Simcenter Nastran Acoustics User's Guide

Contents

Figures

Proprietary & Restricted Rights Notice

List of symbols

Below is a list of the symbols used in this user's guide. Other symbols are defined where they appear.

Letter symbols	Description
A	Specific admittance
A	Area or coupling matrix
$A_f, [A]$	Area or coupling matrix
B	Damping matrix
B	Bulk modulus of air
B_f	Damping or the impedance matrix of the fluid
B_m	Viscous damper
B_s	Damping matrix of the structure
C	Coefficient
c	Speed of sound
c_0	Speed of sound in ambient medium
f	Frequency
$G(\omega)$	Mass density correction factor involving fluid viscous properties in the Johnson-Champoux-Allard model
$G'(\omega)$	Bulk modulus correction factor involving fluid thermal properties in the Johnson-Champoux-Allard model
G_e	Acoustic material damping
$G_e(\omega)$	Equivalent frequency-dependent damping
K	Stiffness matrix
K_f	Stiffness matrix or inverse mass matrix of the fluid
K_m	Grounded Spring
K_s	Stiffness matrix of the structure
K_s	Structure factor or square of tortuosity
k	Wave number
M	Mass matrix

M_f	Mass matrix or compressibility matrix of the fluid
M_m	Mass
M_s	Mass matrix of the structure
N_f	Shape functions for acoustic pressure
N_s	Shape functions for the structure
n_f	Outward normal to the fluid surface
n_s	Outward normal to the structural surface
P	Loading vector
$\mathcal{P}$	Power
Pr	Prandtl number
p	Acoustic pressure
Q_s	Acoustic volume velocity
$\dot{Q}_s$	Strength of acoustic source
q	Velocity potential
q_0	Static viscous permeability
q'_0	Static thermal permeability
$\boldsymbol{r}$	Unit radial vector
R	Gas constant of the acoustic fluid
S	Surface
$\hat{S}$	Monopole amplitude
u	Displacement
u_f	Vector of fluid displacements
$\dot{u}_f$	Acoustic particle velocity
u_{fn}	Vector of fluid displacements normal to the surface with outward normal
u_s	Vector of structural displacements
$\ddot{u}_s$	Accelerations
V	Volume
$W(f)$	Confidence level of the material

$X(f)$	Frequency-dependent reactance
x_s	Source coordinates
x	Spatial location
Z	Specific acoustic impedance
Greek letters	**Description**
α_∞	Tortuosity
β	Bulk modulus
β_e	Equivalent bulk modulus of the porous material
β_{er}, β_{ei}	Real and imaginary parts of bulk modulus of the porous materials
β_0	Bulk modulus of ambient medium
γ	Ratio of specific heats (isentropic expansion factor)
δ	Kronecker's delta
η	Dynamic viscosity
Λ	Viscous characteristic length
Λ'	Thermal characteristic length
ν	Kinematic viscosity
ν'	Thermal diffusivity
ρ	Acoustic fluid density
ρ_e	Equivalent density of the porous material
ρ_{er}, ρ_{ei}	Real and imaginary parts of equivalent density of the porous materials
$\rho_s(\omega)$	Frequency-dependent fluid density
ρ_0	Density of ambient medium
ρ_0	Speed of sound in air
σ	Flow resistivity
$\sigma(f)$	Frequency-dependent flow resistivity
$\left[\Phi_f\right]$	Vector of fluid modes
$\left[\Phi_s\right]$	Vector of structural modes
φ	Porosity
Mathematical operators	**Description**

$\nabla \cdot$	Divergence operator
∇	Gradient operator
Subscripts	**Description**
0	State of the ambient medium at a given temperature (not to be confused with 0 degree temperature)
e	Equivalent
f	Fluid
i	Imaginary
n	Normal
r	Real
S	Structural

Chapter 1: Introduction

1.1 Overview of acoustic capabilities

Simcenter Nastran can be used for both interior and exterior acoustics. It allows you to perform a pure acoustic analysis or a coupled vibro-acoustic analysis.

1.2 Assumptions and limitations

- Analysis is limited to linear acoustics.
- The current acoustic formulations in Simcenter Nastran assume that the medium is quiescent; that is, no mean flow of the ambient medium exists.

Chapter 2: Theory

2.1 Overview of acoustic theory

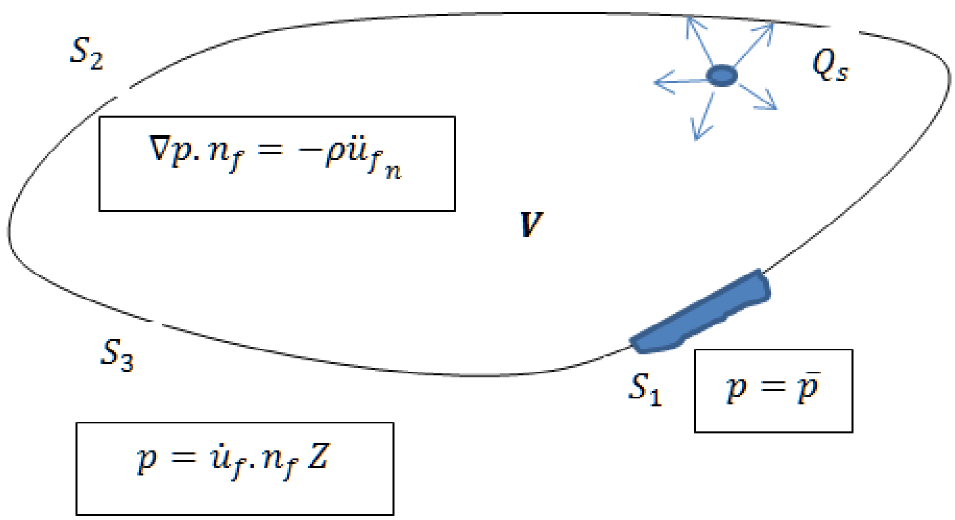


Figure 2-1. Interior Fluid Domain and Boundary Conditions

In Simcenter Nastran, the acoustic equations are based on

- Small linear perturbations with negligible convective momentum terms.
- Linear pressure-density relationship.

Thus Euler's momentum equation is

$$\rho_0 \ddot{u}_f + \nabla p = 0$$

Equation 2-1.

and the mass continuity equation is

$$\dot{\rho} + \rho_0 \nabla \cdot \dot{u}_f = \rho_0 Q_s \delta\left(\boldsymbol{x} - \boldsymbol{x}_s\right)$$

Equation 2-2.

where:

- ρ_0 is the density of the ambient medium.
- $\dot{u}_f$ is the acoustic particle velocity.

- p is the acoustic pressure.
- Q_s is the volume velocity defined on a unit volume.
- δ is the Kronecker's delta function defined such that the value of $\delta(x\text{-}x_s)$ is zero everywhere except at location defined by x_s where the value is unity.

Under linear pressure variation,

$$\frac{p}{\rho} = {c_0}^2$$

Equation 2-3.

where c_0 is the speed of sound in the ambient medium. Substituting Equation 3 in Equation 2 gives

$$\dot{p} = -\beta \nabla \cdot \dot{u}_f + \beta Q_s \delta \left(\boldsymbol{x} - \boldsymbol{x}_s \right)$$

Equation 2-4.

with the compressibility or the bulk modulus β defined as

$$\beta = \rho_0 {c_0}^2$$

Equation 2-5.

Taking the time derivative of Equation 4 and rearranging some terms yields

$$\frac{1}{\beta} \ddot{p} = -\nabla \cdot \ddot{u}_f + \dot{Q}_s \delta \left(\boldsymbol{x} - \boldsymbol{x}_s \right)$$

Equation 2-6.

Similarly, using the ∇ operator on Equation 1 yields

$$\nabla \cdot \ddot{u}_f = -\frac{1}{\rho_0} \nabla \cdot \nabla p$$

Equation 2-7.

or

$$\frac{1}{\beta} \ddot{p} - \frac{1}{\rho_0} \nabla \cdot \nabla p = \dot{Q}_s \delta \left(\boldsymbol{x} - \boldsymbol{x}_s \right)$$

Equation 2-8.

2.2 Boundary conditions

You can specify the following boundary conditions for bounded and unbounded fluid domains.

2.2.1 Imposed pressure (Dirichlet boundary condition)

On surface S_1, you can specify an enforced pressure $p = \overline{p}$

2.2.2 Imposed normal velocity (Neumann boundary condition)

On surface S_2, you can define normal velocities as follows:

$$\nabla p \cdot n_f = -\rho_0 \ddot{u}_{f_n}$$

Equation 2-9.

where:

- n_f is the outward normal.
- $\ddot{u}_{f_n}$ is fluid accelerations (positive along the normal direction).

On rigid boundaries, the velocity and hence acceleration is zero. Therefore, Equation 9 will reduce to pressure gradient being zero.

2.2.3 Imposed impedance or admittance (Robin boundary condition)

On surface S_3, you can define an impedance or admittance boundary condition as follows:

$$p = \dot{u}_f \cdot n_f Z, \qquad \text{or} \qquad \frac{\dot{u}_f \cdot n_f}{p} = A$$

Equation 2-10.

where:

- Z is the specific acoustic impedance.
- A is specific admittance (reciprocal of impedance).

Because of equation $\rho_0 \ddot{u}_f + \nabla p = 0$ (see Overview of acoustic theory), Equation 10 becomes

$$\ddot{u}_f \cdot n_f = -\frac{\nabla p \cdot n_f}{\rho_0} = \frac{\dot{p}}{Z}$$

Equation 2-11.

2.2.4 Open or radiating boundary

An infinite boundary condition requires that the acoustic field vanishes at points farther away from the source. The far field radiation boundary condition is specified using the Sommerfeld boundary condition.

$$\lim_{r\to\infty}\left[r\left(\frac{\partial p}{\partial r}+\frac{1}{c}\frac{\partial p}{\partial t}\right)\right]=0$$

Equation 2-12.

An equivalent statement of Sommerfeld radiation condition is

$$\lim_{r\to\infty}\left[r\left(p-\rho c\dot{u}_r\right)\right]=0 \qquad \text{or} \qquad \lim_{r\to\infty}\left(\frac{p}{\dot{u}_r}\right)=Z=\rho c$$

Equation 2-13.

Thus, the boundary condition at infinity can be replaced by an impedance boundary condition (see Impedance or admittance). This can be accomplished by modeling a fluid region that surrounds the vibrating surface and by applying the impedance boundary condition on the free surface. The number of fluid elements that need to be modeled to approximate infinity depends on the frequency range of interest. At higher frequencies, more fluid exterior to the structure must be modeled. The impedance boundary condition can then be applied using the CAABSF elements.

The Automatically Matched Layer (AML) boundary condition allows defining Equation 12 more accurately.

2.3 Finite element formulation

Using Galerkin weighted residuals, expressing $p=\lfloor N_f \rfloor\{p_i\}$, and using $\lfloor N_f \rfloor^T$ as the weighting function to minimize the residuals, gives

$$\int_V N_f^T\left(\frac{1}{\beta}\ddot{p}-\frac{1}{\rho_0}\nabla\cdot\nabla p\right)dV=\int_V \dot{Q}_s\delta\left(\boldsymbol{x}-\boldsymbol{x}_s\right)dV$$

Equation 2-14.

Because of Gauss divergence theorem, Equation 14 can be expressed as follows:

$$\frac{1}{\beta}\int_V N_f^T\ddot{p}dV+\frac{1}{\rho_0}\int_V \nabla N_f^T\cdot\nabla p dV-\frac{1}{\rho_0}\int_S N_f^T\nabla p\cdot n_f dS=\int_V N_f^T\dot{Q}_s\delta\left(\boldsymbol{x}-\boldsymbol{x}_s\right)dV$$

Equation 2-15.

Equation 15 can be further simplified by substituting equation $\nabla p\cdot n_f=-\rho_0\ddot{u}_{f_n}$ (see Imposed normal velocity (Neumann boundary condition)) and equation

$$\ddot{u}_f \cdot n_f = -\frac{\nabla p \cdot n_f}{\rho_0} = \frac{\dot{p}}{Z}$$

(see Imposed impedance or admittance (Robin boundary condition)) as

$$\frac{1}{\beta}\int_V N_f^T \ddot{p} dV + \frac{1}{\rho_0}\int_V \nabla N_f^T \cdot \nabla p dV + \int_{S_2} N_f^T \ddot{u}_f n_f dS_2 + \frac{1}{Z}\int_{S_3} N_f^T \dot{p} dS = \int_V N_f^T \dot{Q}_s \delta\left(\boldsymbol{x} - \boldsymbol{x}_s\right) dV$$

Equation 2-16.

Given that $p = \lfloor N_f \rfloor \{p_i\}$, Equation 16 reduces to

$$\left[M_f\right]\{\ddot{p}\} + \left[B_f\right]\{\dot{p}\} + \left[K_f\right]\{p\} + \left[A_f\right]\{\ddot{u}_f\} = \{P_f\}$$

Equation 2-17.

where the Compressibility matrix is

$$\left[M_f\right] = \frac{1}{\beta}\int_V N_f^T N_f \, dV$$

Equation 2-18.

Inverse mass or mobility matrix

$$\left[K_f\right] = \frac{1}{\rho_0}\int_V \nabla N_f^T \cdot \nabla N_f \, dV$$

Area matrix

$$\left[A_f\right] = \int_{S_2} N_f^T N_f \lfloor n_f \rfloor dS$$

Impedance matrix

$$\left[B_f\right] = \frac{1}{Z}\int_{S_2} N_f^T N_f dS$$

and Acoustic source vector

$$\{P_f\} = \{\dot{Q}_s\}$$

Note: The contribution to the damping matrix B_f is due to the impedance boundary condition.

2.3.1 Frequency-dependent acoustic materials or absorbers

Acoustic absorbers can be characterized by a fluid that has frequency-dependent acoustic characteristics. For such a case, the density and bulk modulus are expressed in terms of equivalent density and equivalent bulk modulus which are complex quantities as follows:

$$\rho_e(\omega) = \rho_{er}(\omega) + i\rho_{ei}(\omega); \qquad \beta_e(\omega) = \beta_{er}(\omega) + i\beta_{ei}(\omega)$$

Equation 2-19.

where:

- $\rho_e(\omega)$ is the equivalent frequency-dependent density.
- $\beta_e(\omega)$ is the equivalent frequency-dependent bulk modulus.

Then,

$$\frac{1}{\rho_e(\omega)} = \frac{1}{\rho_{er}(\omega) + i\rho_{ei}(\omega)} = \frac{1}{\rho_s(\omega)}\left(1 + iG_e(\omega)\right)$$

Equation 2-20.

where G_e defines the equivalent of structural damping constant for the fluid. The density and G_e can then be determined in terms of ρ_{er} and ρ_{ei} as follows:

$$\rho_s(\omega) = \frac{\rho_{er}^2(\omega) + \rho_{ei}^2(\omega)}{\rho_{er}(\omega)}; \; G_e(\omega) = -\frac{\rho_{ei}(\omega)}{\rho_{er}(\omega)}$$

Equation 2-21.

Similarly,

$$\frac{1}{\beta_e(\omega)} = \frac{1}{\beta_{er}(\omega) + i\beta_{ei}(\omega)} = \frac{\beta_{er}(\omega) - i\beta_{ei}(\omega)}{\beta_{er}^2(\omega) + \beta_{ei}^2(\omega)} = \frac{1 - i\gamma(\omega)}{\beta_s(\omega)}$$

Equation 2-22.

where

$$\beta_s(\omega) = \frac{\beta_{er}^2(\omega) + \beta_{ei}^2(\omega)}{\beta_{er}(\omega)}; \quad \gamma(\omega) = \frac{\beta_{ei}(\omega)}{\beta_{er}(\omega)}$$

Equation 2-23.

Substituting Equation 20 and Equation 22 into equations

$$\left[K_f\right] = \frac{1}{\rho_0}\int_V \nabla N_f^T \cdot \nabla N_f \, dV$$

and

$$\left[M_f\right] = \frac{1}{\beta}\int_V N_f^T N_f \, dV$$

(see Finite element formulation) results in the following for the absorber

$$[K]_{absorber} = \frac{(1+iG_e(\omega))}{\rho_s(\omega)}\int_V \nabla N_f{}^T \cdot \nabla N_f \, dV$$
$$[M]_{absorber} = \frac{1-i\gamma(\omega)}{\beta_s(\omega)}\int_V N_f{}^T N_f \, dV$$

Equation 2-24.

Defining $[K']_{absorber}$ and $[M']_{absorber}$ such that

$$[K']_{absorber} = \frac{1}{\rho_s(\omega)}\int_V \nabla N_f{}^T \cdot \nabla N_f \, dV$$
$$[M']_{absorber} = \frac{1}{\beta_s(\omega)}\int_V N_f{}^T N_f \, dV$$
$$[K_4]_{absorber} = \frac{G_e(\omega)}{\rho_s(\omega)}\int_V \nabla N_f{}^T \cdot \nabla N_f \, dV = G_e(\omega)[K']_{absorber}$$

Equation 2-25.

Equation 24 can be re-written as

$$[K]_{absorber} = [K' + iK_4]$$
$$[M]_{absorber} = [M' - i\gamma(\omega)M']$$

Equation 2-26.

Note: The imaginary part of $[M]_{absorber}$ results in $[B]_{absorber}$ where

$$[B]_{absorber} = [\omega\gamma(\omega)M']$$

Equation 2-27.

2.3.2 Coupling of fluid and structure (vibro-acoustics)

In a coupled vibro-acoustic analysis, the fluid pressure on the structure boundary causes surface tractions on the structure

$$\{F_s\} = \int_S N_s^T \{\varphi\} \, dS$$

Equation 2-28.

where N_S is a shape function for the structure,

$$\varphi = -pn_s$$

Equation 2-29.

and n_s is the outward normal to the structure at the fluid boundary interface. Substituting Equation 29 in Equation 28 gives

$$\{F_s\} = -\int_S N_s^T \, p\{n_s\} \, dS = -\int_S N_s^T N_f \{n_s\} \, dS \{p\} = -[A]\{p\}$$

Equation 2-30.

The structure equations of motion then becomes

$$[M_s]\{\ddot{u}_s\} + [B_s]\{\dot{u}_s\} + [K_s]\{u_s\} = \{P_s\} - [A]\{p\}$$

Equation 2-31.

where

$$[A] = \int_{S_2} N_s^T N_f \{n_s\} \, dS \quad \text{or} \quad [A] = -\int_{S_2} N_s^T N_f \{n_f\} \, dS$$

Equation 2-32.

N_s and N_f are structure and fluid shape functions, and n_s = - n_f .

At the fluid structure interface

$$n_f \cdot \ddot{u}_f + n_s \cdot \ddot{u}_s = 0$$

Equation 2-33.

Equation $[M_f]\{\ddot{p}\} + [B_f]\{\dot{p}\} + [K_f]\{p\} + [A_f]\{\ddot{u}_f\} = \{P_f\}$ (see Finite element formulation) then becomes

$$[M_f]\{\ddot{p}\} + [B_f]\{\dot{p}\} + [K_f]\{p\} - [A^T]\{\ddot{u}_s\} = \{P_f\}$$

Equation 2-34.

The combined fluid structure equations then become

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

Equation 2-35.

where:

- *M* is the mass matrix.

- B is the damping matrix.
- K is the stiffness matrix.
- P is the loading vector.
- Subscript s represents the partitions of the structure.
- Subscript f represents the fluid degrees of freedom.
- Degrees of freedom are displacements, u for the structure and pressure.
- Degrees of freedom are displacements, p for the fluid.
- [A] matrix is the coupling between the fluid and structure degrees of freedom at the wetted interface.

Note, Equation 35 is unsymmetric. This equation is symmetrized by using a variable transformation as follows:

Let a velocity potential q be defined such that

$$p = \dot{q}$$

Equation 2-36.

Substituting Equation 36 in Equation 35, assuming harmonic time dependence, and then integrating the fluid equation in time gives

$$\begin{bmatrix} M_s & 0 \\ 0 & -M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{q} \end{Bmatrix} + \begin{bmatrix} B_s & A \\ A^T & -B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{q} \end{Bmatrix} + \begin{bmatrix} K_s & 0 \\ 0 & -K_f \end{bmatrix} \begin{Bmatrix} u_s \\ q \end{Bmatrix} = \begin{Bmatrix} P_s \\ \dfrac{P_f}{i\omega} \end{Bmatrix}$$

Equation 2-37.

Also, because of equation $\rho_0 \ddot{u}_f + \nabla p = 0$ (see Overview of acoustic theory), it follows that

$$\dot{u}_f = -\frac{1}{\rho_0} \nabla q$$

Equation 2-38.

The pressure and the velocities can therefore be recovered from Equation 36 and Equation 38 respectively.

2.3.3 Modal participation factors

The modal solution for the displacement and the pressures can be written in the form

$$\{u_s\} = [\Phi_s]\{q_s\}; \quad \{p\} = [\Phi_f]\{q_f\}$$

Equation 2-39.

where:

- $[\Phi_s]$ are the uncoupled, undamped structural mode shapes or eigenvectors.
- $[\Phi_f]$ are the uncoupled, undamped rigid-walled acoustic modes.
- The vector $\{q_s\}$ is the modal amplitude or the modal basis function for the structure.
- The vector $\{q_f\}$ is the modal amplitude or the modal basis function for the fluid.

The eigenvectors or the mode shapes allow the transformation from physical space to the modal space. Substituting Equation 39 in unsymmetrical equation

$$[M_f]\{\ddot{p}\} + [B_f]\{\dot{p}\} + [K_f]\{p\} - [A^T]\{\ddot{u}_s\} = \{P_f\}$$

(see Finite element formulation) results in

$$\begin{bmatrix} \Phi_S^T M_s \Phi_s & 0 \\ -\Phi_f^T A^T \Phi_s & \Phi_f^T M_f \Phi_f \end{bmatrix} \begin{Bmatrix} \ddot{q}_s \\ \ddot{q}_f \end{Bmatrix} + \begin{bmatrix} \Phi_S^T B_s \Phi_s & 0 \\ 0 & \Phi_f^T B_f \Phi_f \end{bmatrix} \begin{Bmatrix} \dot{q}_s \\ \dot{q}_f \end{Bmatrix} + \begin{bmatrix} \Phi_S^T K_s \Phi_s & \Phi_S^T A \Phi_f \\ 0 & \Phi_f^T K_f \Phi_f \end{bmatrix} \begin{Bmatrix} q_s \\ q_f \end{Bmatrix} = \begin{Bmatrix} \Phi_S^T P_s \\ \Phi_f^T P_f \end{Bmatrix}$$

Equation 2-40.

or

$$\begin{bmatrix} m_s & 0 \\ -a^T & m_f \end{bmatrix} \begin{Bmatrix} \ddot{q}_s \\ \ddot{q}_f \end{Bmatrix} + \begin{bmatrix} b_s & 0 \\ 0 & b_f \end{bmatrix} \begin{Bmatrix} \dot{q}_s \\ \dot{q}_f \end{Bmatrix} + \begin{bmatrix} k_s & a \\ 0 & k_f \end{bmatrix} \begin{Bmatrix} q_s \\ q_f \end{Bmatrix} = \begin{Bmatrix} f_s \\ f_f \end{Bmatrix}$$

Equation 2-41.

where:

$$\begin{aligned} & m_s = \Phi_S^T M_s \Phi_s; \quad m_f = \Phi_f^T M_f \Phi_f \\ & k_s = \Phi_S^T K_s \Phi_s; \quad k_f = \Phi_f^T K_f \Phi_f \\ & b_s = \Phi_S^T B_s \Phi_s; \quad b_f = \Phi_f^T B_f \Phi_f \\ & a = \Phi_S^T A \Phi_f \\ & f_s = \Phi_S^T P_s; \quad f_f = \Phi_f^T P_f \end{aligned}$$

Equation 2-42.

Assuming a harmonic solution of the form $e^{i\omega t}$, Equation 41 for the fluid degrees of freedom can be reduced to

$$\left[-\omega^2\left[m_f\right]+i\omega\left[b_f\right]+\left[k_f\right]\right]\left\{q_f\right\}=\left\{f_f\right\}-\omega^2\left[a^T\right]\left\{q_s\right\}$$

Equation 2-43.

which in turn can be further rewritten as

$$\left\{q_f\right\}=\left[Z_2\right]\left\{f_f\right\}-\omega^2\left[Z_2\right]\left[a^T\right]\left\{q_s\right\}=\left[Z_2\right]\left\{f_f\right\}-\omega^2\left[Z_2\right]\left[\Phi_f^T\right]\left[A^T\right]\left[\Phi_s\right]\left\{q_s\right\}$$

Equation 2-44.

where

$$\left[Z_2\right]=\left[-\omega^2\left[m_f\right]+i\omega\left[b_f\right]+\left[k_f\right]\right]^{-1}$$

Equation 2-45.

The fluid modal contribution at the grids is therefore,

$$\left[\chi_f\right]=\left[\Phi_f\right]\left[\Lambda_f\right]$$

Equation 2-46.

where $\left[\Lambda_f\right]$ is a diagonal matrix such that $\left[\Lambda_{f_{ii}}\right]=q_{f_i}$

The structure modal contribution for the response is given by

$$\left[\chi_s\right]=-\omega^2\left[\Phi_f\right]\left[Z_2\right]\left[a^T\right]\left[\Lambda_s\right]=-\omega^2\left[\Phi_f\right]\left[Z_2\right]\left[\Phi_f^T\right]\left[A^T\right]\left[\Phi_s\right]\left[\Lambda_s\right]$$

Equation 2-47.

where $\left[\Lambda_s\right]$ is a diagonal matrix such that $\left[\Lambda_{s_{ii}}\right]=q_{s_i}$

The fluid structure panel participation is then

$$\left[\chi_{Panel}\right]=-\omega^2\left[\Phi_f\right]\left[Z_2\right]\left[a_{panel}^T\right]\left[\Lambda_s\right]=-\omega^2\left[\Phi_f\right]\left[Z_2\right]\left[\Phi_f^T\right]\left[A_{panel}^T\right]\left[\Phi_s\right]\left[\Lambda_s\right]$$

Equation 2-48.

and the fluid structure grid participation factor is then

$$\left[\chi_{panel\,grids}\right]=-\omega^2\left[\Phi_f\right]\left[Z_2\right]\left[\Phi_f^T\right]\left[A_{g\,panel}^T\right]\left[\Phi_s\right]\left[\Lambda_s\right]$$

Equation 2-49.

where $A_{g_{panel}}^T$ consists of columns of grids for panel g.

