Simcenter **3D** Engineering Desktop

Integrating geometry and FE modeling to streamline product development

Solution benefits

Provide a platform for multidiscipline simulation

Enable fast, intuitive geometry editing

Deliver comprehensive meshing

Efficiently create and manage finite element assemblies

Quickly pre- and postprocess finite element models for popular common finite element solvers
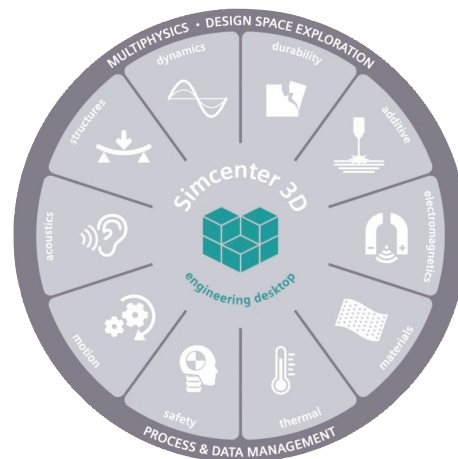
Capture knowledge and automate processes

Simcenter™ 3D Engineering Desktop software is a unified, open and extensible computer-aided engineering (CAE) environment for simulation analysts. Simcenter 3D Engineering Desktop is the hub of the Simcenter 3D platform, and it speeds the simulation process by helping you to efficiently build the model you need from 3D geometry faster than traditional CAE preprocessors.

Providing a platform for multidiscipline simulation

Simcenter 3D Engineering Desktop sits at the core of a broader, integrated multidiscipline simulation environment. Simcenter 3D Engineering Desktop is the centralized working environment for pre-/postprocessing of all Simcenter 3D solutions for structural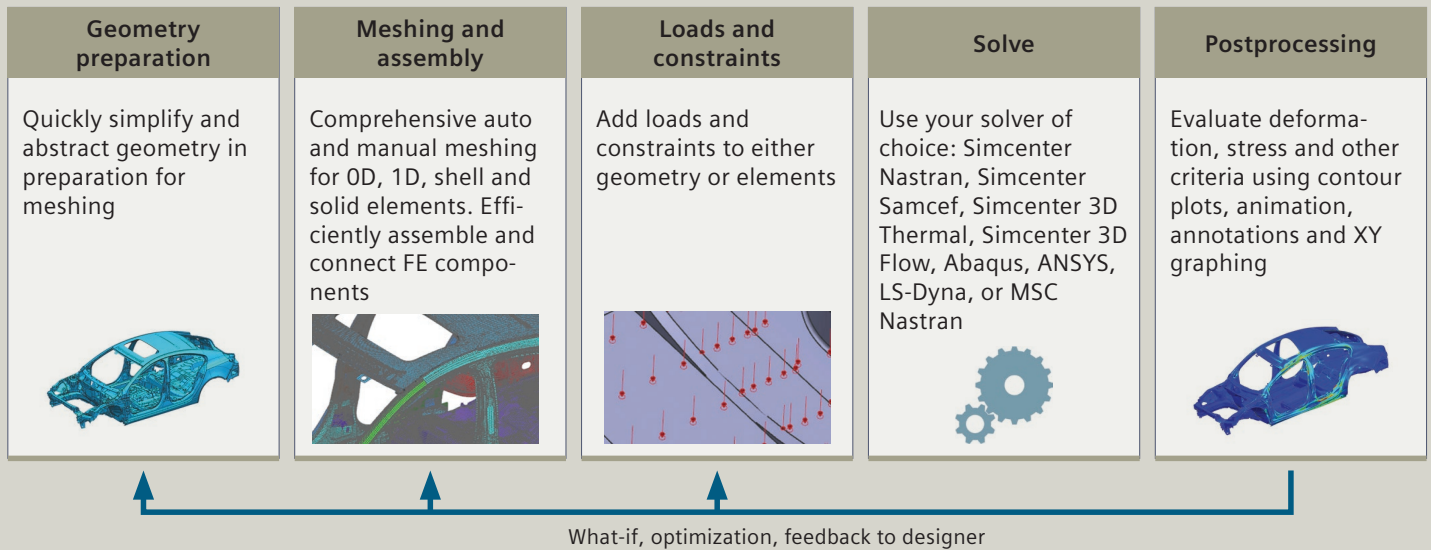, dynamics, composites, durability, acoustics, thermal, flow, motion, optimization, and electromagnetics . This integrated environment helps you to achieve faster CAE processes and streamline multidiscipline simulations that integrate any of the Simcenter 3D solutions together, like thermal-mechanical, vibro-acoustic, or other more complex analyses.

SIEMENS

Simcenter 3D Engineering Desktop

Geometry preparation	Meshing and assembly	Loads and constraints	Solve	Postprocessing
Quickly simplify and abstract geometry in preparation for meshing	Comprehensive auto and manual meshing for 0D, 1D, shell and solid elements. Efficiently assemble and connect FE components	Add loads and constraints to either geometry or elements	Use your solver of choice: Simcenter Nastran, Simcenter Samcef, Simcenter 3D Thermal, Simcenter 3D Flow, Abaqus, ANSYS, LS-Dyna, or MSC Nastran	Evaluate deformation, stress and other criteria using contour plots, animation, annotations and XY graphing

What-if, optimization, feedback to designer

Enabling fast, intuitive geometry editing

Simcenter 3D Engineering Desktop is built on an industry-leading geometry foundation. By using Simcenter 3D Engineering Desktop, you can rapidly clean up and prepare geometry from any computer-aided design (CAD) source using direct modeling. Geometry edits and the complete analysis model remain associated to the base design, which means you can easily update your analysis model each time the design changes. As a result, using Simcenter 3D Engineering Desktop accelerates your design-analysis iterations and improves your overall productivity.

Delivering comprehensive meshing

Simcenter 3D Engineering Desktop includes extensive modeling functions for automatic and manual mesh generation of 0D, 1D, 2D and 3D elements, and numerous techniques for applying loads and boundary conditions. The Simcenter 3D Engineering Desktop integrated environment is unique because it associates the analysis model to its geometry so when the design geometry changes, it rapidly updates the existing analysis geometry, mesh, loads and boundary conditions. This approach greatly reduces downstream modeling time, which results in huge time savings across a project's many design-analysis iterations.

Efficiently create and manage FE assemblies

Unlike traditional CAE preprocessors that were developed for component analysis and require you to build monolithic analysis models, the Simcenter 3D Engineering Desktop assembly finite element (FE) model (AFEM) management tool enables you to create large assembly models by instancing and connecting component models together, like a CAD assembly. When an FE component is updated later in development, Simcenter 3D Engineering Desktop updates all instances of that component in the assembly, eliminating the need to rebuild and connect a new FE assembly.

Supporting multiple common CAE solvers

Take advantage of Simcenter 3D pre- and postprocessing benefits with all your FE solvers. Simcenter 3D Engineering Desktop can be used as the primary pre- and postprocessor for Simcenter Nastran® software and Simcenter Samcef® software, or for third-party solvers such as Abaqus, ANSYS, LS-DYNA and MSC Nastran. This is accomplished with immersive user environments that use the selected solver's terminology, which enables analysts to easily prepare solver-specific analysis models without the need to learn new terminology.

Efficiently launch and monitor remote simulations

Simcenter 3D helps you to efficiently launch and monitor simulations remotely to any workstation running Windows or Linux on your network. You can then monitor the progress of an active solve right from your web browser to ensure it is progressing properly. Once the simulation is over, you can bring the results back to your Simcenter 3D workstation for postprocessing.

Capturing knowledge and automating processes

Simcenter 3D Engineering Desktop allows engineering organizations to capture the expertise of senior analysts and make it available for others in the organization to use in the form of wizards and templates. CAE processes can be captured and automated using NX™ Open software, an open framework for automation and programming. Analysts can capture the steps of a CAE process and then develop scripts and easy-to-use dialog boxes so others can re-use the same process. Since Simcenter 3D Engineering Desktop is built on top of the same platform as NX CAD, designers who use NX can continue to work in a familiar environment when they perform CAE.

Seamlessly connect with simulation data management

Simcenter 3D Engineering Desktop seamlessly integrates with the entire Teamcenter software data management portfolio, including the simulation process management module. Simulation data management capabilities work out-of-the-box (OOTB), and companies can implement a complete environment for managing CAE data, processes and workflow as part of a wider product development environment. This reduces waste by promoting re-use of existing designs and engineering knowledge. It also synchronizes data and makes it readily accessible using data mining, visualization and reporting. Simcenter 3D also includes seamless connection to Xcelerator Share for collaboration with partners outside of Teamcenter.

Industry applications

CAE is used across almost all industries today. However, the CAE process is still quite inefficient as engineers can spend upwards of 80 percent of their time on analysis modeling.

Simcenter 3D Engineering Desktop can help drastically reduce the time you spend on modeling these industry applications.

Aerospace and defense

Engineers that work on airframes and aircraft engines have vastly different needs for simulation modeling. Simcenter 3D can be used to efficiently handle the large assemblies that airframers need to manage, and it can also easily facilitate building the axisymmetric models that are required for aero engines.

