cid,12,5
- MPC Formulation: keyo,cid,2,2
- The application overwrites user-defined contact settings between fluid-fluid and fluid-solid bodies using the above criterion. Refer to Matrix-Coupled FSI Solutions section from the Mechanical APDL Acoustic Analysis Guide for more information.

Note:

Joints, Springs, Bearings, and/or Beams are **not** supported on acoustic bodies.

Establish Analysis Settings

For a **Harmonic Acoustics** analysis, the basic analysis settings include:

Step Controls

Th e Step Controls category enables you to define step controls for an analysis that includes rotational velocities in the form of revolutions per minute (RPMs). You use the properties of this category to define RPM steps and their options. Each RPM load is considered as a load step, such as frequency spacing, minimum frequencies, maximum frequencies, etc. When you select the **Analysis Settings** object, the **Step Controls** category automatically displays in the Worksheet. You can modify certain properties in either the **Worksheet** or in the **Details** pane for the object.

Options

The Options category enables you to specify the frequency range and the number of solution points at which the harmonic analysis will be carried out as well as the solution method to use and the relevant controls.

Only the Direct Integration (**Full**) **Solution Method** is available to perform a Harmonic Acoustics analysis.

Scattering Controls

The Scattering Controls category includes the **Scattered Field Formulation** property. The options for this property include:

- **Program Controlled** (default): The application selects the desired setting.
- **Off**: Scattering controls are off.
- **On**: Selecting this option turns scattering controls on and displays the **Scattering Output Type** property. You use the **Scattering Output Type** property to specify the output type for an acoustic scattering analysis. The options for this property include **Total** and **Scattered**. Select the **Total** option when you wish to output the total pressure field and the **Scattered** option when you want to output the scattered pressure field.

  If you specify an **Incident Wave Source** excitation and also specify the **Incident Wave Location** property as **Inside the Model**, then the application uses the **Total** setting for the **Scattering Output Type** property only.

  For more information, refer to the **ASOL** and **ASCRES** commands in the *Mechanical APDL Command Reference*.

Advanced

The Advanced category includes the property **Far-field Radiation Surface**. Far-field result calculations are based on the Far-field Radiation Surfaces. Therefore, this field controls far-field result definitions and results. The options include:

- **Program Controlled** (default): If your analysis does not include a user-defined **Far-field Radiation Surface** boundary condition object, this setting identifies the Far-field Radiation Surfaces automatically created by the application using the environment option Create Automatic > Far-field Radiation Surface. In this case, the application applies the surface flag MXWF on them. If the analysis does include a user-defined Far-field Radiation Surface object, this settings defined by that object are used.
- **Manual**: This option requires the definition of at least one user-defined Far-field Radiation Surface object.
- **No**: This setting invalidates all Far-field Radiation Surface objects and Far-field Result objects.

Output Controls

Summary

The properties of the Output Controls category enable you to different quantities to be written to the result file for use during post-processing. During acoustics analyses, these quantities are based on the specified Acoustics or Structural Physics Regions.

For specified Acoustics Regions:

By default, the application calculates and stores Acoustic Pressure in the result file. No specific property is associated with this quantity.

In addition, setting the **Calculate Velocity** and **Calculate Energy** properties to **Yes** enables you to request acoustic velocity and acoustic energy.

For specified Structural Regions:

When your analysis is solving an FSI problem in order to control the results calculated on structural domain, by default, only deformations are calculated. You can also request **Stress** and **Strain** results, using the corresponding properties. Furthermore, you can generate node-based force reactions using the **Calculate Reactions** property. This property requires you to set the **Nodal Forces** property to **On**.

General Miscellaneous Property

The General Miscellaneous Property property includes options specific to Acoustics analyses based on the acoustics analysis type, either Harmonic or Modal, and enable you to produce element-based miscellaneous solution data.

Damping Controls

The Damping Controls category is visible when **Structural Physics** is turned **On**. These properties enable you to specify damping for the structure in the Harmonic Acoustics analysis. Properties include: **Structural Damping Coefficient**, **Stiffness Coefficient** (beta damping), and **Mass Coefficient** (alpha damping). They can also be applied as **Material Damping** using the **Engineering Data** workspace.

Element Damping: You can also apply damping through spring-damper elements. The damping from these elements is used only in a Full method harmonic analysis.

Important:

If multiple damping specifications are made the effect is cumulative.

Analysis Data Management

The properties of the Analysis Data Management category enable you to save solution files from the harmonic analysis. The default behavior is to only keep the files required for postprocessing. You can use these controls to keep all files created during the solution or to create and save the Mechanical APDL application database (db file).

Define Physics Region(s)

To specify a **Physics Region** object:

1. Highlight the Environment object and select the Physics Region button on the Environment Context Tab or right-click the Environment object or within the Geometry window and select Insert > Physics Region.

2. Define all of the properties for the new object. For additional information, see the Physics Region object reference section.

   A structural-based **Physics Region** may contain bodies with the Stiffness Behavior set to Rigid. Acoustics Regions do not support a Stiffness Behavior setting of Rigid.

   If the structural region has the **Stiffness Behavior** property set to **Rigid** and if it is in contact with acoustic regions, then fluid-structure interaction may not behave as expected.

   Note:

   You may want to use the following context menu (right-click) options when specifying a **Physics Region** object:

   - **Select Bodies** > **Without Physics Region**.
   - **Select Bodies** > **With Multiple Physics Region**.

Apply Loads and Supports

See the Acoustics Loading Conditions (p. 31) as well as the Boundary Conditions section of the *Mechanical User's Guide* for a listing of all available loads, supports, etc. for this analysis type.

Solve

The Solution Information object provides some tools to monitor solution progress.

The **Solver Output** setting for the **Solution Output** property continuously updates any listing output from the solver and provides valuable information on the behavior of the model during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the Solution Information section.

Review Results

See the Acoustic Results (p. 89) section for descriptions of all supported result types.

Harmonic Acoustic results generally default to the setting **All Acoustic Bodies**. You can individually scope most of the Harmonic Acoustic analysis results to mesh or geometric entities on acoustic bodies.

Additional results are available for structural domain when solving Fluid Structural Interaction (FSI) problems. Refer to the **Review Results** topic in the Harmonic Response Analysis for more information regarding how to set up the harmonic results.

2.2. Harmonic Acoustics Analysis Using Prestressed Structural System

Introduction

Mechanical enables you to perform a FSI Harmonic analysis on a pre-stressed structure using a Static Acoustics Analysis.

Points to Remember

To perform a prestressed Harmonic Acoustics analysis you need to first perform a **Static Acoustics** analysis and properly link it to the Harmonic Acoustics analysis. When performing this type of linked analysis, the **Harmonic Acoustics** analysis uses the **Physics Regions** (**Acoustic** and **Structural**) defined in the **Static Acoustics** analysis. Therefore, you need to remove the **Acoustics Region** from your **Harmonic Acoustics** analysis when you first create the linked systems.

Preparing the Analysis

Because this analysis is linked to (and based on) structural responses, a Static Acoustics (p. 22) analysis is a prerequisite. This setup allows the two analysis systems to share resources, such as material data, geometry, and the boundary condition type definitions that are defined the in the static acoustics analysis.

From the Workbench **Toolbox**, drag a **Static Acoustics** template to the **Project Schematic**. Then, drag a **Harmonic Acoustics** template directly onto the **Solution** cell of the **Static Acoustics** template.

Define Initial Conditions

The Pre-Stress object of the **Harmonic Acoustics** analysis must point to the linked **Structural Acoustics** analysis.

Note:

- All structural loads, including Inertial loads, such as **Acceleration** and **Rotational Velocity**, are deleted from the harmonic analysis portion of the simulation once the loads are applied as initial conditions (via the **Pre-Stress** object). Refer to the Mechanical APDL command **PERTURB,HARM,,,DZEROKEEP** for more details.

- For Pressure boundary conditions in the **Structural Acoustics** analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure acts as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. Pressure loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different results in the follow-on **Harmonic Acoustics** analysis. See the Pressure Load Stiffness topic in the Applying Pre-Stress Effects for Implicit Analysis Help section for more information about using a prestressed environment.

- If displacement loading is defined with Displacement, Remote Displacement, Nodal Displacement, or Bolt Pretension (specified as **Lock**, **Adjustment**, or **Increment**) loads in the **Structural Acoustics** analysis, these loads become fixed boundary conditions for the harmonic solution. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution. If you define a **Nodal Displacement** in the harmonic analysis at the same location and in the same direction as in the structural analysis, it overwrites the previous loading condition and/or boundary condition in the harmonic solution.

Establish Analysis Settings

See the **Establish Analysis Settings** topic in the Harmonic Acoustics (p. 3) section for a complete listing of the Analysis Settings.

Apply Loads and Supports

See the Acoustics Loading Conditions (p. 31) as well as the Boundary Conditions section of the *Mechanical User's Guide* for a listing of all available loads, supports, etc. for this analysis type.

Solve

The Solution Information object provides some tools to monitor solution progress.

The **Solver Output** setting for the **Solution Output** property continuously updates any listing output from the solver and provides valuable information on the behavior of the model during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the Solution Information section.

Review Results

See the Acoustic Results (p. 89) section for descriptions of all supported result types.

Harmonic Acoustic results generally default to the setting **All Acoustic Bodies**. You can individually scope most of the Harmonic Acoustic analysis results to mesh or geometric entities on acoustic bodies.

Additional results are available for structural domain when solving Fluid Structural Interaction (FSI) problems. Refer to the **Review Results** topic in the Harmonic Response Analysis for more information regarding how to set up the harmonic results.

2.3. Harmonic Acoustics One-way Acoustic Coupling Analysis

The Mechanical application enables you to apply velocities from a structural **Harmonic Response** analysis or a FSI **Harmonic Acoustics** analysis as loads in a **Harmonic Acoustics** analysis. Options are available that enable you to import individual velocity loads or to automatically generate multiple velocity loads from the upstream system.

You can also apply velocities from a Motion analysis as loads in a **Harmonic Acoustics** or **Coupled Field Harmonic** analysis. Options are available that enable you to convert time-dependent velocities to frequency-dependent velocities and then to import individual velocity loads or to generate multiple velocity loads automatically from the upstream system.

The load transfer is applicable for the cases where the **Harmonic Response** or FSI **Harmonic Acoustics** and acoustic analyses are solved using different meshes. When different meshes are used, the velocity values are mapped and interpolated between the source and target meshes.

Note:

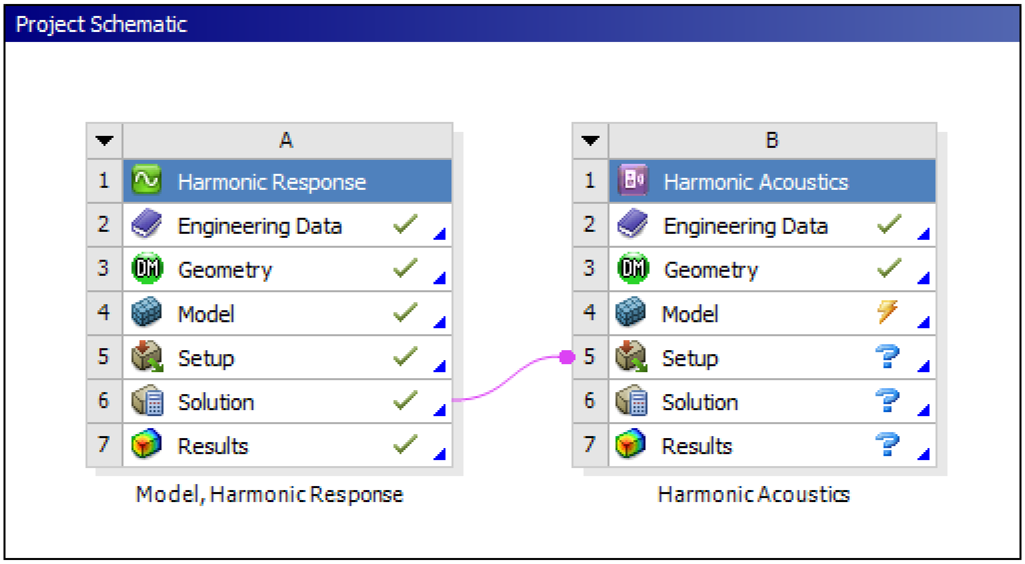
- If the upstream (Structural or FSI Acoustics) system is modified and re-solved after importing the load, a refresh operation on the acoustic system's **Setup** cell is required to notify Mechanical that source data has changed and re-import is required. Alternatively, the source data can be refreshed using the right-click operation on the Imported Load folder and choosing the Refresh Imported Load option.
- If an upstream **Harmonic Acoustics** system is used, it must contain **Structural Physics Region**(s).
- If the upstream system contains **Condensed Parts**, the velocities of these parts are ignored during data transfer.

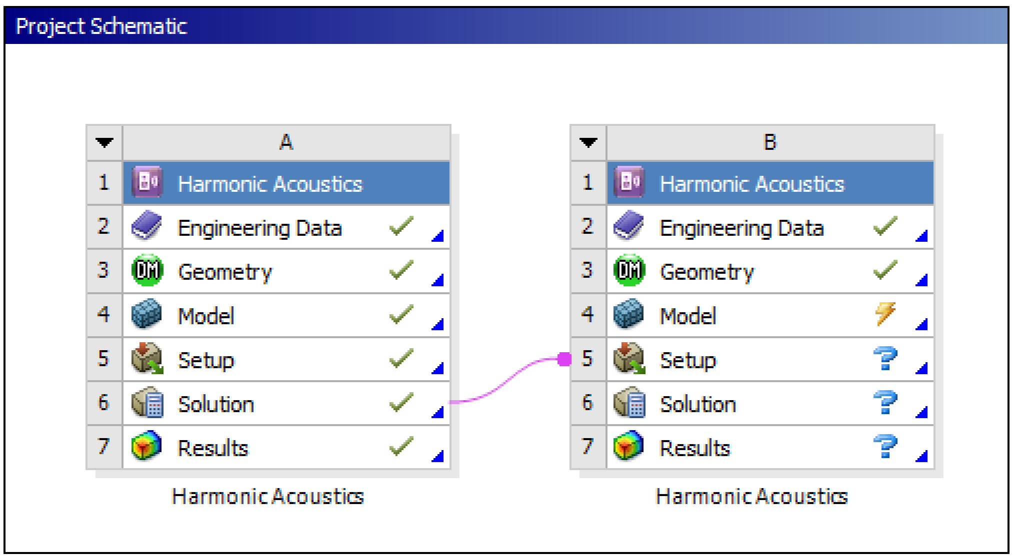
Workflow

Specify Analysis Systems in Workbench

Review the following steps to create and define your upstream system and property configure your downstream acoustics analysis.

