# SIEMENS
## Ingenuity for life

# Simcenter 3D environment for Abaqus

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

### Benefits
- Enables engineers using Simcenter 3D to generate finite element models for the Abaqus solver
- Simplifies the Abaqus modeling process by enabling engineers to create analysis models based on geometry or legacy Abaqus 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 Abaqus environment by using familiar Abaqus terminology and extensive support of Abaqus-specific elements and entities

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

Simcenter 3D's powerful geometry editing and meshing capabilities are ideal for pre- and postprocessing models for Abaqus. 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 pre processors. Adding the Abaqus environment to Simcenter 3D enables you to build Abaqus run-ready input data files, so little or no intermediate processing is ever needed. In addition

to building Abaqus models, the Simcenter 3D environment for Abaqus imports solution results directly from Abaqus results files into Simcenter 3D for post processing. The environment delivers import/export capabilities so you can import Abaqus data decks into Simcenter 3D for modification and then export run-ready decks for solution.

### Import Abaqus models
- Abaqus finite element models can be imported into Simcenter 3D.
- Model information from Abaqus input files is supported.

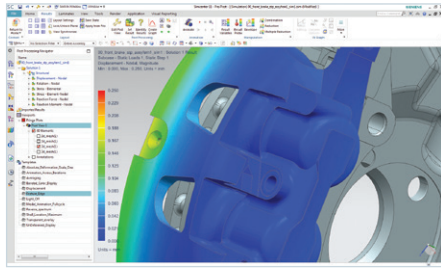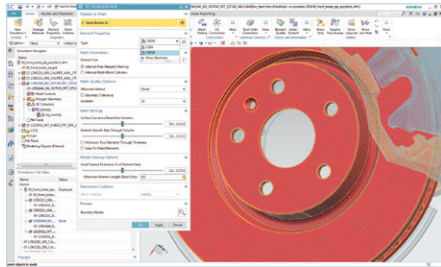### Create and export Abaqus models from Simcenter 3D
- Create and export complete run-ready Abaqus decks including model data (finite element entities) and history data (step definition, solution control, boundary condition set and contact definition).

Supports the following analysis types:
- Structural: statics, buckling and modal
- Steady-state heat transfer
- Visco analyses
- Transient modal dynamic analyses
- Response spectrum analysis
- Complex eigenvalue extraction analysis
- Implicit dynamic stress and displacement analyses
- Dynamic explicit analysis and axisymmetric dynamic explicit analysis
- Direct cyclic analyses

**www.siemens.com/simcenter3d**

# Simcenter 3D environment for Abaqus

- Transient heat analysis in axisymmetric thermal analyses
- Coupled thermal-structural analyses





## Elements and other entities
A wide variety of elements and other model entities are supported:

- Shell and solid elements
- Axisymmetric solid elements
- Spring, beam elements
- Inertia, rotary inertia, point mass, rigid body elements
- Interaction elements including gap, rigid surface
- Abaqus surface-to-surface contact
- Shell and solid laminates
- Line gasket elements
- Connector element
- Point heat capacitance
- Truss element
- Nonlinear springs
- Coupled temperature-displacement elements
- Surface stress elements

## 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
- Gravity, rotational velocities and acceleration loads
- Nodal, elemental and geometry-based thermal heat loads
- Nodal restraints and temperatures
- Traction loads
- Multipoint constraints and kinematics coupling
- Nodal temperature restraint (from Simcenter 3D boundary or from Abaqus FIL-file or other solvers' temperature results)
- Bolt pre-loaded length constraint
- Initial condition
- Enforced acceleration
- Enforced velocity constraint
- General contact
- Tie surface
- Zero-valued boundary constraint

A complete list of Abaqus import/export entity support is provided in the Simcenter 3D online help.
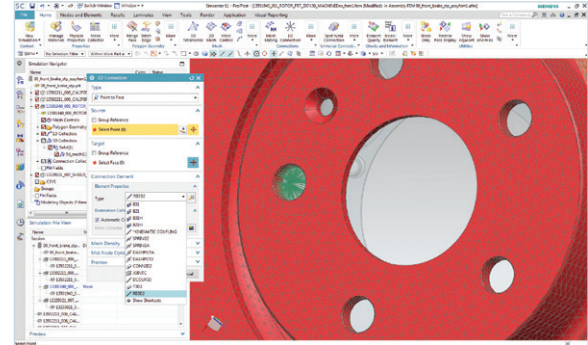
## Compatibility
The Abaqus environment is compatible with the following Abaqus releases:

- Abaqus 2016 or earlier

## Supported hardware/OS
The Abaqus 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 pre-requisite. It is available on all Simcenter 3D-supported