Automotive and transportation

Body-in-white (BIW) modeling is a time-consuming process that requires a connection between hundreds of FE components. Simcenter 3D Engineering Desktop can help automate and reduce the time spent on BIW modeling.
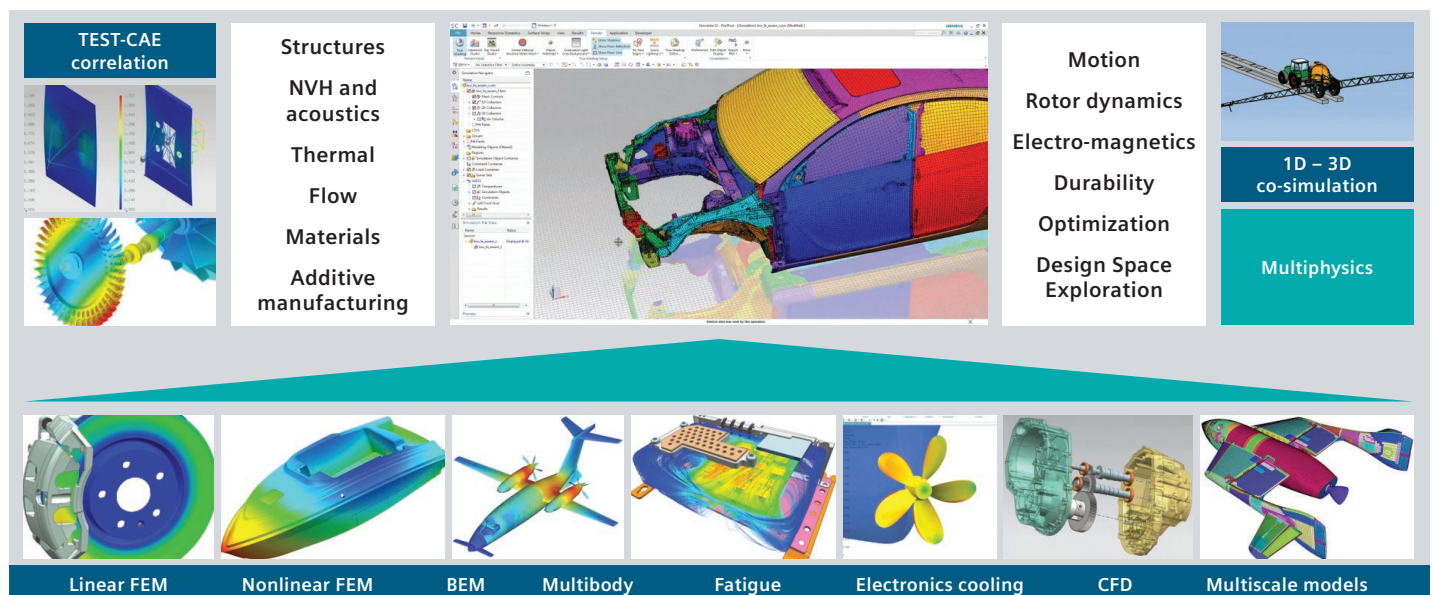
Consumer products

The consumer goods market moves fast, which means your engineers need to move even faster. Simcenter 3D helps you keep pace with rapid design changes so you can quickly simulate the latest iteration of your design.

Industrial machinery

From structural components and rotating equipment to large complex machines, Simcenter 3D Engineering Desktop can help you quickly create the right analysis model.

Marine

Ship models can be quite large, and Simcenter 3D Engineering Desktop has FE assembly management tools that can help you create easy-to-manage structural models.

| TEST-CAE correlation | Structures
NVH and acoustics
Thermal
Flow
Materials
Additive manufacturing | | Motion
Rotor dynamics
Electro-magnetics
Durability
Optimization
Design Space Exploration | 1D – 3D co-simulation
Multiphysics |

| Linear FEM | Nonlinear FEM | BEM | Multibody | Fatigue | Electronics cooling | CFD | Multiscale models |

Simcenter 3D
Engineering Desktop

Simcenter 3D Engineering Desktop is a unified, scalable, open and extensible 3D CAE environment for simulation analysts. It speeds the simulation process by helping you to efficiently build the simulation model you need from 3D geometry faster than traditional CAE preprocessors.
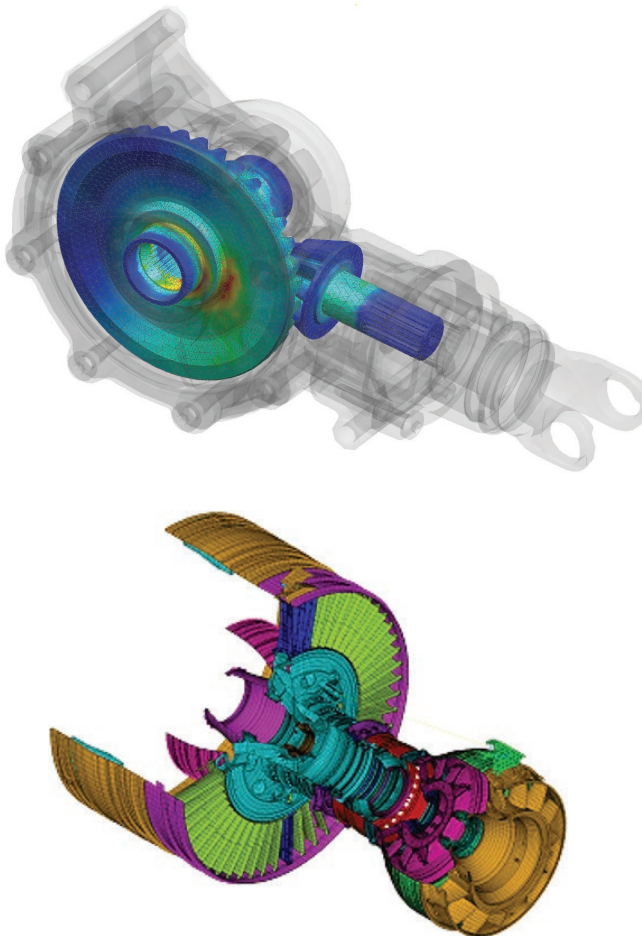
Module benefits
- Speed simulation processes by up to 70 percent
- Increase product quality by rapidly simulating design tradeoff studies
- Lower overall product development costs by reducing costly, late design change orders
- Efficiently manage large, complex FE assemblies
- Capture and automate best practices and commonly used processes
- Provide pre- and postprocess analysis models for the most popular finite element solvers
- Easily add multidiscipline simulation capabilities as your analysis needs grow

Key features
- An industry-leading geometry foundation for faster geometry editing and abstraction
- Comprehensive meshing tools
- Efficient FE assembly management
- Extensive graphics, plotting and reporting capabilities for postprocessing
- An open and extensible platform
- Seamless integration with simulation data management

