SIEMENS
Ingenuity for life

Simcenter 3D environment for Simcenter Nastran

Pre- and postprocessing analysis models in Simcenter 3D for the Simcenter Nastran solver

Benefits

- Enables engineers using Simcenter 3D to generate finite element models for Simcenter Nastran
- Simplifies the Simcenter Nastran modeling process by enabling engineers to create analysis models based on geometry or legacy Simcenter Nastran data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D
- Immerses engineers in the Simcenter Nastran environment through familiar Simcenter Nastran terminology and extensive support of Simcenter Nastran-specific elements and entities

Summary

The Simcenter 3D environment for Simcenter Nastran software enables engineers to build finite element models, define solution parameters and view the solution results for the Simcenter Nastran solver. The environment immerses engineers with familiar Simcenter Nastran language for element definitions, loads and boundary conditions, solution parameters and other common Simcenter Nastran nomenclature. In addition to model definition capabilities, the Simcenter Nastran environment provides bi-directional import/export capabilities that enable you to import current or legacy Simcenter Nastran bulk data files and results as well as export run-ready Simcenter Nastran data files.

Simcenter 3D's powerful geometry editing and meshing capabilities are ideal for pre- and postprocessing models for Simcenter Nastran. Simcenter 3D simplifies the modeling process by integrating high-end finite element modeling tools with world-class geometry capabilities that assist you with developing analysis models faster than with traditional CAE preprocessors.

Adding the Simcenter Nastran environment to Simcenter 3D enables you to build Simcenter Nastran run-ready bulk data decks, so little or no intermediate processing is ever needed. In addition to building Simcenter Nastran models, the Simcenter Nastran environment imports solution results directly from Simcenter Nastran binary results files into Simcenter 3D for postprocessing. The environment delivers import/export capabilities so you can import Simcenter Nastran data decks into Simcenter 3D for modification and then export run-ready decks for solution.

Import/export Simcenter Nastran models

- Import/export complete Simcenter Nastran finite element models including bulk data as well as executive and case controls
- Import model information from either bulk data decks or binary output2 files

Create Simcenter Nastran models in Simcenter 3D

- Create complete run-ready Simcenter Nastran decks including executive and case controls, bulk data
- The Simcenter Nastran environment supports solutions 101, 103, 105, 106, 107, 108, 109, 110, 111, 112, 129, 153, 159, 200, 401, 402, 601/106, 601/129 and 701

Elements and other entities

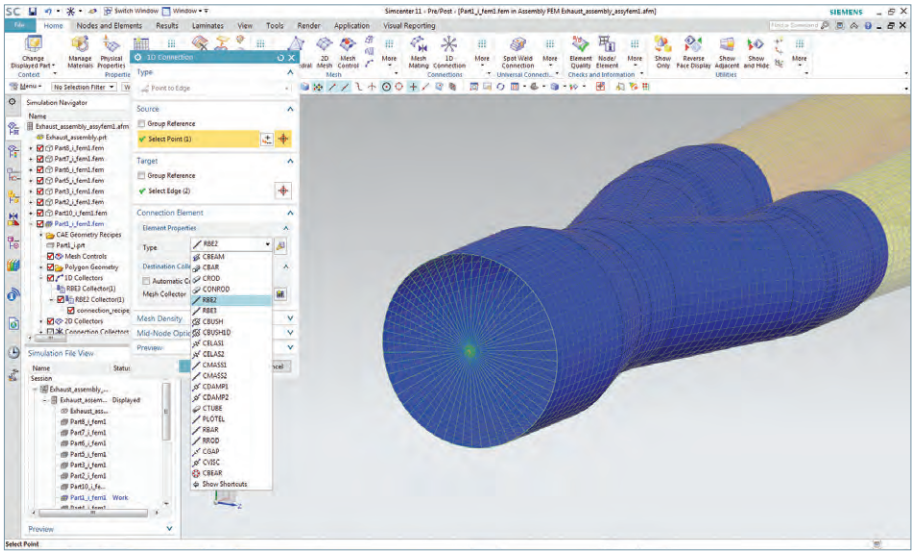
- A wide variety of elements and other model entities are supported.
- Lumped mass, spring, rigid elements
- Axisymmetric solid elements
- Rod, beam and bar elements
- Plane stress and plane strain elements
- Shell and solid elements
- Permanent single-point constraints

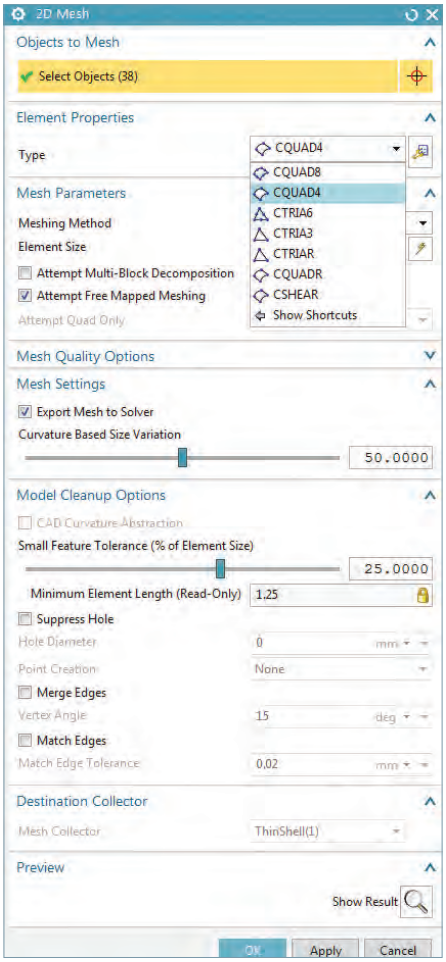
Simcenter 3D environment for Simcenter Nastran

A complete list of Simcenter Nastran import/export entity support is provided in the Simcenter 3D online help.

Loads and boundary conditions
Loads and boundary conditions for structural and thermal analysis are supported.

- Nodal, elemental and geometry-based structural loads
- Beam-concentrated and distributed loads
- Bolt pre-load on solid elements
- Gravity, rotational velocities and acceleration loads
- Nodal, elemental and geometry-based thermal heat loads
- Nodal restraints and temperatures





- Traction loads
- Contact regions and sets
- Time and temperature variations
- Subcase manager to easily manage the loads used in each subcase

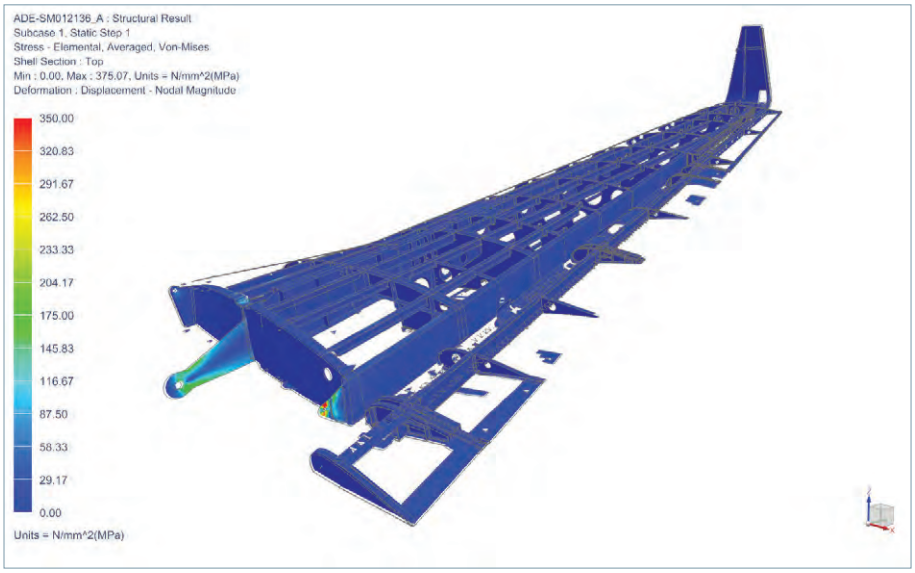
Automatic connection mesh support
Simcenter 3D provides a number of solver-supported methods for connecting different meshes together.

- The mesh mating condition connects individual 2D or 3D meshes together at a specified interface.
- The edge-face connection defines the connection between a set of edges and a set of faces. You can use this feature whenever there are meshes to be connected in T-junction configuration; for example, fins or stiffeners attached to surfaces.
- Edge-to-edge glue connection to define glue connections between selected polygon or element edges
- Weld mesh locates and automates the recognition of weld features (connections) and then automatically creates their FE model representations, including consideration for mid-surfaces. You can use weld mesh to create weld elements (1D mesh) from weld features.
- Contact mesh creates point-to-point contact between two edges or a portion of two edges defined by limiting points.
- Surface contact mesh creates and defines contact elements between two selected faces of a solid or between different components.

