SIEMENS
Ingenuity for life

Simcenter 3D
Meshing for Acoustics

Delivering a high-performance acoustic meshing solution for complex geometries

Benefits
- Accelerate the interior and exterior acoustic meshing process for complex geometries
- Generate an acoustic simulation model from an existing structural mesh model or associative CAD geometry

Summary
The Simcenter™ 3D Meshing for Acoustics application helps you perform acoustic meshing for finite element method (FEM) or boundary element method (BEM) acoustics analysis. Simcenter 3D, which is part of the Simcenter portfolio, contains all the necessary tools to create an acoustic fluid mesh in a user-friendly way from an existing structural mesh or computer-aided design (CAD) geometry.

Surface wrapper for acoustics
Simcenter 3D Meshing for Acoustics software includes the required functionality to efficiently create a fluid domain geometry by surface wrapping a structural mesh and/or a complete CAD assembly model. The solution can be used to prepare acoustic meshes for both BEM and FEM acoustics analysis.

For FEM exterior acoustics, surface wrapping can be used to create the inner boundary of the fluid domain. For FEM interior acoustics, surface wrapping can be used to start creating a cavity mesh (for example, an interior of a car body). Surface wrap resolution, automatic refinement and output element size can be specified for greater flexibility in wrapping the input mesh and generating the output mesh. The enhanced cavity selection tool

www.siemens.com/simcenter3d
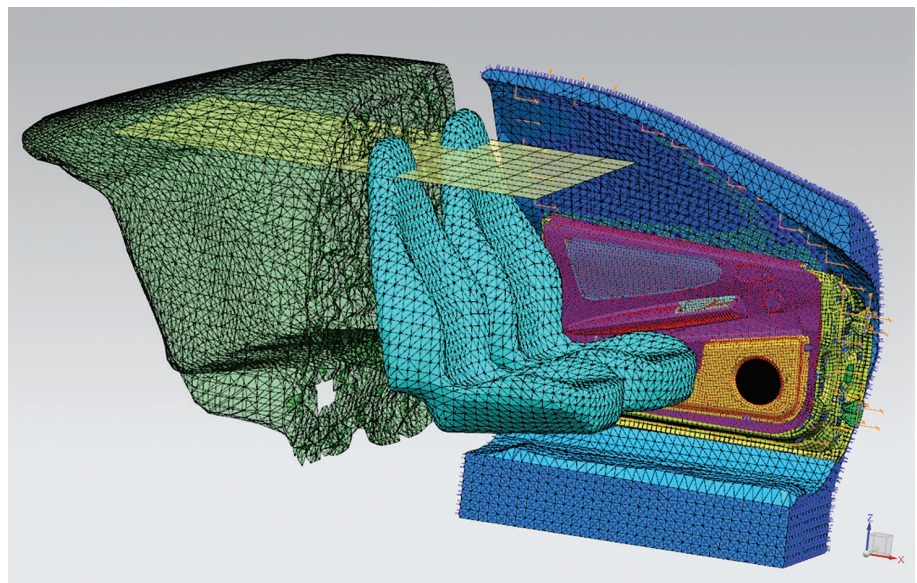
Simcenter 3D Meshing for Acoustics

enables you to detect and list cavities in the model by specifying additional parameters such as minimum size and minimum volume.

Input models can be elements of a structural mesh, polygon geometry of a CAD model or both.

Output models can be a:

- Polygon body to be meshed with Simcenter 3D meshing capabilities

- Polygon body together with its associated 2D mesh (triangular elements)

- 2D mesh ready to be filled by volume elements

Generating a wrap polygon geometry using a CAD model allows you to fully leverage the design-associativity capability of Simcenter 3D.

Local constraints enable the wrapper algorithm to be used to coarsen or refine the mesh size locally.
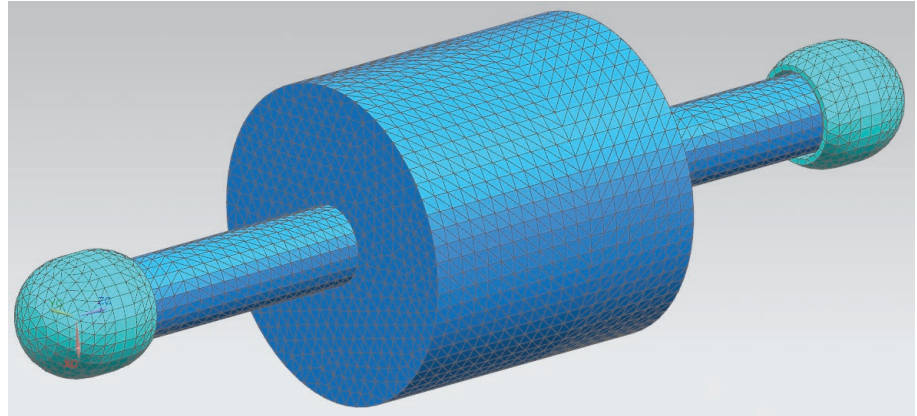
Convex mesher
You can use the convex mesher feature to create a convex surface mesh around an existing part. This convex mesh is used for FEM exterior acoustics that uses automatically matched layer (AML) technology at the exterior boundary of the fluid domain:

- Define input mesh on a 2D surface mesh, a skin mesh of a 3D structural mesh, or a wrapped mesh resulting from surface wrapping of the structural model
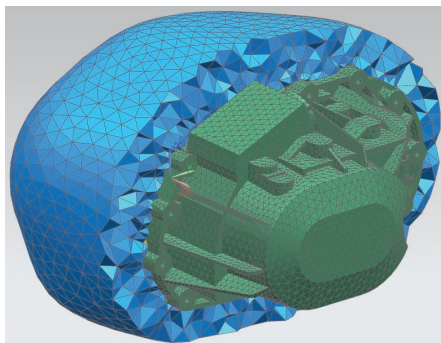
Convex mesh.