Simcenter 3D
Environment for Simcenter Nastran

Included with Simcenter 3D Engineering Desktop, the Simcenter 3D Environment for Simcenter Nastran® software enables you to build finite element models, define solution parameters and view the solution results for the Simcenter Nastran solver. The environment immerses you in familiar Simcenter Nastran language for element definitions, loads and boundary conditions, solution parameters and other common Simcenter Nastran nomenclature. In addition to defining model 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 its run-ready data files.
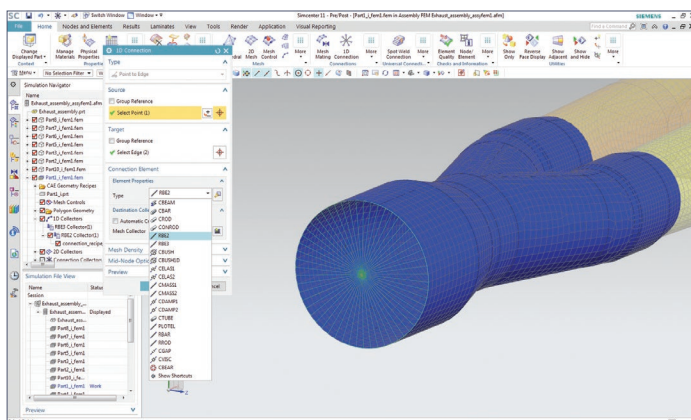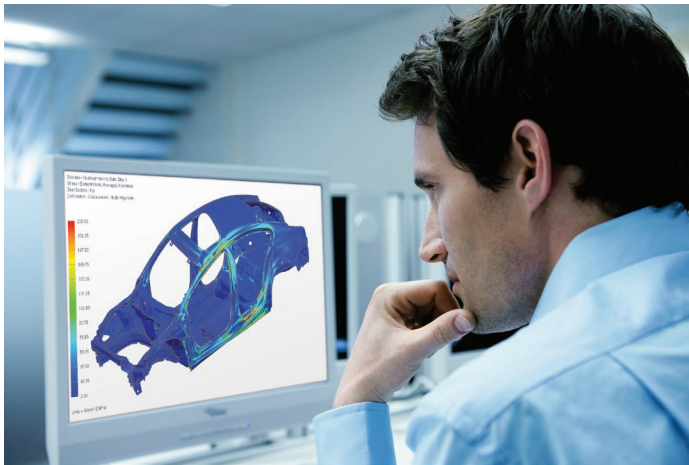
Module benefits

- Simplifies the Simcenter Nastran modeling process by enabling engineers to create analysis models based on geometry or legacy Simcenter Nastran input data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D
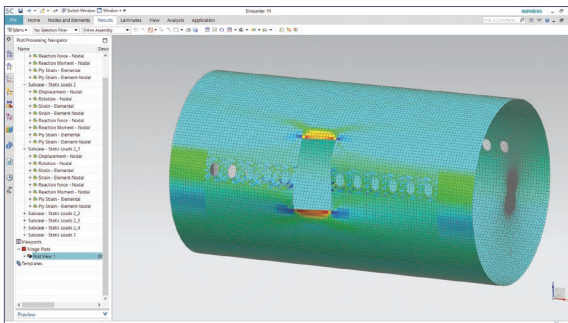
Key features

- Immerses engineers in the Simcenter Nastran environment by using familiar terminology and extensive support of product-specific elements and entities
- The Simcenter Nastran environment supports a variety of solutions:
  - SOL 101 – Linear statics
  - SOL 103 – Normal modes
  - SOL 105 – Buckling
  - SOL 106 – Nonlinear and linear statics
  - SOL 107 – Direct complex eigenvalues
  - SOL 108 – Direct frequency response
  - SOL 109 – Direct transient response
  - SOL 110 – Modal complex eigenvalues
  - SOL 111 – Modal frequency response
  - SOL 112 – Modal transient response
  - SOL 129 – Nonlinear and linear transient response
  - SOL 153 – Static structural and/or steady-state heat transfer analysis with options: linear or nonlinear analysis
  - SOL 159 – Transient structural and/or transient heat transfer analysis with options: linear or nonlinear analysis
  - SOL 200 – Design optimization with option for only sensitivity analysis
  - SOL 401 – Multistep, structural solution that supports a combination of static (linear or nonlinear) subcases and modal (real eigenvalue) subcases
  - SOL 402 – Multistep, structural solution that supports a combination of subcase types (static linear, static nonlinear, nonlinear dynamic, preload, modal, Fourier, buckling) and supports large rotation kinematics

Simcenter 3D
Environment for Simcenter Samcef

The Simcenter 3D Environment for Simcenter Samcef enables you to build FE models, define solution parameters and visualize results for the Simcenter Samcef solver. This environment allows you to take advantage of powerful geometry editing, meshing and general preprocessing capabilities in Simcenter 3D to build analysis models for the Simcenter Samcef faster than with traditional CAE tools.

Module benefits

- Simplifies the modeling process by enabling you to create analysis models based on geometry or legacy Simcenter Samcef input data files

- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D

Key features

- Immerses you in the Simcenter Samcef environment by using familiar terminology and providing extensive support for specific elements and entities of the product

- Includes a variety of supported analysis types:
    - Linear statics with several load cases
    - Modal analysis
    - Buckling analysis with several load cases
    - Nonlinear static and dynamic analysis with several subcases
    - Thermal analysis
    - Chaining of analysis
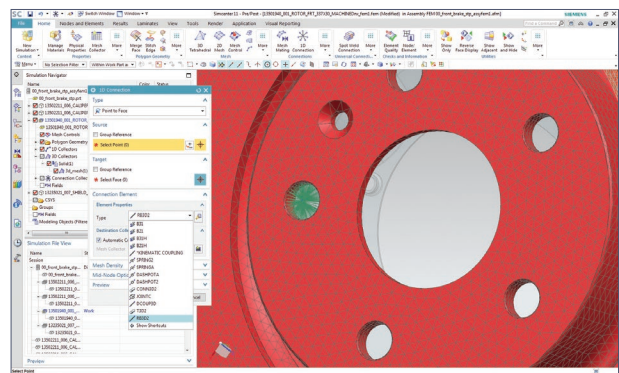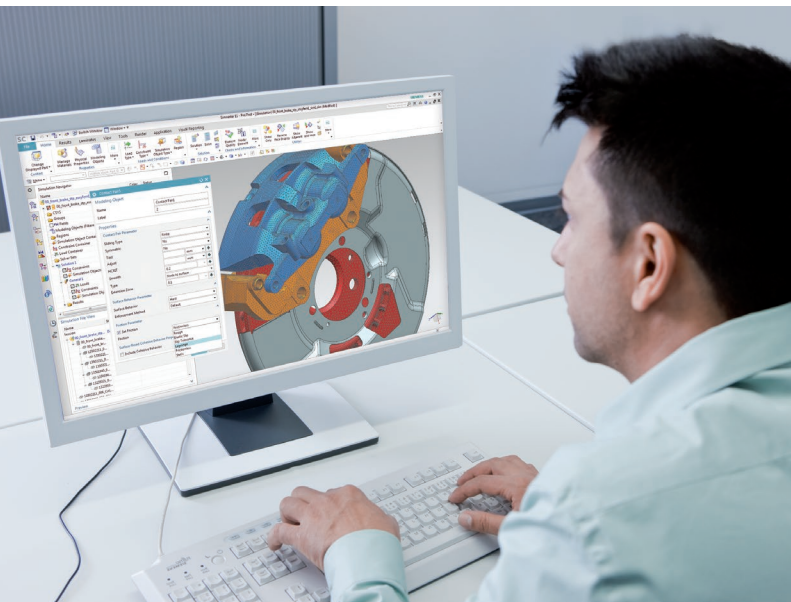
Simcenter 3D
Environment for Abaqus

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

Module benefits

- Simplifies the Abaqus modeling process by enabling you to create analysis models based on geometry or legacy input data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D

Key features

- 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

Simcenter 3D
Environment for ANSYS

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

Module benefits

- Simplifies the ANSYS modeling process by enabling you to create analysis models based on geometry or legacy input data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D

Key features

- Immerses you in the ANSYS environment by using familiar terminology and extensive support of specific elements and entities of ANSYS
- Supported solutions include:
  - 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 Simcenter 3D Motion analysis)
  - Nonlinear buckling
  - Transient dynamic
  - Harmonic
  - Thermal and axisymmetric thermal
  - Thermal transient
  - Thermal-structural multiphysics
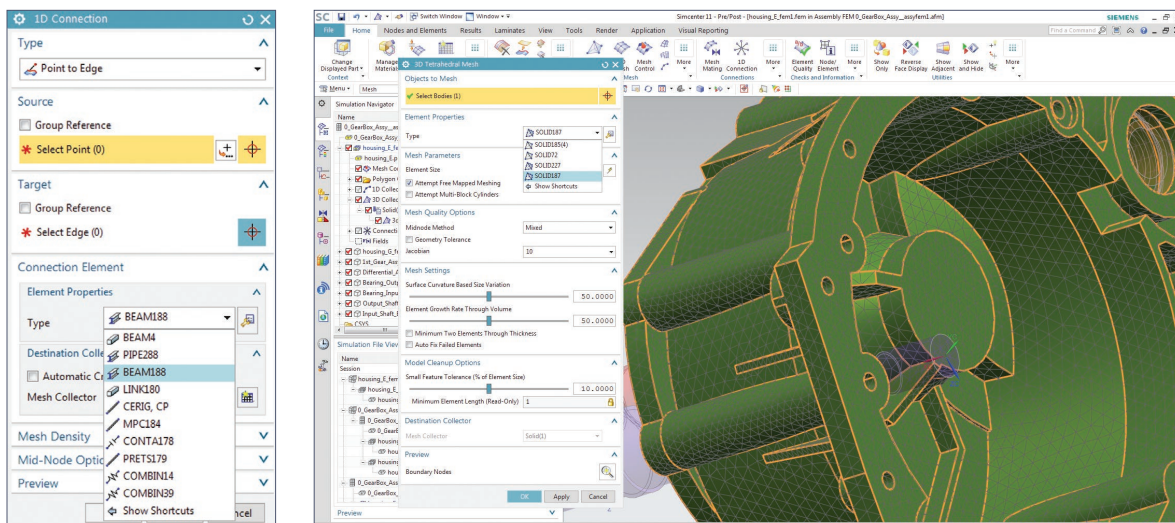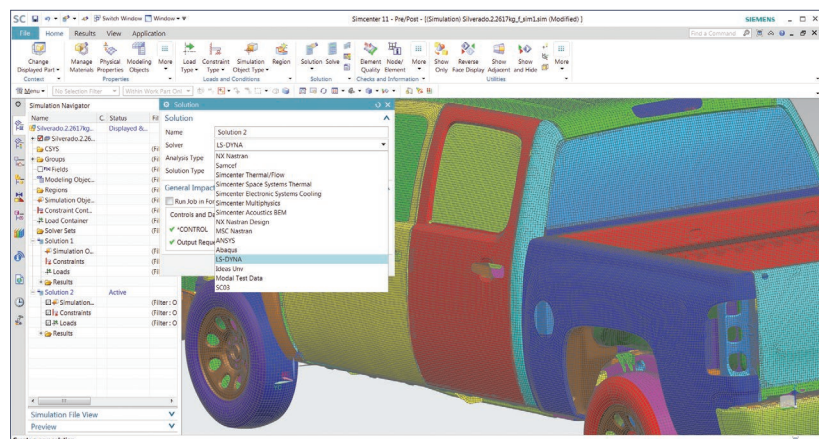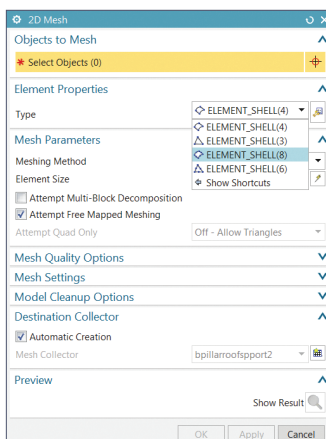
Simcenter 3D
Environment for LS-Dyna

The Simcenter 3D Environment for LS-Dyna enables you to build finite element models and define solution parameters for the LS-Dyna solver. The environment immerses you in familiar LS-Dyna language for element definitions, loads and boundary conditions, solution parameters and other common LS-Dyna nomenclature. In addition to defining model capabilities, the LS-Dyna environment enables bi-directional import/export capabilities that allow you to import current or legacy data files and results, as well as export run-ready LS-Dyna input data files.

Module benefits
- Simplifies the LS-Dyna modeling process by enabling you to create analysis models based on geometry or legacy LS-Dyna input data files

Key features
- Immerses you in the LS-Dyna environment by using familiar terminology and supporting specific product elements and entities
- Supports general structural impact solution

Simcenter 3D
Environment for MSC Nastran

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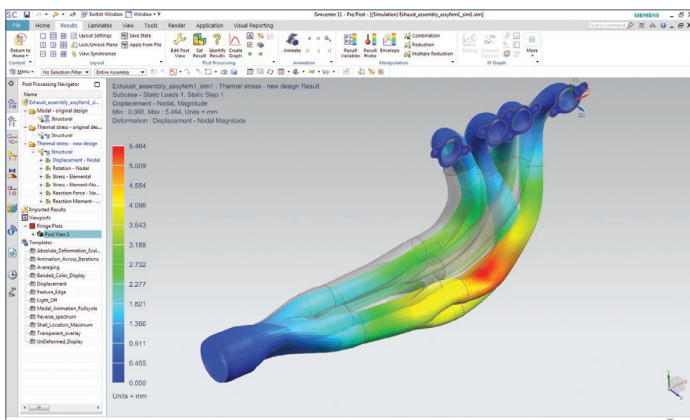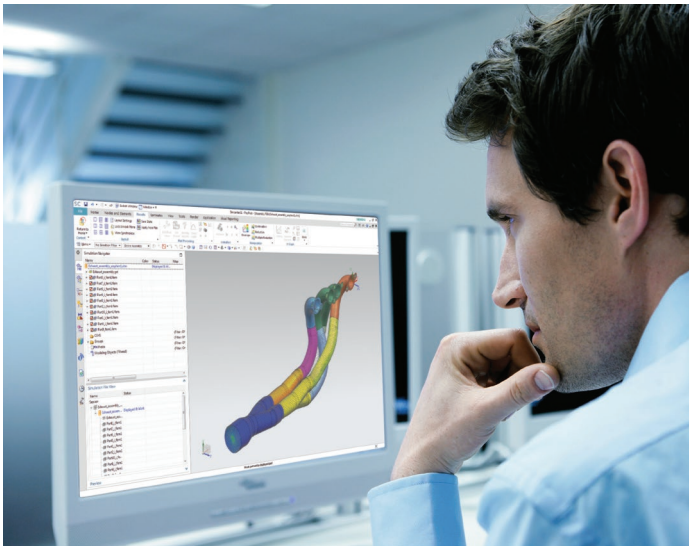
Module benefits

- Simplifies the MSC Nastran modeling process by enabling you to create analysis models based on geometry or legacy MSC Nastran input data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D

Key features

- Immerses you in the MSC Nastran environment by using familiar MSC Nastran terminology and extensive support of product-specific elements and entities
- There are a variety of supported solutions:
  - SOL 101 – Linear statics
  - SOL 103 – Normal modes
  - SOL 105 – Buckling
  - SOL 106 – Nonlinear or linear statics
  - SOL 107 – Direct complex eigenvalues
  - SOL 108 – Direct frequency response
  - SOL 109 – Direct transient response
  - SOL 110 –Modal complex eigenvalues
  - SOL 111 – Modal frequency response
  - SOL 112 –Modal transient response
  - SOL 129 – Nonlinear or linear transient response
  - SOL 153 – Static structural and/or steady-state heat transfer analysis with options: linear or nonlinear analysis
  - SOL 159 – Transient structural and/or transient heat transfer analysis with options: linear or nonlinear analysis
  - SOL 200 – Design optimization with option for only sensitivity analysis
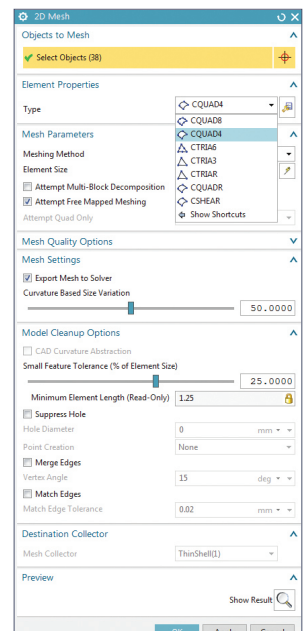
Simcenter 3D
Environment for ADINA