1. From the toolbox, drag and drop a Harmonic Response or Harmonic Acoustics (p. 3) template onto the **Project Schematic**. Open the model in Mechanical and perform all steps to set up a **Harmonic Response** or **Harmonic Acoustics** analysis. Specify mesh controls, boundary conditions, and solution settings as you normally would and solve the analysis.
2. Return to the **Project Schematic** and drag and drop a **Harmonic Acoustics** template onto the **Project Schematic**. Drag the **Solution** cell of the structural or FSI acoustics system onto the **Setup** cell of the acoustic system. Examples are illustrated below.





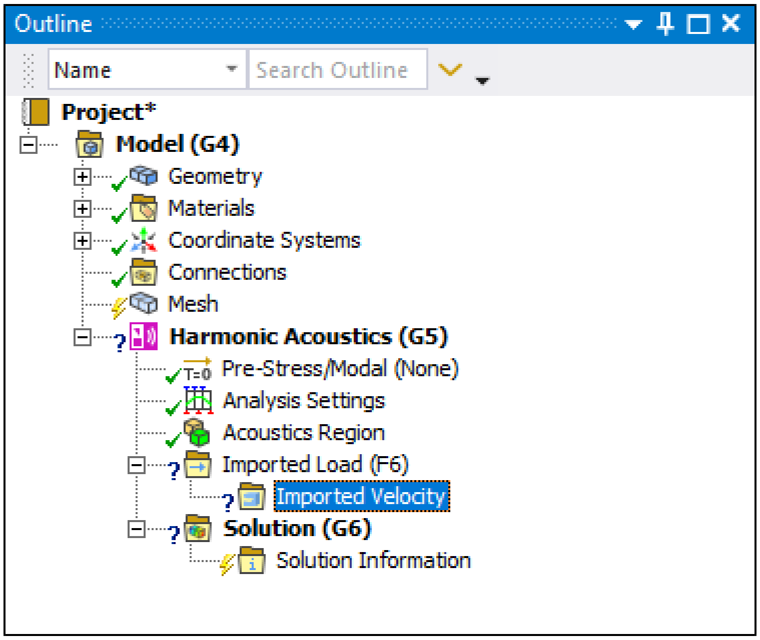
3. Open the downstream system in Mechanical.

Import Velocities

1. You may perform prerequisite property definitions as needed, such as making necessary entries for the **Analysis Setting** and the **Acoustics Region**.

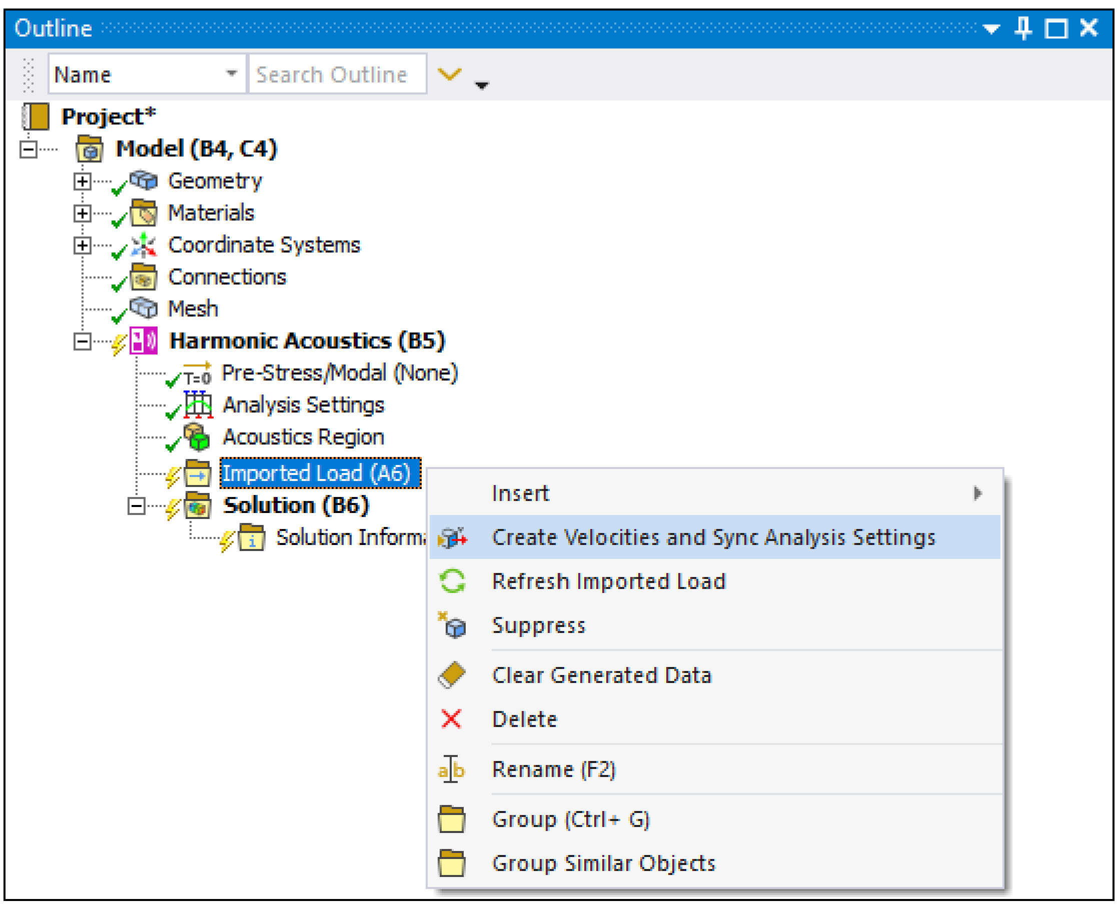
2. Select the **Imported Load** folder/object, and:

   • Open the folder. By default, the application inserts a **Imported Velocity** object. As needed, you can add **Imported Velocity** objects by right-clicking on the Imported Load folder and selecting **Insert** > **Velocity**.



Or...

- Right-click the folder and select the option **Create Velocities and Sync Analysis Settings** to import all of the velocity loads available in the upstream system.



3. Select appropriate geometry in the Details view of the imported velocity object(s) using the **Geometry** or **Named Selection** scoping option.

4. The **Source Bodies** property in the Details view enables you to select the bodies, from the upstream analysis, that makeup the source mesh for mapping the data. The options for this property include:

   - **All**: The source mesh in this case will comprise all the bodies that were used in the upstream analysis.

   - **Manual**: This option enables you to select one or more source bodies to make up the source mesh. The source body selections are made in the **Material IDs** field by entering the material IDs that correspond to the source bodies that you would like to use. Type material IDs and/or material ID ranges separated by commas to specify your selection. For example, type 1, 2, 5–10. The material IDs for the source bodies can be seen in Solution Information object of the source analysis. In the example below, text is taken from a solver output,

     ```
     ***********Elements for Body 1 "coil" ***********
     ***********Elements for Body 2 "core" ***********
     ***********Elements for Body 3 "bar" ************
     ```

     Body "coil" has material ID 1, body "core" has material ID 2 and body "bar" has material ID 3.

   - **Named Selection**: When you set the property to **Named Selection**, the **Source Component** property also displays. This property provides a drop-down menu that enables you to select a Named Selection defined in the upstream system. This option can improve performance by reducing the number of transferred source nodes.

5. If your upstream system includes multiple RPMs, The **RPM Selection** property enables you to select the RPM for which the data is imported.

6. Change any of the columns in the **Data View** pane as needed:

   - **Source Frequency**: Frequency at which the velocities will be imported from the structural analysis.

   - **Analysis Frequency**: Choose the analysis frequency at which the load will be applied.

   Note:

   The **Data View** can automatically be populated with the source and analysis frequencies using the **Source Frequency** property in the Details pane. Use **All** to import data at all frequencies in the source analysis, or **Range** to import data for a range specified by a **Minimum** and **Maximum**. The default Worksheet option requires users to manually input the **Source Frequency** and **Analysis Frequency**.

7. You can transform the source mesh used in the mapping process by using the **Rigid Transformation** properties. This option is useful if the source geometry was defined with respect to a coordinate system that is not aligned with the target geometry system.

You can modify the Mapper Settings to achieve the desired mapping accuracy. Mapping can be validated by using Mapping Validation objects.

8. Right-click the **Imported Velocity** object or on the **Imported Load** folder and click **Import Load** to import the load(s). Following successful import, vectors plot (**All**), or contour plot (**Total**/**X**/**Y**/**Z**) of the real/imaginary components of velocities can be displayed in the Geometry window using the Component property in the details of imported load.

   Note:

   The range of data displayed in the graphics window can be controlled using the Legend controls options. See Imported Boundary Conditions for additional information.

9. If multiple rows are defined in the **Data View**, it is possible to preview imported load vectors/contour applied to a given row or analysis frequency in the **Data view**. Choose **Active Row** or **Analysis Frequency** using the By property under **Graphics Controls** in the details of the imported load and then specify the **Active Row**/**Analysis Frequency** to preview the data.

   Note:

   If you specify the **Analysis Frequency** that does not match the list of analysis frequencies in the **Data View**, the data is displayed at the analysis frequency closest to the specified frequency.

2.4. Modal Acoustics Analysis

Introduction

A Modal Acoustic analysis models a structure and the surrounding the fluid medium to determine frequencies and standing wave patterns within a structure. Examples of acoustics include Sonar (the acoustic counterpart of radar), the design of concert halls, the minimization of noise in a machine shop, noise cancellation in automobiles, audio speaker design, speaker housing design, acoustic filters, mufflers, and Geophysical exploration.

A Modal Acoustic analysis usually involves modeling the fluid medium as well as the surrounding structure in order to determine frequencies and standing wave patterns within a structure. Typical quantities of interest are the pressure distribution in the fluid at different frequencies, pressure gradient, and particle velocity of acoustic waves.

Mechanical enables you to model pure acoustic problems and fluid-structure interaction (FSI) problems. A coupled acoustic analysis accounts for FSI. An uncoupled acoustic analysis simulates the fluid only and ignores any fluid-structure interaction. You can also perform a FSI modal analysis on a prestressed structure using a Static Acoustics Analysis (p. 22).

Points to Remember

Note that:

- This analysis supports 3D geometries only.
- If possible, model your fluid region as a single solid multibody part.
- This analysis requires that the air surrounding the physical geometry be modeled as part of the overall geometry. The air domain can be easily modeled in DesignModeler using the Enclosure feature.
- The Physics Region object(s) need to identify all of the active bodies that may belong to the acoustic and structural (if FSI) physics types. For your convenience, when you open a Modal Acoustics system, the application automatically inserts a **Physics Region** object and scopes it to all bodies. You need to specify the physics selection.
- To perform a prestressed Modal Acoustics analysis you need to first perform a Static Acoustics (p. 22) analysis and properly link it to the Modal Acoustics analysis. When performing this type of linked analysis, the Modal Acoustics analysis uses the Physics Regions (Acoustic and Structural) defined in the Static Acoustics analysis. Therefore, you need to **remove** the **Acoustics Region** from your Modal Acoustics analysis when you first create the linked systems.

Automatic Boundary Condition Detection

In order to assist your analysis, the Environment object contains context menu (right-click) options that enable you to automatically generate interfaces based on physics region definitions. The **Modal Acoustics** analysis includes the option **Create Automatic** > **FSI**. This selection automatically creates a **Fluid Solid Interface** object with all possible Fluid Solid Interface face selections.

Create Analysis System

If you have not already created a **Modal Acoustics** system in the **Project Schematic**, see the Modal Acoustics section in the *Workbench User's Guide* for the steps to create this system.

Define Materials

All of your acoustic bodies must be assigned a material that contains the properties **Density** and **Speed of Sound**.

Important:

The **Fluid Materials** library in the Engineering Data workspace includes the fluid materials **Air** and **Water Liquid**. Each of these materials includes the property **Speed of Sound**. Any other material to be used in the **Acoustics Region** requires you to specify the property Speed of Sound and Density in Engineering Data workspace (**Toolbox** > **Physical Properties**).

Note:

The acoustic damping material properties like Viscosity and/or Thermal Conductivity are applicable only for a damped modal solver. You need to set **Damped** property under **Solver Controls** to **Yes** and select from the available damped modal solver types.

Define Part Behavior

A Structural **Physics Region** may contain bodies with the **Stiffness Behavior** set to **Rigid**. **Acoustics Regions** cannot contain rigid bodies.

If the **Structural Region** has the **Stiffness Behavior** property set to **Rigid** and if it is in contact with acoustic regions, then fluid-structure interaction may not behave as expected.

Define Connections

To define contact between two acoustic bodies or an acoustic and a structural body (FSI contact) which have non-conforming meshes, you must set the:

- **Type** property to **Bonded**.
- **Formulation** property must be set to **MPC**.

For FSI contacts:

- The Contact side must be on the acoustic body and the Target must be on the structural body.
- The Bonded contact type setting and the Pure Penalty formulation is supported in addition to MPC formulation.
- Pure Penalty formulation is not supported for contact conditions between two acoustic bodies.
- The Nodal-Dual Shape Function Projection (keyo,cid,4,4) option, of the Detection Method property, is used by default for FSI contact defined using the Pure Penalty formulation.
- The Combined option (keyo,cid,4,5), of the Detection Method property, is not supported for the MPC formulation type

---

Note:

Contact settings other than Bonded using MPC or Pure Penalty formulation (keyo,cid,2,1) are ignored and are overwritten with the following preferred key options of Bonded and MPC contact:

- For fluid-fluid contact: keyo,cid,1,10 ! select only PRES dof
- For FSI contact:
  - keyo,cid,8,2 ! auto create asymmetric contact
  - keyo,tid,5,2 ! For case of solid-shell body contact
  - keyo,tid,5,1 ! For case of solid-solid body contact
- Bonded Always: keyo,cid,12,5
- MPC Formulation: keyo,cid,2,2

- The application overwrites user-defined contact settings between fluid-fluid and fluid-solid bodies using the above criterion. Refer to Matrix-Coupled FSI Solutions section from the Mechanical APDL Acoustic Analysis Guide for more information.

