SIEMENS
Ingenuity for life

# Simcenter 3D environment for ANSYS

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

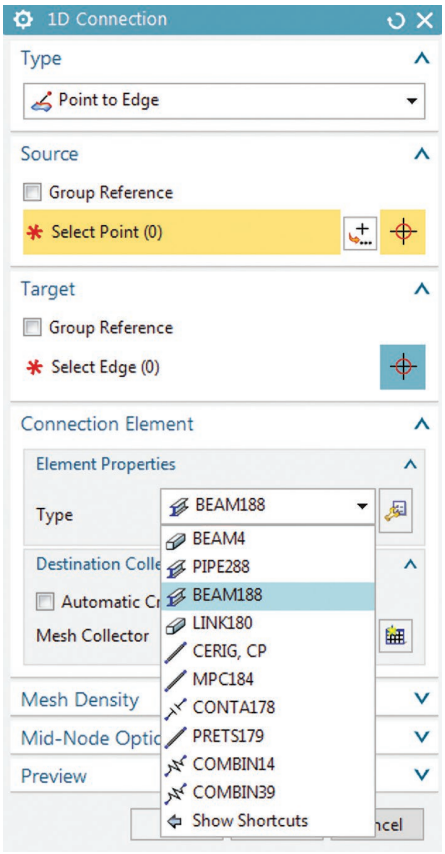
**Benefits**

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

**Summary**

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

Simcenter 3D's powerful geometry editing and meshing capabilities are ideal for pre- and postprocessing models for ANSYS. Simcenter 3D simplifies the modeling process by integrating high-end analyst modeling tools with world-class geometry capabilities that assist you with developing analysis models faster than with traditional CAE preprocessors. Adding the ANSYS environment to Simcenter 3D enables you to build ANSYS run-ready input data files, so little or no intermediate processing is ever needed. In addition to building ANSYS models, the Simcenter 3D ANSYS environment imports solution results directly from ANSYS results files into Simcenter 3D for postprocessing. The environment delivers import/export capabilities so you can import ANSYS data decks into Simcenter 3D for modification and then export run-ready decks for solution.

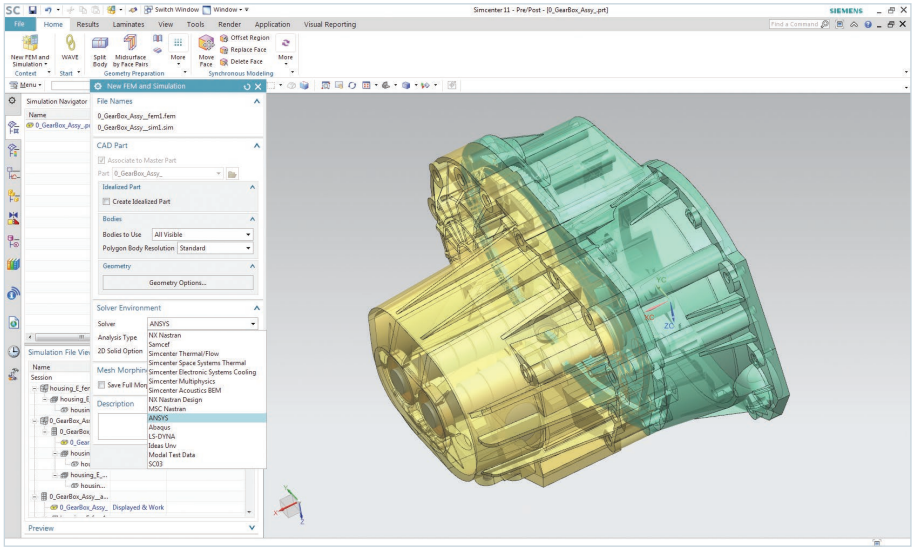



www.siemens.com/simcenter3d

# Simcenter 3D environment for ANSYS

### Import ANSYS models

- Import complete ANSYS finite element models including bulk data as well as solution controls and step controls. PREP7 and CDWRITE formats are supported
- Import beam element cross-section shapes for graphical display
- Import support for nodal thickness values on 2D elements
- Importing material orientation vectors as spatial fields



### Create and export ANSYS models from Simcenter 3D

The following types of analyses are supported:

- Structural linear static, modal, buckling, and nonlinear static
- Cyclic symmetry analysis for linear static, modal, nonlinear static, or harmonic – mode superposition
- Axisymmetric structural linear and nonlinear statics
- Modal flexible body (for use in motion analysis)
- Nonlinear buckling
- Transient dynamic
- Harmonic
- Thermal and axisymmetric thermal
- Thermal transient
- Thermal-structural multiphysics

### Elements and other entities

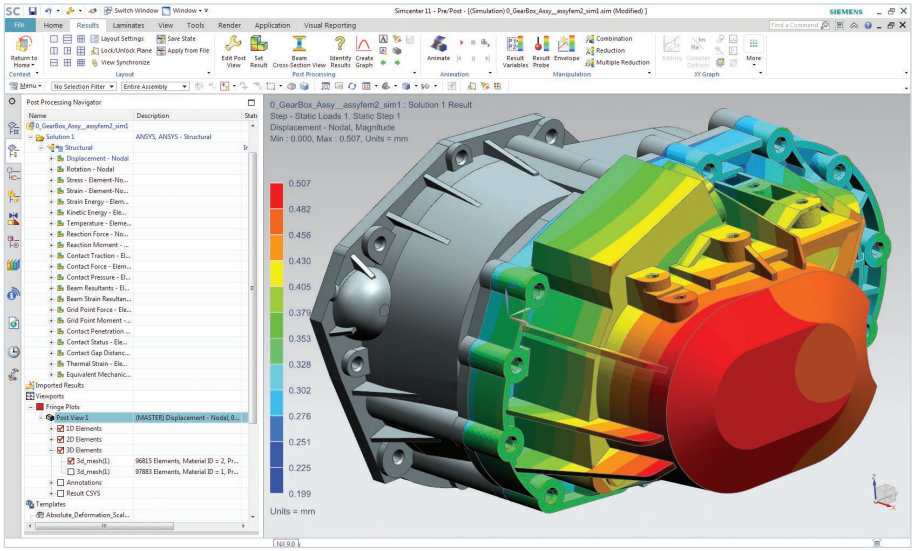
A wide variety of elements and other model entities are supported.

#### Structural element types:

- Rod (BEAM188, LINK180)
- Beam (BEAM44, BEAM188)
- Pipe (PIPE228)
- Axisymmetric solids, plane stress and plane strain (PLANE42, PLANE82, PLANE182, PLANE183, INTER192, SOLID272, SOLID273)
- Thin shell (SHELL63, - 93, -91, -99, -181, -281)
- Solid elements (SOLID45, -92, -95, -185, -186, -187, -191, -226, -227, SOLSH190, INTER195)
- Surface-to-surface contact elements (CONTA174, TARGE170)
- Rigid, constraint, spring, damper, gap and mass elements (CP, CE, CERIG, MPC184, COMBIN14, COMBIN39, COMBIN40, CONTA178, CONTA1789, MASS21)
- Pretension elements (PRETS179)
- Shell and solid laminates

#### Thermal element types:

- Rod (BEAM4) and beam (BEAM44) elements
- 3D conduction bar (LINK33_BAR)
- 3D conduction beam (LINK33_BEAM)
- 1D convective element (LINK34)
- Thin shell (SHELL57, -93, 131,-132)
- Axisymmetric solids (PLANE55, -77)



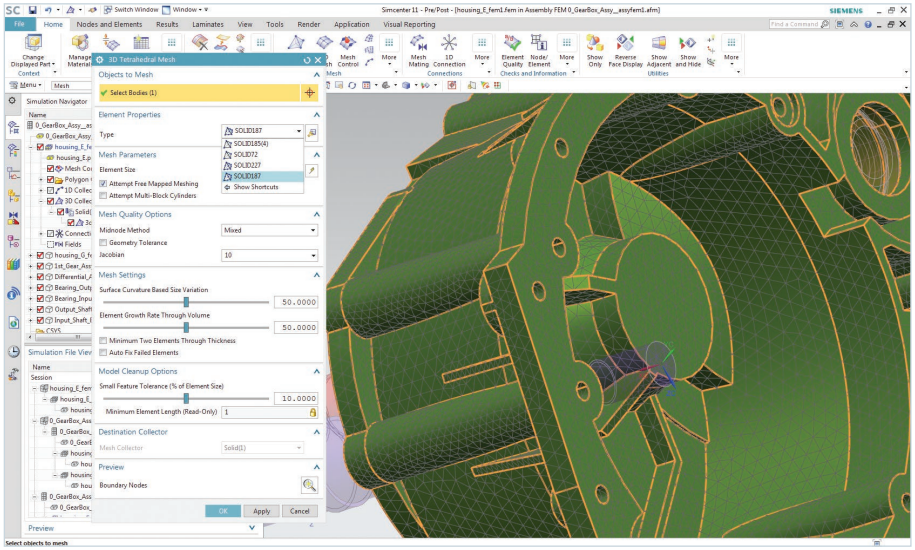
- Solid elements (SOLID70, -87, -90, -226, -227, -272, -273)
- Mass elements (MASS21)
- Thermal-fluid pipe (FLUID116)
- Surface elements (SURF151, -152)
- Cohesive elements (INTER204, -205)

All KEYOPTS and Real Constants are supported in the graphical user interface.

A complete list of ANSYS 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 force
- Nodal temperature and heat source
- Varying nodal pressure loads
- Acceleration loads
- Elemental face and edge pressure
- Elemental face and edge convection
- Elemental heat flux
- Coupled DOF
- Multi-point constraints
- Beam distributed load
- Nodal restraint
- Nodal temperature restraint (Simcenter 3D boundary or from ANSYS temperature results file or other solvers' temperature results)
- Contact in modal and buckling solutions
- Thermal coupling
- Thermal contact




**Compatibility**
The ANSYS environment is compatible with the ANSYS releases v17 or earlier.

**Supported hardware/OS**
The ANSYS 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 PLM Software**
**www.siemens.com/plm**

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3308


10666-A9 2/18 C