2.4 Output forms

Several forms of acoustic pressure are computed in an acoustic or vibro-acoustic solution.

2.4.1 RMS pressure

The mean square and the root mean square (RMS) values of acoustic pressure are defined by

$$p_m^2 = \frac{1}{T}\int p^2(t)\,dt; \qquad p_{rms} = \sqrt{p_m^2}$$

Equation 2-50.

where $p(t)$ is the pressure. For single frequency pressure, Equation 50 becomes

$$p_{rms} = \frac{1}{\sqrt{2}} p_0$$

Equation 2-51.

where p_0 is the peak pressure at a given frequency.

2.4.2 SPL

Generally, sound pressure is expressed on a logarithmic scale as the Sound Pressure Level (SPL) in decibels (dB). The SPL is defined by

$$SPL = 10\log_{10}\frac{p_m^2}{p_{ref}^2} = 20\log_{10}\frac{p_m}{p_{ref}}$$

Equation 2-52.

where p_{ref} is the reference pressure. For example, the reference pressure for air is $p_{ref} = 20\times10^{-6}\,Pa$.

2.4.3 Frequency weighted SPL

Psychoacoustic studies have determined that the perception of loudness in the human ear is frequency dependent. The SPL equation treats the pressure at all frequencies with the same weighting. To compensate for the sensory perception of noise, a transformation function

$$\left(p_m^2\right)_w = \sum_{i=1}^{n} W(f)\,p_m^2(f)$$

Equation 2-53.

is used, where *W* is the weighting function. The most commonly used weighting function is the A-weighting.

Chapter 3: Defining the model

3.1 Defining the fluid volume

The finite element representation of acoustic elements is similar to that in conventional structural finite elements, but differs from the structural elements in the following ways:

- The fluid grid points referenced by acoustic elements must have a value of -1 for the CD field (field 7) of the GRID card. These grids then constitute a set of fluid grids.
- The PSOLID physical property that is referenced by acoustic elements must specify PFLUID for the FCTN field (field 8).
- The PSOLID physical property that is referenced by acoustic elements must in turn reference a fluid material (MAT10, MAT10C, MATF10C, or MATPOR).

The following element types are supported:

- 3 D: CHEXA, CTETRA, CPENTA, and CPYRAM for 3-D elements. These elements must reference either a PSOLID or a PMIC physical property.
- 2 D: CTRIA3 and CQUAD4. These elements must reference only a PMIC physical property.
- 1 D: CROD. These elements must reference only a PMIC physical property.
- 0 D: MICPNT.

3.2 Defining porous or equivalent porous materials

Porous materials are often used to dampen acoustic wave propagation. These materials typically consist of a solid skeleton with pores or interstices through which acoustic waves can propagate. A fully coupled vibro-acoustic solution is then needed to model the poro-elastic material. The limiting case of poro-elastic materials is the case where the elastic skeleton is either completely rigid or limp (almost zero rigidity). Such materials are characterized by various material models. Simcenter Nastran supports the following porous material models:

- **MAT10**:

  You can define acoustic absorber properties using the MAT10 material definition. The values of $\rho_S(\omega)$, $\beta_S(\omega)$, $G_e(\omega)$ and $\gamma(\omega)$ as described in Frequency-dependent acoustic materials or absorbers are specified for RHO, BULK, GE, and GAMMA fields in the MAT10 (see MAT10 in the *Simcenter Nastran Quick Reference Guide*). The frequency-dependent tables are then specified in the continuation line.

- **MAT10C and MATF10C**:

You can also define acoustic absorber properties using the MAT10C or MATF10C material definition. These acoustic material bulk entries let you input complex density and speed of sound for SOL 108 and SOL 111.

- o MAT10C allows you to define constant or nominal properties.
- o MATF10C allows you to define material properties in tabular format.

For example, in a SOL 111 solution, when a modal analysis is first performed, the solver uses the nominal value to calculate the eigenvalue and eigenvector. In the frequency response, however, the actual value is used if MATF10C is specified.

- **MATPOR**:

The MATPOR bulk entry allows you to specify the material models directly, and lets the solver compute the equivalent density and speed of sound.

With the MATPOR bulk entry, you can select the Craggs, Delaney-Bazely/Miki, and Johnson-Champoux-Allard empirical model and enter the relevant parameters.

- o **Craggs**

The wave equation is given by

$$\frac{1}{\rho_0}\nabla^2 p - i\omega\frac{\sigma\varphi}{\rho_0\beta_0}p + \frac{\omega^2 K_s\varphi}{\beta_0}p = 0$$

Equation 3-1.

where:

$$K_S = \alpha_\infty^2$$

Equation 3-2.

where:

- σ is the flow resistivity.
- K_s is the structure factor or square of tortuosity.
- α_∞ is the tortuosity.
- φ is the porosity.

Then

$$\rho_s(\omega) = \rho; \quad G_e(\omega) = 0; \quad \beta_s(\omega) = \frac{\beta_0}{K_s\varphi}; \qquad \gamma(\omega) = \frac{\sigma}{K_s\rho\omega}$$

Equation 3-3.

- o **Delaney-Bazely/Miki**

The wave propagation is constant and the speed of sound is given by the following empirical relationship.

$$k = \frac{\omega}{c_0}\left[1 + C_1 X^{C_2} - iC_3 X^{C_4}\right] \quad \overset{yields}{\rightarrow} \quad c_{eq} = \frac{c_0}{\left[1 + C_1 X^{C_2} - iC_3 X^{C_4}\right]}$$

Equation 3-4.

$$Z = \rho_0 c_0 \left[1 + C_5 X^{C_6} - iC_7 X^{C_8}\right] \quad \overset{yields}{\rightarrow} \quad \rho_{eq} = \rho_0 \left[1 + C_5 X^{C_6} - iC_7 X^{C_8}\right]$$

Equation 3-5.

$$X = \frac{\rho_0 f}{\varphi \sigma}$$

Equation 3-6.

where:

- k is the wavenumber.
- c_0 is the speed of sound in air.
- Z is the characteristic impedance.
- ρ_0 is the density.
- σ is the flow resistivity.

Table 3-1. Coefficients for Delaney-Bazely and Miki

Coefficient	Delaney-Bazely and Miki
C_1	0.122
C_2	-0.618
C_3	0.180
C_4	-0.618
C_5	0.0785
C_6	-0.632

Table 3-1. Coefficients for Delaney-Bazely and Miki

Coefficient	Delaney-Bazely and Miki
C_7	0.120
C_8	-0.632

The values of $\rho_s(\omega)$ and $G_e(\omega)$ can then be computed from equations

$$\rho_s(\omega)=\frac{\rho_{er}^2(\omega)+\rho_{ei}^2(\omega)}{\rho_{er}(\omega)};\;\; G_e(\omega)=-\frac{\rho_{ei}(\omega)}{\rho_{er}(\omega)}$$

(see Frequency-dependent acoustic materials or absorbers)

and Equation 6. Similarly, the real and imaginary values of bulk modulus can be computed using equations

$$\beta_s(\omega)=\frac{\beta_{er}^2(\omega)+\beta_{ei}^2(\omega)}{\beta_{er}(\omega)};\;\; \gamma(\omega)=\frac{\beta_{ei}(\omega)}{\beta_{er}(\omega)}$$

(see Frequency-dependent acoustic materials or absorbers), Equation 4, and Equation 5.

- **Johnson-Champoux-Allard (JCA)**

  The equivalent density and bulk modulus are as follows:

$$\rho_{eq}(\omega)=\rho_0\left(\alpha_\infty+\frac{\varphi\sigma}{i\omega\rho_0}G(\omega)\right)$$

Equation 3-7.

$$B_{eq}(\omega)=B\left(\gamma-\frac{\gamma-1}{1+\dfrac{8\eta}{i\omega\Lambda'^2\rho_0\,\mathrm{Pr}}G'(\omega)}\right)^{-1}$$

Equation 3-8.

where:

$$G(\omega)=\left[1+\left(\frac{2\alpha_\infty\eta}{\varphi\Lambda\sigma}\right)^2\frac{i\omega\rho_0}{\eta}\right]^{\frac{1}{2}}$$

Equation 3-9.

$$G'(\omega) = \left[1 + \left(\frac{\Lambda'}{4}\right)^2 \frac{i\omega\rho_0 \Pr}{\eta}\right]^{\frac{1}{2}}$$

Equation 3-10.

$$q_0 = \frac{\eta}{\sigma}$$

Equation 3-11.

$$q'_0 = \frac{\varphi\Lambda'^2}{8}$$

Equation 3-12.

where:

- φ is the porosity.
- σ is the flow resistivity.
- α_∞ is the tortuosity.
- Λ is the viscous characteristic length.
- Λ' is the thermal characteristic length.
- $q_0 = \frac{\eta}{\sigma}$ is the static viscous permeability.
- η is the dynamic viscosity.
- ρ_0 is the speed of sound in air.
- B is the bulk modulus of air.
- ν is the kinematic viscosity.
- γ is the ratio of specific heats.
- ν' is the thermal diffusivity.
- $q'_0 = \frac{\varphi\Lambda'^2}{8}$ is the static thermal permeability.
- $\Pr$ is the Prandtl number.

- $G(\omega)$ is the mass density correction factor involving fluid viscous properties in the JCA model.
- $G'(\omega)$ is the bulk modulus correction factor involving fluid thermal properties in the JCA model.

You can use MATPOR bulk entries in combination with CHEXA, CPENTA, CPYRAM, and CTETRA elements that reference PSOLID property bulk entries.

For more information on MATPOR bulk entry, see MATPOR bulk entry in the *Simcenter Nastran Quick Reference Guide*.

3.3 Defining spatially varying fluid properties in acoustic analysis

You can use results from a computational fluid dynamics (CFD) general notation system (CGNS) file to define the fluid properties in acoustic radiation and propagation problems. By doing so, you can account for spatial variation in the properties of the fluid across the mesh. This allows you to more accurately model acoustic applications where significant thermal gradients exist. HVAC systems, exhaust systems, and gas turbines are examples of such applications.

This capability is applicable to SOL 108 and 111 acoustic analysis with standard FE and adaptive-order FEMAO meshes.

When you use the results from a CGNS file to define the fluid properties for an acoustic analysis, the software does the following:

1. Reads the property data from a CGNS format file.

   The CGNS format file is typically generated from a CFD mean flow simulation, and the results in the CGNS file are given at either the grid point locations or element centroid locations of the CFD mesh. The software can read the following results from the CGNS file:

   - Temperature only
   - Speed of sound only
   - Temperature and mass density
   - Temperature and pressure
   - Speed of sound and mass density
   - Speed of sound and pressure

   When the software reads a single property from the CGNS file, such as temperature only, it uses a constant value for the pressure that you must specify.

2. Using a maximum distance algorithm, maps the property data in the CGNS file to the grid points of the acoustic mesh. The mapped property data is stored in a SCH5 file.

> **Note**
>
> You cannot modify the parameters that control the maximum distance algorithm from within Simcenter Nastran.

3. If necessary, converts the property data at the grid point locations to speed of sound and mass density values.

4. Using the shape functions for the elements in the acoustic mesh, interpolates the values for speed of sound and mass density at the grid points to the Gauss points of the elements.

During the acoustic analysis solve, the software uses the mapped fluid properties. Thus, when the software performs the integrations to obtain the element matrices during acoustic analysis solve, it accounts for spatial variation in the fluid properties across each element.

Converting properties to speed of sound and mass density

The software uses speed of sound and mass density values when it calculates the element matrices. Thus, if the software reads temperature or pressure data from the CGNS file, the data must be converted to speed of sound and mass density data.

The software uses the following equation to convert temperature data to speed of sound data:

$$c = \sqrt{\gamma RT}$$

where c is the speed of sound, γ is the isentropic expansion factor, R is the gas constant, and T is the (absolute) temperature.

The software uses the following equation to convert pressure data to mass density data:

$$\rho = \frac{\gamma p}{c^2}$$

where ρ is the mass density, γ is the isentropic expansion factor, p is the (absolute) pressure, and c is the speed of sound.

Input file requirements

You use the data in a CGNS file to define the fluid properties for an acoustic analysis by adding the new ACTEMP case control command and bulk entry to the SOL 108 or 111 input file. Because an ACTEMP case control command and bulk entry are present, the software uses the material properties it obtains from the ACTEMP bulk entry specification.

- Use the ACTEMP case control command to select the ACTEMP bulk entry from which the software accesses the fluid property data. The ACTEMP case control command must be placed above the subcases.

- Use the ACTEMP bulk entry to specify the following:

  - The gas constant and isentropic expansion factor for the fluid, and, if applicable, the constant pressure.

  - The combination of properties to retrieve from the CGNS file, such as temperature and pressure, speed of sound and mass density, and so on.

  - o The unit number of the SCH5 file.

    The SCH5 file stores the CFD results that are mapped to the acoustic mesh.

  - o The IDs of the data tables in the CGNS file that you want to access.

- On the PSOLID bulk entry that the acoustic elements reference, in the FCTN field, specify PFLUIDEX.

- In the File Management section, use an ASSIGN statement to associate the SCH5 file with the unit number.

3.4 Defining boundary conditions

3.4.1 Rigid wall

For a rigid wall, no flow occurs across the boundary and hence from Euler's equation the pressure gradient is zero. The surface integral on the rigid boundary therefore vanishes by virtue of equation

$$[A]=\int_{S_2} N_s^T N_f \{n_s\}\, dS \quad \text{or} \quad [A]=-\int_{S_2} N_s^T N_f \{n_f\}\, dS$$

(see Coupling of fluid and structure (vibro-acoustics)).

This is also called the natural boundary condition. This is analogous to a traction free boundary condition, which is the natural boundary condition for the structural problem. No boundary condition needs to be defined for a rigid wall boundary condition.

3.4.2 Enforced pressure

Single-point constraints of the fluid (p = 0) may be enforced on the fluid boundary using the SPC entry with degree of freedom component set to 1. Non-zero enforced pressures can be applied directly using SPCD and RLOADi entries with the TYPE entry for the RLOADi card set to DISP.

3.4.3 Defining the fluid-structure interface boundary condition

The vibro-acoustic coupling interface can be defined in one of two ways:

- Two-way (strong) coupling — That is, the vibration of the structure excites the fluid which in turn causes pressure loading on the structure (see

  $$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix}\begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix}+\begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix}\begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix}+\begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix}\begin{Bmatrix} u_s \\ p \end{Bmatrix}=\begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

  in Coupling of fluid and structure (vibro-acoustics)).

- One-way (weak) coupling — Here, the effect of the fluid on the structure is assumed to be negligible. However, the vibration of the structure on the fluid is assumed to be significant.

Two-way coupling

The coupled fluid-structure interaction is given by equation

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

For more information, see Coupling of fluid and structure (vibro-acoustics).

This can be specified using the ACMODL bulk entry. When no ACMODL is specified, Simcenter Nastran uses default values to compute the wetted surface. As shown below, the fluid and the structure are coupled by the matrix [*A*]. This matrix is called the coupling matrix or the area matrix and is given by equation

$$[A] = \int_{S_2} N_s^T N_f \{n_s\} dS \quad \text{or} \quad [A] = -\int_{S_2} N_s^T N_f \{n_f\} dS$$

For more information, see Coupling of fluid and structure (vibro-acoustics).

Simcenter Nastran allows the structure and fluid mesh to be non-conformal or non-matching at the fluid-structure interface.

For a matching mesh, computation of coupling or the area matrix is straight forward because

the shape function $N_s = N_f$. However, when structure and fluid meshes are non-conformal (non-matching or dissimilar), this computation is quite complex.

The coupling algorithm uses the following multi-step approach to accurately compute the coupling or the area matrix (assuming a non-matching fluid-structure interface).

1. **Free face determination**

   Simcenter Nastran automatically determines the fluid and structural element free faces. If you provide a set of elements, grids, or physical properties, Simcenter Nastran internally converts these to valid element sets. Only those free faces that are associated with this limited set that you provide are then retained for coupling computation.

2. **Pairing**

   Starting with this step, the pairing algorithm works with one fluid face at a time. If you provide a set of elements, grids, or physical properties, Simcenter Nastran internally converts these to valid element sets. Only those free faces that are associated with this limited set that you provide are then retained for coupling computation.

   A bounding box is created around the fluid face as shown in the below *Bounding box* figure to find one or more structural faces that it overlaps. The height of the bounding box is controlled with the NORMAL field (outward normal), and the INTOL field (inward normal) on the ACMODL bulk entry.

   The SRCHUNIT field on the ACMODL bulk entry changes the meaning of the NORMAL and INTOL fields:

   - If SRCHUNIT = REL, NORMAL, and INTOL are a ratio of the search box height to the maximum edge length of the fluid free face (Default).

   - If SRCHUNIT = ABS, NORMAL, and INTOL are a search distance in the model/absolute units.

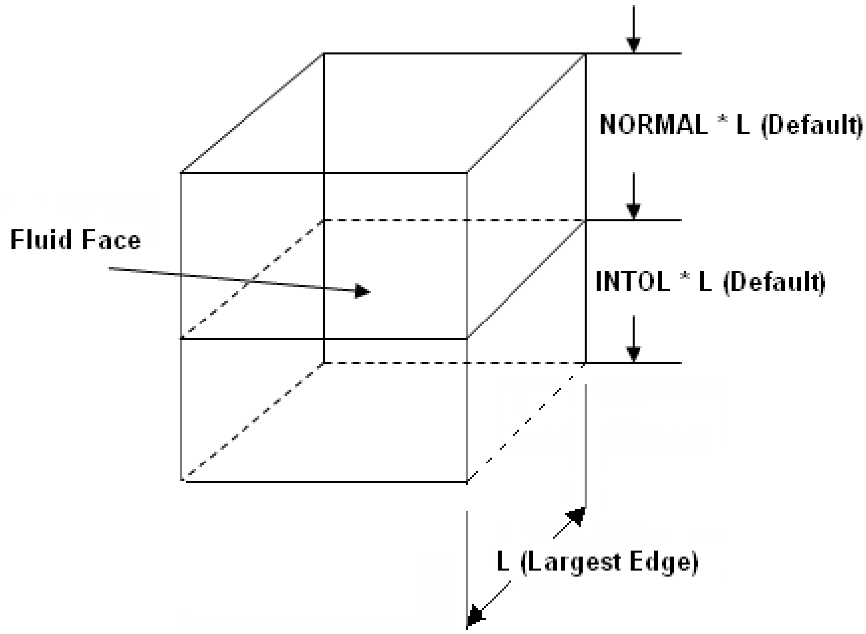


Figure 3-1. Bounding box

3. **Parallel check**

   The structural faces that are within the bounding box must be parallel to the fluid within a specified tolerance. This is checked by ensuring that the normal of the fluid face and structure faces form an angle less than the value of the OVLPANG field on the ACMODL bulk entry (default = 60 degrees).

4. **Subdivision**

   The grids from the structural faces from step 3 are projected onto the fluid face. A mesh is generated on the fluid face from the set of fluid grids and the projected structural grids such that the fluid and structural faces share a matching discretization. This virtual discretization is only used for evaluating the coupling matrix and is not included in the results output.

5. **Trimming**

   Subdivided structural face regions that are outside the fluid face are trimmed and eliminated from coupling with the current fluid face. However, these eliminated structural regions are considered for coupling when the above steps are repeated with other adjacent fluid faces.

6. **Integration**

   The subdivided regions on the fluid face are associated to subdivided regions on the structural faces for integration of matrix [*A*]. If multiple overlapping regions are paired with a single fluid region, the structural region that is the closest to the fluid region is coupled to the fluid face. Because the subdivided fluid and structural regions use a matching discretization, the resulting surface integral, that is, the overlap area between a fluid face and the given overlapping structural face, is exact and independent of structural/fluid mesh discretization.

One-way coupling

In the one-way (weak) coupling, the effect of fluid on the structure is neglected. That is, equation

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

(see Coupling of fluid and structure (vibro-acoustics)) becomes

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & 0 \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

Equation 3-13.

In this case, Equation 13 can be split into two sets of equations

$$M_s\ddot{u}_s + B_s\dot{u}_s + K_s u_s = P_s; \qquad M_f\ddot{p} + B_f\dot{p} + K_f p = P_f + A^T\ddot{u}_s$$

Equation 3-14.

This allows the first equation to be solved independently of the second equation. The resulting acceleration $\ddot{u}_s$ then becomes the load vector for the second set of equations. Because the size of the equations to be solved is now considerably smaller, solve times are faster.

The one-way coupling option can be selected via the CTYPE parameter on the ACMODL bulk entry.

> **Note**
>
> Simcenter Nastran solves the two sets of equations sequentially.

3.4.4 Defining acoustic panel normal velocity and transfer admittance

In SOL 108 and SOL 111, the acoustic panel normal velocity and transfer admittance boundary conditions are required for acoustic workflows such as muffler transmission loss. Transmission loss is a measure of a product's ability to reduce sound.

Acoustic panel normal velocity

Simcenter Nastran supports an acoustic panel normal velocity boundary that you can apply to 2D acoustic element faces, for example, free faces of a 3D fluid element. This velocity boundary can be:

- Used to represent acoustically rigid yet vibrating panels. In this case, the applied panel velocity corresponds with the particle velocity of the fluid in front of the panels.
- Combined with an acoustic impedance or admittance on the same panel. In this case, the acoustic panel normal velocity represents the structural vibration of an acoustically treated and therefore soft panel. Also, the particle velocity of the fluid in front of the panel is different from the structural panel velocity, which is pre-defined.

You use the ACPNVEL bulk entry to specify the boundary. ACPNVEL supports the definition of:

- Magnitude and phase of acoustic velocity.

- Constant or frequency-dependent complex velocity.

Transfer admittance

The admittance can be measured, calculated from Mechel's formula, or derived from transfer matrices. In its most general form, the transfer admittance is expressed as

$$\begin{bmatrix} \upsilon_{n1} \\ \upsilon_{n2} \end{bmatrix} = \begin{bmatrix} \alpha_1 & \alpha_2 \\ \alpha_4 & \alpha_5 \end{bmatrix} \begin{bmatrix} p_1 \\ p_2 \end{bmatrix} + \begin{bmatrix} \alpha_3 \\ \alpha_6 \end{bmatrix}$$

Equation 3-15.

where:

- ν_{n1} is the normal velocity on the nodes of the face selection.
- ν_{n2} is the normal velocity on the nodes of the second face selection.
- p_1 is the pressure on the nodes of the first face selection.
- p_2 is the pressure on the nodes of the second face selection.
- α_1, α_2, α_4, and α_5 are complex admittance coefficients.
- α_3 and α_6 are complex source coefficients.

The six coefficients α_1 through α_6 are determined by the nature of the relation. Because these coefficients have the dimension [velocity/pressure], they are called *transfer admittance coefficients*.

> **Note**
>
> The matrix element values depend on the structure between the nodes defined at two sides of the surface. For example, in a physical problem, the two sides represent two sides of a wall that contains perforations. Instead of modeling the perforations by using many small elements, you model only the volumes on both sides of the wall. Then, you capture the effect of the perforations, which causes the acoustic results between both sides of the wall to be coupled, through the transfer admittance matrix.
>
> In this case, the nature of the relation encompasses the number of holes and their porosity, the viscosity of the fluid in the holes, and so on. These are the parameters in Mechel's formula used to derive the transfer admittance values.

You use the ACTRAD bulk entry that references PACTRAD to specify the transfer admittance.

3.4.5 Duct modes boundary condition

You can apply a rectangular, cylindrical or annular duct modes boundary at the inlet of a duct modeled with fluid elements. Duct modes are supported in an uncoupled acoustics solution running Simcenter Nastran Direct Frequency Response (SOL108) with FEMAO.

A duct mode definition is similar to an acoustic load since it results in a propagating acoustic wave. In duct acoustics, duct modes are important in applications such as HVAC and automotive exhaust systems. Generally, applications in which exhaust noise reduction is important.

In long ducts, each acoustic mode can be expressed as the product of standing waves with a propagating component. A specific duct mode is characterized by the mode orders and the amplitude.

The general solution for a duct mode progagating in a duct with an infinite length (z-direction) is expressed as:

$$S(x, y, z, \omega) = \sum_{m}^{\infty} \sum_{n}^{\infty} A_{m,n}(\omega) \cos\left(\frac{n\pi}{L_x x}\right) \cos\left(\frac{m\pi}{L_y y}\right) e^{\pm i\, k_{m,n} z}$$

where,

$$\cos\left(\frac{n\pi}{L_x x}\right) \cos\left(\frac{m\pi}{L_y y}\right)$$

describes the standing wave in x and y,

$$e^{\pm i\, k_{m,n} z}$$

describes the propagating wave on the duct axis (z-direction),

$$A_{m,n}$$

is the amplitude,

$$m, n$$

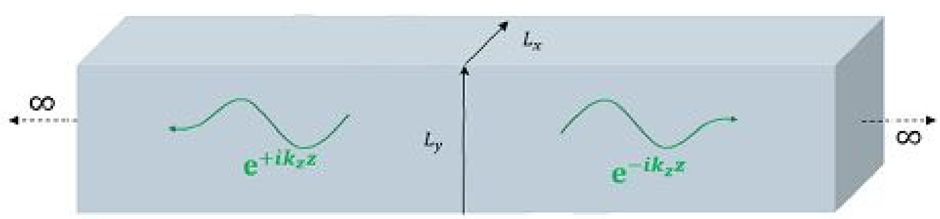
are the mode orders relative to the two directions in the duct cross section,

$$L_x, L_y$$

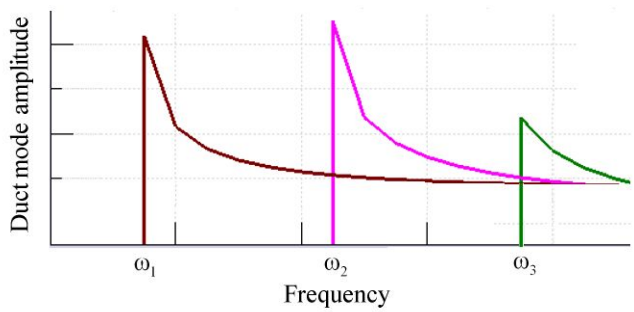
are dimensions of the duct cross section, and

$$k_{m,n}$$

is the wave number.