Open duct mesh.

- Define an offset distance from the input mesh to create the void to be filled later with 3D fluid elements

- Define infinite planes. In this case, the convex mesh includes part of the infinite plane and the remainder of the convex mesh surface makes a 90-degree angle with the infinite plane

- Define the size of the convex mesh elements directly or through computation based on the maximum frequency

Additional functionalities for efficient meshing
Simcenter 3D Meshing for Acoustics provides a set of additional meshing tools to ease the preprocessing step in the workflow to generate the acoustic mesh.

Thicken mesh
The thicken mesh feature provides you with the ability to create 2D mesh with thickness. The thickness parameter specified determines the width of the output 2D mesh, which encompasses the input mesh.

3D hybrid mesh
Using a structured mesh for acoustic computation is made possible by the 3D hybrid mesh functionality. This tool allows you to use tetrahedron elements
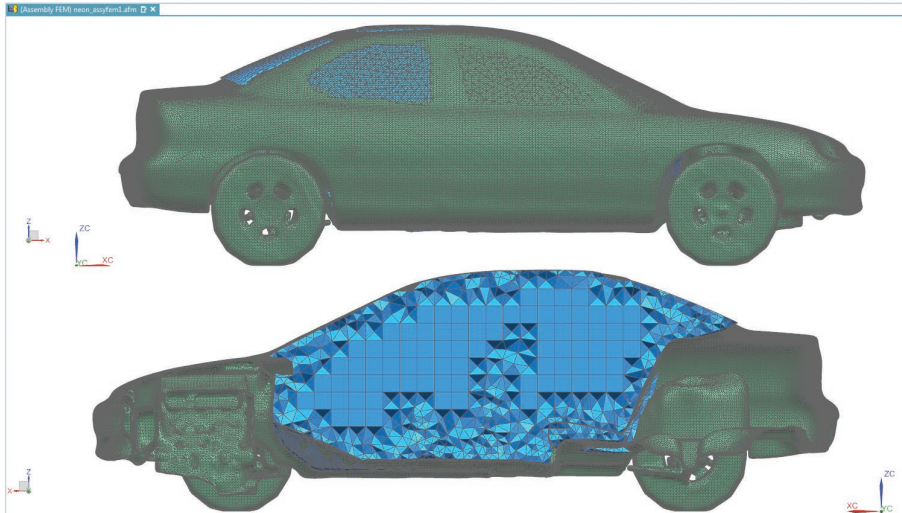
at the boundary of the acoustic mesh and makes use of structured hexahedron elements inside the mesh. Several parameters to control the output mesh are available, such as the element size and the number of layers trimmed at the wall.

Open duct mesh
The simulation of duct acoustic problems and the knowledge of sound radiation from a duct outlet are specific topics that necessitate an adequate tool for the meshing at the duct exit. The open duct mesh functionality is suitable for automatically generating a 2D mesh that will fold back around the duct outlet. The input element edges typically represent the duct outlet, and you can specify the mesh size and the radius of the output open duct surface. A convex mesh is automatically generated in the output 2D mesh as a support for creation of a FEM mesh with AML region in the vicinity of the open duct end.

Ribs removal
The simplification of structural meshes is necessary in the process of generating the acoustic mesh. Small features such as ribs in the structural mesh do not impact the acoustic propagation and may be suppressed from the acoustic mesh. The remove ribs capability allows you to select 2D elements and
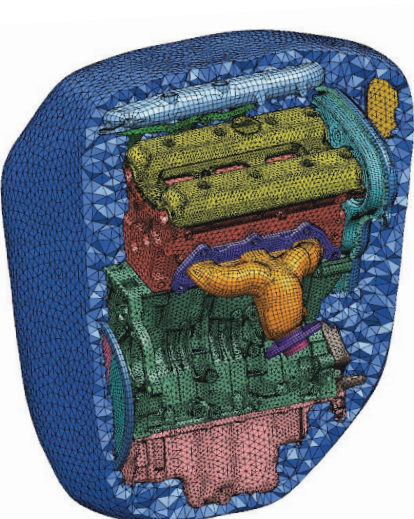
Hybrid mesh.

Check acoustic mesh quality

A series of tools are available to check the quality and consistency of the acoustic mesh.

- You can specify parameters (aspect ratio, skew angle, etc.) and run a global element quality check to detect elements with poor geometrical and topological properties

- Normalizes the mesh to verify the uniformity of the orientation of the elements (2D and 3D) in the mesh

- Lists the duplicate nodes and elements to avoid redundancies in the mesh

specify detection criteria such as maximum elements in the rib and maximum rib area.

The Simcenter 3D Engineering Desktop module provides several acoustic meshing features, including:

Surface coating

The surface coat functionality is used to generate 2D meshes by extracting free faces from 3D meshes or groups of element faces. This is suitable for generating the acoustic envelope mesh from the structural mesh.

Face from mesh

Re-meshing existing meshes to make them coarser or finer is simplified by the face from mesh functionality. This allows for the reworking of an existing structural mesh, which will be used to generate the FEM acoustic mesh. A coarser mesh can also be generated to prepare for the use of the finite element method adaptive order (FEMAO) solver, which allows you to optimize computational efficiency.

Input models can be 2D meshes, 3D element free faces and 2D elements.

Output models are:

- Polygon body

- Editable 2D meshes associated with the polygon body