Acoustics and vibro-acoustic solution support
The Simcenter 3D environment for Simcenter Nastran allows you to set up fully coupled vibro-acoustic FEM analysis of both interior and exterior acoustic problems for the Simcenter Nastran solver, defined as direct (SOL108) of modal (SOL111) frequency response solutions. The following acoustic-specific capabilities supported in the Simcenter 3D environment for Simcenter Nastran:

- Automatically Matched Layer (AML) simulation object to simulate a non-reflecting boundary condition.
- Finite Element Method Adaptive Order (FEMAO) acoustic solver for faster acoustic and vibro-acoustic simulations with more control of accuracy.
- Microphone meshes to define the locations where acoustic results will be computed.

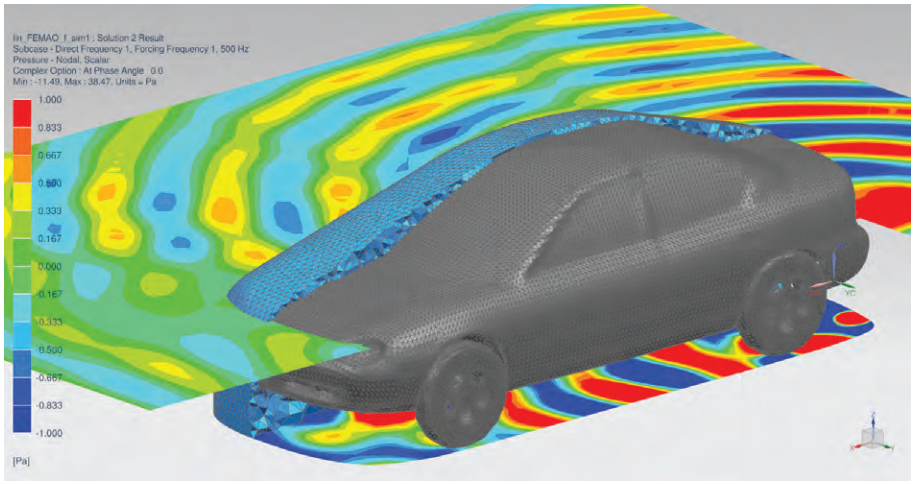
- Acoustic sources and loads such as monopoles, dipoles, plane waves, distributed plane waves, and panel normal velocity.
- Transfer admittance simulation object to capture acoustic transfer relation between two fluids separated by a perforated wall / sheet.
- Rigid and limp porous material modeling using these material models: Craggs (rigid frame), Delany, Bazely and Miki (for limp and rigid frames), and Johnson, Champoux and Allard (for limp and rigid frames).
- Pressure, acoustic particle velocity and acoustic intensity output request at any microphone location.
- Radiated acoustic output power output request.
- Case control output request cards to directly retrieve transmission loss results
- Reference microphone locations as response locations for panel- and modal-contribution analysis
- Equivalent Radiated Power (ERP) output request



Compatibility
The Simcenter Nastran environment is compatible with the following Simcenter Nastran releases:

- Simcenter Nastran v12 or earlier

Supported hardware/OS
The Simcenter Nastran environment is an add-on module within the Simcenter 3D suite. It requires a license of Simcenter 3D Engineering Desktop or Simcenter 3D Structures as a prerequisite. It is available on all Simcenter 3D supported hardware/OS platforms (Windows and Linux).



Siemens Digital Industries Software
siemens.com/plm

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3333

© 2019 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. Femap, HEEDS, Simcenter, Simcenter 3D, Simcenter Amesim, Simcenter FLOEFD, Simcenter Flomaster, Simcenter Flotherm, Simcenter MAGNET, Simcenter Motorsolve, Simcenter Samcef, Simcenter SCADAS, Simcenter STAR-CCM+, Simcenter Soundbrush, Simcenter Sound Camera, Simcenter Testlab, Simcenter Testxpress and STAR-CD are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries or affiliates in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders.
10665-C16 5/19 H