Each duct mode has a characteristic cut-on frequency. As a result, specific duct modes propagate only from a certain frequency onwards. This is demonstrated in the amplitude response plot below where ω_1, ω_2, and ω_3 correspond to the cut-on frequencies of three duct modes, each with their own mode orders (m,n).



The software considers the boundaries you apply duct modes on as reflection-less. That is, wave components that are reflected back to the duct mode boundary are completely absorbed.

There are two options to define duct modes on an inlet boundary:

- You can define a specific duct mode with given phase and amplitude.

  The specific duct mode option is useful when you understand the amplitude and the frequency range associated with the incoming noise. For example, you could know the amplitude of the incident modes from previous experiments. You can define specific duct modes that have a corresponding cut-on frequency within this range.

- You can request the distributed option.

  The distributed duct mode option is useful when your noise source encompasses a broadband of frequencies or you do not have experimental data for the incoming noise source.

  With the distributed option, instead of defining a specific mode, you define the acoustic amplitude which is applied to all modes. For each solution frequency, the software applies all possible cut-on modes as incoherent sources with equal acoustic power. The software computes the response of each mode separately, then sums them to compute the total response.

Multiple duct modes boundary conditions can be defined in the same model, but only one definition can exist at a specific fluid boundary. For example, you could model a system of ducts which includes multiple inlets. A unique duct mode boundary condition could be defined on each inlet.

Duct outlet boundary

To allow the acoustic waves to exit a duct system without reflections, you have two options.

- You can define the Anechoic End Duct (AED) boundary on the outlet. The AED is a reflectionless boundary on a duct outlet. You can use this option when you are only interested in results within the duct system, and not exterior acoustic radiation.

- You can define the Automatically Matched Layer (AML) at the outlet. When the AML is defined, the acoustic energy at the outlet can radiate to exterior microphone locations. You can use this option when you are interested in results within the duct system, and exterior acoustic radiation.

Inputs

- You model a system of ducts with a 3D fluid mesh. The duct inlet and outlet locations in which you plan to define either a duct mode or an anechoic end duct should be modeled rectangular, cylindrical or annular.

- The PACDUCT bulk entry defines the cross sectional properties for the duct mode and the anechoic end duct (AED) boundaries.

- You enter the ID of a BSURFS bulk entry in the BID field. The BSURFS entry selects the fluid element faces where the duct section boundary is applied.
- You specify that your cross section is a circular, annular, or rectangular with the GTYPE field, and enter the dimension of your cross section with the DIMi fields.
- Relative to the coordinate system you select in the CID field, you define the cross section origin with the XLOC,YLOC,ZDOC fields, and the duct axis direction with the XVEC,YVEC,ZVEC fields.
- You can optionally offset the location of the axial position of the duct origin using the OFFSET field.

• You define a duct mode with the ACDUCT bulk entry.

  - You reference the ID of a PACDUCT entry with the PID field.
  - You specify that your boundary is either a specific duct mode, or distributed duct modes with the WTYPE field.
  - You define the units of the duct mode amplitude as pressure, intensity, or power using the MTYPE field.
  - When you are defining a specific duct mode, you will define the mode number pair (m,n) using the MODX1, MODY1 fields.
  - For both the specific duct mode and the distributed duct modes, you will define the amplitude as a complex data pair. You designate the form of this data pair, (real and imaginary) or (magnitude and phase), with the FORM field.

• You reference the ID of your duct mode defined with the ACDUCT bulk entry from the case control with the ALOAD case control command.

• You define your solution frequencies with the FREQi bulk entries. You select these solution frequencies from the case control with the FREQUENCY case control command.

• You define an anechoic end duct (AED) boundary with the ACNDUCT bulk entry. You reference the ID of a PACDUCT entry with the PID field.

• The DUCTFMAX parameter is available to define the maximum duct mode frequency using the product (DUCTFMAX * maximum excitation frequency). This value determines the truncation of the duct modes wavebasis. It is typically defined slightly over the maximum frequency of interest. The DUCTFMAX default is 1.2.

Duct mode output

Two output options are available when a duct modes boundary condition is defined on a duct inlet, and either an anechoic end duct boundary or an AML is defined on a duct outlet.

• The DMTRLOSS case control command is available to request the duct modes transmission loss output.

Transmission loss is computed as the power introduced by the duct mode boundary conditions divided by the sum of the modal transmitted power coefficients at the anechoic end duct boundary or the power at the AML radiation surface. Mathematically it is written as:

$$\text{Transmission Loss} = \frac{\text{Power In}}{\text{Power Out}}$$

- The DMTRCOEF case control command is available to request the duct modes transmission coefficients.

  Transmission Coefficients represent the amplitude of the output modes expressed in terms of pressure, intensity, or power.

Input file example:

```
...
$Case Control
...
***Solution frequency selection
FREQUENCY = 100

***Duct tranmission loss output request
DMTRLOSS(PRINT) = YES

SUBCASE 1
***ALOAD command selects the duct mode with a set ID of 2.
ALOAD = 2

BEGIN BULK
***These parameters request FEMAO
ACADAPT STANDARD
ACORDER  MINIMUM       1
ACORDER  MAXIMUM       1

***Solution frequency definition
FREQ1        100100.00005.000000     180

***Anechoic end duct definition
ACNDUCT        1       2

***Anechoic end duct cross sectional properties
PACDUCT        2       2       CIRCULAR 28.5956              1       0+
+        5171.33603.8923 1003.400.4762721.7123-40.879298

***Description referenced by the anechoic end duct
DESC           1Duct Outlet

***Fluid element face selection for anechoic end duct boundary
BSURFS         2                       332342   91799   88669   91798+
+        332418   88688   91799   91800  332464   91797   88671   91796+
+        332623   91796   91795   91814  337443   91795   88671   88672+
+        337444   91801   91821   91822  337797   88687   88688   91800

***Duct mode definition
ACDUCT         2       4SPECIFIC   PRESS    REAL                        +
```

```
+              0       1 1000.00  0.0000

***Duct mode cross sectional properties
PACDUCT        4       4  0.0000CIRCULAR 25.3987               2       0+
+        1823.95-167.270 1357.170.9493285.4486-50.314286

***Description referenced by the duct mode
DESC           2Duct Inlet

***Fluid element face selection for duct mode boundary
BSURFS         4                        155804   98888   98889   98906+
+         156115  103130   98882   98884 157499  103364   98885   98883+
+         157502  102356   98884   98901 157503  102401   98884  102356+
+         157661   98883   98902   98882 158279   98887   98888   98906+
+         158293   98903   98883   98885 158894   98886  103409   98887+
+         158897   98902   98883   98903 159463   98904   98906   98905
...
```

3.4.6 Coupled FEMAO

You can define coupling of a structure to an acoustic fluid in a vibro-acoustic FEMAO solution.

You can couple a structure to a fluid in a FEMAO (FEM Adaptive Order) solution. A FEMAO solution is a higher-order polynomial method for acoustic and vibro-acoustic analyses. It provides more accurate results and faster solution times by adapting the computational effort to the complexity of the analysis. By using a FEMAO solution, the fluid elements can be relatively large, yet result in accurate results at high frequencies.

In a vibro-acoustic simulation, the structure is expressed in modal DOFs and the acoustic fluid in physical DOFs.

For FEMAO, SOL 108 Direct Frequency Response has been enhanced, and SOL 111 Modal Frequency Response is a new solution. The enhanced functionality can be used in Simcenter Nastran FEM vibro-acoustics solutions, such as:

- An engine in a vehicle engine compartment radiating sound both inside the engine bay and outside the vehicle.

  This type of solution could be used to model an automotive engine in a vehicle, surrounded by an acoustic fluid which incorporates the geometric features of other structures, such as the body-in-white and the passenger acoustic cavity. You can excite the engine, which radiates sound into the fluid, and capture the sound response both inside the engine bay, and outside of the vehicle where it contributes to pass-by noise.

- A panel transmission loss simulation, using modal coordinates for the structure and fluid meshes with AML (AMLREG) in physical coordinates to represent reverberant and anechoic side on the front and back of the panel.

Fluid-structure interaction

You can now couple structural FEMs to acoustic fluids as component FEMs in an assembly FEM, and solve the solution using FEMAO.

Structural FEMs:

- Can be coupled to acoustic fluids as component FEMs in an assembly FEM and solved using FEMAO.
- Can now be solved with fluids for acoustics and vibro-acoustics in both FEM and FEMAO solutions SOL 108 and SOL 111.

SOL 111 FEMAO now supports:

- Two-way (strong) fluid-structure coupling.
- One-way (weak) fluid-structure coupling with only force excitation of the structure

You define the minimum and maximum order for the elements in the entire model.

Examples:

ACADAPT FINE

ACORDER MAXIMUM 10

ACORDER MINIMUM 2

You set the FEMAO options in the Bulk Data.

You define the adaption rule with the ACADAPT bulk entry.

You define the element order with the ACORDER bulk entry.

3.4.7 Impedance or admittance

The acoustic absorber element CAABSF defines frequency-dependent impedance or admittance boundary conditions. It allows POINT, LINE, or AREA impedance to be specified on the free fluid surface. The element topology is specified by CAABSF, and the frequency-dependent impedance values are specified on the PAABSF entry.

Simcenter Nastran also supports constant real and imaginary impedance and admittance on a 2D fluid free surface when CAABSF references the physical property PAABSF1.

If only one grid point is specified in CAABSF element, then the impedance

$$Z(f) = Z_R(f) + iZ_i(f)$$

is the total impedance at the point. If two grids are specified, then the impedance is per unit length. If three or four points are specified, then the impedance is per unit area or the specific impedance. The PAABSF bulk entry allows you to define impedance as follows:

$$Z(f) = Z_R(f) + iZ_i(f); \quad Z_R(f) = TZREID(f) + B; \; Z_i(f) = TZRIMD(f) - \frac{K}{2\pi f}$$

The resistance represents a damper quantity B that is frequency dependent. The reactance represents a quantity of the type (ωM − K/ω). The impedance is defined by equation

$$p = \dot{u}_f \cdot n_f Z, \qquad \text{or} \qquad \frac{\dot{u}_f \cdot n_f}{p} = A$$

For more information, see Imposed impedance or admittance (Robin boundary condition).

The scale factor S is used in computing element stiffness and damping terms as:

$$[B] = \frac{1}{Z} \int_{S_2} N^T N \, dS = \frac{Z_R - iZ_i}{Z_R^2 + Z_i^2} \int_{S_2} N^T N \, dS$$

Equation 3-16.

The imaginary part of $[B]$ can then be construed as frequency-dependent stiffness matrix. Therefore,

$$[B] = \frac{Z_R}{Z_R^2 + Z_i^2} \int_{S_2} N^T N \, dS$$

$$[K] = \frac{i\omega Z_i}{Z_R^2 + Z_i^2} \int_{S_2} N^T N \, dS$$

Equation 3-17.

The absorption coefficient is then

$$\alpha = \frac{\dfrac{4Z_R}{\rho c}}{\left(1 + \dfrac{Z_R}{\rho c}\right)^2 + \left(\dfrac{Z_I}{\rho c}\right)^2}$$

Equation 3-18.

Note

- Parabolic element faces are also supported because only corner grids are used to define the element faces.
- Point and line impedance are not supported in the acoustic method introduced in NX Nastran 11, which includes AML, porous materials, and microphone elements.
- Point and line impedance are only supported through CAABSF that references PAABSF if the system cell 617 (ACFORM) is set to 0. ACFORM = 0 reverts to the NX Nastran10 acoustics behavior.

- Although PAABSF is not supported by the FEM Adaptive Order (FEMAO) method, PAABSF1 is supported by both the standard FEM and the FEM Adaptive Order (FEMAO) method.

3.4.8 Defining infinite boundary

See Open or radiating boundary for details on infinite boundary condition.

3.5 Defining acoustic loads

3.5.1 Defining acoustic loads

The following types of loading are available in Simcenter Nastran for fluid elements:

- Constant, frequency-dependent, or time-dependent enforced pressure at the grid points.
- An acoustic source characterized by a volumetric flow rate and corresponding to a power spectral density function.

An acoustic source is assumed to be a pulsating sphere in infinite space and is defined on the ACSRCE bulk entry. The ACSRCE entry is selected by the DLOAD Case Control command and contains the material properties of the source and references a DAREA and TABLEDi entry. The TABLEDi entry defines the power-versus-frequency curve characterizing the acoustic source. The ACSRCE entry may also define a delay time and phase angle which is useful whenever multiple sources are present.

In addition to the above acoustic loads, you can also apply any typical dynamic loads to the structural portion of the model.

3.5.2 Enforced vibration loading at fluid-structure interface

When surface vibrations (displacements, velocities, or accelerations) on the structure are pre-defined, the structural degrees of freedom are mainly used to compute the fluid loading using the coupling matrix [A].

This reduces equation

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & 0 \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

(see One-way coupling) as follows:

$$\varphi = -p n_s$$

Equation 3-19.

$$M_f \ddot{p} + B_f \dot{p} + K_f p = P_f + A^T \ddot{u}_s$$

Equation 3-20.

The SPCSTR parameter can be used to instruct Simcenter Nastran to constrain the structural degrees of freedom in the analysis set (after transferring the enforced vibration to the fluid).

The parameter SPCSTR (see *Parameters and Parameter Applicability Tables* chapter in the *Quick Reference Guide*) is used for fluid-structure analysis only.

3.5.3 Enforced motion loading from file

You can read frequency-dependent enforced motion loads (displacements, velocities, or accelerations that can be translational, rotational, or both) from an OUTPUT2 (*.op2) file or external SC_H5 (*.sc_h5) file in HDF5 format for a frequency response analysis. The SC_H5 file contains pre-computed motion loads that may originate from an Abaqus or ANSYS file. This allows you to more efficiently solve vibro-acoustic problems.

Typically, enforced motion loading is used when motion (displacement, velocity, or acceleration) is specified instead of, or in conjunction, with applied loads.

This capability is supported by SOL 108 and SOL 111 solutions that use the standard (fixed low-order) FEM method or the FEM Adaptive Order (FEMAO) method, which is a higher-order polynomial method.

Workflow for load extraction from SC_H5 file and enforced motion load application:

1. Use the ASSIGN file management statement to select the SC_H5 file that contains the motion results from a previous frequency response analysis.

2. Use the SPC case control command to reference SPC1 bulk entries that constrain nodes in all degrees of freedom (DOFs).

3. Use the DLOAD case control command to combine dynamic loads defined with RLOAD1 bulk entries.

   > **Note**
   >
   > This command is even required for one dynamic load.

4. Use the SPCF bulk entry to specify the following:

   - Identification number of the SPCF bulk entry.

   - Identification number of a SET1 bulk entry that contains a list of structural grid points at which the loads are to be extracted and applied to for all DOFs of grids.

     > **Note**
     >
     > When SET1ID is blank, the software distributes the motion results to all structural grids contained in the SC_H5 file and applies enforced motion loads.

- Subcase number of motion results in the SC_H5 file.

> **Note**
>
> When the SC_H5 file only contains one subcase, you can leave the SUBC field blank.

- Logical unit number that was assigned by the ASSIGN statement to the SC_H5 file.
- File type to indicate that the type is SC_H5.
- Load descriptor of a loading condition contained in the SC_H5 file.

5. Use the DTI,DISTL bulk entry to reference the loading description contained in the SC_H5 file.
6. Use the RLOAD1 bulk entry to reference the SPCF bulk entry.

For more information, see the ASSIGN case control command, the SPCF, DTI,DISTL, and RLOAD1 bulk entries.

Input file example:

```
...
$File Management
ASSIGN SC_H5='filename of HDF5 file .sc_h5 extension typical',
UNIT=201, DEFER

$Executive Control
...
SOL 108

$Case Control
...
$***Solution frequency selection
FREQUENCY = 100
$***Displacement output request
DISPLACEMENT(PLOT,REAL) = ALL

$***Subcases
SUBCASE 1
   DLOAD = 301
...SPC = 1

BEGIN BULK
...
$***Definition of discrete excitation frequencies
FREQ2        100 10.0000500.0000       5
$***Constraint definition
SPC1           1  123456       5
$***Combination of dynamic loads
DLOAD        301     1.0     1.0       2
$***Enforced node definition
SET1           4       5
$***Enforced displacement loading from external file
SPCF           3       4       1     201       1       6
```

```
$***Description contained in the SC_H5 file
DTI        DISTL       6       6DISPDATA  ENDREC

$***Dynamic loading definition
RLOAD1         2       3                                       1

...
```

3.5.4 Enforced acoustic pressure with complex data input

When acoustic pressures on certain grids are known (for example, the pressures are identified through measurement or previous calculation), you can apply them using the enforced acoustic pressure card ACPRESS. SOL 108 and SOL 111 acoustic and vibro-acoustic solutions support ACPRESS.

ACPRESS supports complex data and the definition of:

- Magnitude and phase of pressure on multiple grids.
- Constant and frequency dependency.

> **Note**
>
> Simcenter Nastran adds the degrees of freedom referenced by ACPRESS to the s-set. For detailed information on the s-set, see the *Understanding Sets and Matrix Operations* chapter in the *User's Guide*.

3.5.5 Enforced force, pressure, or moment loading from file

You can read frequency-dependent dynamic forces, pressures, or moments for a frequency response analysis from an external file that is in the open-source HDF5 format and commonly has a *.sc_h5 extension. This allows you to more efficiently solve vibro-acoustic problems. Typically, these loads are spatially and time-varying computational fluid dynamics, acoustic, force loads, or stochastic pressure loads that were transformed to Simcenter Nastran loads and stored in the external file by pre-processing/post-processing software, such as Simcenter 3D Pre/Post.

For example, you may want to transform deterministic aerodynamic forces from Simcenter Star-CCM+ that cause an automotive side window to vibrate and use these loads in a vibro-acoustic solution to examine the acoustic response at a driver's ear to the structural excitation.

This capability is supported by SOL 108 and SOL 111 solutions with standard (fixed low-order) FEM or FEM Adaptive Order (FEMAO) method, which is a higher-order polynomial method.

Force loading from file

This workflow shows a random response run. The workflows for enforced pressure or moments are similar.

1. Use the ASSIGN file management statement to select the .sc_h5 file that contains the Simcenter Nastran loads.

2. Use the RLOADEX bulk entry to define frequency-dependent dynamic forces obtained from a file of the form:

$$\{P(f)\} = \{A\,[F(f)]e^{i\{\theta-2\pi f\tau}\}$$

3. Use the RANDPEX bulk entry to reference a PSD specification that is a simple PSD across all degrees of freedom for use in a random analysis.

4. Use the DLOAD case control command to apply the dynamic forces to the response problem.

5. Use the DLOAD bulk entry to define a dynamic loading condition for the RLOADEX bulk entry.

6. Use the DTI,DISTL bulk entry for the load description of the RLOADEX and RANDPEX bulk entries

7. Use the RANDOM case control command to select the identification number of a RANDPEX bulk entry.

8. Depending on the organization of your subcases for a random analysis, you may need to include ANALSIS = RANDOM in the subcases that contain the RANDOM case control command.

For more information, see the ASSIGN case control command, the RANDPEX, RLOADEX, and DTI,DISTL bulk entries.

For more information on ANALYSIS = RANDOM, see the RANDOM case control command.

3.5.6 Fan noise boundary condition

You can define a fan noise inside an acoustic domain. These aeroacoustic simulations for subsonic fluid excitations, such as from a rotating fan or propeller, are solved using a FEM Adaptive Order (FEMAO) acoustic solution. The method can be used to solve a variety of aeroacoustic simulations, such as marine propeller sound propagation, cooling fan noise, or HVAC blower and duct acoustic interaction.

A technique has been implemented to solve for the acoustic response using aeroacoustic excitations. Previously solved CFD simulation surface pressures as a function of time are imported from a CGNS (CFD General Notation System) file, converted to a blade force excitation, and are then applied as a fan blade force using equivalent rotating acoustic forces.

You can read transient or frequency-dependent dynamic forces from spatially and time varying computational fluid dynamics, acoustic, force loads, or stochastic pressure loads that were transformed to Simcenter Nastran loads and stored in the external *.sc_h5 file by a pre/post software, such as Simcenter 3D Pre/Post.

You create a SOL108 FEMAO Acoustics solution and then include single or multiple fan noise sources in your simulation. The rotation speed (RPM) for all fans must be identical.

A fan noise can have a single blade, for which all blade forces will be assumed to be periodically identical. If the fan noise has multiple blades, then blade forces can be unique for every blade. Fan noises must be located on or inside the acoustic fluid boundary.

Fan excitation can be:

- Tonal Noise, which only includes blade passing frequencies and noise.

> **Note**
>
> If the imported CFD fan pressures are defined for only a single blade, you can only apply Tonal Noise.

- If you have imported CFD fan pressures with varying forces on several blades, then you can apply Tonal and Broadband noise, which includes blade passing frequencies and broadband turbulent noise.

You use a fan noise load to excite the acoustic fluid around your structure.

In the solution Case Control, you specify the forcing frequencies as harmonics and sub-harmonics to set the solution frequencies. Harmonics include only the fan. Subharmonics are available for both the fan and shaft.

You create a fan noise load with the ACFAN bulk entry.

You define the fan noise properties with the PACFAN bulk entry.

You define fan noise harmonics with the FREQH bulk entry

Input file example:

```
...
$* FILE MANAGEMENT
ASSIGN SC_H5='D:\WorkDir\Project\nx13_LMS1306 Fan Noise\generated_fan_s,
egments.sc_h5' UNIT=201
ASSIGN SC_H5='D:\WorkDir\Project\nx13_LMS1306 Fan Noise\generated_fan_s,
egments.sc_h5' UNIT=202
SOL 108
CEND
$* CASE CONTROL
SUBCASE 1
  LABEL = Subcase - Direct Frequency 1
  ALOAD = 1
  FREQUENCY = 101
SUBCASE 2
  LABEL = Subcase - Direct Frequency 2
  ALOAD = 2
  FREQUENCY = 102
$* BULK DATA
BEGIN BULK

$REQH         SID      TYPE   HTYPE      PID   START
FREQH         101 SINGLE   SHAFT    101            1
$ACFAN        PID    TYPE       RPM     NBLADES
PACFAN        101 TONAL 2000.00                  1

$REQH         SID     TYPE   HTYPE      PID   START      END     STEP     NSUB
FREQH         102   LSUB   BLADE    102          10         50          4            2
$ACFAN        PID   TYPE        RPM   NBLADES
PACFAN        102 TONAL 2166.67                1


$*  Load: Fan Noise(1)
$CFAN        SID       PID     LUNF     LDID     LCID   WINDOW
```

```
ACFAN          1       101      201             1          1  HANNING
$TI          NAME     LDID     LDID      DESCRIPT    "ENDREC"
DTI           DISTL              1           1   Fan1/Forces  ENDREC

$*  Load: Fan Noise(2)
$CFAN        SID       PID     LUNF     LDID     LCID  WINDOW
ACFAN           2       102      202            2           2  RECTANG
$TI          NAME    LDID     LDID       DESCRIPT    "ENDREC"
DTI          DISTL            2            2    Fan1/Forces  ENDREC
```

3.5.7 Acoustic source loading

An acoustic source is assumed to be a sphere that oscillates in an infinite acoustic field. The strength of the source is characterized by a frequency-dependent flux of volume velocity defined by (surface area times velocity)

$$\dot{Q}_s e^{i\omega t} = \int_S \dot{u}_f^T \boldsymbol{r}\, dS$$

Equation 3-21.

where:

- $\dot{Q}_s$ is the strength of the acoustic source.
- *r* is the unit radial vector.

The power $\mathcal{P}$ radiated from the source is related to its strength by:

$$\mathcal{P} = \frac{2\pi}{\rho_0 c_0}\left|\hat{S}^2\right|; \qquad \dot{Q}_S = \frac{4\pi\hat{S}}{i\omega\rho_0}$$

Equation 3-22.

where:

- ρ_0 is the density of the fluid.
- c_0 is the speed of sound in the ambient medium at a given temperature.
- $\hat{S}$ is the monopole amplitude.

Equation

$$[M_s]\{\ddot{u}_s\}+[B_s]\{\dot{u}_s\}+[K_s]\{u_s\}=\{P_s\}-[A]\{p\}$$

(see Coupling of fluid and structure (vibro-acoustics)) can be re-arranged to yield

$$P_f = \dot{Q}_s = \frac{1}{2\pi f}\left(\frac{8\pi c_0 \mathcal{P}}{\rho_0}\right)^2$$

Equation 3-23.

The acoustic source strength can be specified explicitly as a dynamic nodal load (see $\{P_f\} = \{\dot{Q}_s\}$ in Finite element formulation) or can be computed given the source radiated power (using equation

$$[A] = \int_{S_2} N_s^T N_f \{n_s\} dS \quad \text{or} \quad [A] = -\int_{S_2} N_s^T N_f \{n_f\} dS$$

in Coupling of fluid and structure (vibro-acoustics)).

Acoustic source can be defined in terms of:

- Monopole amplitude.
- Source strength amplitude.
- Acoustic power.

If the source strength amplitude is known, this data can be specified directly as a dynamic load using the RLOADi bulk entry. If the acoustic power is known then this data can be specified using the ACSRCE bulk entry. If only the monopole amplitude is known, then either the acoustic power or the source strength amplitude can be computed and in turn either the ACSRCE or RLOADi bulk entry can then be used to specify the acoustic source information.

3.5.8 Monopole, dipole, plane wave, and surface dipole acoustic sources

Simcenter Nastran supports dedicated acoustic monopole, dipole, and plane wave sources in SOL 108 and SOL 111.

Acoustic monopole source

An acoustic monopole is a pulsating sound source that radiates equally in all directions.

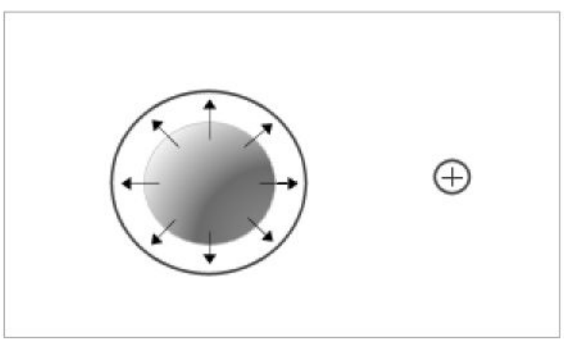


Figure 3-2. Acoustic monopole source

> **Note**
>
> The plus sign means that the monopole source expands.