Note:

Joints, Springs, Bearings, and/or Beams are **not** supported on acoustic bodies.

Establish Analysis Settings

Basic Analysis Settings for this analysis include the following:

Options

Using the **Max Modes to Find** property, specify the number of frequencies of interest. The default is to extract the first 6 natural frequencies. The number of frequencies can be specified in two ways:

The first N frequencies (N > 0).

Or...

The first N frequencies in a selected range of frequencies.

Note:

The **Limit Search to Range** property is set to **Yes** by default and the **Range Minimum** property is set to greater than or equal to **`0.01`** Hz.

Solver Controls

This \ category includes the following properties:

- **Damped**: Use this property to specify if the modal system is undamped (**No**) or damped (**Yes**). Depending upon your selection, different solver options are provided. The default setting of the **Damped** property is **No**, which assumes that the modal acoustics system is an undamped system.
- **Solver Type**: It is generally recommended that you allow the application to select the solver type (**Program Controlled**) for your analysis, be it an undamped and damped system.

Output Controls

The properties of the **Output Controls** enable you to different quantities to be written to the result file for use during post-processing. During acoustics analyses, these quantities are based on the specified Acoustics or Structural Physics Regions.

For specified Acoustics Regions:

By default, the application calculates and stores Acoustic Pressure in the result file. No specific property is associated with this quantity.

In addition, setting the **Calculate Velocity** and **Calculate Energy** properties to **Yes** enables you to request acoustic velocity and acoustic energy.

For specified Structural Regions:

When your analysis is solving an FSI problem in order to control the results calculated on structural domain, by default, only mode shapes are calculated. You can also request **Stress** and **Strain** results, using the corresponding properties. These properties only show the relative distribution of stress in the structure and are not real stress values. Furthermore, you can generate node-based force reactions using the **Calculate Reactions** property. This property requires you to set the **Nodal Forces** property to **On**.

General Miscellaneous Property: This property includes options specific to Acoustics analyses based on the acoustics analysis type, either Harmonic or Modal, and enable you to produce element-based miscellaneous solution data.

Damping Controls

The properties of the Damping Controls category depend upon the setting of the Damped property of the Solver Controls category.

Undamped System

When the **Damped** property is set to **No** the **Ignore Acoustic Damping** property displays. This property provides the options **No** (default) and **Yes**. Setting this property to **Yes** instructs the application to ignore material properties that create damping effects, specifically **Specific Heat**, **Thermal Conductivity**, and **Viscosity**. Ignoring these material-based damping effects enables the application to use undamped eigensolvers without the need to suppress these material properties in Engineering Data.

Damped System

When the **Damped** property is set to **Yes** (**Full Damped**) and the **Structural** property of the **Environment** (**Modal Acoustics**) object is set to **Yes**, the **Stiffness Coefficient Defined By** property displays. The options for this property include **Direct Input** (default) or **Damping vs. Frequency**. The options of this property enable you to define the method used to define the **Stiffness Coefficient**. If you select **Damping vs. Frequency**, the **Frequency** and **Damping Ratio** properties display and require you to enter values to calculate the **Stiffness Coefficient**. Otherwise, you specify the **Stiffness Coefficient** manually. The **Mass Coefficient** property also requires a manual entry.

Analysis Data Management

These properties enable you to define whether or not to save the Mechanical APDL application database as well as automatically delete unneeded files.

Define Pre-Stress Conditions

You can point to a **Static Acoustics** analysis in the **Pre-Stress** environment field if you want to include pre-stress effects. A typical example is the large tensile stress induced in a turbine blade under centrifugal load that can be captured by a static structural analysis. This causes significant stiffening of the blade. Including this pre-stress effect will result in much higher, realistic natural frequencies in a modal analysis.

If the Modal analysis is linked to a **Static Acoustics** analysis for initial conditions and the parent static analysis has multiple result sets (multiple restart points at load steps/sub steps), you can start the Modal analysis from any restart point available in the Static Acoustics analysis. By default, the values from the last solve point are used as the basis for the modal analysis. See the **Restarts from Multiple Result Sets** topic in the Applying Pre-Stress Effects for Implicit Analysis section for more information.

Note:

- When you perform a prestressed Modal analysis, the support conditions from the static analysis are used in the Modal analysis. You cannot apply any new supports in the Modal analysis portion of a prestressed modal analysis. When you link your Modal analysis to a **Structural Acoustics** analysis, all structural loading conditions, including Inertial loads, such as **Acceleration** and **Rotational Velocity**, are deleted from the Modal portion of the simulation once the loads are applied as initial conditions (via the Pre-Stress object). Refer to the Mechanical APDL command **PERTURB,HARM,,,DZEROKEEP** for more details.

- For **Pressure** boundary conditions in the **Static Acoustics** analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure acts as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different modal results in the follow-on Modal Analysis. See the **Pressure Load Stiffness** topic in the Applying Pre-Stress Effects for Implicit Analysis section for more information about using a prestressed environment.

- If displacement loading is defined with **Displacement**, **Remote Displacement**, **Nodal Displacement** , or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the **Static Acoustics** analysis, these loads become fixed boundary conditions for the **Modal Acoustics** solution.

Define Physics Region(s)

To specify a **Physics Region** object:

1. Highlight the Environment object and select the **Physics Region** button on the Environment Context Tab or right-click the **Environment** object or within the Geometry window and select **Insert** > **Physics Region**.

2. Define all of the properties for the new object. For additional information, see the Physics Region object reference section.

   A structural-based **Physics Region** may contain bodies with the **Stiffness Behavior** property set to **Rigid**. **Acoustics Regions** do not support a **Stiffness Behavior** setting of **Rigid**.

If the structural region has the **Stiffness Behavior** property set to **Rigid** and if it is in contact with acoustic regions, then fluid-structure interaction may not behave as expected.

Note:

You may want to use the following context menu (right-click) options when specifying a **Physics Region** object:

- **Select Bodies** > **Without Physics Region**.
- **Select Bodies** > **With Multiple Physics Region**.

Apply Loads and Supports

See the Acoustics Loading Conditions (p. 31) as well as the Boundary Conditions section of the *Mechanical User's Guide* for a listing of all available loads, supports, etc. for this analysis type.

Solve

The Solution Information object provides some tools to monitor solution progress.

Review Results

This analysis type does not provide Acoustic Results. All structural result types are available. You can use a Solution Information object to track, monitor, or diagnose problems that arise during a solution.

Once a solution is available you can contour the results or animate the results to review the response of the structure.

As a result of a nonlinear static analysis you may have a solution at several time points. You can use probes to display the variation of a result item as the load increases. An example might be large deformation analyses that result in buckling of the structure. In these cases it is also of interest to plot one result quantity (for example, displacement at a vertex) against another results item (for example, applied load). You can use the Charts feature to develop such charts.

2.5. Static Acoustics Analysis

Introduction

You use the Static Acoustics analysis as a method for applying stresses to a downstream analysis. This is a Fluid-Structure Interaction (FSI) analysis incorporating two different physics phenomena that can then interact with one another. The static analysis can be linear or nonlinear. It creates a pre-stress environment for the downstream dynamic acoustics analysis.

The Acoustics Regions of the Static Acoustics analysis do not effect the results of the downstream Modal or Harmonic Acoustics analysis, except that the mesh can be morphed during the solution.

Points to Remember

Note that:

- This analysis supports 3D geometries only.
- If possible, model your fluid region as a single solid multibody part.
- The Physics Region object(s) need to identify all of the active bodies that may belong to the acoustic and structural physics types. For your convenience, when you open a Static Acoustics system, the application automatically inserts a **Acoustics Region** object and a **Structural Region** object.
- Only Structural Results are supported for this analysis type.

Automatic Boundary Condition Detection

In order to assist your analysis, the Environment object contains context menu (right-click) options that enable you to automatically generate interfaces based on physics region definitions. The **Static Acoustics** analysis includes the option **Create Automatic** > **FSI**. This selection automatically creates a **Fluid Solid Interface** object with all possible Fluid Solid Interface face selections.

Create Analysis System

If you have not already created a **Static Acoustics** system in the **Project Schematic**, see the Static Acoustics section in the *Workbench User's Guide* for the steps to create this system.

Define Materials

All of your acoustic bodies must be assigned a material that contains the properties **Density** and **Speed of Sound**.

Important:

The **Fluid Materials** library in the Engineering Data workspace includes the fluid materials Air and Water Liquid. Each of these materials includes the property Speed of Sound. Any other material to be used in the **Acoustics Region** requires you to specify the property Speed of Sound and Density in Engineering Data workspace (**Toolbox** > **Physical Properties**).

Define Part Behavior

A Structural **Physics Region** may contain bodies with the **Stiffness Behavior** set to **Rigid**. **Acoustics Regions** cannot contain rigid bodies.

If the **Structural Region** has the **Stiffness Behavior** property set to **Rigid** and if it is in contact with acoustic regions, then fluid-structure interaction may not behave as expected.

Define Connections

To define contact between two acoustic bodies or an acoustic and a structural body (FSI contact) which have non-conforming meshes, you must set the:

- **Type** property to **Bonded**.
- **Formulation** property must be set to **MPC**.

For FSI contacts:

- The Contact side must be on the acoustic body and the Target must be on the structural body.
- The Bonded contact type setting and the Pure Penalty formulation is supported in addition to MPC formulation.
- Pure Penalty formulation is not supported for contact conditions between two acoustic bodies.
- The Nodal-Dual Shape Function Projection (keyo,cid,4,4) option, of the Detection Method property, is used by default for FSI contact defined using the Pure Penalty formulation.
- The Combined option (keyo,cid,4,5), of the Detection Method property, is not supported for the MPC formulation type

---

Note:

Contact settings other than Bonded using MPC or Pure Penalty formulation (keyo,cid,2,1) are ignored and are overwritten with the following preferred key options of Bonded and MPC contact:

- For fluid-fluid contact: keyo,cid,1,10 ! select only PRES dof
- For FSI contact:
  - keyo,cid,8,2 ! auto create asymmetric contact
  - keyo,tid,5,2 ! For case of solid-shell body contact
  - keyo,tid,5,1 ! For case of solid-solid body contact
- Bonded Always: keyo,cid,12,5
- MPC Formulation: keyo,cid,2,2
- The application overwrites user-defined contact settings between fluid-fluid and fluid-solid bodies using the above criterion. Refer to Matrix-Coupled FSI Solutions section from the Mechanical APDL Acoustic Analysis Guide for more information.

Note:

Joints, Springs, Bearings, and/or Beams are **not** supported on acoustic bodies.

Establish Analysis Settings

For simple linear static analyses, you typically do not need to change the default **Analysis Settings**. For more complex analyses the basic **Analysis Settings** include:

Large Deflection

Large Deflection is typically needed for slender structures. Use large deflection if the transverse displacements in a slender structure are more than 10% of the thickness.

Small deflection and small strain analyses assume that displacements are small enough that the resulting stiffness changes are insignificant. Setting the **Large Deflection** property to **On** will take into account stiffness changes resulting from changes in element shape and orientation due to large deflection, large rotation, and large strain. Therefore, the results will be more accurate. However, this effect requires an iterative solution. In addition, it may also need the load to be applied in small increments. As a result, the solution may take longer to solve.

You also need to turn on large deflection if you suspect instability (buckling) in the system. Use of hyperelastic materials also requires large deflection to be turned on.

Step Controls for Static and Transient Analyses

Step Controls are used to i) control the time step size and other solution controls and ii) create multiple steps when needed. Typically analyses that include nonlinearities such as large deflection or plasticity require control over time step sizes as outlined in the Automatic Time Stepping section. Multiple steps are required for activation/deactivation of displacement loads or pretension bolt loads. This group can be modified on a per step basis.

Note:

Time Stepping is available for any solver.

Output Controls

Output Controls enable you to specify the time points at which structural results should be available for postprocessing. In a nonlinear analysis it may be necessary to perform many solutions at intermediate load values. However i) you may not be interested in all the intermediate results and ii) writing all the results can make the results file size unwieldy. This group can be modified on a per step basis except for **Stress** and **Strain**.

Nonlinear Controls

Nonlinear Controls enable you to modify convergence criteria and other specialized solution controls. Typically you will not need to change the default values for this control. This group can be modified on a per step basis. If you are performing a nonlinear Static Acoustics analysis, the **Newton-Raphson Type** property becomes available. This property only affects nonlinear analyses. Your selections execute the Mechanical APDL **NROPT** command. The default option, **Program Controlled**, allows the application to select the appropriate **NROPT** option or you can make a manual selection and choose **Full**, **Modified**, or **Unsymmetric**.

See the **NROPT** command in the *Mechanical APDL Command Reference* for additional information about the operation of the **Newton-Raphson Type** property.

Damping Controls

When you pre-stress a **Modal Acoustics** analysis with a **Static Acoustics** analysis, the Damping Controls category of the **Analysis Settings** displays. It includes the property **Ignore Acoustic Damping**. This property provides the options **No** (default) and **Yes**. Setting this property to **Yes** instructs the application to ignore material properties that create damping effects, specifically Specific Heat, Thermal Conductivity, and Viscosity in your downstream Modal system. Ignoring these material-based damping effects enables the application to use undamped eigensolvers without the need to suppress these material properties in Engineering Data.

Analysis Data Management

Settings enable you to save specific solution files from the **Static Acoustics** analysis for use in other analyses. You can set the **Future Analysis** property to **Pre-Stressed Analysis** if you intend to use the static acoustics results in a subsequent Modal or Harmonic analysis. If you link a structural system to another analysis type in advance, the **Future Analysis** property defaults to **Pre-Stressed Analysis**.

Note:

Scratch Solver Files, **Save Ansys db**, **Solver Units**, and **Solver Unit System** are applicable to static systems only.

Define Physics Region(s)

To specify a **Physics Region** object:

1. Highlight the Environment object and select the **Physics Region** button on the Environment Context Tab or right-click the **Environment** object or within the Geometry window and select **Insert** > **Physics Region**.

2. Define all of the properties for the new object. For additional information, see the Physics Region object reference section.

   A structural-based **Physics Region** may contain bodies with the **Stiffness Behavior** property set to **Rigid**. **Acoustics Regions** do not support a **Stiffness Behavior** setting of **Rigid**.

   If the structural region has the **Stiffness Behavior** property set to **Rigid** and if it is in contact with acoustic regions, then fluid-structure interaction may not behave as expected.

Note:

You may want to use the following context menu (right-click) options when specifying a **Physics Region** object:

- **Select Bodies** > **Without Physics Region**.
- **Select Bodies** > **With Multiple Physics Region**.

Apply Loads and Supports

See the Acoustics Loading Conditions (p. 31) as well as the Boundary Conditions section of the *Mechanical User's Guide* for a listing of all available loads, supports, etc. for this analysis type.

Solve

The Solution Information object provides some tools to monitor solution progress.

Review Results

This analysis type does not provide Acoustic Results. All structural result types are available. You can use a Solution Information object to track, monitor, or diagnose problems that arise during a solution.

Once a solution is available you can contour the results or animate the results to review the response of the structure.

As a result of a nonlinear static analysis you may have a solution at several time points. You can use probes to display the variation of a result item as the load increases. An example might be large deformation analyses that result in buckling of the structure. In these cases it is also of interest to plot one result quantity (for example, displacement at a vertex) against another results item (for example, applied load). You can use the Charts feature to develop such charts.

2.6. LS-DYNA Acoustics Analysis

For a general discussion of acoustics analyses in Mechanical, see Acoustics Analysis Overview (p. 1).

Introduction

Like the Harmonic Acoustics (p. 3) analysis, the LS-DYNA Acoustics analysis is used to determine the frequency response of a structure and the surrounding fluid medium to loads and excitations that vary with frequency. However, the fluid region in the LS-DYNA analysis is not meshed. The solver uses BEM (Boundary Element Method), which is a numerical computational method of solving linear partial differential equations which have been formulated as integral equations (in boundary integral form).

The integral equation may be regarded as an exact solution of the governing partial differential equation. The boundary element method attempts to use the given boundary conditions to fit boundary values into the integral equation, rather than values throughout the space defined by a partial differential equation. Once this is done, in the post-processing stage, the integral equation can then be used again to calculate numerically the solution directly at any desired point in the interior of the solution domain.

Note:

Only Surface/Shell bodies are supported for LS-DYNA Acoustics analyses.

Points to Remember

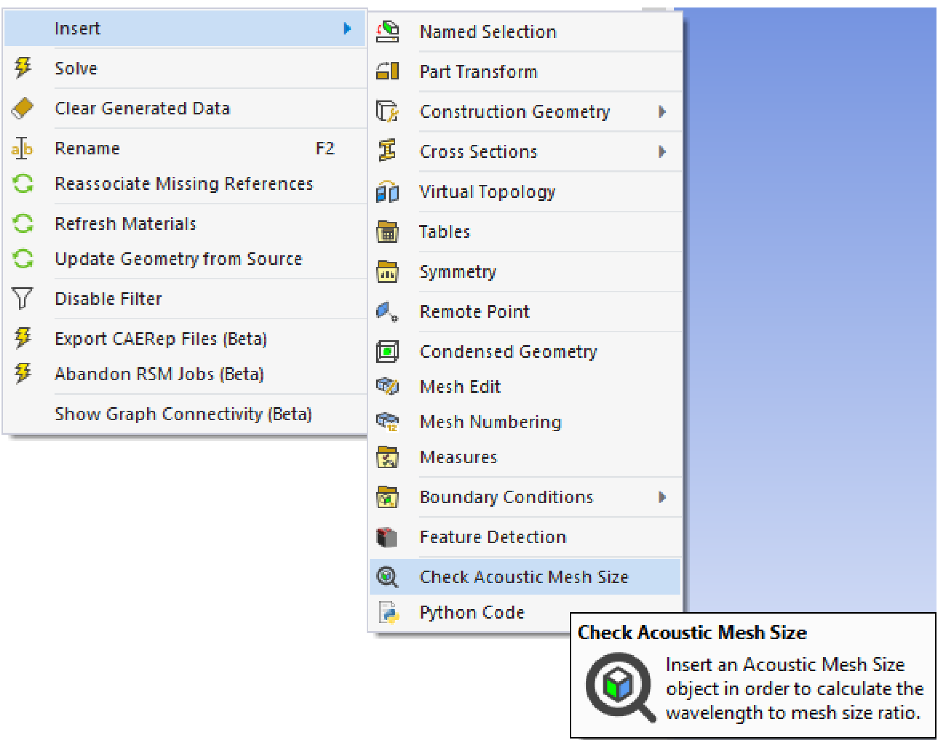
Note that:

- This analysis type supports 3D geometries only.
- Only shell bodies can be used. Material and thickness of shell bodies have no influence on the result. The fluid domain is not represented.
- The results supported for this type are Contour (p. 89), (Pressure, Total Velocity, Sound Pressure Level) and Frequency (p. 93) (Pressure, Sound Pressure Level, Directional Velocity).

Check Acoustic Mesh Size

The **Check Acoustic Mesh Size** object enables you to evaluate the ratio of the element length to acoustic wavelength. The element length is a property of each element of the mesh. The acoustic wavelength is defined as Sound speed/frequency. It is recommended that you use a ratio of less than 1/8.

To insert a **Check Acoustic Mesh Size** object, right click the **Model** object and select **InsertCheck Acoustic Mesh Size**. Alternatively, you can click the **Check Acoustic Mesh Size** icon found in the Acoustics section of the Environment tab of the ribbon.



Once you have inserted the object, scope it to the body of interest and enter the **Sound Speed** and **Frequency** values. The **Minimum** and **Maximum** fields show the minimum and maximum ratios of the Element lengths to the Wavelength.

Details of "Check Acoustic Mesh Size"	
Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Sound Speed	300 m/s
Frequency	1100 Hz
Plot	Element Length / Wavelength
Minimum	0.00718502220239013
Maximum	0.00719015557700108

Establish Analysis Settings

The **Analysis Settings** object contains several fields that are specific to performing and acoustics analysis using LS-DYNA.

Solver	
Method	Collocation BEM
Output Fields	Interior
Maximum Number Of Iterations	1000
Tolerance	1E-06
Number of Domain Decompositions	8

Method

The options available for this property are:

- **Program Controlled**

  Program controlled sets the **Method** based on the **Output Fields** value:

  – for an Exterior output field, the solver uses the Variational Indirect BEM method.

  – for an Interior output field, the solver uses the Collocation BEM method.

- **Variational Indirect BEM**

  An indirect BEM implemented by variational formulation for solving the Helmholtz integral equation. In this method, the primary variables are discontinuity of acoustic pressure and the normal gradient of the pressure across the boundary elements. This method is flexible and can work with acoustic problems with closed boundary and those with openings and appendices.

- **Collocation BEM**

  A direct BEM implemented by collocation formulation for solving the Helmholtz integral equation. In this method, the primary variables are the acoustic pressure and acoustic velocity. It works for closed boundary acoustic problems. The acoustic results (acoustic pressure, normal velocity, and acoustic intensity) can be obtained on the boundary elements directly.

- **Collocation BEM with Burton-Miller formulation**

  Although this option appears in the field, you should not use it.

Output Fields

The space in which the output is calculated. Specify Interior if you have an enclosed domain or Exterior if you have an infinite domain.

Maximum Number of Iterations

Specify the maximum number of iterations to reach convergence. The default value is 1000.

Tolerance

Specify the convergence tolerance. The default value is 0.000001.

Number of Domain Decompositions

Specify the number of domains that you want to divide the geometry into during the solution. Domain decompositions is used to save memory. For large problems, the boundary mesh is decomposed into the number of domains specified for reduced memory allocation. The default value is 8.

Note:

If the **Multiple Steps** property is enabled under **Step Controls**, the Worksheet will be read only. The values are populated from the **Imported Velocity** object in the Outline. Only one active object is allowed and will populate the worksheet.

Worksheet

Analysis Settings

Properties	Step 1	Step 2	Step 3	Step 4	Step 5
Step Controls					
RPM Value	10.472	20.944	31.416	41.888	52.36
Step Frequency Spacing	Linear	Linear	Linear	Linear	Linear
Step Frequency Range Minimum	0.	100.	200.	300.	400.
Step Frequency Range Maximum	100.	200.	300.	400.	500.
Step Solution Intervals	2.	3.	4.	5.	6.

When you modify RPM properties in the upstream system (Harmonic Response), you must perform these 2 steps in order to update the RPM properties in the LS-DYNA Acoustics system:

1. Update the **Setup** cell of LS-DYNA Acoustics system in the project Schematic.

2. Right-click the **Imported Load Group** in the LS-DYNA Acoustics System and select Create velocities and Sync Analysis Settings.

.

Chapter 3: Acoustics Loading Conditions

This section describes all of the supported acoustic-based loading and boundary conditions. They are separated into groups based on type.

Acoustic Excitations

Mass Source (p. 32)

Surface Velocity (p. 34)

Diffuse Sound Field (p. 37)

Incident Wave Source (p. 40)

Port In Duct (p. 43)

Mass Source Rate (p. 45)

Surface Acceleration (p. 1)

Acoustic Loads

Temperature (p. 50)

Impedance Sheet (p. 53)

Static Pressure (p. 55)

Acoustic Boundary Conditions

Pressure (p. 57)

Impedance Boundary (p. 59)

Absorption Surface (p. 63)

Radiation Boundary (p. 65)

Absorption Element (p. 67)

Free Surface (p. 69)

Thermo-Viscous BLI Boundary (p. 71)

Rigid Wall (p. 73)

Symmetry Plane (p. 75)

Port (p. 77)

Far-field Radiation Surface (p. 80)

Fringe Plot (p. 82)

Acoustic Models

Transfer Admittance Matrix (p. 83)
Low Reduced Frequency (p. 86)

3.1. Mass Source

A **Mass Source** excitation is used to create a sound wave.

Analysis Types

The **Mass Source** excitation is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the excitation, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Mass Source** excitation include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Mass Source**.

- Body: *Supported*.
- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*.
- Nodes: *Supported*.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The **Mass Source** boundary condition's loading is defined by **Magnitude** only.

Loading Data Definition: Enter loading data using one of the following options.

- Constant
- Tabular (Frequency Varying)

Boundary Condition Application

To apply a **Mass Source**:

1. On the **Environment** Context tab: click **Acoustic Excitations**>**Mass Source**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustic**>**Mass Source**.
2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the Body selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only bodies specified in the acoustics **Physics Region(s)** can be selected or defined.
3. Define the **Magnitude**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Body, 3 Bodies, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Mass Source**.

Category	Fields/Options/Description
	Magnitude **Important:** Changing the geometry scoping may change the Unit system used for the magnitude. For example, the unit system for the **Magnitude** of a face is kg/m^2 × s whereas the unit for an edge selection is kg/m × s. Furthermore, if you change the scoping from geometry to nodes, in addition to the unit system changing to kg/s, the value of the **Magnitude** property also changes. That is, a magnitude of 10 kg/m^2 × s (face scoping) could become 5 kg/s (edge scoping). In addition, if you change from a geometry selection to a node-based selection, the **Magnitude** property automatically resets to zero (0) and the field highlights in yellow to indicate you need to make an entry. **Phase Angle** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Acoustic Mass Source is applied using the **MASS** label of **BF** command.

API Reference

See the Mass Source section of the ACT API Reference Guide for specific scripting information.

3.2. Surface Velocity

A **Surface Velocity** loading condition applies a velocity to a surface on your model.

Analysis Types

The **Surface Velocity** excitation condition is available for Coupled Field Harmonic, Harmonic Acoustics (p. 3), and LS-DYNA Acoustics (p. 27) analyses.

Common Characteristics

This section describes the characteristics of the excitation condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Surface Velocity** boundary condition include:

- Solid: **Supported**. (*Not Supported* for LS-DYNA analyses.)
- Surface/Shell: *Not Supported*. (**Supported** for LS-DYNA analyses.)
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Surface Velocity**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Normal To**
- **Components**

Loading Data Definition: Enter loading data as:

- **Constant**
- **Tabular** (Frequency Varying)

Boundary Condition Application

To apply a **Surface Velocity**:

1. On the **Environment** Context tab: click **Acoustic Excitations**>**Surface Velocity**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustic**>**Surface Velocity**.
2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the Body selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only bodies specified in the acoustics **Physics Region(s)** can be selected or defined.
3. Select the method used to define the load: **Normal To** (default) or **Components**.

4. Define the **Magnitude**.

Note:

When you define multiple Surface Velocities there is no cumulative loading effect. Therefore, if you define multiple Surface Velocities on the same face, only the last Surface Velocity, as defined in the input file, is used to calculate results.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry (Face only) selections must be made on faces specified in the acoustic Physics Region. *(For LS-DYNA Acoustics systems the geometry is scoped to shell bodies and there is no acoustics physics region.)* Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Surface Velocity**. **Define By**, options include: • **Normal To**: This option requires **Magnitude** and **Phase Angle** entries. • **Components**: this option defines the loading type as components in the Global Coordinate System or a user-defined local coordinate system. *(The Components option is not supported for LS-DYNA Acoustics analyses.)* Requires the specification of at least one of the following inputs: – **X Component**: Defines magnitude in the X direction.

Category	Fields/Options/Description
	– **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Phase Angle** – **Y Phase Angle** – **Z Phase Angle** **Magnitude** **Phase Angle**: **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Application of Acoustic Surface Velocity is based on how you define it:

- If **Define By** = **Normal To**, the application uses the **SHLD** label of **SF** command. This defines surface normal velocity.
- If **Define By** = **Components**, the application uses the **VELO** label of **BF** command.

For more information, refer to the Outward Normal Velocity and the Arbitrary Velocity sections in the *Mechanical APDL Acoustic Analysis Guide*.

API Reference

See the Surface Velocity section of the ACT API Reference Guide for specific scripting information.

3.3. Diffuse Sound Field

You use the **Diffuse Sound Field** excitation condition to create random excitation waves. The diffuse sound field is approached by the asymptotic model summing a high number of uncorrelated plane waves with random phases from all directions in free space. For additional information, see the **DFSWAVE** section in the Mechanical APDL Command Reference.

Analysis Types

The **Diffuse Sound Field** excitation condition is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Diffuse Sound Field** excitation condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Diffuse Sound Field**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Data Definition: The **Diffuse Sound Field** excitation condition is defined as a constant.

Boundary Condition Application

To apply a **Diffuse Sound Field**:

1. On the **Environment** Context tab: select **Acoustic Excitations** > **Diffuse Sound Field**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Diffuse Sound Field**.
2. Define the **Scoping Method**: options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the **Face** selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only faces specified in the structural **Physics Region**(s) can be selected or defined.
3. Define **Coordinate System** such that the +Z axis of the Cartesian coordinate system is consistent with the panel's outward normal unit vector on the panel's incident diffuse sound field side.

Details View Properties

The selections available in the Details view are described below.

<table>
<tr><th>Category</th><th>Fields/Options/Description</th></tr>
<tr><td>Scope</td><td>Scoping Method - Options include:

• Geometry Selection: This is the default setting and indicates that the boundary condition is applied to a geometry or geometries, you select using a graphical selection tools.

– Geometry: Visible when the Scoping Method is set to Geometry Selection. Displays the type of geometry (Face selection only) and the number of geometric entities (for example: 1 Face, 2 Faces) to which the boundary has been applied using the selection tools.

• Named Selection: This option Indicates that the geometry selection is defined by a Named Selection.

– Named Selection: Visible when the Scoping Method is set to Named Selection. This field provides a drop-down list of available user-defined Named Selections.</td></tr>
<tr><td>Definition</td><td>Type: Read-only field that displays boundary condition type - Diffuse Sound Field.

Coordinate System: Drop-down list of available coordinate systems. Global Coordinate System is the default. For this Coordinate System selection, the positive Z axis (Cartesian) must align with the normal of the selected face or faces (the outward normal unit vector).

Radius of Reference Sphere Define By, the options for this property include:

• Program Controlled: The application selects the Radius automatically.

• User Defined: This option enables you to specify the Radius.

Reference Power Spectral Density: Entry field. The default value is <code>1 W/Hz</code>.

Maximum Incident Angle: Entry field. The default value is <code>0°</code>.

Suppressed - Include (No - default) or exclude (Yes) the boundary condition.</td></tr>
<tr><td>Base Medium</td><td>Material Assignment: Select a material to define material properties (Mass Density and Speed of Sound) of the Base Medium.

Mass Density: Read-only field whose value is updated based on the selected Material Assignment.

Speed of Sound: Read-only field whose value is updates based on the selected Material Assignment.</td></tr>
<tr><td>Advanced</td><td>Number of Divisions on the Reference Sphere: Entry field. The default value is <code>20</code>.

Random Sampling Type: the options for this property include All (default), Multiple, and Single.

Number of Samplings: Entry field. The default value is <code>1</code>.</td></tr>
</table>

Category	Fields/Options/Description
	Norm Convergence Tolerance: Entry field. The default value is `0.05`. **Frequency of Norm Convergence Check**: Entry field. The default value is **5**.

Mechanical APDL References and Notes

The Mechanical APDL command **DFSWAVE** is used to apply the acoustic **Diffuse Sound Field** excitation condition. For more information, refer to the Random Excitation with Diffuse Sound Field section in the *Mechanical APDL Acoustic Analysis Guide*.

Important:

Acoustic analyses use the **MSOLVE** command for the solution. This command creates multiple solutions for the analysis. As a result, the result file contains multiple solution sets for the loading condition. For the result types that have the **By** property under the **Definition** category, such as **Total Deformation**, the **By** property has only one option, **Set**. You can evaluate these results for a specific **Set Number** as desired.

API Reference

See the Diffuse Sound Field section of the ACT API Reference Guide for specific scripting information.

3.4. Incident Wave Source

An acoustic **Incident Wave Source** excitation condition is used to create an incident wave.

Analysis Types

The acoustic **Incident Wave Source** excitation condition is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Important:

- This excitation is not supported if the Solution Method property (**Analysis Settings**) is set to **Krylov**.
- When you select the **Back Enclosed Loudspeaker** option for the **Wave Type** property, the **Pressure** setting for the **Excitation Type** property is not supported.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Topology: No topology selection is made for the Incident Wave Source. Its location is defined using the **Source Origin X/Y/Z** coordinates.

Loading Data Definition: The **Incident Wave Source** excitation condition is defined as a constant.

Boundary Condition Application

To apply an **Incident Wave Source**:

1. On the **Environment** Context tab: select **Acoustic Excitations** > **Incident Wave Source**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Incident Wave Source**.
2. Define **Wave Type**, **Excitation Type**.
3. Define **Incident Wave Location**. Options include **Wave Type** other than **Planar Wave**.
4. Define **Pressure Amplitude** for **Excitation Type** = **Pressure**.
5. Define **Velocity Amplitude** for **Excitation Type** = **Velocity**.
6. Define the **Material Assignment** property (**Base Medium**). A selection fly-out menu is provided.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Definition	**Type**: Read-only field that describes the object - **Incident Wave Source**. **Wave Type**: This property provides the following options. • **Planar Wave** • **Monopole** • **Dipole** • **Back Enclosed Loudspeaker** • **Bare Loudspeaker** **Incident Wave Location**: This property provides the following options. • **Outside The Model** • **Inside The Model**

Category	Fields/Options/Description
	• Read-only for **Wave Type** = **Planar Wave** and selects the **Outside The Model** option. **Excitation Type**: This property provides the following options. • **Pressure** • **Velocity** Specification of the following inputs are required based on **Wave Type** and **Excitation Type**. • **Pressure Amplitude**: Input available for **Excitation Type** = **Pressure** to specify excitation pressure amplitude. • **Velocity Amplitude**: Input available for **Excitation Type** = **Velocity** to specify excitation velocity amplitude. • **Phase Angle**: Input to specify the phase angle • **Angle Phi (From X Axis Toward Y Axis)**: Input available for **Wave Type** = **Planar Wave**. • **Angle Theta (From Z Axis Toward X Axis)**: Input available for **Wave Type** = **Planar Wave**. • **Radius of Pulsating Sphere**: Input available for Wave Type = Monopole, Dipole, Back Enclosed Loudspeaker, and Bare Loudspeaker. The Radius must be greater or equal to zero. Specification of the following inputs are required for **Wave Type** = **Dipole** and **Bare Loudspeaker**. • **Dipole Length** • **X Component of Unit Dipole Vector** • **Y Component of Unit Dipole Vector** • **Z Component of Unit Dipole Vector** **Calculate Incident Power**: This property provides the options **No** and **Yes**. **Port Selection**: Select a port from drop-down list of available valid ports. **Suppressed**: Include (No - default) or exclude (Yes) the boundary condition.
Base Medium	**Material Assignment**: Select a material to define material properties (Mass Density and Speed of Sound) of the Base Medium. **Mass Density**: Read-only field whose value is updated based on the selected **Material Assignment**.

Category	Fields/Options/Description
	Speed of Sound: Read-only field whose value is updates based on the selected **Material Assignment**.
Location	When you set **Wave Type** to either **Monopole**, **Dipole**, **Back Enclosed Loudspeaker**, and **Bare Loudspeaker**, define location of wave source using the following properties: • **Source Origin X**: Input field. • **Source Origin Y**: Input field. • **Source Origin Z**: Input field. • **Source Location**: Geometry Selection button.

Mechanical APDL References and Notes

The Mechanical APDL command **AWAVE** is used to apply the **Incident Wave Source** excitation condition. Refer to the Analytic Incident Wave Sources section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

API Reference

See the Incident Wave Source section of the ACT API Reference Guide for specific scripting information.

3.5. Port In Duct

You use the **Port In Duct** excitation condition is used to create an incident wave and acoustic duct ports.

Analysis Types

The **Port In Duct** loading condition is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Scoping

The **Port In Duct** excitation scoped using the Port (p. 77) object.

Loading Data Definition

The **Port In Duct** excitation condition is defined as a constant only.

Boundary Condition Application

To apply a **Port In Duct**:

1. On the **Environment** Context tab: select **Acoustic Excitations** > **Port In Duct**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Port In Duct**.

2. Define Port Attribution, Wave Type, Coordinate System.

3. Define Pressure Amplitude & Phase Angle

4. Define Width, Height, and Mode indices along width and height for Wave Type = Rectangular Duct

5. Define Radius, and Mode indices along azimuth and radii for Wave Type = Circular Duct

6. Select a Port from the available Ports

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Definition	**Type**: Read-only field that describes the object – **Port In Duct**. **Port Attribution**: This property provides the options **Inlet** and **Outlet**. **Wave Type**: This property provides the following options. • **Planar Wave** • **Rectangular Duct** • **Circular Duct** **Coordinate System**: Drop-down list of available coordinate systems. Global Coordinate System is the default. The Local Coordinate System should be chosen such that the cross section of **Port in Duct** lies in X-Y Plane. Specification of the following inputs are required based on **Wave Type**. • **Pressure Amplitude**: Input specify excitation pressure amplitude. Only available for Port Attribution Inlet. • **Phase Angle**: Input to specify the phase angle. • **Angle Phi (From X Axis Toward Y Axis)**: Input available for **Wave Type** = **Planar Wave**.

Category	Fields/Options/Description
	• **Angle Theta (From Z Axis Toward X Axis)**: Input available for **Wave Type** = **Planar Wave**. For **Wave Type** = **Rectangular Duct**, define the following rectangular duct properties: • **Width** • **Height** • **Mode index for pressure variation along the width** • **Mode index for pressure variation along the height** For **Wave Type** = **Circular Duct**, define the following circular duct properties: • **Radius** • **Mode index for pressure variation along the azimuth** • **Mode index for pressure variation along the radii** **Port Selection**: Select a port from drop-down list of available valid ports. The **Vibro** option for **Port Behavior** is supported. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The command **APORT** is used to apply the Acoustic Port In Duct excitation condition. For more information, refer to the Specified Mode Excitation in an Acoustic Duct section in the *Mechanical APDL Acoustic Analysis Guide*.

API Reference

See the Port In Duct section of the ACT API Reference Guide for specific scripting information.

3.6. Mass Source Rate

A **Mass Source Rate** excitation is used to create a sound wave.

Analysis Types

The **Mass Source Rate** excitation is available for a Coupled Field Transient analysis.

Common Characteristics

This section describes the characteristics of the excitation, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Mass Source Rate** excitation include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Mass Source Rate**.

- Body: **Supported**.
- Face: **Supported**.
- Edge: **Supported**.
- Vertex: **Supported**.
- Nodes: *Not Supported*.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The **Mass Source Rate** boundary condition's loading is defined by **Magnitude** only.

Loading Data Definition: Enter loading data using one of the following options.

- Constant
- Tabular (Time Varying)

Boundary Condition Application

To apply a **Mass Source Rate**:

1. On the **Environment** Context tab: click **Acoustic Excitations**>**Mass Source Rate**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustic**>**Mass Source Rate**.
2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the Body selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only bodies specified in the acoustics **Physics Region(s)** can be selected or defined.
3. Define the **Magnitude**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Body, 3 Bodies, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Mass Source Rate**. **Magnitude** **Important:** Changing the geometry scoping may change the Unit system used for the Magnitude. For example, the Unit system for the **Magnitude** of a Face is kg $m^{-2}s^{-2}$ whereas the Unit for an Edge selection is kg $m^{-1}s^{-2}$. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Acoustic **Mass Source Rate** is applied using the **MASS** label of **BF** command.

API Reference

See the Mass Source Rate section of the ACT API Reference Guide for specific scripting information.

3.7. Surface Acceleration

A **Surface Acceleration** loading condition applies an acceleration to a surface on your model.

Analysis Types

The **Surface Acceleration** excitation is available for a Coupled Field Transient analysis.

Common Characteristics

This section describes the characteristics of the excitation condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Surface Acceleration** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Surface Acceleration**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Normal To**
- **Components**

Loading Data Definition: Enter loading data as:

- **Constant**
- **Tabular** (Time Varying)

Boundary Condition Application

To apply a **Surface Velocity**:

1. On the **Environment** Context tab: click **Acoustic Excitations**>**Surface Acceleration**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustic**>**Surface Acceleration**.
2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the Body selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only bodies specified in the acoustics **Physics Region(s)** can be selected or defined.
3. Select the method used to define the load: **Normal To** (default) or **Components**.
4. Define the **Magnitude**.

Note:

When you define multiple Surface Accelerations there is no cumulative loading effect. Therefore, if you define multiple Surface Accelerations on the same face, only the last Surface Acceleration, as defined in the input file, is used to calculate results.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry (Face only) selections must be made on faces specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).

Category	Fields/Options/Description
Definition	**Type**: Read-only field that describes the object - **Surface Acceleration**. **Define By**, options include: • **Normal To**: This option requires a **Magnitude** entry. • **Components**: This option defines the loading type as components in the Global Coordinate System or a user-defined local coordinate system. Requires the specification of at least one of the following inputs: – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. **Magnitude**: Specify the magnitude. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Application of Acoustic Surface Acceleration is based on how you define it:

- If **Define By** = **Normal To**, the application uses the **SHLD** label of **SF** command. This defines surface normal Acceleration.
- If **Define By** = **Components**, the application uses the **VELO** label of **BF** command.

For more information, refer to the Outward Normal Acceleration and the Arbitrary Acceleration sections in the *Mechanical APDL Acoustic Analysis Guide*.

API Reference

See the Surface Accelerator section of the ACT API Reference Guide for specific scripting information.

3.8. Temperature

This boundary condition applies a constant or spatially varying temperature. Geometry selections for this load must be made on bodies specified in the acoustic Physics Region(s).

Note:

The application does not support data transfer for this loading condition.

Analysis Types

The **Temperature** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), and Modal Acoustics (p. 16) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Temperature** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Temperature**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Types: The **Temperature** boundary condition's loading is defined by **Magnitude** only.

Loading Data Definition: Enter loading data using one of the following options.

- **Constant**: **Supported**.
- **Tabular** (Spatially Varying): **Supported**.
- **Function** (Spatially Varying): **Supported**.

Boundary Condition Application

To apply a **Temperature**:

1. On the **Environment** Context tab: select **Acoustic Loads**>**Temperature**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Temperature**.

2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the Body selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only bodies specified in the acoustics **Physics Region(s)** can be selected or defined.