The Simcenter 3D Environment for ADINA enables you to build finite element models, define solution parameters and view the solution results for the ADINA solver using the Nastran interface. The environment immerses you in familiar ADINA language for element definitions, loads and boundary conditions, solution parameters and other common ADINA nomenclature. In addition to defining model capabilities, the ADINA environment enables bi-directional import/export capabilities that enable you to import its current or legacy data files and results, as well as export run-ready ADINA input data files.

Module benefits
- Simplifies the modeling process by enabling you to create analysis models based on geometry or legacy input data files for ADINA in the Nastran format
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from Simcenter 3D
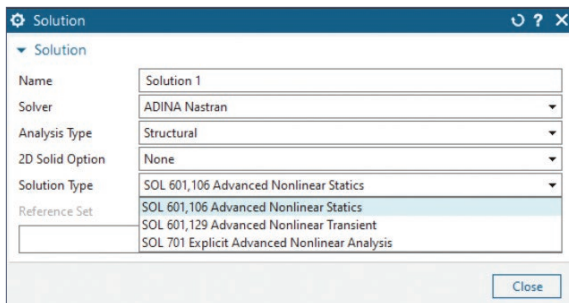
Key features
- Immerses you in the ADINA environment by using familiar terminology and extensive support of specific elements and entities of ADINA
- Solve geometric, material, and contact nonlinearities in static or dynamic (implicit or explicit) analyses
- Supported solutions include:
  - SOL 601,106 Advanced Nonlinear Statics (implicit static)
  - SOL 601,129 Advanced Nonlinear Transient (implicit transient dynamic)
  - SOL 701 Explicit Advanced Nonlinear Analysis (explicit transient dynamic for full structural analyses only, not axisymmetric structural)

mhl1_dvm_assembly_s : 601,129 Result
Subcase - Nonlinear Implicit, Increment 6, Time = 6.000e-004 s
Acceleration - Nodal, Magnitude
Min : 9810.0, Max : 9810.1, Units = mm/sec^2
Deformation : Displacement - Nodal Magnitude

Units = mm/sec^2
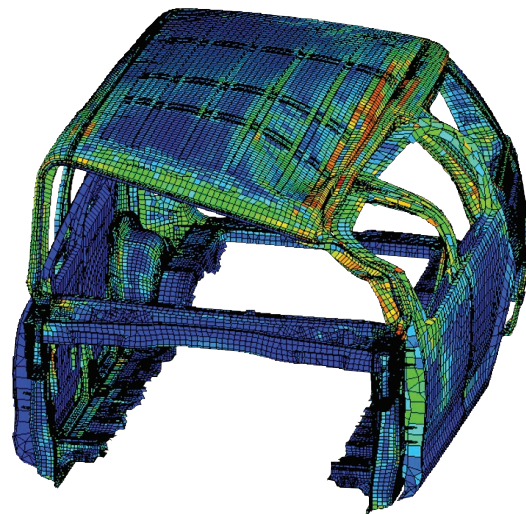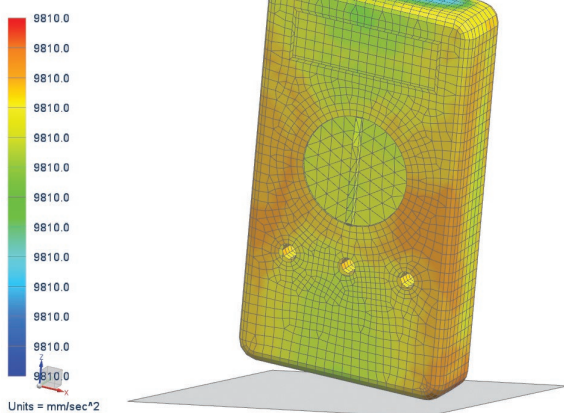
Capabilities chart

Simcenter 3D Engineering Desktop

General capabilities	Specific capabilities	Simcenter 3D Engineering Desktop
	Pre-processing	
Geometry import	Neutral geometry transfer (IGES, STEP, the JT™ data format, Parasolid® software)	•
	Direct geometry transfer (CATIA V4, CATIA V5, Pro/E)	+
Geometry modeling	Parasolid® software geometry kernel	•
	Parametric solid and surface modeling	•
	Direct modeling with synchronous technology	•
	Feature modeling	•
	Assembly structure creation	•
	Interpart relationship	•
	Configurations	•
	Convergent modeling (facet body editing)	+
Geometry editing prior to meshing	Direct editing with synchronous technology	•
	Defeature tools (geometry repair, feature suppression, stitch surface, remove hole/fillet, partitioning)	•
	Non-manifold topology generation for volumes	•
	Mid-surfacing (constant and variable thickness)	•
	Automatic topology abstraction	•
	Manual topology modification tools	•
	Create surface from mesh face	•
	Accept convergent model as input, push convergent model to NX CAD module	•
Meshing and modeling	Beam modeling	•
	Automated bolt connection meshing	•
	Automated shell meshing	•
	2D mapped meshing	•
	Automated tetrahedral meshing	•
	Swept hexahedral meshing	•
	Surface coating	•
	Transition meshing	•
	Axisymmetric meshing	•

General capabilities	Specific capabilities	Simcenter 3D Engineering Desktop
Meshing and modeling (continued)	Batch meshing	•
	Mesh quality checks	•
	Material property creation and management	•
	Mass property calculations	•
	Physical property creation and management	•
	Variable element thickness	•
	Grouping	•
Finite element assemblies	Hierarchical assembly management	•
	Efficient connection modeling	•
	Superelement support	+
	FE and test-based data hybrid modeling	+
Boundary conditions	Apply on geometry	•
	Apply to local coordinate system	•
	Apply on FE entities	•
	Apply on groups	•
	Axisymmetric boundary conditions	•
	Automatic contact detection and setup	•
	Automated load transfer from motion analysis	•
	Import test measurement data from and map to finite element model (load recipe)	•
External solver data deck support (import and export)	Simcenter Nastran	•
	Simcenter Samcef	+
	Abaqus	+
	ANSYS	+
	LS-Dyna	+
	MSC Nastran	+
	ADINA - Nastran interface	+
Analysis model to design associativity	Automatic analysis model update based on geometry change	•

Legend:

• = included in module
+ = additional product required

General capabilities	Specific capabilities	Simcenter 3D Engineering Desktop
	Postprocessing	
	Contour displays	•
	Beam cross-section contour displays	•
	Vector displays	•
	Isosurface displays	•
	Cutting planes	•
	Advanced lighting control	•
	Animations	•
	Complex dynamic response results	•
	Multiple viewports	•
	Probing of results on nodes	•
	Postprocessing data table with sort/criteria	•
	Results listings	•
	Transparency display	•
	Local coordinate system	•

General capabilities	Specific capabilities	Simcenter 3D Engineering Desktop
	XY graphing	•
	Bar graphing	•
	Synchronized contour and XY plotting displays	•
	Annotated graphs	•
	Output (JT, postscript, tif, etc.)	•
	Enhanced acoustics and NVH postprocessing	+
External solver result file support	Simcenter Nastran	•
	Simcenter Samcef	+
	Abaqus	+
	ANSYS	+
	LS-Dyna	+
	MSC Nastran	+
	ADINA - Nastran interface	+

Legend:

• = included in module
+ = additional product required

Simcenter 3D solver environments

General capabilities		Specific capabilities	Simcenter 3D Environment for Simcenter Nastran	Simcenter 3D Environment for Simcenter BEM Acoustics	Simcenter 3D Environment for Simcenter Samcef	Simcenter 3D Environment for Abaqus	Simcenter 3D Environment for ANSYS	Simcenter 3D Environment for LS-Dyna	Simcenter 3D Environment for MSC Nastran	Simcenter 3D Environment for ADINA
		Supported analysis types								
Structural analysis		Linear statics	•		•	•	•		•	
		Normal modes	•		•	•	•		•	
		Buckling	•		•	•	•		•	
		Nonlinear	•		•	•	•		•	•
		Axisymmetric structural	•		•	•	•	•	•	•
Structural dynamics		Modal frequency response	•						•	
		Direct frequency response	•			•			•	
		Modal transient response	•			•			•	
		Direct transient response	•			•			•	
		Random response	•							
		Complex modes	•			•				
		Superelements	•			•			•	
		Flexible bodies (for motion simulation)	•			•	•		•	
		Dynamic explicit				•		•		
		Axisymmetric dynamic explicit				•		•		
		Response spectrum				•				

General capabilities	Specific capabilities	Simcenter 3D Environment for Simcenter Nastran	Simcenter 3D Environment for Simcenter BEM Acoustics	Simcenter 3D Environment for Simcenter Samcef	Simcenter 3D Environment for Simcenter Abaqus	Simcenter 3D Environment for ANSYS	Simcenter 3D Environment for LS-Dyna	Simcenter 3D Environment for MSC Nastran	Simcenter 3D Environment for ADINA
Supported analysis types *(continued)*									
Advanced dynamics	Coupled fluid-structure (vibro-acoustic) analysis	•							
	Frequency response functions (FRF)	+							
	Recursive domain normal modes (RDMODES)	•							
	Fast frequency response (FASTFR)	•							
	Modal representation	+							
Thermal	Steady-state heat transfer	•		•	•	•		•	
	Transient heat transfer	•		•	•	•		•	
	Axisymmetric thermal	•		•	•	•			
Acoustics	Acoustic eigenvalues (normal modes, natural freq)	•							
	Frequency response	•	•						
	Transient response		•						
	Vibro-acoustics	•	•						
Multiphysics	Coupled thermal-mechanical	+			•	•			
	Coupled fluid-structure interaction	+							
	Coupled thermal-flow-structural	+							

Legend:

• = included in module
+ = additional product required

Note: Simcenter 3D Engineering Desktop is a minimum prerequisite for all Simcenter 3D products. Other dependency or prerequisites may apply for individual products.