In Simcenter Nastran FEM acoustics, you can create acoustic monopoles:

- Inside the meshed fluid volume at a fluid grid point or inside an element.
- Outside the meshed fluid volume.

The monopole source generates an incident sound field at a location of a distance *R* due given by

$$p_{inc} = \hat{S}\frac{e^{-ikR}}{R}$$

Equation 3-24.

where:

- $\hat{S}$ is the monopole amplitude.
- *R* is the distance from the source.
- *k* is the wave number.

The monopole source can also be specified by its volume velocity Q_S. The relationship between the monopole amplitude $\hat{S}$ and volume velocity is given by

$$Q_s = \frac{4\pi\hat{S}}{i\omega\rho}$$

Equation 3-25.

Another method you can use to specify the monopole source is to use its acoustic power.

$$W = \frac{\rho c k^2 Q_s^2}{8\pi}$$

Equation 3-26.

where:

- ρ is the fluid density.
- *c* is the speed of sound.

You use the bulk entry ACPOLE1 to specify a monopole source.

- The bulk entry supports power and monopole amplitude inputs that can be constant or frequency dependent.

- In ACPOLE1, the source location is defined by the coordinate that can be inside or outside of the meshed fluid volume. For sources outside the FEM mesh, you specify an AMLREG on all or part of the free faces of the fluid model. This captures the effect of the monopole incident field inside the fluid domain.

- ACPOLE1 inherits the density and speed of sound from the location of the monopole source. If the source is outside the meshed fluid volume, the fluid properties are acquired by averaging the properties used in those fluid elements that their free faces are used in the definition of the AMLREG.

Acoustic dipole source

The dipole source can be visualized as an oscillating sphere with no deformation as shown in the image below. The fluid near the source moves back and forth to produce sound. Unlike a monopole source, the sound does not radiate equally in all directions.

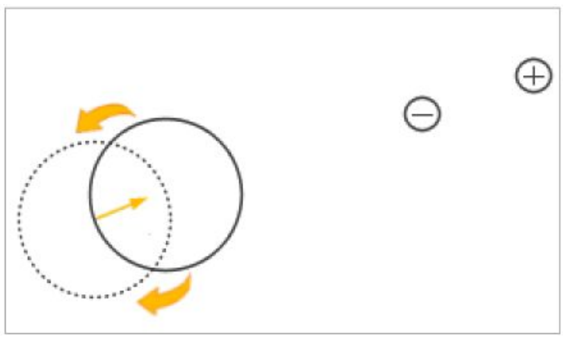


Figure 3-3. Acoustic dipole source

Also, the dipole source can be visualized as two out-of-phase monopole sources separated by a distance, where one monopole contracts (- sign) and the other one (+ sign) expands. In this case, the dipole generates a sound field that can be written as

$$\left|p(r,\theta,t\right| = \frac{Q\rho ck}{4\pi r} kd \cos\theta$$

Equation 3-27.

(1) Evaluation point

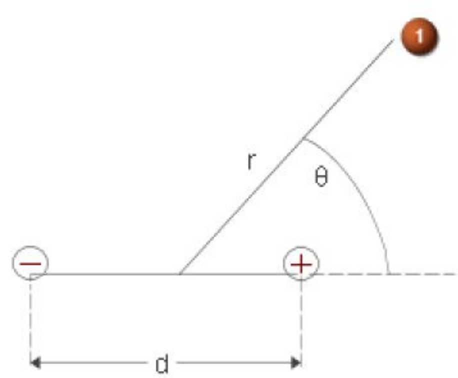


Figure 3-4. Dipole sound field

where:

- *d* is the distance between two out of phase monopole sources.
- *r* is the line that connects the midpoint between the monopole sources and the evaluation point (1).
- θ is the angle between the line that connects those monopole sources and the line that connects the evaluation point (1) and the midpoint between the monopole sources.

To specify the dipole source in your simulation, you define the dipole moment *Sd*

where:

- *S* is the source strength.
- *d* is the direction vector.

You use the ACPOLE2 bulk entry to specify a dipole source. The bulk entry supports:

- The definition of the dipole moment in a coordinate system.
- The source location inside or outside the meshed fluid volume. If the source is outside the meshed fluid volume, you must define an AMLREG bulk entry to account correctly for the incident field from the dipole.

Acoustic plane wave source

A plane wave source generates a plane wave on only one side of the space, in the positive direction of the source vector.

(1) Incident field

(2) Vector

(3) Position

(4) Non-incident field

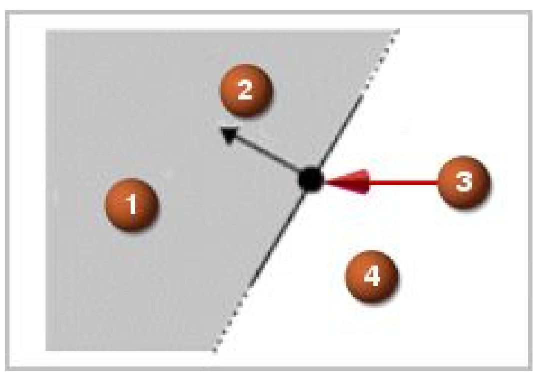


Figure 3-5. Acoustic plane wave source

The incident acoustic pressure p_i due to a plane wave source is

$$p_i = Ae^{-ikd}$$

Equation 3-28.

where:

- *A* is the amplitude of the plane wave.
- *k* is the wave number.

- *d* is the perpendicular distance from the plane source.

> **Note**
>
> - Simcenter Nastran uses surface integration to compute acoustic power. For a monopole source, you can create a spherical microphone mesh that encloses the source, and the software computes the acoustic power on this sphere. This also means that you can define acoustic power for a monopole source. However, for plane waves, you cannot define acoustic power because the plane waves are infinitely large in terms of direction, and the surface integration would result in infinite numbers.
>
> - You can use a preprocessor, such as Simcenter 3D Pre/Post, to create a collection of discrete plane wave sources equally distributed in space that model an acoustic diffuse field. This collection is usually used for a random vibro-acoustic analysis.

You use the bulk entry ACPLNW to specify a plane wave acoustic source and define location, direction, and amplitude that can be constant or frequency dependent.

If the monopole, dipole, and plane wave sources are outside of the meshed fluid volume, you must define an AMLREG bulk entry to correctly account for the incident field of these sources inside the meshed fluid volume. Also, in this case, an incident-scattered formulation is used in Simcenter Nastran FEM acoustics.

Surface dipole source

For aeroacoustic analyses, you can create an equivalent surface dipole acoustic source as a boundary condition. A surface dipole is an acoustic source that you use to model the sound generated by rigid surfaces located in a low-speed flow field application. In Simcenter Nastran, surface dipoles are based on the Neumann formulation, which assumes that the mass density and speed of sound around the rigid surfaces are uniform.

To model the sound generated by the interaction of a compressible or incompressible flow with a rigid surface, define the surface dipole boundary condition with fluid pressures.

The aerodynamic or hydrodynamic fluid pressures are typically computed by computational fluid dynamics (CFD) software, such as Simcenter Star-CCM+, and processed by a pre/post software for later use by Simcenter Nastran. A pre/post software may read, map, Fourier transform, time-segment, and write the dynamic fluid pressures to an sc_h5 file (*.sc_h5) in HDF5 format.

For example, you may want to examine sound generated by the interaction of low-speed flow with surrounding surfaces, such as HVAC duct walls, or an automotive side mirror. In contrast to solving the entire acoustics problem with volumetric flow information and generating an equivalent point source for each CFD cell, you use a surface dipole source on, for example, a side mirror surface as a boundary condition in your analysis. This reduces the data transfer from the CFD software to the pre/post software, and consequently the computational effort by Simcenter Nastran.

This capability is supported in the following solutions and requires the FEM Adaptive Order (FEMAO) method:

- SOL 108 for uncoupled acoustic and coupled vibro-acoustic analyses.

- SOL 111 for coupled vibro-acoustic analyses.

> **Note**
>
> The coupling can be weak or strong.

Surface dipole workflow:

1. Use the ASSIGN file management statement to select the .sc_h5 file that contains the processed dynamic fluid pressures.
2. Use the ACSPO2 bulk entry to define a surface dipole for the frequency response analysis.
3. Use the PACSPO2 bulk entry to define the parameters of the surface dipole for the frequency response analysis.
4. Use the ALOAD case control command to apply the dynamic loads to the response problem.
5. Use the ALOAD bulk entry to define a dynamic loading condition for the ACSPO2 bulk entry.
6. Use the DTI,DISTL bulk entry for the load description of the ACSPO2 bulk entry.

For more information, see the ACSPO2 and PACSPO2 bulk entries.

3.6 Defining microphone meshes

You can request acoustic results at arbitrary locations exterior or interior to the fluid. These locations are defined with the 0D, 1D, 2D, or 3D microphone mesh. A microphone mesh is also known as a field point mesh.

When a microphone mesh is interior or on the Automatically Matched Layer (AML) boundary, Simcenter Nastran interpolates the results from the fluid grids to the microphone location. However, when the microphone mesh is exterior to the AML boundary, Simcenter Nastran uses the acoustic results at the boundary of the FE domain and a boundary integral to obtain the acoustic response.

For more information on AML and microphone meshes exterior to the AML boundary, see Acoustic analysis using automatically matched layer.

You can request pressure output with the PRESSURE case control command, acoustic intensity with the ACINTENSITY case control command, and acoustic velocity with the ACVELOCITY case control command for the fluid grids referenced by a 0D, 1D, 2D, and 3D microphone mesh. You can also request acoustic power for the fluid grids referenced by a 2D microphone mesh with the ACPOWER case control command.

Note

When you define a 2D microphone mesh for power output over multiple acoustic domains and you also use acoustic incident or scattered sources, Simcenter Nastran can only approximate the power for the 2D microphone faces crossing from one domain to another.

The following summarizes the 0D, 1D, 2D, and 3D microphone mesh definitions on fluid grid points:

- You can specify the 0D microphone mesh using the MICPNT bulk entry.
- You can specify the 1D microphone mesh using the CROD entry, which references the PMIC bulk entry for the physical property.
- You can specify the 2D microphone mesh using the CTRIA3 and CQUAD4 entries, which reference the PMIC bulk entry for the physical property.
- You can specify the 3D microphone mesh using the CTETRA, CHEXA, CPENTA, and CPYRAM entries, which reference the PMIC bulk entry for the physical property.

3.7 Using pre-computed modes from file

You can read an SC_H5 (*.sc_h5) file in binary format that contains a mode set component representation (that is, structural modes from a previous frequency analysis). You use the mode set component representation along with element and grid point ID offsets in the residual portion of a system model. Solving a reduced model in binary format takes less time and requires fewer computer resources.

A typical application is to compute structural modes with your favorite FE solver, such as ANSYS or Abaqus, convert the result file to an SC_H5 file with a pre/post software, such as Simcenter 3D Pre/Post, and use the calculated modes in a vibro-acoustic analysis to compute vibro-acoustic responses.

Workflow for modes extraction:

1. Use the ASSIGN file management statement to select the SC_H5 file that contains the structural modes results from a previous frequency response analysis.
2. Use the MDSBULK entry to specify the following:
   - Logical unit number that was assigned by the ASSIGN statement to the SC_H5 file.
   - Element and grid point ID offsets.
   - Descriptor identification number of a description contained in the SC_H5 file for translational mode shapes.
   - Descriptor identification number of a description contained in the SC_H5 file for rotational mode shapes.
3. Use the DTI,DISTL bulk entry to reference the loading description contained in the SC_H5 file.

For more information, see the ASSIGN case control command, the MDSBULK, and DTI,DISTL bulk entries.

Input file example:

```
...
$File Management
...
$***Mode Set
ASSIGN SC_H5='filename of HDF5 file .sc_h5 extension typical',
UNIT=201, DEFER

$Executive Control
...
SOL 108

$Case Control
...

BEGIN BULK
...
$***Definition of logical unit number for the SC_H5 file
$***and any element and grid point ID offsets
$*
MDSBULK              201       0       0       1

$***Description contained in the SC_H5 file
DTI        DISTL       1       1STRUCTURAL_MODESENDREC
...
```

Chapter 4: Solution methods

4.1 Overview of solution methods

Direct and modal

Acoustic and vibro-acoustic analyses can be performed using both direct and modal methods.

- Direct dynamic analysis can be performed using solution sequences 107 through 109 and solution 200 with ANALYSIS = DFREQ.

> **Note**
>
> In a direct frequency response solution (SOL 108):
>
> o You can request the global iterative solver. The solve can result in significant computational savings through preconditioner reuse. To activate the global iterative solver in SOL 108, set either `NASTRAN ITER = YES` in the Nastran input file or add `iter = yes` to the Nastran command line.
>
> o You can request the Padé via Lanczos (PvL) approach for vibro-acoustic models in which the structure and fluid are strongly coupled. The PvL approach utilizes factorization of the matrix at selected frequencies for all responses and therefore significantly reduces the computational time for problems with large number of frequencies.
>
> To request the PvL approach, set either the new keyword `krylov` or the new system cell `679` (krylov) to YES.
>
> ```
> NASTRAN KRYLOV=YES
> ```
>
> The iterative solver support for frequency-dependent solutions and the Pade via Lanczos (PvL) approach are mutually exclusive. If you request both PvL and global iterative solver solution, the PvL approach will take precedence over the frequency-dependent iterative solver approach.

- Modal analysis can be performed using solution sequences 103, 110, 111, 112, and 200 with ANALYSIS = MFREQ and ANALYSIS = MTRAN.

> **Note**
>
> In modal solutions (103, 110, 111, 112, and 200), the normal modes for the structure and the fluid are computed independently. The uncoupled modes of the fluid and structure are then used to compute modal frequency response. Thus, the separately evaluated fluid and structural modes from, for example, a SOL 103 run, can be used to perform a restart into a modal response calculation.

Modal

When you create a vibro-acoustic analysis with SOL 111, you can optionally use modal coordinates for the structure and physical coordinates for the fluid.

When you specify METHOD(STRUCTURE) instead of METHOD(FLUID) in the case control section, the solver:

1. Computes the normal modes for the structure.

2. Does not compute the modes for the fluid. The fluid is retained in the physical degrees of freedom.

You use this mixed-coordinates approach for frequency-dependent fluid. For example, the fluid contains porous material or has an Automatically Matched Layer (AML) boundary condition.

For information on AML, see Acoustic analysis using automatically matched layer.

> **Note**
>
> You can also use modal coordinates for the fluid and physical coordinates for the structure. However, you rarely use this approach.

Chapter 5: Input data requirements

5.1 Executive control commands

No extra executive control commands are necessary to perform an acoustic or a vibro-acoustic solution. Both modal and direct solutions are available for selection.

5.2 Case control commands

The standard case control commands for the modal and direct solutions can be used for both acoustics as well as vibro-acoustics. Single and multipoint constraints (SPC and MPC) can be imposed on the model and the dynamics loads referenced via the DLOAD entry.

5.2.1 Acoustic load, boundary condition, and source control

Use the case control command ALOAD to specify the enforced acoustic pressure load with:

- Complex data input (ACPRESS).
- Panel normal velocity boundary condition (ACPNVEL).
- Monopole source (ACPOLE1).
- Dipole source (ACPOLE2).
- Plane wave sources (ACPLNW).

ALOAD can optionally reference a corresponding bulk entry ALOAD.

For more information on the ALOAD bulk entry, see Acoustic loads, boundary condition, and sources.

5.2.2 Import coupling or the area matrix

The case control command A2GG can be used to import the coupling or the area matrix in the DMIG format.

5.2.3 Fluid-structure interaction control

The FLSTCNT case control command (see FLSTCNT in the *Simcenter Nastran Quick Reference Guide*) can be used to specify the following:

- Formulation to use symmetric or asymmetric, that is, equation

  $$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

  or equation

$$\begin{bmatrix} M_s & 0 \\ 0 & -M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{q} \end{Bmatrix} + \begin{bmatrix} B_s & A \\ A^T & -B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{q} \end{Bmatrix} + \begin{bmatrix} K_s & 0 \\ 0 & -K_f \end{bmatrix} \begin{Bmatrix} u_s \\ q \end{Bmatrix} = \begin{Bmatrix} P_s \\ \dfrac{P_f}{i\omega} \end{Bmatrix}$$

For more information, see Coupling of fluid and structure (vibro-acoustics).

Setting **ACSYM = YES** requests a symmetric formulation and **ACSYM = NO** requests an asymmetric formulation.

- The type of pressure output, peak or RMS. RMS output is available in Modal and Direct Frequency Response analysis only (SOLs 111, 108, and 200).

- The reference pressure to use for computing pressure in decibels. A peak reference pressure must be specified for PREFDB. Pressure is expressed in decibels as follows dB = 20 log(P/PREFDB). For example, the reference pressure for air is PREFDB = 2.0E−05 Pascals.

- Whether a coupled solution of structure and fluid needs to be performed. The default is **ASCOUP = YES**. Specifying **ASCOUP = NO** disables the coupled solution.

- If the coupling or the area matrix needs to be punched out. This is controlled by **AGGPCH** on the **FLSTCNT** case control command. If **AGGPCH = YES**, Simcenter Nastran writes the area or coupling matrix in the DMIG format to the punch file.

  You can then import and optionally scale this AGG matrix in a consecutive solution that includes the same model definition at the coupled interface. Use the A2GG case control command and the ASCOUP describer on the FLSTCNT case control command to request this import.

  See the FLSTCNT case control command in the *Simcenter Nastran Quick Reference Guide*.

- The output of the coupling information file is controlled by the SKINOUT parameter. Depending on the option used, a Simcenter Nastran punch file *.pch, coupling information output file *.dat, and/or OACCQ output data block is created. The name of the files is the base name of the deck appended with _coupling_info.pch and _coupling_info.dat. For example, if the original input file is test.dat, the debug files are test_coupling_info.pch and test_coupling_info.dat.

  The OACCQ output data block contains distance information for the coupled and uncoupled fluid and structure faces. The output is in addition to the coupling information written to the .dat and .pch files.

  You can use this information to display the coupling data as follows.

(1) Uncoupled Structural Faces (red)

(2) Coupled Structural Faces (blue)

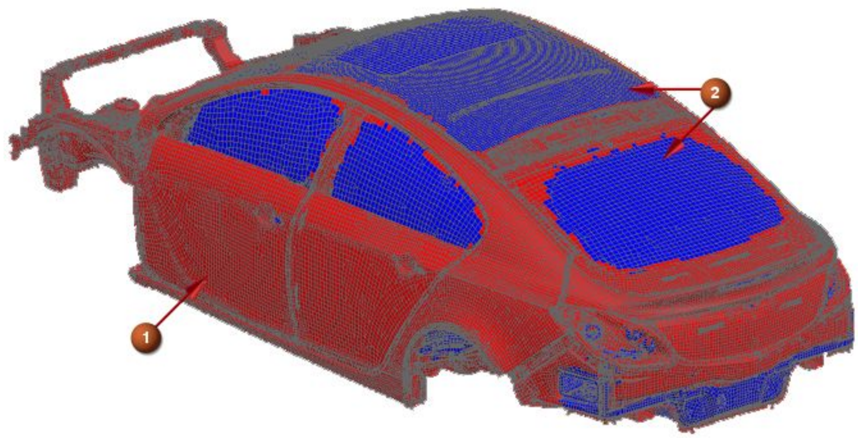


Figure 5-1. Coupled and uncoupled structural faces

For detailed information on the OACCQ data block, see the *Simcenter Nastran DMAP Programmer's Guide*.

The SKINOUT input options are:

- **SKINOUT = PUNCH**:

  A punch file, a coupling information file and an OACCQ output data block are output. The punch file (*.pch) contains a list of the original structural and fluid element IDs that participated in the coupling. The debug data deck (*.dat) contains:

  - Dummy shell elements that represent the coupled structural faces, and are assigned to a dummy pshell with ID = 1.
  - Dummy shell elements that represent the coupled fluid faces, and are assigned to a dummy pshell with ID = 2.

- **SKINOUT = FREEFACE**:

  A coupling information file and an OACCQ output data block are created (no punch file). The coupling information file contains:

  - Dummy shell elements that represent the coupled structural faces, and are assigned to a dummy pshell property with ID = 1.
  - Dummy shell elements that represent the coupled fluid faces, and are assigned to a dummy pshell property with ID = 2.
  - Dummy shell elements that represent the uncoupled structural free faces, and are assigned to a dummy pshell property with ID = 3.
  - Dummy shell elements that represent the uncoupled fluid free faces, and are assigned to a dummy pshell property with ID = 4.

- **SKINOUT = STOP**:

Works like **SKINOUT = PUNCH**, except the solution stops immediately after the debug files are created. It is recommended that you use SKINOUT,STOP in the first run, then review the coupling interface in a pre/post system before continuing with the remainder of the solution.

- o **SKINOUT = NONE** (default):

  No coupling information files are created.

- To request if the SFE Akusmod fort.70 acoustic coupling matrix needs to be read into Simcenter Nastran. When **SFEF70 = YES**, Simcenter Nastran does not compute the coupling, and instead uses the coupling definition from the external file. When **SFEF70 = NO** (default), the external coupling file is not imported. Simcenter Nastran expects the AKUSMOD file in the same directory where the job is being run, and expects the file name to be fort.70. An ASSIGN statement which uses UNIT = 70 must be defined in the file management section of your input file if the coupling file is not named fort.70 or if it is in a location other than where the job is run.

  Example

  ASSIGN OUTPUT2 = '/directory_path/user_file_name.70' UNIT = 70

5.2.4 Random acoustic analysis

For distributed acoustic plane wave problems, you can perform power spectral density (PSD) random acoustics analysis. This capability is supported for both direct and modal solutions.

To request that the software calculate the random response, in your input file include the RANDOM case control command. Then, organize the subcases as follows:

1. Create a frequency response subcase for each acoustic loading. Make sure that all of these subcases reference the same set of frequencies.

   The acoustic loads can include acoustic monopole, dipoles, or plane waves. A typical acoustic loading, is that of an acoustic diffuse field. You can represent an acoustic diffuse field with a number of subcases that each contain an acoustic plane wave source. Orient each plane wave source differently with respect to the object that sees the acoustic diffuse field. For example, the object might be a panel or a spacecraft. To define a diffuse acoustic field, you can define these sources as uncorrelated sources in a RANDOM subcase by specifying the diagonal (auto-PSD) factors in the RANDPS bulk entries only.

2. Below the frequency response subcases, include a random analysis subcase for each PSD function that you want to evaluate. Include ANALYSIS = RANDOM in these subcases.

   The software uses the frequency response functions that are calculated in the immediately preceding frequency response subcases as the inputs to the random calculations.

To specify cross-power spectral density and cross-correlation functions, use the RCROSS or RCROSSC bulk entries. When you do so, set the RTYPE to PRESS to obtain acoustic pressure output.

Example

Suppose that the fluid in a vibro-acoustic problem is excited by three plane wave loadings, and you want to evaluate the acoustic random response for two PSD functions. You can organize the subcase structure of the input file as follows:

```
SUBCASE 1
$
$ Subcase 1 calculates the frequency response function for the
$ loading specified by ALOAD 111 at the frequencies specified by
$ FREQUENCY set 13
$
FREQUENCY=13
ALOAD=111
$
SUBCASE 2
$
$ Subcase 2 calculates the frequency response function for the
$ loading specified by ALOAD 211 at the frequencies specified by
$ FREQUENCY set 13
$
FREQUENCY=13
ALOAD=211
$
SUBCASE 3
$
$ Subcase 3 calculates the frequency response function for the
$ loading specified by ALOAD 311 at the frequencies specified by
$ FREQUENCY set 13
$
FREQUENCY=13
ALOAD=311
$
SUBCASE 4
$
$ Subcase 4 uses the frequency responses from Subcases 1-3 to
$ calculate the random response of the structure for the PSD function
$ specified by RANDOM 100
$
ANALYSIS=RANDOM
RANDOM=100
$
SUBCASE 5
$
$ Subcase 5 uses the frequency responses from Subcases 1-3 to
$ calculate the random response of the structure for the PSD function
$ specified by RANDOM 200
$
ANALYSIS=RANDOM
RANDOM=200
```

5.2.5 Output requests

You can use the following case control commands to request results output from acoustic and vibro-acoustic types of frequency response analysis:

Table 5-1. Acoustic and vibro-acoustic analysis output requests			
Case control command	**Output description**	**Direct or modal solutions**	**Acoustic or vibro-acoustic analysis**
ACINTENSITY	Acoustic intensity at fluid grids referenced by microphone elements	Both	Both
ACPOWER	Acoustic power for 2D microphone surfaces and AML regions	Both	Both
ACVELOCITY	Acoustic velocity at fluid grids referenced by microphone elements	Both	Both
DMTRLOSS	Requests the duct modes transmission loss output.	Direct	Acoustic
DMTRCOEF	Requests the duct modes transmission coefficients.	Direct	Acoustic
GRDCON	Contributions of grids in structural panels to acoustic response	Both	Vibro-acoustic
INPOWER	Incident acoustic power for 2D microphone surfaces and acoustic free faces	Both	Both
MODCON	Contributions of structural modes to acoustic response	Modal	Both
PANCON	Contributions of structural panels to acoustic response	Both	Vibro-acoustic
PEAKOUT	Limits the output to the peak acoustic responses	Both	Both
PRESSURE	Acoustic pressure at fluid grids or fluid grids referenced by microphone points	Both	Both
TRLOSS	Acoustic transmission loss	Both	Both
TRPOWER	Transmitted acoustic power for 2D microphone surfaces and AML regions	Both	Both

To minimize noise radiated from a structure without performing an acoustic or vibro-acoustic analysis, you can use the ERP case control command to help identify the areas of a structure that are the primary sources of the noise due to structural vibration.

For more information on the ERP case control command, see the Equivalent Radiated Power Output section of the *Simcenter Nastran Basic Dynamic Analysis User's Guide*.

Modal, panel, and grid contribution output

You can use the MODCON, PANCON, and GRDCON case control commands to produce output that relates the acoustic response at specified locations at each excitation frequency to structural or fluid modes, structural panels, or grids in structural panels.

- Use MODCON to examine how each structural mode contributes to the acoustic response, how each fluid mode contributes to the structural response, or how with the PANELMC describer, each structural mode contributes to the acoustic response on a structural panel-by-structural panel basis.

  For more information on the MODCON case control command including applications to structural only frequency and transient response analysis, see the Frequency Response Solution Control and Output section of the *Simcenter Nastran Basic Dynamic Analysis User's Guide*.

- Use PANCON to examine how the motion of structural panels contribute to the acoustic response.
- Use GRDCON to examine how the motion of grids in structural panels contribute to the acoustic response.

Note

PANCON and GRDCON results are only valid when the excitation is from structural loads and not acoustic loads.

PANCON case control command

The PANCON case control command output quantifies how the motion of structural panels contributes to the acoustic response at a specific location in the fluid.

Structural panels are surfaces at the interface of the structure and fluid. To define structural panels, use the PANEL bulk entry.

When you use the PANCON case control command, you can specify whether the output is in SORT1 or SORT2 format and whether the output is normalized.

For SORT1 output, at each excitation frequency (as specified by FREQ) and response (as specified by SETMC grid and DOF entries) combination, the software ranks the contribution that each panel makes to the response based on the results of the following equation:

$$MAG_i\left(j\right) = \left|\frac{X_i\left(j\right)}{X\left(j\right)}\right|$$

where:

$MAG_i(j)$ = magnitude of contribution at excitation frequency j from panel i (real)

$X_i(j)$ = panel contribution to response at excitation frequency j from panel i (real or complex)

$X(j)$ = total response at excitation frequency j (real or complex)

You can use the TOPP describer to specify the number of top contributing panels that the software outputs for each excitation frequency and response combination. In the output, the panels are listed from the highest contributor to the lowest contributor.

For example, if you specify TOPP = 10, for each excitation frequency and response combination, the software outputs the 10 panels that contribute the most. If you specify TOPP = ALL, the software outputs the contribution of all the panels listed from the highest contributor to the lowest contributor. If you do not specify the TOPP describer, by default, the software outputs the 5 panels that contribute the most.

You can use the ABS, NORM, or BOTH describers to specify if the output is normalized.

For example, specify NORM to output normalized panel contributions and specify ABS to output non-normalized panel contributions. Specify BOTH to output both normalized and non-normalized panel contributions.

For SORT1 output, the software calculates the normalized panel contributions as follows:

$$NMC_i\left(j\right) = \frac{X_i\left(j\right)}{X\left(j\right)}$$

where $NMC_i(j)$ is the normalized panel contribution to the response at excitation frequency *j* from panel *i*.

For SORT2 output, at each panel and response (as specified by SETMC grid and DOF entries) combination, the software ranks the contribution that the panel makes at each excitation frequency (as specified by FREQ) to the response based on the results of the following equation:

$$MAG_i = \frac{\sqrt{\frac{1}{N}\sum_{j=1}^{N} X_i\left(j\right) * X_i^*\left(j\right)}}{\sqrt{\frac{1}{N}\sum_{j=1}^{N} X\left(j\right) * X^*\left(j\right)}}$$

where:

$MAG_i(j)$ = magnitude of contribution at excitation frequency *j* from panel *i* (real)

$X_i(j)$ = panel contribution to response at excitation frequency *j* from panel *i* (real or complex)

$X_i^*(j)$ = complex conjugate of $X_i(j)$

$X(j)$ = total response at excitation frequency *j* (real or complex)

$X^*(j)$ = complex conjugate of $X(j)$

N = total number of excitation frequency increments

You can use the TOPP describer to specify the number of excitation frequencies where the panel contributes most to the response for the software to output for each panel and response combination. In the output, the excitation frequencies are listed from the highest contributor to the lowest contributor.

For example, if you specify TOPP = 10, for each panel and response combination, the software outputs the contribution of the panel at the 10 excitation frequencies where the contribution of the panel is greatest. If you specify TOPP = ALL, the software outputs the contribution of the panel at all frequencies listed from the highest contributor to the lowest contributor. If you do not specify the TOPP describer, by default, the software outputs the contribution of the panel at the 5 excitation frequencies where the contribution of the panel is greatest.

You can use the ABS, NORM, or BOTH describers to specify if the output is normalized.

For example, specify NORM to output normalized panel contributions and specify ABS to output non-normalized panel contributions. Specify BOTH to output both normalized and non-normalized panel contributions.

For SORT2 output, the software calculates the normalized panel contributions as follows:

$$NMC_i\left(j\right) = \frac{X_i\left(j\right)}{\sqrt{\frac{1}{N}\sum_{j=1}^{N} X\left(j\right) * X^*\left(j\right)}}$$

where $NMC_i(j)$ is the normalized panel contribution to the response at excitation frequency j from panel i.

Example SORT1 input:

```
...
$ CASE CONTROL
SETMC 99 = PRES/198(T1)
SET 500 = TOP, BOTTOM
PANCON(SORT1,PRINT,ABS,TOPP=3,PANEL=500) = 99
FREQ = 1
...
BEGIN BULK
FREQ1,1,86.0,27.0,2
...
ENDDATA
```

Example SORT1 output:

```
...
              COMPLEX ABSOLUTE PANEL CONTRIBUTIONS
FREQUENCY =  8.600000E+01      PRESSURE, GRID =        198
CONTRIBUTIONS FROM STRUCTURAL MODES
    PANEL                  REAL           IMAGINARY

    TOTAL              -1.649896E-09    -1.889964E-08
    BOTTOM              8.855658E-10    -5.356264E-09

              COMPLEX ABSOLUTE PANEL CONTRIBUTIONS
FREQUENCY =  1.130000E+02      PRESSURE, GRID =        198
CONTRIBUTIONS FROM STRUCTURAL MODES
    PANEL                  REAL           IMAGINARY

    TOTAL              -9.694239E-09    -1.137355E-08
    BOTTOM             -1.736440E-09    -3.504814E-09
    TOP                -3.024307E-09    -2.259977E-09
...
```

GRDCON case control command

The GRDCON case control command output quantifies how the motion of grid points in structural panels contributes to the acoustic response at a specific location in the fluid.

When you use the GRDCON case control command, you can specify whether the output is in SORT1 or SORT2 format and whether the output is normalized.

For SORT1 output, at each excitation frequency (as specified by FREQ) and response (as specified by SETMC grid and DOF entries) combination, the software ranks the contribution that each grid in a structural panel makes to the response based on the results of the following equation:

$$MAG_i(j) = \left| \frac{X_i(j)}{X(j)} \right|$$

where:

$MAG_i(j)$ = magnitude of contribution at excitation frequency j from grid i (real)

$X_i(j)$ = grid contribution to response at excitation frequency j from grid i (real or complex)

$X(j)$ = total response at excitation frequency j (real or complex)

You can use the TOPG describer to specify the number of top contributing grids that the software outputs for each excitation frequency and response combination. In the output, the grids are listed from the highest contributor to the lowest contributor.

For example, if you specify TOPG = 10, for each excitation frequency and response combination, the software outputs the 10 grids that contribute the most. If you specify TOPG = ALL, the software outputs the contribution of all the grids listed from the highest contributor to the lowest contributor. If you do not specify the TOPG describer, by default, the software outputs the 5 grids that contribute the most.

You can use the ABS, NORM, or BOTH describers to specify if the output is normalized.

For example, specify NORM to output normalized grid contributions and specify ABS to output non-normalized grid contributions. Specify BOTH to output both normalized and non-normalized grid contributions.

For SORT1 output, the software calculates the normalized grid contributions as follows:

$$NMC_i(j) = \frac{X_i(j)}{X(j)}$$

where $NMC_i(j)$ is the normalized grid contribution to the response at excitation frequency j from grid i.

For SORT2 output, at each grid and response (as specified by SETMC grid and DOF entries) combination, the software ranks the contribution that the grid makes at each excitation frequency (as specified by FREQ) to the response based on the results of the following equation:

$$MAG_i = \frac{\sqrt{\frac{1}{N}\sum_{j=1}^{N} X_i(j) * X_i^*(j)}}{\sqrt{\frac{1}{N}\sum_{j=1}^{N} X(j) * X^*(j)}}$$

where:

$MAG_i(j)$ = magnitude of contribution at excitation frequency j from grid i (real)

$X_i(j)$ = grid contribution to response at excitation frequency j from grid i (real or complex)

$X_i^*(j)$ = complex conjugate of $X_i(j)$

$X(j)$ = total response at excitation frequency j (real or complex)

$X^*(j)$ = complex conjugate of $X(j)$

N = total number of excitation frequency increments

You can use the TOPG describer to specify the number of excitation frequencies where the grid contributes most to the response for the software to output for each grid and response combination. In the output, the excitation frequencies are listed from the highest contributor to the lowest contributor.

For example, if you specify TOPG = 10, for each grid and response combination, the software outputs the contribution of the grid at the 10 excitation frequencies where the contribution of the grid is greatest. If you specify TOPG = ALL, the software outputs the contribution of the grid at all frequencies listed from the highest contributor to the lowest contributor. If you do not specify the TOPG describer, by default, the software outputs the contribution of the grid at the 5 excitation frequencies where the contribution of the grid is greatest.

You can use the ABS, NORM, or BOTH describers to specify if the output is normalized.

For example, specify NORM to output normalized grid contributions and specify ABS to output non-normalized grid contributions. Specify BOTH to output both normalized and non-normalized grid contributions.

For SORT2 output, the software calculates the normalized grid contributions as follows:

$$NMC_i(j) = \frac{X_i(j)}{\sqrt{\frac{1}{N}\sum_{j=1}^{N} X(j) * X^*(j)}}$$

where $NMC_i(j)$ is the normalized grid contribution to the response at excitation frequency j from grid i.

Example SORT1 input:

```
...
$ CASE CONTROL
SETMC 99 = PRES/198(T1)
SET 700 = 10544 10305 10110 10777
GRDCON(SORT1,PRINT,ABS,TOPG=2,GRID=700) = 99
FREQ = 1
...
BEGIN BULK
FREQ1,1,86.0,27.0,2
...
ENDDATA
```

Example SORT1 output:

```
...
              COMPLEX ABSOLUTE GRID CONTRIBUTIONS
FREQUENCY =  8.600000E+01      PRESSURE, GRID =        198
CONTRIBUTIONS FROM STRUCTURAL MODES
     GRID                   REAL           IMAGINARY

    TOTAL               -5.846166E-10    -1.356268E-09
    10305                3.565532E-15     7.451980E-15
    10777               -5.422780E-17    -7.540371E-17

              COMPLEX ABSOLUTE GRID CONTRIBUTIONS
FREQUENCY =  1.130000E+02      PRESSURE, GRID =        198
CONTRIBUTIONS FROM STRUCTURAL MODES
     GRID                   REAL           IMAGINARY

    TOTAL               -7.430111E-10    -4.562482E-10
    10305                2.862127E-15    -7.918542E-15
    10777               -1.468421E-17     1.060137E-17
...
```

Incident and transmitted acoustic power and acoustic power transmission loss output

For direct and modal acoustic frequency response analysis, you can request results output for acoustic power, incident acoustic power, transmitted acoustic power, and acoustic power transmission loss in deterministic and random acoustic analysis.

To output acoustic power for 2D microphone surfaces and AML regions, use the ACPOWER case control command. Because acoustic power results include both incident and scattered (reflected) components, you can use the INPOWER, TRPOWER, and TRLOSS case control command to examine acoustic results in more detail and compute acoustic transmission loss.

- Use INPOWER to isolate the acoustic power that is incident on a surface like a 2D microphone mesh or a set of acoustic free faces (BSURFS) and that is attributable to acoustic sources like monopoles or acoustic plane waves, but not reflections.

- Use TRPOWER to request the acoustic power that is transmitted through an automatically matched layer (AML) region or a 2D microphone mesh that includes both incident and scattered components.

- Use TRLOSS to request the acoustic power transmission loss. The software calculates the acoustic power transmission loss from the INPOWER and TRPOWER results as follows:

$$TRLOSS = 10\log_{10}\left(\frac{INPOWER}{TRPOWER}\right)$$

Calculating acoustic power transmission loss through a structural panel

As an example, you can use TRLOSS to calculate the acoustic power transmission loss through a structural panel.

In a laboratory setting, you can measure the acoustic power transmission loss through a structural panel by mounting the panel in an aperture of a wall between two rooms. One room is reverberant and contains a diffuse acoustic plane wave source. The other room is anechoic and has no pure acoustic sources. The anechoic room also contains the instrumentation for measuring the acoustic power. Because the room is anechoic, you measure the acoustic power that radiates from the panel, and the measurement is not influenced by reflections.

A simulation for such a laboratory setup is shown below:

(1) AMLREG1 (reverberant side)

(2) Structural panel

(3) Acoustic free faces (reverberant side)

(4) AMLREG2 (anechoic side)

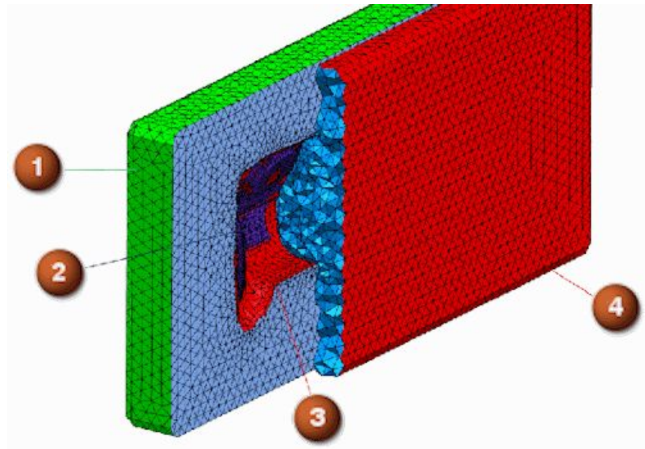


Figure 5-2. Panel test

In the simulation, both rooms are represented by fluid meshes using AML regions. AMLREG1 represents the reverberant room where the fluid domain opens to the half space in front of the panel. The acoustic power source within the reverberant room can be modeled by a set of random plane waves typically outside the fluid mesh. AMLREG2 represents the anechoic room where the fluid domain opens to the half space behind the panel.

1. Use INPOWER to calculate the incident acoustic power on the free acoustic faces on the reverberant room side that couple with the panel.

2. Use TRPOWER to calculate the acoustic power flowing through AMLREG2.

3. Use TRLOSS to calculate the acoustic power transmission loss through the panel.

Random results

If you request random results with the INPOWER, TRPOWER, and TRLOSS case control commands, only real output is supported.

Limiting acoustic output to peak responses

In a SOL 108 or SOL 111 acoustic or vibro-acoustic frequency response analysis, you can limit the output to frequencies where peak responses occur. This capability is called *peakout*.

For more information on peakout, see the Limiting Output to Peak Responses section of the *Simcenter Nastran Basic Dynamic Analysis User's Guide*.

5.3 Bulk entries

The following describes the bulk entries used for acoustic solutions.

5.3.1 Acoustic grids

GRID bulk data entries (see GRID in the *Simcenter Nastran Quick Reference Guide*) are used to specify the geometric coordinates and coordinate system of the fluid degrees of freedom. Acoustic fluid grids are defined with CD = -1 and acoustic elements reference only fluid grids. At the boundary between the structure and the fluid, the grids can be coincident. However, structural elements cannot connect to fluid grids and vice versa.

5.3.2 Acoustic elements

Acoustic elements are 3-D and are modeled using the CHEXA, CTETRA, CPENTA, and CPYRAM elements. These elements reference a PSOLID entry (see also PSOLID in the *Simcenter Nastran Quick Reference Guide*) to define the physical property.

The FCTN field must be set to “PFLUID” for acoustic elements. In addition the MID field must reference a MAT10, MAT10C, MATF10C, or MATPOR material.

The 2-D elements CTRIA3 and CQUAD4 are used to define microphone elements (see Microphone mesh). In addition, you can define CROD and MICPNT to define a 1-D and 0-D element respectively. The CROD, CTRIA3, and CQUAD4 elements in this case must reference PMIC physical property

5.3.3 Acoustic material definition

Material properties for acoustic elements are defined using MAT10, MAT10C, MATF10C, or MATPOR bulk entries.

In the MAT10 entry, the density is specified in the RHO field and the speed of sound in the C field. Porous materials with equivalent properties can be specified using GE and GAMMA fields, and the frequency-dependent tables via the continuation line entries. Equations

$$\rho_s(\omega) = \frac{\rho_{er}^2(\omega) + \rho_{ei}^2(\omega)}{\rho_{er}(\omega)}; \quad G_e(\omega) = -\frac{\rho_{ei}(\omega)}{\rho_{er}(\omega)}$$

and

$$\beta_s(\omega) = \frac{\beta_{er}^2(\omega) + \beta_{ei}^2(\omega)}{\beta_{er}(\omega)}; \quad \gamma(\omega) = \frac{\beta_{ei}(\omega)}{\beta_{er}(\omega)}$$

(see Frequency-dependent acoustic materials or absorbers) that define ρ_s, β_s, G_e, and γ are then specified as entries in the MAT10.

For detailed information on MAT10, MAT10C, MATF10C, or MATPOR, see the *Simcenter Nastran Quick Reference Guide*.

5.3.4 Acoustic panel normal velocity

You use the ACPNVEL bulk entry to specify the acoustic panel normal velocity boundary. ACPNVEL supports the definition of:

- Magnitude and phase of acoustic velocity.
- Constant or frequency-dependent complex velocity.

5.3.5 Transfer admittance

You use the ACTRAD bulk entry that references PACTRAD to specify the transfer admittance between two sets of acoustic free faces.

5.3.6 Impedance or admittance

Frequency-dependent absorbers are specified using CAABSF elements (see CAABSF in the *Simcenter Nastran Quick Reference Guide*). CAABSF allows POINT, LINE, or AREA impedance to be specified on the free fluid surface. The grids G1 through G4 can reference only fluid grids. The CAABSF elements define the element topology and in turn reference the physical property PAABSF or PAABSF1.

The PAABSF physical property table is used to specify constant complex impedance or frequency-dependent impedance, or a combination of both.

The PAABSF1 physical property table is used to specify constant real and imaginary impedance and admittance on a 2D fluid free surface.

5.3.7 Acoustic loads, boundary condition, and sources

- Use the ACSRCE bulk entry (see ACSRCE in the *Simcenter Nastran Quick Reference Guide*) to define the acoustic source loading where you define the source by acoustic power as a function of frequency. Alternatively, you can directly define acoustic volume acceleration using the RLOAD1 or RLOAD2 entries.
- Use the ALOAD bulk entry to specify a combination of the following:
    - Monopole source using the ACPOLE1 bulk entry.
    - Dipole source using the ACPOLE2 bulk entry.
    - Plane wave source using the ACPLNW bulk entry.
    - Enforced acoustic pressure load with complex data input using the ACPRESS bulk entry.
    - Panel normal velocity boundary condition using the ACPNVEL bulk entry.

> **Note**
>
> The ALOAD bulk entry does not support scaling of loads.

5.3.8 Modeling structure-acoustic interface

The ACMODL bulk entry (see ACMODL in the *Simcenter Nastran Quick Reference Guide*) can be used to model the structure-acoustic interface. The search parameters for coupling are specified via NORMAL and OVLPANG fields. The units for the search parameters can be in absolute or relative units (specified using SRCHUNIT field). Structural and fluid elements that need to be coupled can be specified using FSET and SSET fields respectively. The input can be a set of grids, elements, or physical properties. Simcenter Nastran converts the input into relevant elements.

> **Note**
>
> Because a FSET or SSET field can select a SET3 bulk entry, you can also use 2D PLOTEL elements in the ACMODL bulk entry. PLOTEL elements are elements that you can add to your model for plotting and visualization in a preprocessor, such as Simcenter 3D.

Chapter 6: Acoustic and vibro-acoustic analyses

6.1 Overview of acoustic and vibro-acoustic analyses

- The 3-D elements CHEXA, CPENTA, CPYRAM, or CTETRA can be used to model the fluid.
- The 2-D elements CTRIA3 and CQUAD4 are used to model microphone elements.
- The rod element CROD is used to model the 1-D microphone element.
- The MICPNT element can be used for modeling a 0-D microphone element or microphone point.
- 3-D frequency-dependent absorbers, in which the absorptive properties are included on the MAT10 or MAT10C bulk entry (fluid material).
- 2-D/1-D frequency-dependent absorbers (CAABSF).
- In a coupled vibro-acoustic problem, the interface between the fluid and structure can be modeled with coincident or non-coincident unconnected grids. The fluid grids and structural grids can be coincident but unconnected.

> **Note**
>
> For more information on various coupling options, see Defining the fluid-structure interface boundary condition.

- Coupled vibro-acoustic analysis is available in the direct dynamic solution sequences 107 through 109 and 200 with ANALYSIS = DFREQ, and the modal dynamic solution sequences 103, 110, 111, 112, and 200 with ANALYSIS = MFREQ and ANALYSIS = MTRAN. Note that in solutions 103, 110, 111, 112, and 200, the normal modes are computed separately for the fluid and structural parts of the model. In other words, the uncoupled modes of the fluid and structure are used in the modal formulation of the stiffness, mass, and damping. In SOLs 110 through 112, the SDAMPING Case Control command and the parameters G and W3 are applied only to the structural portion of the model. Design Sensitivities may be computed in SOL 200.

6.1.1 Defining the fluid

You define fluid grid points by specifying a value of -1 for CD in field 7 on the GRID bulk entry.

Fluid elements are defined using the CHEXA, CPENTA, CPYRAM, and CTETRA bulk entries. However, in the PSOLID entry, you must specify the character value PFLUID for FCTN in field 8 and MID, field 3, must reference a MAT10, MAT10C, MATF10C, or MATPOR material entry.

The MAT10 entry defines the bulk modulus and the mass density properties of the fluid, along with optional absorber properties.

Microphone elements, which are also fluid elements, are defined using the CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CQUAD4, CROD, and MICPNT bulk entries. Except for MICPNT, these bulk entries must reference the PMIC bulk entry for the physical property.

Additionally, you can model different fluid domains and connect them using a surface-to-surface glue definition.

> **Note**
>
> Surface-to-surface gluing can be used between acoustic mesh faces. Similar to structural glue conditions, the meshes between the acoustic glued faces can be dissimilar. The acoustic-to-acoustic glue inputs are consistent with the structural-to-structural capability, except that on the BGPARM bulk entry, GLUETYPE, PENTYPE, and PENT are ignored. PENN is used to calculate the acoustics penalty matrix [K].
>
> Refer to the *Gluing Elements* chapter in the *Simcenter Nastran Simcenter Nastran User's Guide* for more information.

6.1.2 Absorber elements

The acoustic absorber elements are special elements needed to model soundproofing materials on the structural surface.

See Impedance or admittance for more information.

6.1.3 Acoustic loading

Loads on fluid elements are, in most cases, analogous to enforced displacements on structural elements. The types of loading available for fluid elements are detailed in Defining acoustic loads.

6.1.4 Single-point constraints

Single-point constraints of the fluid (P = 0.0) may be enforced on the fluid boundary using the SPC entry or the PS field on the GRID entry. The fluid pressure degree-of-freedom is defined as component 1 on the SPC entry.

6.1.5 Output requests (overview)

The following output can be requested on the fluid:

- Pressure and peak sound pressure levels at the fluid points, sound pressure level in dB and dBA.
- RMS sound pressure level in some Nastran solutions.
- Separate output for the fluid and structural portion of the model.
- Particle velocity at element centroid.
- Modal and panel contribution output.
- Equivalent radiated power for some shell elements and output equivalent radiated power for panels that contain shell elements.

- Coupling information files with the FLSTCNT case control command.

See the Output requests and Fluid-structure interaction control sections in the *Input Data Requirements (Summary)* chapter for more information.

6.1.6 Vibro-acoustic coupling punch output

You can request the output of the vibro-acoustic coupling matrix AGG to the punch file with the AGGPCH describer on the FLSTCNT case control command.

For more information, see Fluid-structure interaction control.

6.1.7 Vibro-acoustic coupling

The model for fluid/structural systems has an explicit FE mesh for both the structural components and the interior fluid cavity. The equation of motion for the coupled system is shown by equation

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

(see Coupling of fluid and structure (vibro-acoustics)).

The coupling algorithm uses a multi-step approach, which is detailed in Two-way coupling.

6.1.8 Superelement analysis for vibro-acoustic analysis

Superelements may be used with the restrictions listed in the Superelements section.

6.1.9 Running the job

Diagnostic messages

Diagnostic messages are generated automatically by the software. The following messages indicate the error factors in area for free body motions. Nonzero numbers indicate a hole in the model as indicated by the T2 value.

```
 ^^^ DMAP INFORMATION MESSAGE 9055 (SEMG) -  THE FL./STR. INTERFACE CHECK IS FORCES
 AND MOMENTS RESULTING FROM A UNIT INCREASE IN PRESSURE, OR CHANGES IN THE FLUID PRESSURE
RESULTING FROM RIGID BODY MOTIONS OF THE STRUCTURE. THESE VALUES ARE DIRECTLY PROPORTIONAL
 TO THE OPEN SURFACE OF THE FLUID.

                                   FL./STR. INTERFACE CHECK
         T1              T2             T3              R1              R2              R3
 1   2.3554080E-18 -8.4260993E-02 -1.4653091E-28   6.0271138E-10   2.6100844E-19   1.8405548E-10
```

Diagnosing problems

Some recommended techniques are as follows:

- Carefully check the special diagnostic messages. Holes in the boundaries cause nonzero resultant area factors.

- You can include the SKINOUT describer on the FLSTCNT case control command to request coupling information files. See *Requesting coupling information files* below.

- Run tests without the boundary coupling to estimate the frequencies.

- Temporarily switch to an alternate eigenvalue method and/or a smaller range to ensure modes are correct.
- If a model is large, try subdividing it into smaller sections that can be checked more conveniently.

Requesting coupling information files

You can include the SKINOUT describer on the FLSTCNT case control command to request debug files. Depending on the option used, a punch file *.pch and/or debug data deck *.dat is created. The resulting files have _acdbg appended to their name. For example, if the original input file is test.dat, the debug files are test_acdbg.dat and test _acdbg.pch.

The SKINOUT input options are described in detail in the Fluid-structure interaction control section in the *Input Data Requirements* chapter.

6.2 Loudspeaker example (simplified version)

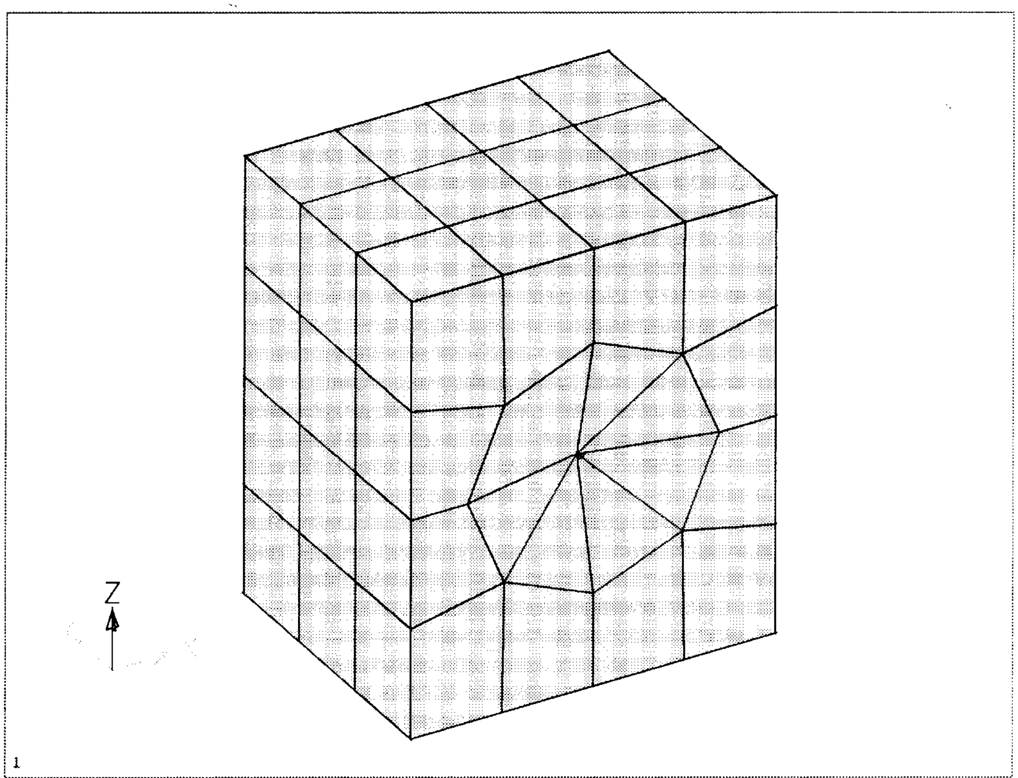


Figure 6-1. Acoustic suspension loudspeaker

Test problem description

A simple test problem illustrates the fundamentals of acoustic analysis for enclosed containers with flexible walls. The physical problem represents an initial attempt at design and analysis of an enclosure for an acoustic suspension loudspeaker system. The objective was to calculate the resonant frequencies and responses of the system without the mass and impedance of the speaker coil and magnet components.

Physical description

The structure consists of a simple rectangular box with wood walls as shown by the plate model in Figure 6-1. A single cutout is provided for the speaker and a thin polyethylene cone was modeled with triangular shells. The physical properties of the model are listed below. Note the use of the MKS system of units, which illustrates the flexibility of Simcenter Nastran.

Width:	0.5 M	Depth:	0.4 M
Height:	0.6 M	Hole Diam.	0.345 M
Box E Modulus:	11.61E9	Box Density:	562 Kg/M**3
Wall thickness:	0.015 M		
Air Wave speed:	344 M/Sec	Air Density:	1.11KG/M**3
Cone E Modulus:	3.4E9	Cone Density:	450.0
Cone thickness:	0.1E-3 M	Cone Depth:	0.04 M

Executive and case control

The configuration shown below is a direct complex modal solution using the Lanczos method to extract eigenvalues.

Bulk data

The key data of note in the Bulk Data section are as follows:

- ACMODL bulk entry is not explicitly listed in the input deck. However, this means that Simcenter Nastran uses internally ACMODL entry with defaults.
- The speaker cone consists of a light, thin plastic material, and the box is made from wood.
- The cone was meshed with CTRIA3s and the box with CQUAD4s, while CHEXAs represent the air.

Results from Simcenter Nastran

Three runs (out of many) are described below. They illustrate a recommended sequence for the analysis process.

Run 1: Real eigenvalue analysis

Before the coupled structure is analyzed, it is important to understand the behavior of the structural and fluid models separately. Fortunately they can be included in the same data file and the Real Eigenvalue Solution Sequence (SOL 103) may by used. The results for the uncoupled speaker box and the acoustic modes are shown below. Note that the natural frequencies for both systems occur in the same range, which indicates that the enclosure modes will interact with the acoustic cavity resonances. Results for the acoustics were checked by one-dimensional wave solutions. Results for the box were checked against calculated natural frequencies of simply-supported plates.

Mode no.	Frequency, Hz	Type
1	0.0	Air- Constant Pressure (fictitious)
2	292	Air- 1st z (up/down)
3	297	Box- u(y) Back panel
4	351	Air- 1st x (Left/Right)
5	359	Box- u(x) Sides in-Phase

Mode no.	Frequency, Hz	Type
6	379	Box- u(x) Sides out-of-phase
7	450	Box- u(z) Top/bottom
8	453	Air- 1st y (Fore/Aft)

Run 2: Complex eigenvalue analysis, coupled modes

The coupled natural frequencies must be obtained from an unsymmetrical matrix equation which requires a Complex Eigenvalue method even in the undamped case. The CLAN method is recommended for most large-order non-superelement jobs. The only changes from the Real Modes job was the addition of an ACMODL input and the changes in eigenvalue method.

Mode no.	Frequency	Type of dominant motion
1	0.0	Air- Constant Pressure
2	291	Air- u(z)
3	295	Coupled- u(y)- back panel
4	336	Coupled- u(x)- sides
5	371	Coupled- u(y)- cone
6	373	Coupled- u(x)- sides
7	445	Coupled- u(z)- top/bottom
8	453	Coupled- u(y)- cone

Results above are interpreted by examining the frequency shifts from the uncoupled system. The modes with u(y) fore-aft motion are most likely to be excited by the speaker. Modes 1, 2, 3, 4, 6, and 7 are close to their uncoupled equivalent. Modes 5 and 8 are new combinations of higher modes. The results were confirmed by several methods:

1. The job was rerun using the INV method on the EIGC Bulk Data and produced nearly identical results.

2. A printout of the interface area matrix was obtained by using a DMAP alter and verified by hand calculations.

3. The natural symmetry of the geometry produced symmetric and antisymmetric results relative to the natural structural planes of symmetry (except for roots with close frequencies).

Excerpt of an input file demonstrating the 2nd run:

```
ACOUSTIC TEST PROBLEM DATA FILES
ID SPEAK39F,DNH
SOL 107
DIAG 8,12 $ PRINTS MATRIX TRAILERS AND ROOT-TRACKING MESSAGES.
CEND
TITLE=SPEAKER BOX -WITH CONE, SIMPLE CORNER SUPPORTS
SUBTITLE = COUPLED BOUNDARY, NON-MATCHING ELEMENTS
ECHO= UNSORT
SEALL=ALL
SPC=20
$ USES MKS SYSTEM
CMETHOD = 7  $LANCZOS
SET 20= 3,13,23,43,82,83,84,91,93,95,103,113,123,
```

```
    131,135,153,163,171,173,175,183,
    1013,1023,1043,1082,1083,1084,1091,1095,1113,
    1131,1135,1163,1173,1183,1193
DISP= 20  $ FOR MINIMUM PRINTOUT
$ DISP(PLOT)=ALL
$
BEGIN BULK
$
PARAM,POST,0
PARAM,COUPMASS,1
$
$ BOX PROPERTIES - WOOD
MAT1,11,11.61+9,,0.3,562.0
PSHELL,1000,11,.015,11,,11
$
$ SPEAKER CONE
MAT1,3,3.4+9,,0.3,450.0
PSHELL,10,3,0.1-3,3,,,0.223
$
$ PROPERTIES OF AIR
MAT10,100,131.94+3,1.115
PSOLID,100,100,,2,,1,PFLUID
$
$ EIGEN METHODS
EIGR,20,MGIV,1.0,600.0
EIGC,7, CLAN,,,,,,,+CLAN
+CLAN,0.0,10.0,0.0,1600.0,100.0,,20
$ USE INVERSE POWER TO CHECK LOW ROOTS
EIGC,107,INV,MAX,,,,,,+EC1
+EC1,0.0,100.0,0.0,1800.0,100.0,12,9
$
$ FIX BOX AT BOTTOM CORNERS
SPC1,20,123,1001,1005,1031,1035
$ COORDINATE SYSTEM AT CENTER OF HOLE
CORD1C,83,283,113,85
$
$
$FLUID GRID POINTS
$ NOTE VALUE OF -1 IN FIELD 7 INDICATES 1 DOF.
GRID,1,, -.25, -.2, -.3,-1
GRID,2,,-.125, -.2, -.3,-1
GRID,3,,  0.0, -.2, -.3,-1
GRID,4,, .125, -.2, -.3,-1
GRID,5,,  .25, -.2, -.3,-1
GRID,11,, -.25, -.0667, -.3,-1
GRID,12,,-.125, -.0667, -.3,-1
GRID,13,,  0.0, -.0667, -.3,-1
GRID,14,, .125, -.0667, -.3,-1
GRID,15,,  .25, -.0667, -.3,-1
GRID,21,, -.25, 0.0667, -.3,-1
GRID,22,,-.125, 0.0667, -.3,-1
GRID,23,,  0.0, 0.0667, -.3,-1
GRID,24,, .125, 0.0667, -.3,-1
.25, -.2, 0.,-1
-
ETC.
```

```
-
$ STRUCTURE GRIDS
GRID,1001,, -.25, -.2, -.3
GRID,1002,,-.125, -.2, -.3
GRID,1003,,  0.0, -.2, -.3
GRID,1004,, .125, -.2, -.3
GRID,1005,,  .25, -.2, -.3
GRID,1011,, -.25, -.0667, -.3
GRID,1012,,-.125, -.0667, -.3,,6
GRID,1013,,  0.0, -.0667, -.3,,6
-
ETC.
-
$ OPTIONAL ASET DATA TO USE WITH HESS METHOD
$ASET1,1,1,THRU,195
$ASET1,123,1003,1012,1014,1022,1024,1033
$ASET1,123,1163,1172,1174,1182,1184,1193
$ASET1,123,1042,1043,1044,1081,1082,1083,1084
$ASET1,123,1065,1122,1123,1124
$ASET1,123,1072,1074,1112,1113,1114,1152,1154
$ASET1,123,1051,1061,1131,1141
$ASET1,123,1055,1065,1135,1145
$
$
$SOLID ELEMENTS FOR AIR
CHEXA   1       100     1       2       12      11      41      42      +001
+001    52      51
CHEXA   2       100     2       3       13      12      42      43      +011
+011    53      52
CHEXA   3       100     3       4       14      13      43      44      +021
+021    54      53
CHEXA   4       100     4       5       15      14      44      45      +031
+031    55      54
CHEXA   11      100     11      12      22      21      51      52      +041
+041    62      61
CHEXA   12      100     12      13      23      22      52      53      +051
+051    63      62
CHEXA   13      100     13      14      24      23      53      54      +061
+061    64      63
CHEXA   14      100     14      15      25      24      54      55      +071
+071    65      64
CHEXA   21      100     21      22      32      31      61      62

+081
+081    72      71
-
ETC
-
$STRUCTURAL ELEMENTS
CQUAD4  1001    1000    1001    1002    1012    1011
CQUAD4  1002    1000    1002    1003    1013    1012
CQUAD4  1003    1000    1003    1004    1014    1013
CQUAD4  1004    1000    1004    1005    1015    1014
CQUAD4  1011    1000    1011    1012    1022    1021
CQUAD4  1012    1000    1012    1013    1023    1022
CQUAD4  1013    1000    1013    1014    1024    1023
```

```
CQUAD4  1014    1000    1014    1015    1025    1024
-
ETC
-
$
$SIMPLE CONE
$
CTRIA3, 5042,10,1042,1043,1083
CTRIA3, 5043,10,1043,1044,1083
CTRIA3, 5044,10,1044,1084,1083
CTRIA3, 5084,10,1084,1124,1083
CTRIA3, 5124,10,1124,1123,1083
CTRIA3, 5123,10,1123,1122,1083
CTRIA3, 5122,10,1122,1082,1083
CTRIA3, 5082,10,1082,1042,1083
ENDDATA
```

Run 3: Complex eigenvalue analysis, modal formulation

A more efficient method for solving a coupled matrix problem is to reduce the size of the matrices by using modal coordinates. In this case, both the structural displacements and fluid pressures are replaced in the matrix solution by generalized coordinates representing the uncoupled real modes. For the modal cases, the real modes (25 total) below 800 Hz. were used. In the coupled cases, 13 modes were obtained below 500 Hz. All results compared to within 1%.

On a Sun SPARCstation 1+ the CPU time comparisons are:

Type solution	Solution size	CPU time (seconds)
Uncoupled Modes	538	110.2
Direct, No Modes	538	586.3
Structural Modes Only	114	143.2
All Modes	25	106.9

The conclusion is that the modal method (SOL 110) reduces the costs with very little effect on accuracy. It is recommended that you use a liberal number of real modes (twice the number of coupled modes) to represent the system.

The complete input file **spkall.dat** for the modal method is available in *install_directory*\nxnr\nast\misc\doc\advdynamics\.

Chapter 7: Acoustic analysis using automatically matched layer

7.1 Exterior acoustics using automatically matched layer

Modeling exterior acoustics using the finite element method (FEM) has some challenges. In particular, a non-reflecting (anechoic) boundary condition is required at the free fluid boundary.

In previous releases, you defined acoustic absorbers on the free fluid mesh boundary with a characteristic impedance (density x speed of sound). To prevent reflections with this method, this boundary had to be several wavelengths from the vibrating source. As a result, these models tended to be large, and perfect absorption was still difficult for a range of frequencies.

Another recommended method of representing the non-reflective acoustic boundary condition is the Automatically Matched Layer (AML) method. The AML method uses a reflectionless artificial layer that absorbs outgoing waves regardless of their frequency and angle of incidence. As depicted in the following figure, the AML is defined on a convex shape boundary, which uses the AMLREG bulk entry.

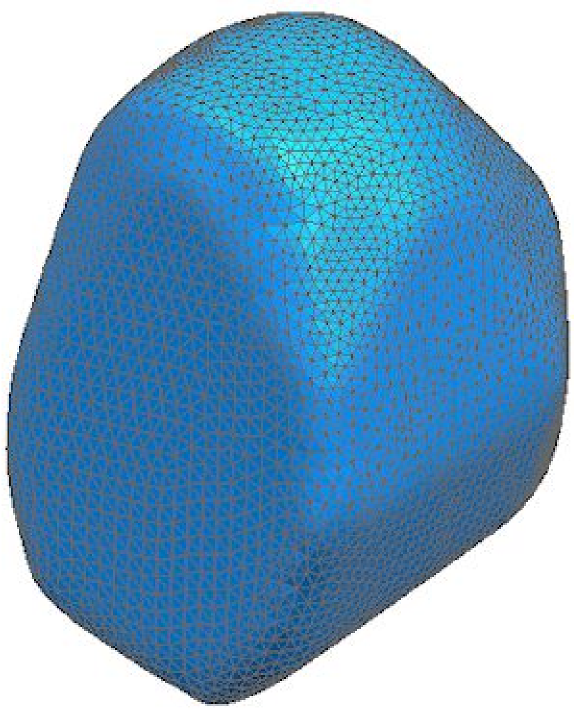


Figure 7-1. Convex mesh/surface

An AML region can be modeled close to the vibrating structure or acoustic source with good accuracy resulting in much smaller FE models.

The following example demonstrates how an AML can be specified to represent the radiation from a vibrating gearbox.

(1) Structural Mesh

(2) Fluid Mesh

(3) AML

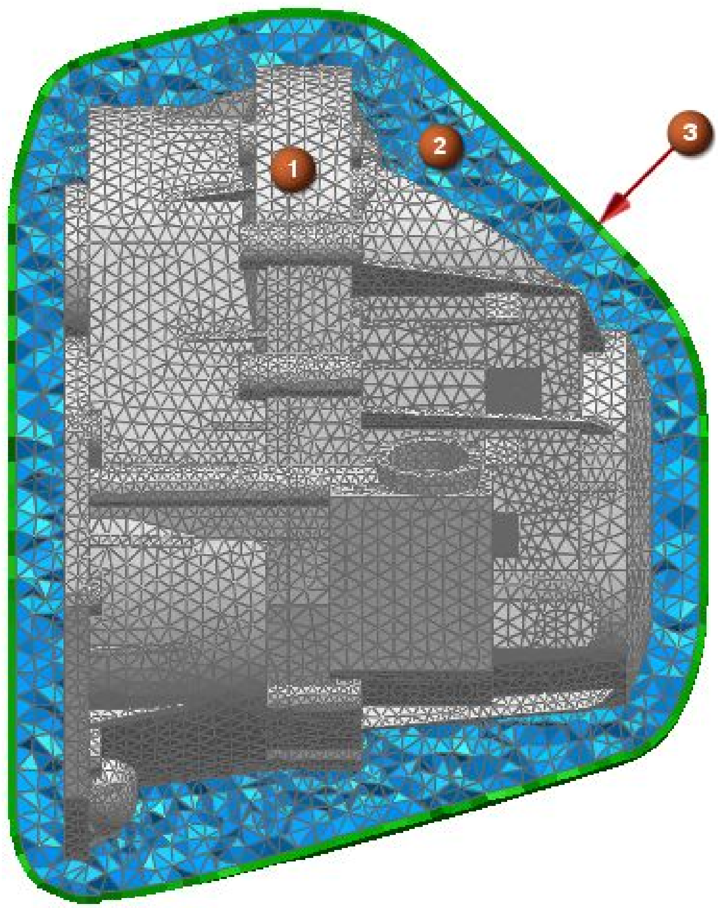


Figure 7-2. Gearbox

The AML produces accurate results for the FE domain, yet the FE domain represents a small part of the fluid, which in reality is infinite. You can request acoustic results at arbitrary locations exterior or interior to the fluid. For output requests exterior to the meshed volume, Simcenter Nastran uses the acoustic results at the boundary of the FE domain and a boundary integral to obtain the acoustic response. These exterior and interior locations are defined with the microphone mesh. See Microphone mesh.

The AML can be specified in a direct frequency response solution (SOL 108) or in a modal frequency response solution (SOL 111). When an AML is defined in a modal frequency response solution, the structure is reduced to modal coordinates, although the acoustic fluid remains in physical coordinates.

7.2 Assumptions and limitations of AML

- The AML boundary condition can be specified only in a direct frequency response SOL 108 or in a modal frequency response SOL 111.

> **Note**
>
> Because the AML boundary condition creates frequency-dependent element matrices, the structure is modeled using modal coordinates in SOL 111. The fluid, however, is still retained in physical coordinates.

- The AML surface must be defined on fluid faces only.
- The AML surface must be convex.
- The AML region must not protrude through an infinite plane that represents a reflection boundary.
- The face of an AML surface must not overlap or intersect with an infinite plane.
- AML faces that contact the infinite plane must be perpendicular to it.
- The infinite plane must be parallel to the plane of two global axes within a tolerance.

For more information about infinite planes, see Infinite planes..

7.3 Convex AML surface

The AML surface must be convex so that modal behavior between parts of a structure can be accounted for.

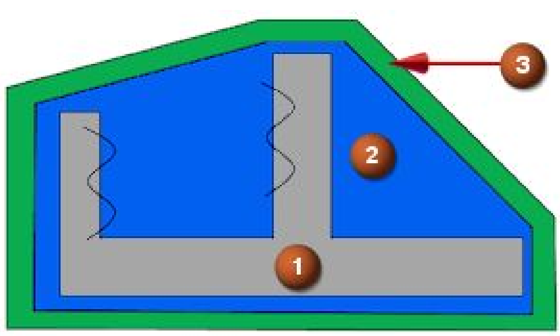
The following examples illustrate convex and non-convex AML surfaces.

Legend for all examples:

(1) Structural Mesh (grey)

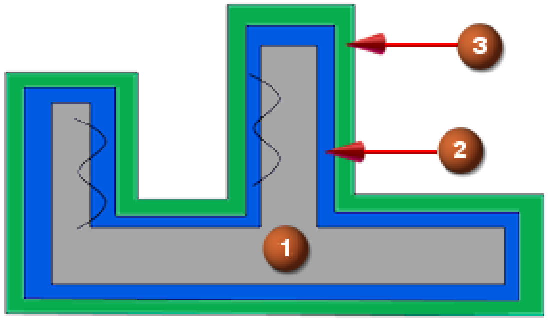
(2) Fluid Mesh (blue)

(3) AML (green)



The waves radiating from the structure are not absorbed, as no non-convex regions exist between the two vibrating surfaces.

Figure 7-3. Convex AML surface



The waves radiating from the structure are absorbed in the non-convex region between the two vibrating surfaces.

Figure 7-4. Non-convex AML surface

7.4 AML faces and infinite plane

AML faces that contact the infinite plane must be perpendicular to it so that the solver computes accurate results. The following examples illustrate perpendicular and non-perpendicular AML faces.

Legend for all examples:

(1) Structural Mesh (grey)

(2) Fluid Mesh (blue)

(3) AML (green)

(4) Microphone point (yellow)

(5) Infinite Plane (purple)

(6) Local surface normal (black)

(7) Outer boundary of reflectionless artificial layer (green dashes) formed by the solver when it extrudes the AML surface.

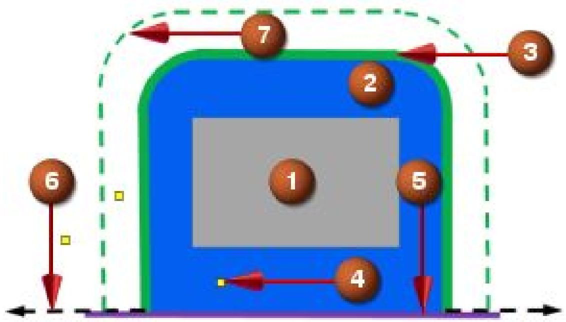


Figure 7-5. AML faces (3) that contact the infinite plane are perpendicular

This model produces an accurate solution.

- The modeled fluid domain matches a half-space where both the FE mesh and the reflectionless artificial layer respect the infinite plane.

> **Note**
>
> In half-space, the sound radiation is only half of the radiation in a free field because of the acoustic reflection boundary.

- The solver accounts for all microphone points outside the FE mesh including points close to the infinite plane.

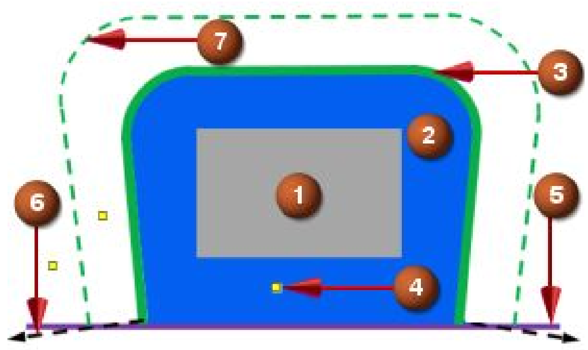


Figure 7-6. AML faces (3) that contact the infinite plane are not perpendicular.

This model may not provide an accurate solution for exterior microphone points because the reflectionless artificial layer intersects the infinite plane. This may also adversely affect the accuracy of interior microphone point results.

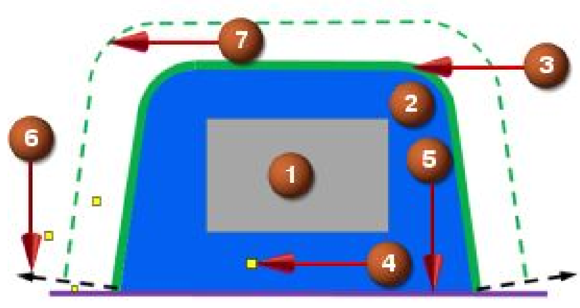


Figure 7-7. AML faces (3) that contact the infinite plane are not perpendicular and the entire reflectionless artificial layer is not in contact with the infinite plane.

This model may not provide an accurate solution because of the following issues:

- The solver may not follow the domain boundaries.
- The solver may not account for microphone points that are outside the FE mesh and close to the infinite plane.
- For interior microphone points, the entire reflectionless artificial layer is not in contact with the infinite plane.

7.5 AML with non-homogeneous elements

The elements that define the AML surface do not have to be homogeneous.

Although the acoustic results derived in the interior will be still accurate, the results on the exterior will not be. This is because Simcenter Nastran computes the acoustic radiation on the exterior using a free space Green's function that assumes homogeneity.

7.6 AML and fluid damping coefficient

The software supports the definition of a user-specified, uniform fluid damping coefficient with the GFL parameter, even when an automatically matched layer (AML) is present.

In a coupled vibro-acoustic analysis, you apply GFL to the fluid portion of a model.

For more information on the GFL parameter, see G - Parameters.

7.7 Microphone mesh

Microphone mesh

Microphone locations can be:

- Exterior to the convex AML boundary.
- Interior to the convex AML boundary.
- On the convex AML boundary.

When a microphone mesh is:

- Interior or on the AML boundary, Simcenter Nastran interpolates the results from the fluid grids to the microphone location.
- Exterior to the AML boundary, Simcenter Nastran uses the pressure and velocity on the AML (or on the physical boundary if requested), and a boundary integral, to compute the results at the exterior microphone locations.

For information on output requests and microphone mesh definitions, see Defining microphone meshes.

The following example demonstrates a 2D microphone mesh.

(1) 2D microphone mesh

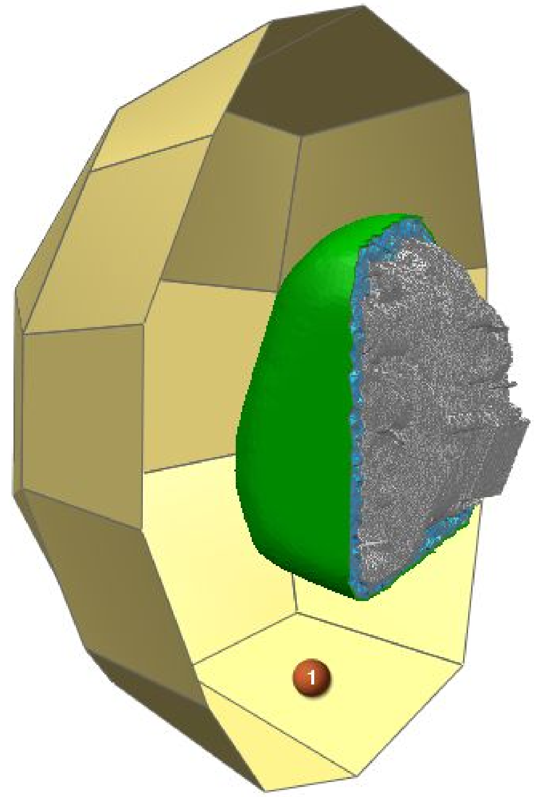


Figure 7-8. Microphone mesh

Infinite planes

Acoustic results for microphone meshes exterior to the AML region are derived using boundary integrals. You can define additional boundaries called infinite planes exterior to AML region to influence the derived acoustic results.

