

Abaqus for CATIA V5 R19 SP3

CATIA V5 Integration

Integration Features

- Availability of linear and nonlinear static, dynamic, and thermal analysis capabilities within the CATIA V5 environment
- Complete geometric associativity
- Support for Knowledgeware, publications, and results sensors
- