



**SIEMENS**

*Ingenuity for life*

# Simcenter 3D environment for Simcenter Samcef Solver Suite

## Accelerating pre- and postprocessing for an efficient FE solver

### Benefits

- Enables engineers using Simcenter Engineering Desktop and Simcenter Structures to generate finite element models for the Simcenter Samcef Solver Suite
- Simplifies and speeds the modeling process by enabling engineers to create analysis models based on geometry
- Immerses engineers in the Simcenter Samcef Solver Suite environment by using familiar Simcenter terminology and providing extensive support for Simcenter Samcef-specific elements and entities
- Delivers unique capabilities to predict nonlinear phenomena, material nonlinearities, bolt pre-load delamination of laminated composite structures

### Summary

The Simcenter™ 3D environment for Simcenter Samcef™ Solver Suite software enables engineers to build finite element (FE) models, define solution parameters and visualize results using the Simcenter Samcef Solver Suite. This environment allows you to take advantage of the powerful geometry editing, meshing and general preprocessing capabilities in Simcenter 3D to build analysis models for the Simcenter Samcef Solver Suite faster than with traditional CAE tools. The environment also allows engineers using Simcenter 3D to employ advanced solutions offered in the Simcenter Samcef Solver Suite. With the Simcenter Samcef Solver Suite, you can run linear and transient nonlinear static and dynamic analyses as well as modal and buckling/post-buckling analyses. Thermal analysis can also be performed, including advanced thermomechanical analysis of curing and crystallization to simulate

manufacturing processes of composites. The Simcenter Samcef Solver Suite also includes unique capabilities for post buckling analysis and the prediction of complex, nonlinear phenomena, such as progressive damage in the unidirectional and woven fabric plies of a laminated composite structure, and delamination with coupling to the damage inside the plies.

**Using Simcenter 3D to create Simcenter Samcef Solver Suite models**  
Simcenter 3D preprocessing is ideal for creating Simcenter Samcef Solver Suite models and solutions. Using 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 computer-aided engineering (CAE) preprocessors.

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

# Simcenter 3D environment for Simcenter Samcef Solver Suite

## Importing Simcenter Samcef Solver Suite models into Simcenter 3D

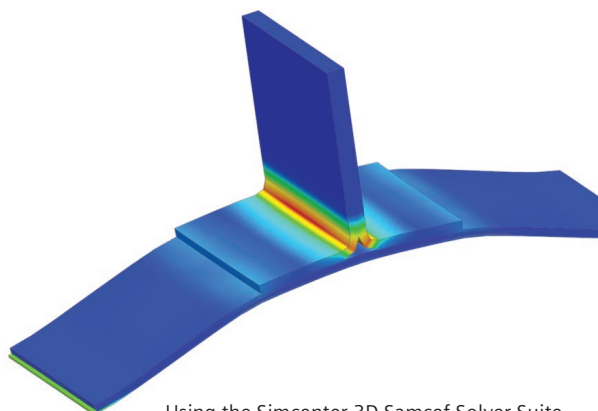
By using the Simcenter 3D environment for the Simcenter Samcef Solver Suite you can import Simcenter Samcef Solver Suite finite element models defined using either the ASCII or binary format database files. The following entities are supported:

- Elements
  - 1D
  - 2D: fully linear or parabolic
  - 3D: fully linear or parabolic
  - 3D cohesive element
- Material
  - Mechanical and thermal isotropic
  - Mechanical and thermal
  - Orthotropic
  - Anisotropic
  - Composite damaged materials (damaged ply, enhanced damaged ply, damaged woven ply)
  - Damaged interface
  - Elastoplastic material
  - Hyperelastic
  - Viscoelastic
  - Viscoplastic
  - Thermal – isotropic
  - Thermal – orthotropic
  - Thermal – composites
  - Thermal – fluid
- Physical
  - Shell thickness, shell offset
  - Nonstructural mass
  - Shell element axis
  - Composite materials
  - Damping properties
  - Material orientation
  - Sections, rod area
  - Coordinate systems
- Utility element definitions
  - Bushing
  - Capacity element
  - Mean and rigid element
  - Concentrated mass
- Rigid and flexible contact conditions
- Loads
  - Nodal forces and moments
  - Pressure
  - Angular acceleration
  - Acceleration
  - Normal pressure by component
  - Prescribed temperature
  - Flux
- Constraints
  - Fixed
  - Prescribed displacement
  - Prescribed velocities
  - Predefined composite damage
  - Initial temperature
  - Prescribed temperature at time 0
  - Cavity definition
- Beam profile definition
- Curing simulation
  - Thermal material properties – curing
  - Loads: degree of cure, glass transition temperature
- Groups of nodes, elements, face
- Explicitly defined function
- Coordinate system definition
- Output request

## Creating and exporting Simcenter Samcef Solver Suite models from Simcenter 3D

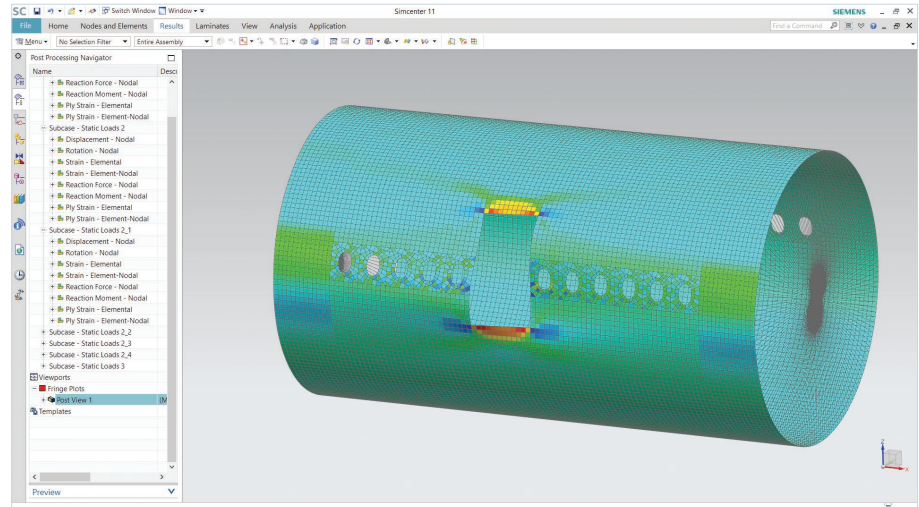
After you have created an FE model in Simcenter 3D, you can export complete ready-to-run Simcenter Samcef Solver Suite input decks with supported FE element and solver parameters. The Simcenter 3D environment for the Simcenter Samcef Solver Suite supports export for the following analysis types, elements and other entities:

- Analysis types
  - Linear statics with several load cases
  - Modal analysis
  - Buckling analysis (from linear statics analysis) with several load cases
  - Nonlinear static and dynamic analysis with several subcases
  - Thermal analysis
  - Chaining of analysis
- General algorithm control parameters (contact/friction, reordering, eigenvalue extraction, etc.)
- Strategy parameters for nonlinear computation
- Elements
  - 0D lumped mass, grounded bushing and spring elements
  - 1D beam and rod elements with predefined or user-defined profile
  - 1D connection elements: rigid links and averaged kinematic constraints, bushes, springs and bolts
  - 2D Mindlin thin shell, heterosis shell, 2D membrane or shell membrane, metallic or composite
  - 3D solid and solid shell, metallic and composite elements
  - 3D cohesive elements
  - Thermal 1D elements: bar, fluid
  - Thermal 2D element
  - Thermal 3D solid element
  - Rigid surface, contact element



Using the Simcenter 3D Samcef Solver Suite environment with Simcenter 3D Laminate Composites to model and simulate delamination.

- Ply and laminate characteristics and draping
- Rigid bodies
- Glue conditions
  - Nodes-to-surface gluing
  - Solid-to-edge gluing
  - Edge-to-edge gluing
  - Surface-to-surface gluing
  - Edge-to-surface gluing
- Contact conditions with option (such as sliding)
  - Surface-to-surface contact
  - Edge-to-surface contact
  - Rigid-flexible contact
  - Ancillary display preview for contact and glue conditions displayed in FE elements
- Definition of joints elements: bushing, spring
- Definition of joints properties: linear and nonlinear laws for bushing
- Materials
  - Isotropic with thermal dependency
  - Orthotropic with thermal dependency and stress limits
  - Anisotropic
  - Composite damaged materials (damaged ply, enhanced damaged ply, damaged woven ply)
  - Damaged interface
  - Elastoplastic material with different hardening measures
  - Hyperelastic
  - Viscoelastic
  - Viscoplastic
  - Thermal – isotropic
  - Thermal – orthotropic
  - Thermal – composites
  - Thermal – fluid



Modal analysis of an automotive panel using the Simcenter Samcef Solver Suite.

- Composite damage models
  - General orthotropic ply (Simcenter Samcef unidirectional ply damage)
  - Enhanced ply for non-crimp fabrics (NCF) and Simcenter Samcef enhanced unidirectional ply damage
  - Woven fabrics material (Simcenter Samcef woven ply damage)
  - Damage interface for cohesive element
- Damping properties
- Definition of local axes at the nodes
- Creation of groups of nodes, cells, interfaces, faces
- Loads and boundary conditions (including with time dependency)
  - Speed, acceleration
  - Nodal force, nodal temperature, rotations, moments, initial temperature
  - Pressure
  - Bolt preload
- Thermal loads and boundary conditions
  - Heat load
  - Heat flux
  - Volume flux
  - Radiation
  - Temperature
  - Convection definition
  - Cavity convection definition
  - Imposed flow rate
- Curing simulation
  - Thermal material properties – curing
  - Loads: degree of cure, glass transition temperature
- Fluid network definition

- Constraints
  - Initial speed, position
  - Initial damage for composites
  - Mean displacement
  - Fixed, enforced displacement and rotations
  - Symmetric
  - Manual coupling
  - Gluing node to volume faces or shell element
  - Planarity condition
- Explicitly defined functions

### Postprocessing Simcenter Samcef Solver Suite results in Simcenter 3D

The Simcenter 3D environment for the Simcenter Samcef Solver Suite reads the \*.des/\*.fac result files directly so you can visualize and evaluate product performance. Simcenter 3D can be used to postprocess all classical structural results (scalar, vector, tensor and complex values), such as displacements, stresses, strains, shell resultants and beam results and also temperature and flux in thermal analysis. For laminate structures, you can visualize stresses including interlaminar, damage and other native failure criteria results. Results can be displayed in absolute, material, native or user-defined coordinate systems, and can be plotted in XY graphs. Specific results related to curing simulation can also be displayed.

### Compatibility

The Simcenter 3D environment for the Simcenter Samcef Solver Suite supports the following Simcenter Samcef releases:

- Simcenter Samcef 17.3 (build V17.01-04) and higher

### Supported hardware/operating systems

The Simcenter Samcef Solver Suite environment is an add-on module within the Simcenter suite. It requires a license for Simcenter Engineering Desktop or Simcenter Structures. It is available on all Simcenter-supported hardware/OS platforms.

Siemens Digital Industries Software  
[siemens.com/plm](https://www.siemens.com/plm)

Americas +1 314 264 8499

Europe +44 (0) 1276 413200

Asia-Pacific +852 2230 3333