The planes are created with the IPLANE bulk entry, which includes the TYPE field to designate them as a zero velocity or a zero pressure acoustic reflection boundary. You then select up to three infinite planes with the AMLREG entry. If you select multiple infinite planes, they must be perpendicular to one another.

- An infinite plane with TYPE = 0 defines a rigid, reflective boundary in which the velocity is zero. This is also known as a symmetric acoustic boundary. For example, air-to-ground is a zero velocity reflective boundary.

- An infinite plane with TYPE = 1 defines a pressure release reflective boundary in which the pressure is zero. This is also known as an anti-symmetric acoustic boundary. For example, air-to-water is a zero pressure reflective boundary.

You should only define a microphone mesh on the side of an infinite plane in which the fluid elements are defined. The software will not compute results on the non-fluid side.

Location of pressure and pressure gradients

You can use the RADSURF field on the AMLREG entry to select the location of pressure and pressure gradients for computing the acoustic results on microphone locations exterior to the AML region.

- When RADSURF = AML, the pressure and pressure gradients at the AML are used to compute the results exterior to the AML region.
- When RADSURF = PHYB, the pressure and pressure gradients on the physical boundary are used to compute the results exterior to the AML region. The physical boundary is defined as all fluid free faces excluding those on the AML and those on an infinite plane.

The following examples illustrate the use of AML, RADSURF (field of the AMLREG bulk entry), and infinite plane.

Legend for all examples:

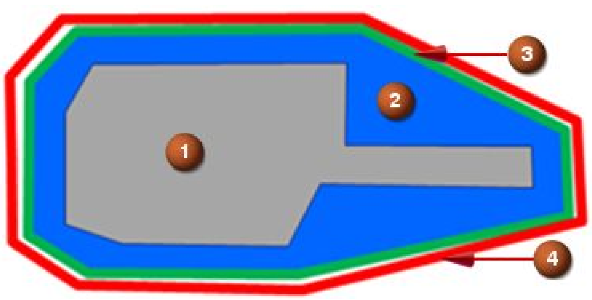
(1) Structural Mesh (grey)

(2) Fluid Mesh (blue)

(3) AML (green)

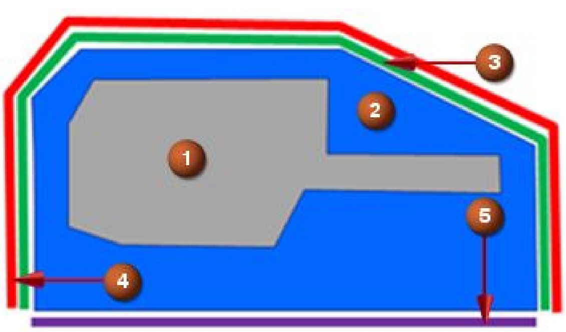
(4) RADSURF (red)

(5) Infinite Plane (purple)



- RADSURF = AML surface
- No infinite plane

Figure 7-9. Acoustic radiation from a machine



- RADSURF (red) = AML
- FE fluid domain modeled to the infinite plane implies AML surface is not closed

Figure 7-10. Acoustic radiation from a machine

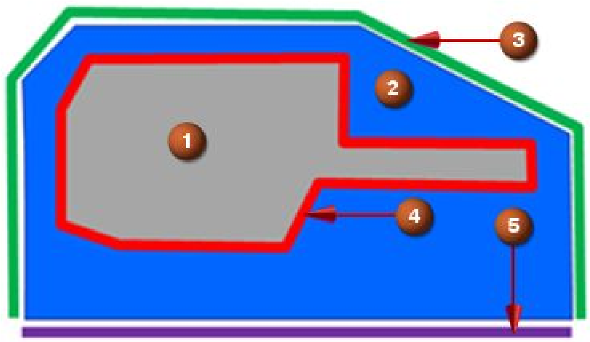


Figure 7-11. Acoustic radiation from a machine

- RADSURF (red) = PHYB (should yield same results as with RADSURF = AML)

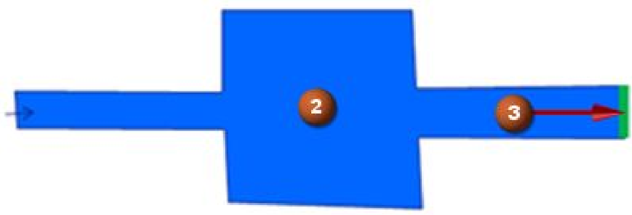


Figure 7-12. Anechoic duct

- Duct ends at the right end and continues indefinitely
- RADSURF is not needed as interest is on what happens inside the duct

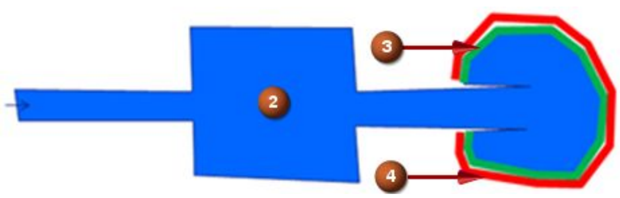


Figure 7-13. Open duct at the right end

- RADSURF (red) = AML
- RADSURF is needed, if the results are required outside FE domain

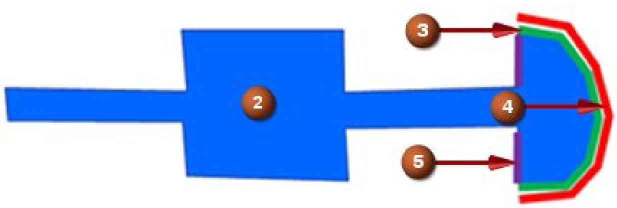
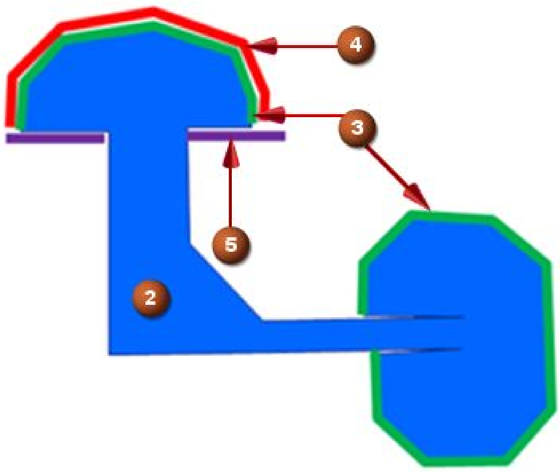


Figure 7-14. Open duct at the right end and ends in a baffle

- Duct ends in a baffle (ground, wall, and so on) acting as an infinite plane



- No RADSURF at the open duct on the right end
- RADSURF is only required if sound is outside the conventional fluid domain

Figure 7-15. Open duct at the right end and top duct ends in a baffle

7.8 Acoustic computation process

The computation for acoustic results in the conventional FE domain and at microphone points involves two steps:

1. The results are computed for only the conventional FE domain.

2. In a post-processing step in Simcenter Nastran, the results at microphone points are computed as follows:

   - Points inside or on the conventional FE domain:

     Nodal results are used to interpolate acoustic pressure results at fluid grid points referenced by microphone elements.

   - Points outside the conventional FE domain:

     Kirchhoff-Helmholtz boundary integral is used in conjunction with the acoustic pressure and the gradients from either the AML surface or from the physical boundary.

     The choice of using the AML boundary or physical boundary is user selectable.

7.9 Gearbox example

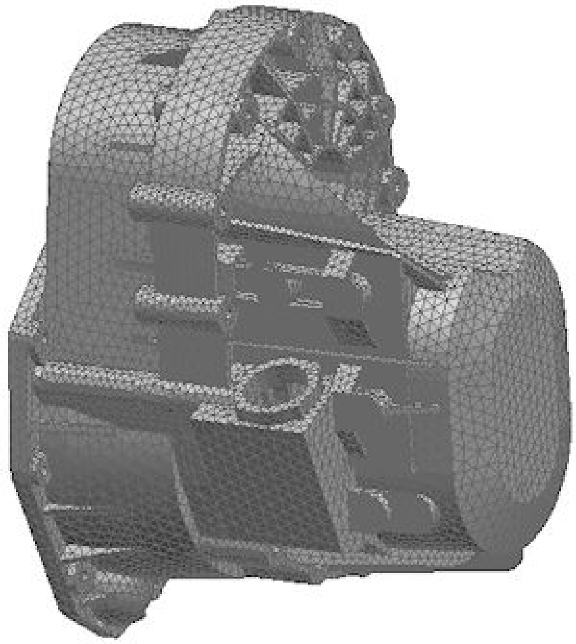


Figure 7-16. Acoustic radiation from a vibrating gearbox

Test problem description

A test problem illustrates the fundamentals of an exterior acoustic analysis using an automatically matched layer. The physical problem represents an initial attempt at design and analysis of an acoustic radiation from a vibrating gearbox. The objective was to carry out a weakly coupled vibro-acoustic response due to the vibrations of the gearbox.

Frequency-dependent forces and moment loads on the structural mesh were applied at the connection between the gearbox shafts and the housing parts, and nodes on the gearbox flange were constrained to simulate a connection between the gearbox and engine. However, only the forces coming from the engine shaft were considered.

Physical description

The ready-to-be-solved structure consisted of a simplified gearbox made from steel without internal shafts and gears as shown by the model in the following figure.

(1) Structural Mesh

(2) Fluid Mesh

(3) AML

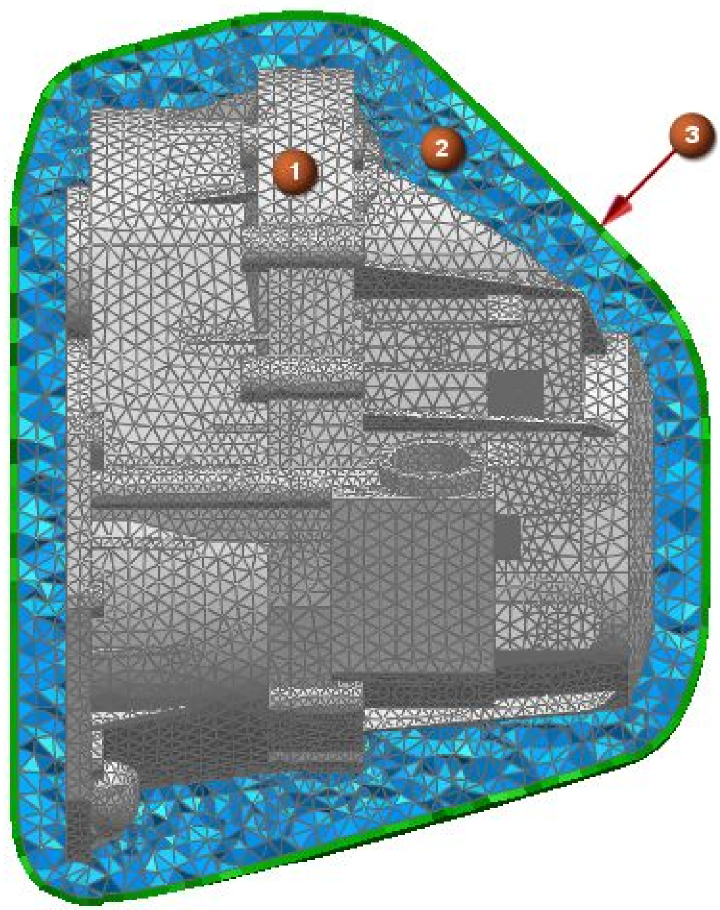


Figure 7-17. Gearbox

The microphone mesh as shown in the following figure was used to place microphones in space (exterior to the convex AML boundary), where acoustic pressure results were requested.

(1) 2D microphone mesh

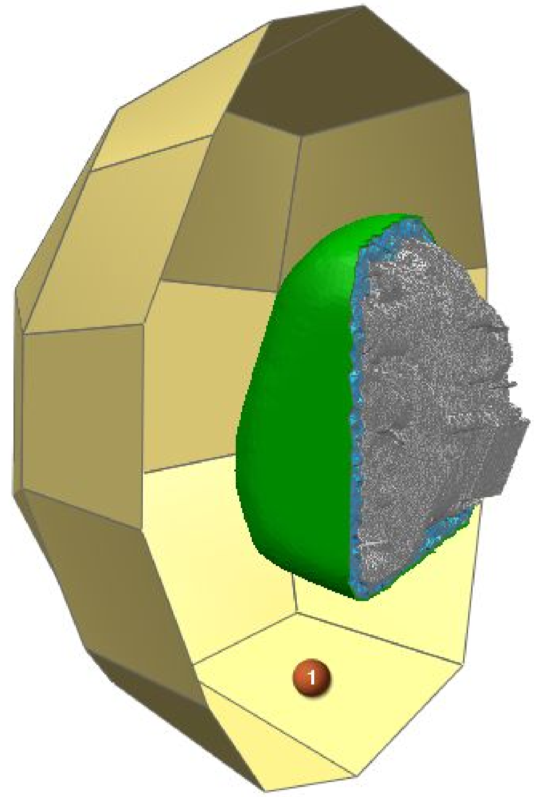


Figure 7-18. Microphone mesh

Some physical properties of the model are listed below.

Material	Properties	
Air	Density: 1.225 kg/m^3	Speed of Sound: 340 m/sec
Steel	Density: 7.829e-006 kg/mm^3	E Modulus: 2.0694e+008 mN/mm^2

Executive and case control

The SKINOUT describer on the FLSCNT case control command was used to request coupling information. Several subcase runs with different RPMs using the same model were made to investigate the effects of increased RPM. The configuration shown below was a direct frequency response SOL 108 solution.

Bulk data

The key data of note in the Bulk Data section are as follows:

- The FREQ1 card was used to apply several forcing frequencies ranging from 1000 to 2000Hz to the model and all subcases.
- The structure and the fluid were meshed with linear tetrahedral solids, and the spherical microphone mesh was modeled with linear quadrilaterals.
- The PMIC property was used as dummy property for the microphone elements.

- The structure was made from steel and the fluid material was air with a constant mass density and speed of sound.
- The CTYPE describer on the ACMODL card, where you can set fluid-structure interface modeling parameters, was used to specify the fluid-structure coupling type.
- The AML was defined on a convex shape boundary using the AMLREG bulk entry. The SID field of the AMLREG bulk entry referenced a BSURFS bulk entry. The element faces on the BSURFS bulk entry were all on fluid elements.

Results from Simcenter Nastran

Frequency, Hz	Acoustic Power [microW] radiated through the AML at 3000 RPM	
	Real	**Imaginary**
1000	7.894135E+00	3.093594E+00
1100	5.114406E+00	9.429158E-01
1200	6.318355E+03	8.360485E+02
1300	4.341093E+01	5.550558E+00
1400	1.160105E+01	4.358886E-01
1500	1.253860E+02	-8.275199E-01
1600	3.911594E+00	-1.572538E-02
1700	3.019994E+02	1.949584E+00
1800	6.132312E+00	4.249183E-01
1900	3.751575E+01	1.073381E+00
2000	4.130656E+01	-1.386058E+00

Excerpt of the input file:

```
$*
$*                   Simcenter Nastran VERSION 11.0
$*
$*         ANALYSIS TYPE: Vibro-Acoustic
$*         SOLUTION TYPE: SOL 108 Direct Frequency Response
$*     SOLVER INPUT FILE: gearbox.dat
$*
$*
$*                 UNITS: mm (milli-newton)
$*                       ... LENGTH : mm
$*                       ... TIME   : sec
$*                       ... MASS   : kilogram (kg)
$*                       ... TEMPERATURE : deg Celsius
$*                       ... FORCE  : milli-newton
$*                       ... THERMAL ENERGY : mN-mm (micro-joule)
$*
NASTRAN FREQVM=1
ID,NASTRAN,gearbox
SOL 108
CEND
$*
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$*
$* CASE CONTROL
```

```
$*
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$*
ECHO = NONE
FLSTCNT ACSYM=YES ACOUT=PEAK ASCOUP=YES PREFDB=2.0-8 SKINOUT=FREEFACE ,
AGGPCH=NO SFEF70=NO
SPC = 181
OUTPUT
ACPOWER(AMLREG=ALL,PRINT) = YES
DISPLACEMENT(PLOT,REAL) = ALL
$*  Step: RPM=3000 - Data Source 1
SUBCASE 1
LABEL = RPM=3000 - Data Source 1
DLOAD = 401
FREQUENCY = 201
$*  Step: RPM=3100 - Data Source 1
SUBCASE 2
LABEL = RPM=3100 - Data Source 1
DLOAD = 402
FREQUENCY = 201
$*  Step: RPM=3200 - Data Source 1
SUBCASE 3
LABEL = RPM=3200 - Data Source 1
DLOAD = 403
FREQUENCY = 201
$*
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$*
$* BULK DATA
$*
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$*
BEGIN BULK
$*
$* SOLUTION CARDS
$*
$*  Modeling Object: Forcing Frequencies - Direct1
FREQ1        201 1000.00100.0000      10
$*
$* PARAM CARDS
$*
PARAM          G  0.0000
PARAM        GFL  0.0000
PARAM      K6ROT100.0000
PARAM     OIBULK     YES
PARAM    OMACHPR     YES
PARAM       POST      -2
PARAM    POSTEXT     YES
PARAM     SPCSTR      NO
PARAM    UNITSYS   MN-MM
$*
$* GRID CARDS
$*

...
```

```
$*
$* ELEMENT CARDS
$*
...

$*  Mesh Collector: Solid(1)
$*  Mesh: 3d_mesh(1)
CTETRA    178102       1   53999   61516   66858   61413
CTETRA    178103       1   53999   61516   61413   54954

...

$*  Mesh Collector: AcousticalMesh_ForTutorial_finished_Microphone Surface(1)
$*  Mesh: AcousticalMesh_ForTutorial_finished_Sphere Primitive Mesh (1)
CQUAD4    469922      19  111668  111672  111673  111669
CQUAD4    469923      19  111668  111669  111685  111684

...

CQUAD4    469974      19  111706  111722  111723  111707
CQUAD4    469975      19  111703  111707  111723  111719
$*
$* PROPERTY CARDS
$*
$*  Property: PSOLID1
PSOLID         1       1       0                           SMECH
$*  Property: AcousticalMesh_ForTutorial_finished_PSOLID - Acoustic Fluid1
PSOLID        18       5                                  PFLUID
$*  Property: AcousticalMesh_ForTutorial_finished_PMIC1
PMIC          19
$*
$* MATERIAL CARDS
$*
$*  Material: Steel
MAT1           12.0694+8        0.2880007.8290-61.1280-5
MATT1          1       1               2               3
TABLEM1        1                       1                +
+        20.00002.0694+8 21.11002.0694+8    ENDT
TABLEM1        2                       1                +
+        20.00000.288000 21.11000.288000 23.89000.288000 37.78000.288000+
+        51.67000.289000 65.56000.289000 79.44000.290000 93.33000.290000+
+       107.22000.291000121.11000.291000135.00000.291000148.89000.292000+
+       162.78000.292000176.67000.293000190.56000.293000204.44000.293000+
+       218.33000.294000232.22000.294000246.11000.294000260.00000.295000+
+       273.89000.295000287.78000.296000301.67000.296000315.56000.296000+
+       329.44000.297000343.33000.297000357.22000.298000371.11000.298000+
+       385.00000.298000398.89000.299000412.78000.299000426.67000.300000+
+       440.56000.300000454.44000.301000468.33000.301000482.22000.302000+
+       496.11000.302000510.00000.303000523.89000.304000537.78000.304000+
+       551.67000.305000565.56000.306000579.44000.307000593.33000.308000+
+       607.22000.309000621.11000.310000635.00000.311000648.89000.312000+
+       662.78000.313000676.67000.314000690.56000.316000704.44000.317000+
+           ENDT
TABLEM1        3                       1                +
+        20.00001.1280-5 93.33001.1790-5107.22001.1880-5121.11001.1970-5+
+       135.00001.2060-5148.89001.2132-5162.78001.2222-5176.67001.2312-5+
```

```
+       190.56001.2402-5204.44001.2492-5218.33001.2582-5232.22001.2672-5+
+       246.11001.2762-5260.00001.2852-5273.89001.2942-5287.78001.3032-5+
+       301.67001.3122-5315.56001.3212-5329.44001.3302-5343.33001.3392-5+
+       357.22001.3464-5371.11001.3554-5385.00001.3644-5398.89001.3734-5+
+       412.78001.3806-5426.67001.3896-5440.56001.3986-5454.44001.4058-5+
+       468.33001.4148-5482.22001.4238-5496.11001.4310-5510.00001.4400-5+
+       523.89001.4472-5537.78001.4544-5551.67001.4616-5565.56001.4688-5+
+       579.44001.4742-5593.33001.4796-5607.22001.4832-5621.11001.4886-5+
+       635.00001.4904-5648.89001.4922-5662.78001.4922-5676.67001.4922-5+
+       690.56001.4904-5704.44001.4886-5718.33001.4850-5732.22001.4796-5+
+       746.11001.4724-5760.00001.4634-5773.89001.4472-5787.78001.4256-5+
+       801.67001.4004-5815.56001.3680-5    ENDT
$*  Material: Air_acoustic
MAT10          5        1.2250-93.4000+5
$*
$* FLUID-STRUCTURE INTERFACE
$*
$*  Modeling Object: Fluid-Structure Interface Modeling Parameters1
ACMODL                                  1.000000         60.0000     REL+
+       0.500000       0                    WEAK
$*
$* LOAD AND CONSTRAINT CARDS
$*
$*  Constraint: Fixed(1)
SPC          181   68337  123456  0.0000
SPC          181   68338  123456  0.0000
SPC          181   68339  123456  0.0000
SPC          181   68340  123456  0.0000
SPC          181   68341  123456  0.0000
SPC          181   68342  123456  0.0000
SPC          181   68343  123456  0.0000
SPC          181   68344  123456  0.0000
SPC          181   68348  123456  0.0000
SPC          181   68349  123456  0.0000
SPC          181   68350  123456  0.0000
SPC          181   68351  123456  0.0000
$*
$*  Simulation Object: Automatically Matched Layer(1)
AMLREG       182       1Automatically Matched Layer(1)               +
+              5     AML       0       0       0
$*  Region: AmlRegion1
BSURFS         1                          254119   79946   81121   79948+
+         254222   79416   79418   79419  254283   78598   78594   78596+
+         254307   78655   78656   78657  254337   82861   80250   80251+

...

$*  Solver Set: Force - Node on Node 68306:DOF1(2)
RLOAD1         2       1  0.0000  0.0000       1       2       0
TABLED1        1                       1                               +
+         0.0000 -1722.36.243900 1699.62 12.4878 1059.11 18.7317 -92.390+
+        24.9756 1370.47 31.2195448.2240 37.4634 1004.60 43.7073 1321.89+
+        49.9512549.0030 56.1951216.8290 62.4390397.1410 68.6829877.1630+
+        74.9268 1173.46 81.1707 1028.60 87.4146993.3700 93.6585 1653.23+
+        99.9024 1945.41106.1460 1133.06112.3900502.3700118.6340 42.9523+
+       124.8780105.0700131.1220167.1880137.3660187.0870143.6100 -70.311+
```

```
...

$*  Load: Force - Node on Node 68347:DOF6(36)
$*  Solver Set: Force - Node on Node 68347:DOF6(179)
MOMENT       179   68347       01.000000  0.0000  0.00001.0000+6
DLOAD        4031.0000001.000000     1221.000000     1241.000000     126+
+       1.000000     1281.000000     1301.000000     1321.000000     134+
+       1.000000     1361.000000     1381.000000     1401.000000     142+
+       1.000000     1441.000000     1461.000000     1481.000000     150+
+       1.000000     1521.000000     1541.000000     1561.000000     158+
+       1.000000     1601.000000     1621.000000     1641.000000     166+
+       1.000000     1681.000000     1701.000000     1721.000000     174+
+       1.000000     1761.000000     1781.000000     180
ENDDATA
```

A comparable, complete input file and a detailed workflow is available from the Simcenter 3D AML acoustic activity in the *Self-Paced Catalog* on the Learning Advantage website. However, the input file in that activity may produce slightly different results.

Chapter 8: FEM adaptive order

8.1 Finite Element Method Adaptive Order (FEMAO)

Finite Element Method Adaptive Order (FEMAO) is a higher-order polynomial method for acoustic and vibro-acoustic analyses. It provides more accurate results and faster solve times by adapting the computational effort to the complexity of the analysis. You can use FEMAO with SOL 108 or SOL 111.

Working principles of the FEMAO method

The FEMAO method adapts each element's basis of shape functions at each frequency to provide an accurate representation of the acoustic pressure inside the element.

The FEMAO method uses:

- Higher-order acoustic shape functions for high frequencies, large elements, or a combination of both.
- Lower-order acoustic shape functions for low frequencies, small elements, or a combination of both.

The order of shape functions in an element can be as high as polynomial order 10. At order 1, an element that uses linear shape functions can span only 1/8 to 1/6 of a wavelength. With the standard (fixed low-order) FEM method, you need 6 to 8 elements per wavelength. The maximum admissible frequency would be *x* Hz. However, at order 10 with FEMAO, you need only about a 6/10 element per wavelength. The maximum admissible frequency would be 10*x* Hz. Thus, if you use FEMAO, you can use the same mesh to compute frequencies more than 10 times higher than with standard FEM.