3. Define the **Magnitude**, **Coordinate System**, and/or **Direction** of the thermal boundary condition based on the above selections.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Body selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry (Body only) selections must be made on bodies specified in the acoustic Physics Region(s). Use the Body selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Body, 3 Bodies, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (body-based only).
Definition	**Type**: Read-only field that describes the object - **Temperature**. **Magnitude**: Enter a temperature value. The default value is 22°C. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Temperatures are applied using the **BF** command.

- **Magnitude** (constant, tabular, and function) is always represented as a table in the input file.

Also see the Non-Uniform Ideal Gas Material section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

API Reference

See the Temperature section of the ACT API Reference Guide for specific scripting information.

3.9. Impedance Sheet

You can specify the 2 x 2 transfer admittance matrix with continuous pressure and discontinuous normal velocity across an acoustic impedance sheet using this object. For additional details, refer to Impedance Sheet topic in the *Mechanical APDL Acoustic Analysis Guide*.

Analysis Types

The **Impedance Sheet** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), and Modal Acoustics (p. 16) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Impedance Sheet** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Impedance Sheet**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.

- Element Face: **Not Supported**.

Loading Types: This boundary condition's loading is applied as a constant only.

Loading Data Definition: Loading data specified as a constant only.

Boundary Condition Application

To apply a **Impedance Sheet**:

1. On the **Environment** Context tab: click **Acoustic Loads**>**Impedance Sheet**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Impedance Sheet**.
2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the Face selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only faces specified in the acoustics **Physics Region** can be selected or defined.
3. Define **Resistance** and **Reactance** based on the above selections.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. You use this property to specify the face or faces of bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Impedance Sheet**. **Resistance**: Specify the resistance. This property can be designated as a parameter.

Category	Fields/Options/Description
	Reactance: Specify the reactance. This property can be designated as a parameter. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The application applies Impedance Sheet using the **IMPD** label of **BF** command.

API Reference

See the Impedance Sheet section of the ACT API Reference Guide for specific scripting information.

3.10. Static Pressure

An **Static Pressure** load applies a constant pressure to one or more bodies in the acoustic fluid regions.

Note:

If both Static Pressure and Temperature are defined, the ideal gas model is activated.

Analysis Types

The **Static Pressure** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), and Modal Acoustics (p. 16) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Static Pressure** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Static Pressure**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Types: This boundary condition's loading is only applied to the volume of the selected topology.

Loading Data Definition: Loading data specified as a constant only.

Boundary Condition Application

To apply a **Static Pressure**:

1. On the **Environment** Context tab: click **Acoustic Loads**>**Static Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Static Pressure**.
2. Define the **Scoping Method**. Options include **Geometry Selection** (default) and **Named Selection**. For either scoping type, you must use the **Body** selection filter (on the Graphics Toolbar) for geometry selection or Named Selection definition. Only bodies specified in the acoustics **Physics Region(s)** can be selected or defined.
3. Define the **Magnitude** of the pressure.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Body selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry (Body only) selections must be made on bodies specified in the acoustic Physics Region. Use the Body selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Body, 3 Bodies, etc.).

Category	Fields/Options/Description
	• **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (body-based only).
Definition	**Type**: Read-only field that describes the object - **Static Pressure**. **Magnitude**: Enter a constant pressure value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Static Pressure is applied using the **SPRE** label of **BF** command. Also see the Non-Uniform Ideal Gas Material section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

API Reference

See the Static Pressure section of the ACT API Reference Guide for specific scripting information.

3.11. Pressure

You can specify pressure on desired fluid regions in an acoustic analysis by inserting an **Pressure** object.

This boundary condition can be scoped only to face/edge/vertex of a body in acoustic domain. Refer to Pressure Boundary section of *Mechanical APDL Acoustic Analysis Guide* for more information.

Analysis Types

The **Pressure** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), Modal Acoustics (p. 16), and LS-DYNA Acoustics (p. 27) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Pressure** boundary condition include:

- Solid: **Supported**. (*Not Supported* for LS-DYNA analyses.)
- Surface/Shell: *Not Supported*. (**Supported** for LS-DYNA analyses.)
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Pressure**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: *Supported*. (**Not Supported** for LS-DYNA analyses.)
- Vertex: *Supported*. (**Not Supported** for LS-DYNA analyses.)
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Types: This boundary condition's loading is only applied to a vertex, edge, or area of the selected topology.

Loading Data Definition: Loading data specified as a constant only.

Boundary Condition Application

To apply a **Pressure**:

1. On the **Environment** Context tab: click **Acoustic Boundary Conditions**>**Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Pressure**.
2. Define the **Scoping Method**.
3. Define the **Magnitude** of the pressure.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. • **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Specified geometric entities must be in the acoustic Physics Region. *(For LS-DYNA Acoustics systems the geometry is scoped to shell bodies and there is no acoustics physics region.)*

Category	Fields/Options/Description
	Use the selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. This property displays the type of geometry (Face, Edge, etc.) and the number geometric entities (1 Face, 3 Faces, etc.) to which you have applied the boundary condition. *(For LS-DYNA Acoustics systems only Faces are supported.)* • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Pressure**. **Magnitude**: Enter a constant pressure value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The Pressure boundary condition is applied using the **PRES** label of **D** command.

3.12. Impedance Boundary

This boundary condition enables you to specify a complex form of impedance on an acoustic surface. It requires Resistance, Reactance, and Frequency entries. Refer to the Surface Impedance Boundary section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

The application defines complex impedance on a selected surface based on the inputs to the Acoustic Impedance Boundary using the command: **SF**,**Nlist**,**IMPD**,**VALUE**,**VALUE2**.

The application calculates **VALUE** and **VALUE2** using the following impedance and admittance definitions:

Impedance

The equation for Impedance:

$$Z = Z_r + jZ_i$$

where:

Z_r = Resistance

Z_i = Reactance

Admittance

The equation for Admittance (Y) = Inverse of Impedance:

$$Y = Z^{-1} = (Z_r + Z_i)^{-1} = \frac{1}{Z_r^2 + Z_i^2}(Z_r - jZ_i)$$

$$Y = Y_r + jY_i$$

where:

$Y_r = \text{Conductance}$

$Y_i = \text{Susceptance}$

$$Y_r = \frac{Z_r}{Z_r^2 + Z_i^2}$$

$$Y_i = \frac{-Z_i}{Z_r^2 + Z_i^2}$$

In Mechanical, you can use **Impedance Boundary** to define impedance or admittance (Modal Acoustics). The **VALUE** and **VALUE2** fields for the **SF** command are based upon the Resistance and Reactance entries.

Harmonic Acoustics

VALUE = Resistance

VALUE2 = Reactance

Modal Acoustics

If **Reactance** = 0 (Impedance):

VALUE = Resistance

VALUE2 = 0

If **Reactance** != 0 (Admittance):

$$VALUE = -Y_r$$

$$VALUE2 = 2\pi * Frequency * Y_i$$

Analysis Types

The **Impedance Boundary** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), Modal Acoustics (p. 16), and LS-DYNA Acoustics (p. 27) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Impedance Boundary** include:

- Solid: **Supported**. (*Not Supported* for LS-DYNA analyses.)
- Surface/Shell: *Not Supported*. (**Supported** for LS-DYNA analyses.)
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Impedance Boundary**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Data Definition: Enter loading data using one of the following options.

- Constant
- Tabular (Frequency Varying): *Supported for Harmonic Acoustics and LS-DYNA Acoustics only.*

Boundary Condition Application

To apply a **Impedance Boundary**:

1. On the **Environment** Context tab: click **Acoustic Boundary Conditions**> **Impedance Boundary** . Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Impedance Boundary**.
2. Define the **Scoping Method**.
3. Define the **Resistance**, **Reactance**, and **Frequency** of the Acoustic Impedance Boundary based on the above selections.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. *(For LS-DYNA Acoustics systems the geometry is scoped to shell bodies and there is no acoustics physics region.)* Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Impedance Boundary**. **Resistance**: Specify the resistance. This property can be designated as a parameter. **Reactance**: Specify the reactance. This property can be designated as a parameter. **Frequency** (Modal Acoustics): Specify a frequency value. This property is displayed for Modal Acoustics analyses when the Reactance property is specified as greater than zero or if it is parameterized. This property can be designated as a parameter. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Impedance Boundary is applied using the **IMPD** label of **SF** command.

API Reference

See the Impedance Boundary section of the ACT API Reference Guide for specific scripting information.

3.13. Absorption Surface

The **Absorption Surface** boundary condition enables you to dampen the effect of sound pressure on the impedance boundary as well as to approximate infinity.

Analysis Types

The **Absorption Surface** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), and Modal Acoustics (p. 16) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Absorption Surface** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Absorption Surface**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Data Definition: Enter loading data using one of the following options.

- Constant

- Tabular (Frequency Varying): *Supported for Harmonic Acoustics only.*

Boundary Condition Application

To apply a **Absorption Surface**:

1. On the **Environment** Context tab: click **Acoustic Boundary Conditions**>**Absorption Surface**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics Absorption Surface**.
2. Define the **Scoping Method**.
3. Define an **Absorption Coefficient**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Absorption Surface**. **Absorption Coefficient**: Specify an **Absorption Coefficient** value for the selected boundary. This property can be designated as a parameter. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is applied using the **SF** command. Refer to the Boundary with Absorption Coefficient description in the *Mechanical APDL Acoustic Analysis Guide* for more information.

API Reference

See the Absorption Surface section of the ACT API Reference Guide for specific scripting information.

3.14. Radiation Boundary

The **Radiation Boundary** boundary condition enables you to dampen the effect of sound pressure on the impedance boundary as well as to approximate infinity.

Analysis Types

The **Radiation Boundary** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), and Modal Acoustics (p. 16) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Radiation Boundary** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Radiation Boundary**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.

- Element Face: **Not Supported**.

Boundary Condition Application

To apply a **Radiation Boundary**:

1. On the **Environment** Context tab: click **Acoustic Boundary Conditions**>**Radiation Boundary**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Radiation Boundary**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Radiation Boundary**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is applied using the **INF** label of **SF** command. Refer to the Surface Impedance Boundary section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

API Reference

See the Radiation Boundary section of the ACT API Reference Guide for specific scripting information.

3.15. Absorption Element

An exterior acoustics problem typically involves an infinite, homogenous, inviscid fluid surrounding a given geometric entity. The pressure waves generated by this geometric entity must satisfy the radiation condition on the surrounding space. To simulate this infinite fluid, you enclose your model with the **Absorption Element** boundary condition. This approach truncates the unbounded domain by introducing a second-order absorbing boundary condition so that an outgoing pressure wave reaching the boundary of the model is "absorbed" with minimal reflections back into the fluid domain.

Note:

- For a 3-D acoustic analysis, the absorbing element must be scoped to spherical faces having the same radius and center and that are connected to bodies with the same material assignment. For more information, refer to Absorbing Boundary Condition (ABC) section in the *Mechanical APDL Acoustic Analysis Guide*.
- Currently, the **Absorption Element** boundary condition is not supported if you wish to scope it to an Acoustic region that is defined using frequency dependent material properties. However, you can resolve this issue by specifying the **TBCOPY** command using a Commands object at the Solution level.

Analysis Types

The **Absorption Element** boundary condition is available for the following analysis types:

- Coupled Field Harmonic
- Coupled Field Modal
- Coupled Field Static
- Coupled Field Transient
- Harmonic Acoustics (p. 3)
- Modal Acoustics (p. 16)
- Static Acoustics (p. 22)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Absorption Element** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Absorption Element**.

- Body: **Not Supported**.
- Face: *Supported* (spherical face only).
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Boundary Condition Application

To apply an **Absorbing Element**:

1. On the **Environment** Context tab: click **Acoustic Boundary Conditions**>**Absorption Element**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Absorption Element**.
2. Define the **Scoping Method**.

Note:

The application calculates the radius of the sphere internally based upon the geometry you have selected. Material property requirements are derived based on the assigned material using the **MPCOPY** command.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Face, Edge, etc.) and the number of geometric entities (for example: 1 Face, 2 Edges) to which the boundary has been applied using the selection tools. These geometry selections must be made on bodies specified in one of the acoustic Physics Regions. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Absorption Element**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The Absorption Element is applied using the FLUID130 element.

3.16. Free Surface

This boundary condition enables you to specify a plane as a free surface in order to consider sloshing effects on your model. You should define an Acceleration load as well with Gravitational Acceleration values to properly define the sloshing problem.

Note:

The free surface must be aligned with the coordinate plane in the Global Cartesian Coordinate System. The gravitational acceleration input should always be positive regardless of how the model is set up.

Analysis Types

The **Free Surface** loading condition is available for Coupled Field Harmonic, Coupled Field Modal, Coupled Field Transient, Harmonic Acoustics (p. 3), and Modal Acoustics (p. 16) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Types Supported

- 3D Simulation: **Supported**.
- 2D Simulation: *Not Supported*.

Geometry Types: Geometry types supported for the **Free Surface** boundary condition include:

- Solid: **Supported**.
- Surface/Shell: *Not Supported*.
- Wire Body/Line Body/Beam: *Not Supported*.

Topology: The following topology selection options are supported for **Free Surface**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element: **Not Supported**.
- Element Face: **Not Supported**.

Loading Data Definition: Not applicable.

Boundary Condition Application

To apply a **Acoustic Free Surface**:

1. On the **Environment** Context tab: click **Acoustic Boundary Conditions**>**Free Surface**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Acoustics**>**Free Surface**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Free Surface**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is applied using the **FREE** label of the **SF** command. Refer to the Free Surface (Sloshing Effect) Sources section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

3.17. Thermo-Viscous BLI Boundary

Acoustic waves propagating in viscous-thermal media include a complex propagating constant in the frequency domain. The attenuation of the acoustic wave is proportional to the shear and bulk viscosity and the thermal conduction coefficient of the media. The interaction between the viscous fluid and rigid walls is taken into account using the **Thermo-Viscous BLI Boundary** condition.

Analysis Types

The **Thermo-Viscous BLI Boundary** loading condition is available for Coupled Field Harmonicand Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Thermo-Viscous BLI Boundary** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Thermo-Viscous BLI Boundary**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition: Not applicable.

Boundary Condition Application

To apply an **Thermo-Viscous BLI Boundary**:

1. On the **Environment** Context tab: select **Acoustic Boundary Conditions** > **Thermo-Viscous BLI Boundary**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Thermo-Viscous BLI Boundary**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools.

Category	Fields/Options/Description
	– **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Thermo-Viscous BLI Boundary**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is applied using the **BLI** label of **SF** command. For additional information, see the Boundary Layer Impedance (BLI) Model topic in the *Mechanical APDL Acoustic Analysis Guide*.

API Reference

See the Thermo-Viscous BLI Boundary section of the ACT API Reference Guide for specific scripting information.

3.18. Rigid Wall

This boundary condition enables you to specify a face as rigid wall (Neumann boundary).

Analysis Types

The **Acoustic Rigid Wall** loading condition is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Acoustic Rigid Wall** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Acoustic Rigid Wall**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition: Not applicable.

Boundary Condition Application

To apply an **Rigid Wall**:

1. On the **Environment** Context tab: select **Acoustic Boundary Conditions** > **Rigid Wall**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Rigid Wall**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry

Category	Fields/Options/Description
	selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Acoustic Rigid Wall**. **Coordinate System**: Drop-down list of available coordinate systems. Global Coordinate System is the default. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This excitation condition is applied using **RIGW** label of the **SF** command. For more information, refer to the Rigid Wall Boundary section in the *Mechanical APDL Acoustic Analysis Guide*.

API Reference

See the Rigid Wall section of the ACT API Reference Guide for specific scripting information.

3.19. Symmetry Plane

This boundary condition enables you to define the symmetry plane (Neumann boundary).

Analysis Types

The **Symmetry Plane** loading condition is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Symmetry Plane** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Symmetry Plane**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition: Not applicable.

Boundary Condition Application

To apply a **Symmetry Plane**:

1. On the **Environment** Context tab: select **Acoustic Boundary Conditions** > **Symmetry Plane**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Symmetry Plane**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry

Category	Fields/Options/Description
	selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Symmetry Plane**. **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **Symmetry Axis**: Read-only field with the setting **Normal to Selected Faces**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is consumed for Far-field result calculations. You will not see any change to the input file if the **HFSYM** command is used for the Far-field calculation.

API Reference

See the Symmetry Plane section of the ACT API Reference Guide for specific scripting information.

3.20. Port

An acoustic **Port** enables you to define an exterior or interior acoustic surface in order to reuse this surface as an excitation location, a boundary, or a surface on which result are evaluated. For example, a **Port** can be used to launch acoustic modes or define transfer admittance (p. 83) connections.

Analysis Types

The **Port** loading condition is available for the following analysis types:

- Coupled Field Harmonic
- Coupled Field Static

- Harmonic Acoustics (p. 3)
- Static Acoustics (p. 22)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Port** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Port**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition: The **Port** boundary condition is defined as a constant only.

Boundary Condition Application

To apply a **Port**:

1. On the **Environment** Context tab: select **Acoustic Boundary Conditions** > **Port**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Port**.
2. Define the attributes of the Port.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Port Surfaces	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only). **Port Position**: Options include **On Exterior Face** (default) and **On Interior Face**.
Inside Surface Bodies	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Body selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Body selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Body, 3 Bodies, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (body-based only).
Definition	**Type**: Read-only field that describes the object – **Port**.

Category	Fields/Options/Description
	Port Behavior: Options include **Transparent** (default) and **Vibro**. **Suppressed**: Include (No - default) or exclude (Yes) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is applied using the **PORT** label of **SF** (Exterior port) or **BF** (Interior port) commands. Refer to the Surface Port section in the *Mechanical APDL Acoustic Analysis Guide* for more information.

API Reference

See the Port section of the ACT API Reference Guide for specific scripting information.

3.21. Far-Field Radiation Surface

This boundary condition enables you to define the Maxwell surface for Far-field parameters.

Analysis Types

The Acoustic **Far-Field Radiation Surface** loading condition is available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Far-Field Radiation Surface** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Far-Field Radiation Surface**.

- Body: *Supported*.
- Face: *Supported*.

- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition: Not applicable.

Boundary Condition Application

To apply an **Far-field Radiation Surface**:

1. On the **Environment** Context tab: select **Acoustic Boundary Conditions** > **Far-Field Radiation Surface**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Far-Field Radiation Surface**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Equivalent Surface Selection	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).

Category	Fields/Options/Description
Inside Surface Bodies	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** (Face selection only): Visible when the **Scoping Method** property is set to **Geometry Selection**. Geometry selections must be made on bodies specified in the acoustic Physics Region. Use the Face selection filter to pick your geometry, click in the **Geometry** field, and then click the **Apply** button that displays. After you select the geometry, this property displays the geometric entities (1 Face, 3 Faces, etc.). • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections (face-based only).
Definition	**Type**: Read-only field that describes the object - **Far-Field Radiation Surface**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

This boundary condition is applied using **MXWF** label of the **SF** command. For more information, refer to the Equivalent Surface Source section in the *Mechanical APDL Acoustic Analysis Guide*.

API Reference

See the Far-field Radiation Service section of the ACT API Reference Guide for specific scripting information.

3.22. Fringe Plot

The area selected for the fringe plot will be treated as a result only (results will be reported for the nodes on the scoped body). No boundary conditions will be applied.

Analysis Types

Fringe Plot is an **Acoustics Load** that is available for LS_DYNA Acoustics (p. 27) analyses only.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

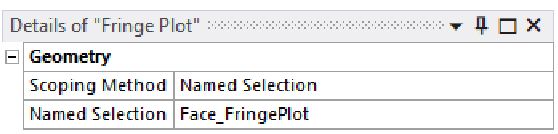
Boundary Condition Application

To apply a **Fringe Plot**:

1. On the **Environment** Context tab: select **Acoustic Boundary Conditions** > **Fringe Plot**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Fringe Plot**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.



Scope the object using Geometry Selection or Named Selection Scoping Methods.

3.23. Transfer Admittance Matrix

This acoustic model enables you to specify a transfer admittance matrix to trim structures such as a complex perforated structure, a square grid structure, or a hexagonal grid structure. The connection at the interface between the two entities is between two acoustic regions or a structural and uncoupled acoustic region. The transfer admittance matrix can be specified using two Ports and if one interface is an FSI interface, then Port 1 is assigned to that interface. For additional information, see the Trim Element with Transfer Admittance Matrix topic in the *Mechanical APDL Acoustic Analysis Guide*.

Analysis Types

Transfer Admittance Matrix is an **Acoustics Model** that is available for the following analysis types.

- Coupled Field Harmonic
- Coupled Field Static
- Harmonic Acoustics (p. 3)
- Static Acoustics (p. 22)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Note:

Only Transparent Ports on Exterior Face are allowed in Transfer Admittance Matrix.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Scoping: The **Transfer Admittance Matrix** is scoped using **Port** (p. 77) objects.

Loading Data Definition: Enter loading data using one of the following options.

- Constant
- Tabular (Frequency Varying): *Supported for Harmonic Acoustics only.*

Boundary Condition Application

To apply a **Transfer Admittance Matrix**:

1. On the **Environment** Context tab: select **Acoustic Models** > **Transfer Admittance Matrix**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Transfer Admittance Matrix**.
2. Define the **Port1** and **Port2** properties.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Definition	**Type**: Read-only field that describes the object - **Transfer Admittance Matrix**. **Model Type**: This property specifies the type of structure to be trimmed. It provides a drop-down with the options: **Perforated Plate** (default), **Square Grid Structure**, and **Hexagonal Grid Structure**. Based on the entry of the **Model Type** property, the following properties are available.

<table>
<tr><th>Category</th><th>Fields/Options/Description</th></tr>
<tr><td></td><td>
<table>
<tr><th>Perforated Plate</th><th>Square Grid Structure or Hexagonal Grid Structure</th></tr>
<tr><td>Port1
Y11 [Real]
Y11 [Imag]
Y12 [Real]
Y12 [Imag]
Alpha1 [Real]
Alpha1 [Imag]
Port2
Y21 [Real]
Y21 [Imag]
Y22 [Real]
Y22 [Imag]
Alpha2 [Real]
Alpha2 [Imag]</td><td>Port1
Port2
Hole Radius
Grid Period
Structure Thickness
Mass Density of the Fluid
Dynamic Viscosity of the Fluid
Ratio of Inner and Outer Radius</td></tr>
</table>
Suppressed: Include (No - default) or exclude (Yes) the boundary condition.</td></tr>
</table>

Mechanical APDL References and Notes

Transfer Admittance Matrix Models of Perforated Structures is applied using the command:

TB,PERF,,,,TBOPT

Where:

- **TBOPT** = **YMAT** for **Model Type** = **Perforated Plate**. This corresponds to the case of General transfer admittance matrix.
- **TBOPT** = **SGYM** for **Model Type** = **Square Grid Structure**.
- **TBOPT** = **HGYM** for **Model Type** = **Hexagonal Grid Structure**.

API Reference

See the Transfer Admittance Matrix section of the ACT API Reference Guide for specific scripting information.

3.24. Low Reduced Frequency Model

This acoustic model enables you to account for the interaction between an acoustic pressure wave in a viscous fluid and a rigid wall for specific structures according to Low Reduced Frequency (LRF) approximation. For additional information, see the Low Reduced Frequency (LRF) Model topic in the *Mechanical APDL Acoustic Analysis Guide*.

Guidelines

See Low Reduced Frequency for guidelines in using a **Low Reduced Frequency Model** in an acoustic simulation.

Analysis Types

Low Reduced Frequency Model is an **Acoustics Model** that is available for the following analysis types.

- Coupled Field Harmonic
- Coupled Field Static
- Harmonic Acoustics (p. 3)
- Static Acoustics (p. 22)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Scoping: The **Low Reduced Frequency Model** model is scoped bodies.

Loading Data Definition: Enter loading data using one of the following options.

- Constant
- Tabular (Frequency Varying): *Supported for Harmonic Acoustics only.*

Boundary Condition Application

To apply a **Low Reduced Frequency**:

1. On the **Environment** Context tab: select **Acoustic Models** > **Low Reduced Frequency Model**. Or, right-click the **Environment** tree object or click in the **Geometry** window and select **Insert** > **Acoustics** > **Low Reduced Frequency Model**.
2. Define body scoping.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Definition	**Type**: Read-only field that describes the object - **Low Reduced Frequency Model**. **Model Type**: This property specifies the type of structure to be represented. It provides a drop-down with the options: **Thin Layer** (default), **Rectangular Tube**, and **Circular Tube**. Based on the **Model Type** property selection, one of the following properties displays: • **Thickness Of Layer**: Enter a thickness value. • **Width Of Rectangle** and **Height Of Rectangle**: enter width and height values. • **Radius Of Circle**: Enter a radius value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The Low Reduced Frequency Model is applied using the command:

TB,AFDM,,,,TBOPT

Where:

- **TBOPT** = **THIN** for **Model Type** = **Thin Layer**.
- **TBOPT** = **RECT** for **Model Type** = **Rectangular Tube**.
- **TBOPT** = **CIRC** for **Model Type** = **Circular Tube**.

API Reference

See the Low Reduced Frequency section of the ACT API Reference Guide for specific scripting information.

Chapter 4: Acoustic Results

Mechanical supports the following result types. The availability of the result type may vary based on the type of acoustics analysis you are performing.

4.1. Acoustics Contour Results

4.2. Acoustic Far-field Results

4.3. Acoustic Frequency Response

4.4. Acoustic Power Loss Results

4.5. Acoustic Diffuse Sound Transmission Loss

4.6. Acoustic Waterfall Diagrams

Also, refer to the Reviewing Analysis Results section of the *Mechanical APDL Acoustic Analysis Guide* for more information.

4.1. Acoustics Contour Results

The following contour result options are supported for acoustic analyses:

Contour Result Option	Description
Pressure	Nodal pressures are calculated as part of the overall nodal solution. This corresponds to Mechanical APDL command **PLNSOL**,**PRES**.
Total Velocity	The acoustic total velocity is calculated by the Mechanical APDL solver as the corresponding sum of the pressure gradient. This corresponds to Mechanical APDL command **PLNSOL**,**PG, SUM**.
Directional Velocity	The acoustic directional velocity is calculated by the Mechanical APDL solver as the corresponding component of the pressure gradient. The available directions are X, Y, and Z. This corresponds to Mechanical APDL command **PLNSOL**,**PG**, **X/Y/Z**.
Kinetic Energy	Acoustic kinetic energy in element. This corresponds to Mechanical APDL command **PLNSOL**,**KENE**.
Potential Energy	The acoustic potential energy in the element. This corresponds to Mechanical APDL command **PLNSOL**,**MENE**.

The following result options are supported Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

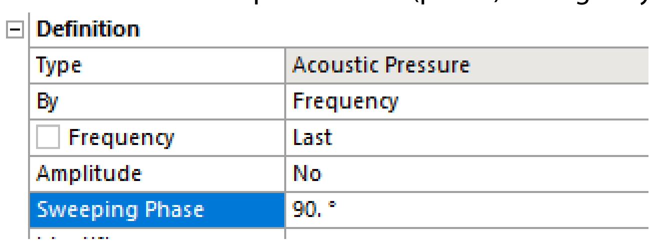
Contour Result Option	Description
Sound Pressure Level	The Acoustic Sound Pressure Level is calculated as part of the overall nodal solution. This corresponds to Mechanical APDL command **PLNSOL**,**SPL**.
A-Weighted Sound Pressure Level	The Acoustic A-Weighted Sound Pressure Level is calculated as part of the overall nodal solution. This corresponds to Mechanical APDL command **PLNSOL**,**SPLA**.
Frequency Band SPL	The Acoustic Frequency Band SPL is calculated for the requested nodes. This corresponds to Mechanical APDL command **PRAS**,**BSPL**.
A-Weighted Frequency Band SPL	The Acoustic A-Weighted Frequency Band SPL is calculated for the requested nodes. This corresponds to Mechanical APDL command **PRAS**,**BSPA**.

The following contour results are supported for an LS-DYNA Acoustics (p. 27) analysis:

Contour Result Option	Description
Pressure	Nodal pressures are calculated as part of the overall nodal solution.
Total Velocity	The acoustic total velocity is calculated as the corresponding sum of the pressure gradient.
Sound Pressure Level	The Acoustic Sound Pressure Level is calculated as part of the overall nodal solution.

Note:

The contour results plot real*cos(phase) - imaginary*sin(phase).

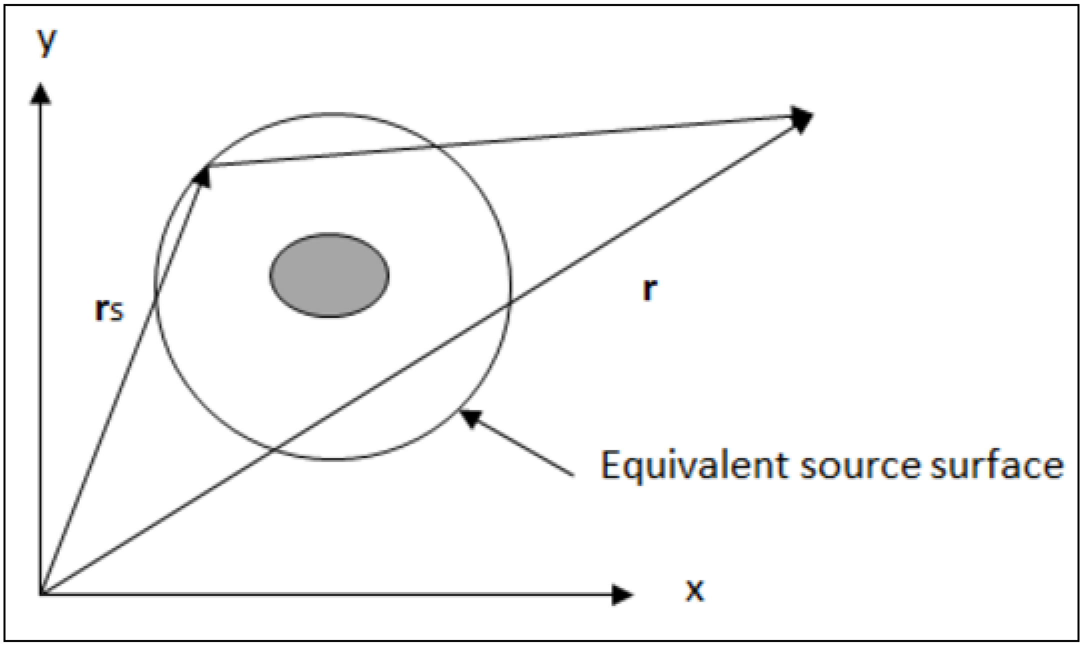


4.2. Acoustic Far-field Results

The far sound pressure field and far-field parameters (for example, radiation patterns, directivity, radiated power, radiation efficiency, and target strength) are essential for sound radiation or sound scattering analysis. The equivalent source surface principle using Green's function allows us to evaluate these parameters.

The surface equivalence principle enables you to calculate the pressure fields beyond the FEA domain. It states that the pressure field exterior to a given surface can be exactly represented by an equivalent source placed on that surface and allowed to radiate into the region external to that surface. Refer to

Far Sound Pressure Field and Far-Field Parameters topic in the *Mechanical APDL Acoustic Analysis Guide* for more information.



Where:

r = far-field observation position.

$\mathbf{r_s}$ = equivalent source position on the enclosed surface.

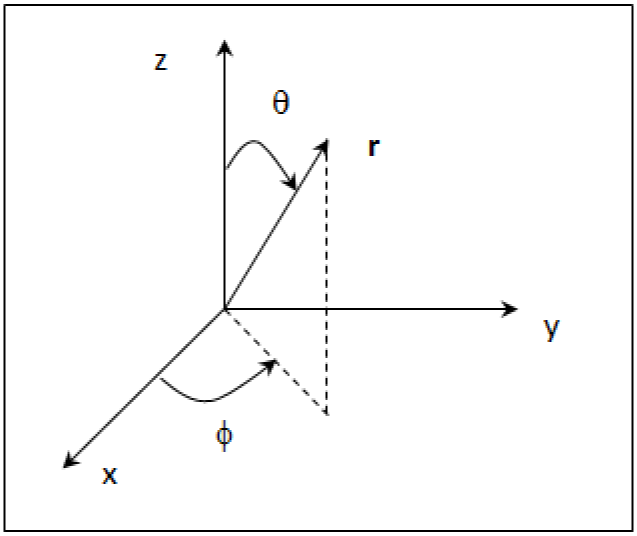
The following Far-field results are available in Mechanical:

Polar Plot Results	Microphone Results
• **Far-field SPL** • **Far-field A-Weighted SPL** • **Far-field Maximum Pressure** • **Far-field Phase** • **Far-field Directivity** • **Far-field Maximum Scattered Pressure** • **Far-field Target Strength** • **Far-field Sound Power Level**	• **Far-field SPL Mic** • **Far-field A-Weighted SPL Mic** • **Far-field Maximum Pressure Mic** • **Far-field Maximum Phase Mic**

See the Results and Result Tools (Group) object reference page for additional information about the **Details** view properties for these results.

Polar Plot Results

Polar plot results enable you to evaluate acoustic quantities on a spherical arc or surface defined by phi (from x to y), theta (from z to x-y plane), and r (radius) as illustrated in the following global polar coordinate system. You enter these values. In addition, these values are located outside the model mesh.



The arc or spherical surface on which the application calculates the results is displayed in the Geometry window prior to result generation.

Microphone Results

Microphone results enable you to evaluate acoustic quantities outside of the mesh by defining coordinates corresponding to the microphone location. Using the properties of the result object, you define either a singular microphone location in the Details view or up to nine microphone locations using the **Worksheet**.

Notes

Important:

If your Harmonic Acoustics analysis specifies an **Incident Wave Source** (p. 40) excitation:

- The Far-field Sound Power Level and Far-field Directivity results are not supported when the **Incident Wave Location** property is set to **Outside the Model** (default).
- In order to post process the Far-field Sound Power Level and Far-field Directivity results, the **Scattering Field Formulation** property must be set to **On**. Refer to the Scattering Controls section for additional information.

- The Far-field Maximum Scattered Pressure and Far-field Target Strength results are not supported when the **Incident Wave Location** property is set to **Inside the Model**.

Note:

- Currently, Far-field results are not:
  - Supported for animation.
  - Supported for user Defined Results.
  - Calculated at points on the finite element model. The Geometry window legend displays the minimum and maximum values but no contours are displayed on the elements.
- Except for the **Sound Power Level Far-field** and Microphone results, all Far-field Results are evaluated for a single frequency or a single set specified by the user. If a specified frequency is not contained in the result file frequency history, then the nearest frequency from the file is used.
- The **Far-field Sound Power Level** and Microphone results evaluate at all frequencies and the minimum and maximum displayed in the Geometry window legend represent the extremes over all frequencies.
- When you specify the **Model Type** property as either **2.5D Z-Direction Extrusion** or **2.5D Y-Axis Rotation Extrusion**, the location of microphone results is projected onto X-Y plane.
- The following capabilities are disabled if you have imported or restored an archive file (that does not include a result file).
  - Exporting Far-field Results to an ASCII file.
  - The Worksheet and Tabular Data upon clicking a Far-field Result.

  Re-solve the project in order to restore these post processing capabilities.

4.3. Acoustic Frequency Response

These Frequency Response options enable you to chart Acoustics results and display how the response varies with the frequency. Frequency Response options include the following acoustics-based results (p. 89):

- **Pressure**
- **Directional Velocity**
- **Kinetic Energy**
- **Potential Energy**
- **Sound Pressure Level**

- **A-Weighted Sound Pressure Level**

You can view these options as a value graphed along a specified frequency range. The plot includes all of the frequency points at which a solution was obtained. When you generate frequency response results, the default plot (Bode) shows the amplitude and phase angle. These Acoustic Frequency Response results are available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses only. The application evaluates the corresponding acoustic quantities using the following equation (Equation (2) of Frequency Response).

$$\{u(t)\} = \{u_{real}\}\cos(\Omega t) - \{u_{imag}\}\sin(\Omega t)$$

Where:

$$u_{real} = u_{\max}\cos\phi$$

$$u_{imag} = u_{\max}\sin\phi$$

$$u_{\max} = \sqrt{u_{real}^2 + u_{imag}^2}$$

$$\phi(\text{in degrees}) = \tan^{-1}\frac{u_{imag}}{u_{real}}$$

The LS-DYNA Acoustics (p. 27) system supports Frequency Response options for the following acoustics-based results:

- **Pressure**
- **Directional Velocity**
- **Sound Pressure Level**

Results may be scoped to a freestanding remote point.

4.4. Acoustic Power Loss Results

The following acoustic power loss results are available for Coupled Field Harmonic and Harmonic Acoustics (p. 3) analyses.

- **Transmission Loss**: This result solves for the Sound Power Level loss between the Incident Power on the Inlet Port and the Transmitted Power at Outlet Port.

  $$TL = 10\log_{10}\left(\frac{P_{in}}{P_t}\right)(dB)$$

- **Absorption Coefficient**: This result solves for the ratio of sound energy absorbed by a given Inlet Port to incident sound energy upon the surface

  $$\mathrm{ALPHA} = (\mathrm{I_{in}}\text{-}\mathrm{I})/\mathrm{I_{in}}$$

- **Return Loss**: This result solves for the difference between the Incident Sound Power and the Reflected Sound Power on Inlet Port.

$$RL = 10\log_{10}\left(\frac{P_{in}}{P_r}\right)(dB)$$

These results use a pre-defined Input Port (p. 77) and/or Output Port (p. 77) in order to derive acoustic power quantities based on the surface loading conditions applied during the analysis.

Port definition requires that the:

- **General Miscellaneous** property of the Analysis Setting be active (default).
- **Port Position** property of the selected Port (p. 77) object be set to **On Exterior Surface** (default setting).
- **Port Behavior** property of the **Port** be set to **Transparent** (default setting).

Refer to the **PLAS** command in the *Mechanical APDL Command Reference* for more information.

4.5. Acoustic Diffuse Sound Transmission Loss

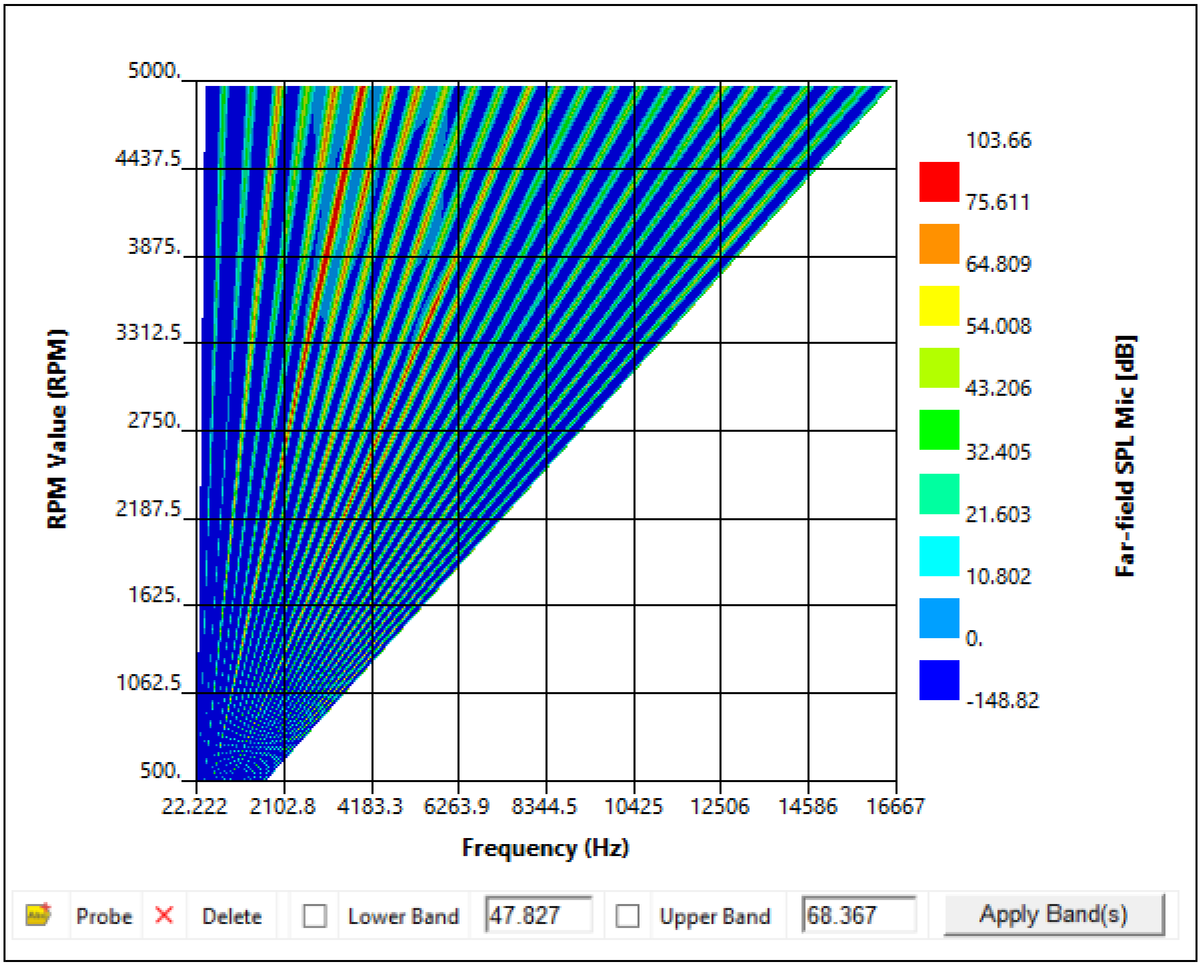
When you have defined a **Diffuse Sound Field** excitation condition, the **Diffuse Sound Field Transmission Loss** result enables you to predict the transmission loss of the structural panel specified by the excitation. This result calculates the average transmission loss for multiple sampling phases at each frequency over the frequency range. Refer to the **PLAS** command in the *Mechanical APDL Command Reference* for more information.

4.6. Acoustic Waterfall Diagrams

A Coupled Field Harmonic or a Harmonic Acoustics (p. 3) analysis that includes multiple RPM loading conditions, provides the following result options:

- **Far-field Sound Power Level Waterfall Diagram**
- **Far-field SPL Mic Waterfall Diagram**

These results calculate diagrams for Far-field Sound Power Level and SPL Microphone for all RPM loading conditions to display waterfall diagrams. An example is illustrated below.



Because Waterfall Diagrams can plot large amounts of data, the application provides a zoom feature as well as other display features in the diagram window. See the Waterfall Diagram Display Features section for the steps to use these features.