The following example shows the same 2D mesh with different element orders for two frequencies:

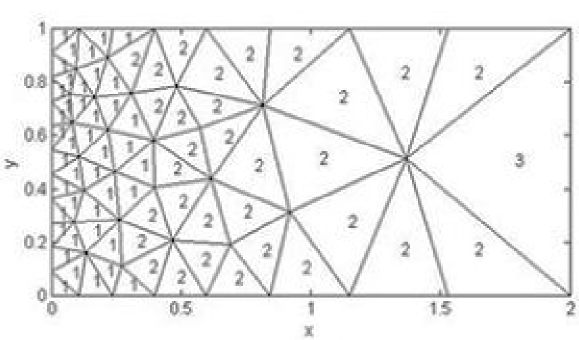


Figure 8-1. f = 100Hz

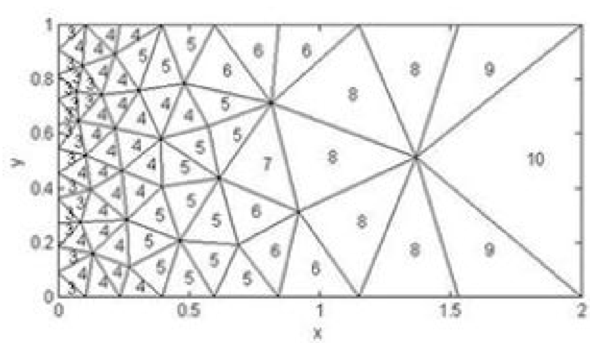


Figure 8-2. f = 1000Hz

When you use FEMAO, you specify the level of accuracy you want in a Simcenter Nastran adaptation rule. To ensure the chosen accuracy, the FEMAO method chooses the optimal polynomial order of shape functions per element. The order adaptation is based on the following parameters:

Parameter	Modeling considerations	Solver behavior
Element size	You can use larger elements for voids and local mesh refinement near geometric boundaries to accurately capture the geometric of the acoustic domain (for example, an engine surface) or where local variations of the fluid properties exist.	FEMAO automatically chooses the order per element by taking the element size into account.
Frequency	You can use larger elements even if you need to capture a high maximum frequency of interest in the mesh.	FEMAO uses lower-order shape functions (lower number DOFs per element) at low frequencies, and higher-order shape functions (higher number DOFs per element) at high frequencies. The solver can return the maximum admissible frequency as a quality result.
Local speed of sound	You can define the speed of sound using the Simcenter Nastran material bulk entries MAT10, MAT10C, MATF10C, and MATPOR.	FEMAO incorporates the specified speed of sound per element to determine the element order (shape functions).
Adaptation rule	You can use the Simcenter Nastran bulk entry ACADAPT to define the adaptation rule, and set the rule to COARSE, STANDARD, or FINE.	The FINE adaptation rule uses more higher-order elements at lower frequencies, and for the same mesh, the COARSE adaptation rule uses more lower-order elements at lower frequencies.
Polynomial order	You can use the Simcenter Nastran bulk entry ACORDER to control the lowest and highest allowed polynomial order for the entire model.	FEMAO automatically chooses 2 for the lowest and 10 for the highest polynomial order unless limits are specified. The default value for the minimum order number is 2 instead of 1 for the following reasons: • Some acoustic analyses require a pressure gradient (spatial derivative of the pressure) next to pressure computation. On the surface of a first-order element mesh, where the mesh represents a set of combined elements, only the computed

Parameter	Modeling considerations	Solver behavior
		pressure field is continuous. The pressure gradient field, which is associated with acoustic particle velocity, is piecewise constant with abrupt changes between the element faces. However, for a surface with free second-order element faces, the pressure gradient is continuous. Thus, the use of second-order element faces results in increased computational accuracy. • Incident acoustic pressure from, for example, acoustic monopole sources outside of the FEM domain, is also more accurately captured with second-order elements.

Shape functions

A large number of polynomial shape functions is required to represent the pressure field within each element. In first-order, linear elements, the number of shape functions and DOFs is the same as the number of physical nodes in the element. In higher-order elements, the number of shape functions and DOFs is much higher. Because FEMAO adapts the order on a per element and per frequency basis, it allows a more efficient representation of the pressure field.

> **Note**
>
> FEMAO uses the hierarchical Lobatto shape functions, which are more flexible and efficient than the Lagrangian shape functions used in standard (low-order) FEM. (For detailed information on shape functions, see Reference.)

The following table outlines typical numbers of shape functions defined on a tetrahedral element as a function of the order.

Order	External (bubble)	Internal (vertex + edge + face)	Total shape functions
1	4	0	4
2	10	0	10
3	20	0	20
4	34	1	35
5	52	4	56
6	74	10	84

Order	External (bubble)	Internal (vertex + edge + face)	Total shape functions
7	100	20	120
8	130	35	165
9	164	56	220
10	202	84	286

Four types of shape functions are shown below on a tetrahedral element:

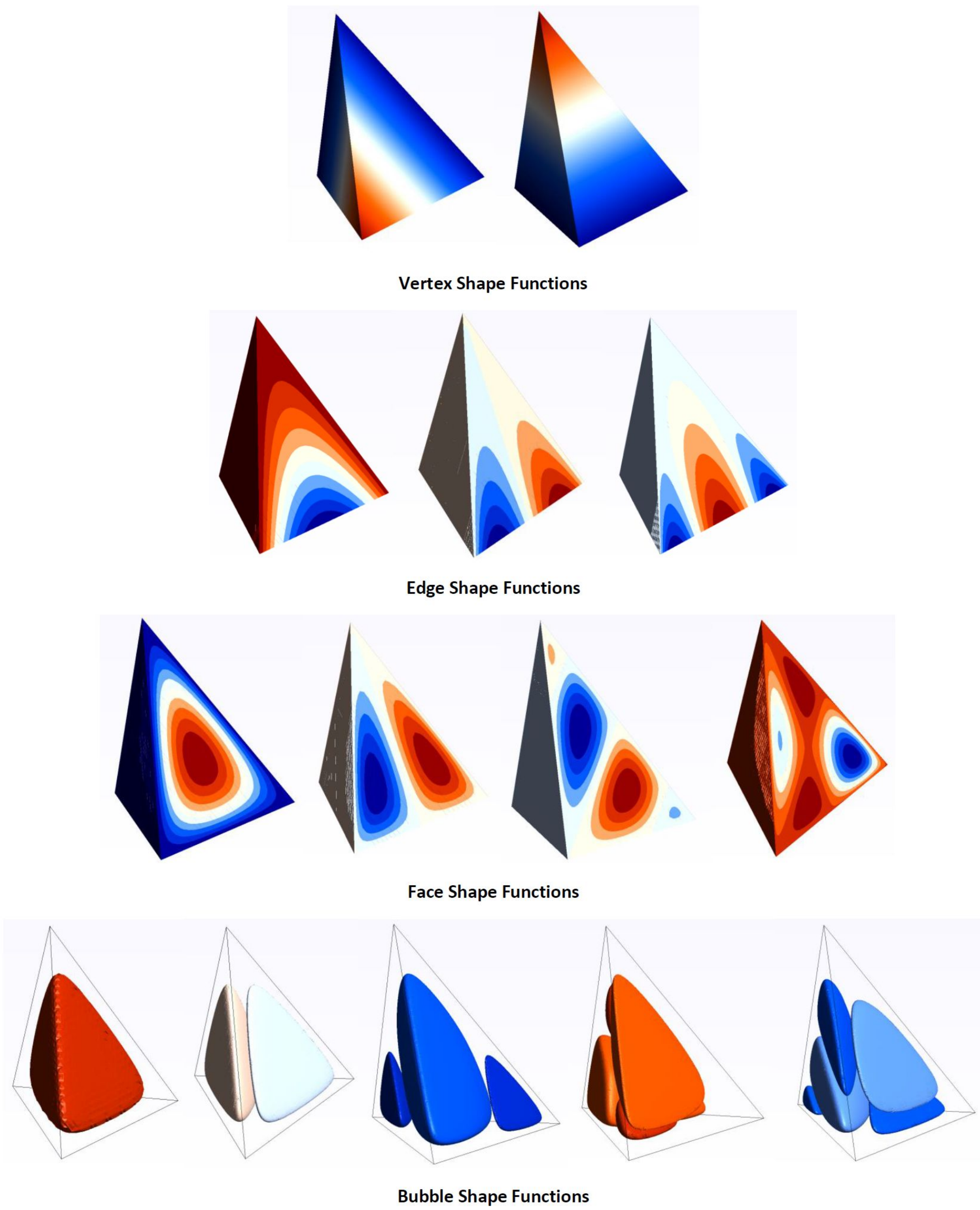


Figure 8-3. Shape function types

FEMAO benefits

- Accuracy through element order

  Standard FEM does not adjust the element order (shape functions). In standard FEM, you typically use a single mesh for the full frequency range of the analysis, which results in an over-discretized model at lower frequencies. At higher frequencies, the standard FEM model becomes under-discretized and less accurate. It may even miss the targeted accuracy because no automatic correction is in place.

  FEMAO, however, adjusts the element order automatically, and the model is represented each time with the right number of DOFs for each frequency to reach the desired accuracy.

- Performance through adaptivity

  An over-discretized model results in longer solve times.

  In FEMAO, however, the order adaptation for each frequency guarantees the optimal model size and consequently the optimal solve time for each frequency. This yields much faster computation times compared to the fixed number of DOFs approach that standard FEM uses.

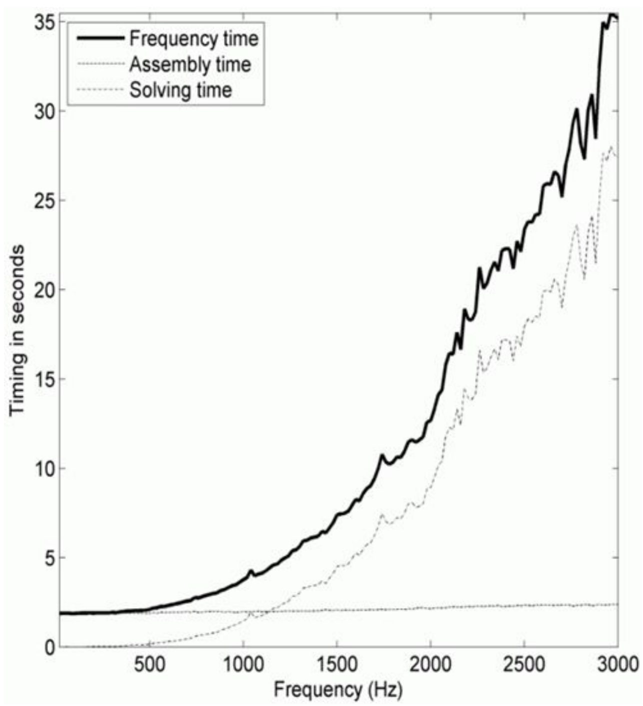


Figure 8-4. Computation by FEMAO as a function of time and frequency

- Performance through shape function efficiency

  The higher-order shape function basis is also more efficient in capturing the acoustic pressure field within each element compared to a standard first-order or second-order elements-based FEM approach.

  For example, if you fill a volume with large elements using higher-order shape functions, which results in more DOFs and shape functions in a single element, and then fill the same volume with small elements using first-order or second-order shape functions as in standard FEM, the total number of DOFs to reach the same accuracy is higher for the standard FEM than for the FEMAO method. Thus, FEMAO becomes more efficient as the frequency increases.

- Pre-processing

  FEMAO allows you to use large elements in the acoustic domain. This results in a lean FEMAO model that contains fewer elements than an equivalent standard FEM model. This also means that you can mesh the model faster in any pre-processor, and a coarser-meshed model improves graphics performance.

  The below images show the differences between a typical FEM mesh, a FEMAO mesh, and a FEMAO mesh with local refinement. For a 1 meter box, all are designed for a frequency of 3000 Hz.

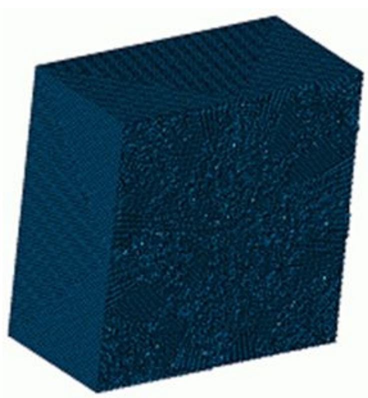


Figure 8-5. FEM mesh

Figure 8-6. FEMAO mesh

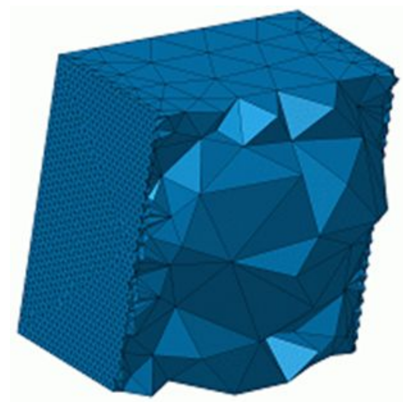


Figure 8-7. FEMAO mesh with local refinement

A coarse mesh may not represent the boundary accurately or provide an accurate solution, so a standard FEM model must use many small elements for an accurate high-frequency solution.

In FEMAO, you should ensure that the spatial variations of the geometry, fluid properties (sound velocity and density), and boundary conditions (velocity and admittance) are well represented by the geometrically linear or the quadratic mesh. For this, you can use local refinement.

Example

Consider a velocity boundary condition, which is applied as acoustic panel velocity on the fluid mesh, with spatial variations of the order of a centimeter required on two faces of the unit box. The FEM mesh in Figure 8-6 typically yields a poor representation. Therefore, you should use mesh local refinements as shown in Figure 8-7.

However, if the vibrations originate from a meshed structural panel next to the free fluid faces, the FEMAO mesh in Figure 8-6 with coarse elements at the fluid structure interface is supported. In this case, the coupling matrix also includes the higher-order DOFs of the coarse fluid elements. The structural mesh must be discretized finely enough to capture the spatial variations in the velocity boundary condition it imposes on the fluid.

Maximum frequency

- The maximum frequency of a FEMAO mesh is reached when the order of any element P_e^f is greater than 10. If you assume 8 elements per wavelength for linear elements, the edge size to use is

  h < (1/8) * (c_0 / f_{max})

  For higher-order elements this becomes

  h < (P_e^f / 8) * (c_0 / f_{max})

  This means that the maximum frequency can be estimated by

$$f_{\max} < \frac{P_e^f c_0}{8h}$$

Equation 8-1.

where:

- o P_e^f is the order of the element.
- o c_0 is the speed of sound in the ambient medium.
- o h is the element dimension.

However, the Lobatto shape function basis of FEMAO is more efficient compared to standard FEM. This means that at higher orders, fewer DOFs can be used per wavelength to represent the acoustic field accurately.

When the ACADAPT adaptation rule is:

- o Coarse, 2.2 times fewer DOFs can be used per wavelength.
- o Standard, 2 times fewer DOFs can be used per wavelength.
- o Fine, 1.6 times fewer DOFs can be used per wavelength.

The maximum frequency criterion therefore becomes

$$f_{\max,\text{coarse}} = \frac{c_0}{0.36h}\;; \qquad f_{\max,\text{standard}} = \frac{c_0}{0.41h}\;; \qquad f_{\max,\text{fine}} = \frac{c_0}{0.51h}$$

Equation 8-2.

Bulk entries

When you use FEMAO, make sure that your model uses the correct bulk entries:

- For a simple acoustic source, use the acoustic monopole source ACPOLE1. Do not use ACSRCE.
- For a damping and stiffness absorber, use CAABSF with the bulk entry PAABSF1. Do not use the CAABSF/PAABSF combination.

- For gluing features, use the ACTRAD bulk entry to enforce acoustic continuity across two surfaces of acoustic meshes. Do not use BGSET and BGADD.

> **Note**
>
> The software ignores unsupported bulk entries.

Fluid to structure interaction

The FEMAO method supports the coupling of a structure to a fluid. In an analysis where you request the computation of vibro-acoustic transfer vectors (VATVs), FEMAO only supports strong coupling. Couplings in an analysis without VATV computation may be weak or strong.

> **Note**
>
> For a SOL 108 vibro-acoustic strong coupling solution, the software can also solve input files where elements are constrained at fluid-structure coupling surfaces.

For more information on coupling, see Defining the fluid-structure interface boundary condition.

For more information on VATVs, see Vibro-Acoustic Transfer Vector (VATV).

Vibro-acoustic output requests

Available outputs with fluid-structure coupling and FEMAO are:

- Pressure, velocity, intensity, acoustic power, and transmission loss.
- Panel pressure and power, modal pressure and power, and grid pressure contribution as listed in the table.

Vibro-acoustic output requests computed by FEMAO						
		Panel contribution		Modal contribution		C
Solution	Fluid-structure Interaction	Pressure	Power	Pressure	Power	
SOL 108	Strong			*	*	
SOL 108	Weak	Yes	Yes	*	*	
SOL 111	Strong	Yes	Yes	Yes	Yes	
SOL 111	Weak	Yes	Yes	Yes	Yes	
* SOL 108 does not use mode shapes to compute a response. It computes structural response at discrete ex by solving a set of coupled matrix equations using complex algebra. Thus, SOL 108 cannot compute any m						

> **Note**
>
> - o Panel pressure contribution with the exception of the number of structural panels (TOPP describer).
> - o Modal pressure contribution with the exception of the number of structural modes (TOPS describer). The contributions are computed on all panels. You cannot specify a subset of panels.
> - o The solver computes the quantity power in the selected microphone element.
> - o The solver does not convert SORT1 to SORT2 data and vice versa. However, the solver sorts data based on subcase, frequency, and so on.

Performance checking

The solver writes performance indicators, such as computation time (in seconds) and memory (in GB) per frequency per subcase to the .log file and OUTPUT2 (.op2) file for post-processing in a pre/post software, such as Simcenter 3D.

Quality checking

For quality assessment, element order per element results from an acoustic analysis will be written to the .op2 file.

Optionally, you can use the ACORDCHK case control command to validate the quality of elements before you proceed with the computation-intensive part of the solution. When you submit an acoustic or vibro-acoustic solution with the ACORDCHK case control command, the solver:

1. Outputs the maximum valid frequency and element order information.
2. Terminates the solution before the computation-intensive frequency response computation.

> **Note**
>
> In a FEMAO vibro-acoustic analysis, when the SKINOUT = STOP describer setting is specified on the FLSTCNT case control command, despite of ACORDCHK = STOP the software still writes the coupling quality information to the OP2 file before terminating the solution.

Workflow

1. Create a SOL 108 acoustic or vibro-acoustic solution.
2. Set the following bulk entries:
   a. ACADAPT to invoke the FEMAO solution and to define the adaptation (refinement) rule.

> **Note**
>
> The RULE parameter ensures that the numerical error stays within an acceptable range. This allows you to perform and compare multiple solutions with the same mesh, but different RULE values.

b. ACORDER to specify the lowest and highest allowed polynomial order for FEMAO.

Reference

Efficient implementation of high-order finite elements for Helmholtz problems, Hadrien Bériot, Albert Prinn, and Gwénaël Gabard, International Journal for Numerical Methods in Engineering, 2015.

Chapter 9: Transfer vectors

9.1 Acoustic Transfer Vector (ATV)

An Acoustic transfer vector (ATV) is a frequency-dependent acoustical representation of fluid volume that you can use in SOL 108 and 111 solutions to solve external acoustics problems. ATVs relate the response at microphone points and 2D microphone elements to excitation at the fluid-structure (coupling) interface. The primary benefit of ATVs is computational efficiency.

ATVs are similar to output transformation matrices (OTMs). They both use a matrix to relate the response at specific spatial locations or over specific regions to the excitation at specific locations or from specific regions.

For more information on OTMs, see the Output Transformation Matrices section of the *Simcenter Nastran Basic Dynamic Analysis User's Guide*.

For example, suppose you want to examine how the acoustic response at specific locations to structural excitation varies with respect to structural materials, loads, and so on. Rather than solving the entire acoustics problem repeatedly, you can solve the acoustics problem once to create an ATV, and then use the ATV repeatedly to examine how the response varies.

To use the ATV capability, two Simcenter Nastran runs are required.

- The first run is the ATV computation run. During this run, the software creates and writes the matrix representation of the ATV to an OP2 file. The ATV computation run must be SOL 108.
- The second run is the ATV response run. During this run, the software retrieves and uses the matrix representation of the ATV to calculate the acoustic response at microphone points and elements to the excitation of the structural portion of the model. The ATV response run can be either SOL 108 or 111 and can use one ATV only.

> **Note**
>
> Only a SOL 108 ATV response run supports an empty A-set. For example, the A-set is empty when you constrain the structure in all degrees of freedom.

In the ATV computation run:

- Use the ATVOUT case control command to trigger the creation of the ATV, reference the ATVFS bulk entry that specifies the coupling interface, and reference the SET case control command that specifies the microphone elements that comprise 2D microphone meshes.
- Use the ATVFS bulk entry to specify the BSURFS bulk entries that define the coupling interface.
- Use a FREQi bulk entry to specify the frequencies at which the software calculates the matrix representation of the ATV.

Note

During the ATV response run, the software interpolates ATV data over the frequency range defined by the FREQi bulk entry. Because the software does not extrapolate ATV data, a fatal error occurs if the frequency is out of the range defined by the FREQi bulk entry.

In the ATV response run:

- Use the ATVBULK bulk entry to select the OP2 file that contains the matrix representation of the ATV and to offset the element, grid point, and property IDs in the ATV to ensure that they are unique from those in the structural portion of the model.
- Use the ATVBK bulk entry to compute the PANCON, GRDCON, and MODCON contributions. You use ATVBK to select the OP2 file that contains the matrix representation of the ATV and to offset the element, grid point, and property IDs in the ATV to ensure that they are unique from those in the structural portion of the model.
- Use an ACMODL bulk entry to specify the coupling interface parameters.
- Use the PRESSURE case control command to request acoustic pressure results at microphone points.
- Use the PANCON case control command to request the acoustic pressure at microphone points that are attributable to the vibration of selected structural panels.
    - Use PANEL bulk entries to define the structural panels.
    - Use SETMC case control commands to specify the microphone points.

      For microphone points, specify RTYPE = PRES.
- Use the GRDCON case control command to request the response at microphone points to vibration at grids in structural panels.
- Use the MODCON case control command to request the contribution of modes to the acoustic pressure at microphone points. This is valid for a SOL 111 ATV response run only.
    - Use SETMC case control commands to specify the microphone points.

      For microphone points, specify RTYPE = PRES.

When you use an ATV, acoustic particle velocity and acoustic intensity results cannot be computed, and the ACINTENSITY and ACVELOCITY case control commands are ignored if they are specified.

9.2 Vibro-Acoustic Transfer Vector (VATV)

The Vibro-acoustic transfer vectors (VATVs) let you efficiently compute vibro-acoustic pressure due to aerodynamic or acoustic loads on flexible structures in the context of two-way (strong) coupled vibro-acoustic problems. VATVs contain matrices that link the structural grids of a flexible structure with microphone points or 2D microphone elements.

The VATVs are independent of the loading and you can reuse them for various loads in multiple vibro-acoustic problems as long the structure, acoustic fluid, and the location of microphones remain the same.

For example, suppose you want to examine how the acoustic response at specific locations to structural excitation varies with respect to loads. Rather than solving the entire acoustics problem repeatedly, you can solve the vibro-acoustics problem once to create a VATV, and then use the VATV repeatedly to examine how the response varies.

Using a VATV in your vibro-acoustic problems involves two runs:

1. The solver computes the VATV using SOL 108 or SOL 111. During this run, Simcenter Nastran writes the VATV matrices, which stores the results in pressure format for fluid grid points referenced by microphone elements and normal nodal force format for structural grids, to a *name*_vatv.op2 file.

2. You use the computed VATV in a frequency response or random analysis. During this run, the solver retrieves and uses the matrix representation of the VATV to calculate the acoustic response at microphone points and elements to the excitation of the structural portion of the model. The VATV response analysis can be only SOL 108 and can use one VATV only.

In the VATV computation run:

- Use the VATVOUT case control command to trigger the creation of the VATV. The command references a VATVFS bulk entry and specifies the Fortran unit number for the .op2 file to which the VATV results are written.

- Use the VATVFS bulk entry to specify the BSURF or BSURFS bulk entries that define the structural element faces of a pressure boundary. The structural element faces describe the free surface region.

- Use an ACMODL bulk entry to specify the coupling interface parameters.

- Use a FREQi bulk entry to specify a frequency range at which the software calculates the matrix representation of the VATV.

Note

During the VATV response run, the software derives the exact frequencies from the VATV matrix computed frequencies in the specified frequency range.

In the VATV response run:

- Use the VATVBK bulk entry to select the .op2 file that contains the matrix representation of the VATV.

- Use the FREQV bulk entry to specify a frequency range of interest.

- Use the PRESSURE case control command to request acoustic pressure at microphone points.

- Use the ACPOWER case control command to request acoustic power at 2D microphone elements.

- Use the PANCON case control command to request the acoustic pressure or power at microphone points that are attributable to the vibration of selected structural panels.
  - Use PANEL bulk entries to define the structural panels.
  - Use SETMC case control commands to specify the microphone points and response type.

    Specify RTYPE = PRES for acoustic pressure.
- Use the GRDCON case control command to request the response at microphone points to vibration at grids in structural panels.

When you use a VATV, acoustic particle velocity and acoustic intensity results cannot be computed, and the ACINTENSITY and ACVELOCITY case control commands are ignored if they are specified.

For more information, see the VATVOUT case control command, and the VATVFS, FREQV, and VATVBK bulk entries.

Chapter 10: Other considerations

10.1 Superelements

Superelements may be used with the following restrictions:

1. A superelement may contain either fluid or structural points, but not both. The residual structure may contain both.

2. The grid points at the fluid-structure interface may be assigned to the residual structure only. This requires the specification of q-set points using the SEQSETi and EIGR or EIGRL Bulk Data entries and the METHOD case control command.

3. Superelements must be linear. That is, any non-linear or frequency-dependent effects must be in the residual. Consequently, the presence of the Exterior acoustics using automatically matched layer or elements that reference porous materials must be defined in the residual. This applies to the AML boundary condition, CAABSF absorber element (frequency-dependent), and elements that reference MAT10 material definition pointing to frequency-dependent tables.

Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing, and electronics design meet tomorrow. Our solutions help companies of all sizes create and leverage digital twins that provide organizations with new insights, opportunities, and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit siemens.com/software or follow us on LinkedIn, Twitter, Facebook, and Instagram.

Siemens Digital Industries Software —
Where today meets tomorrow.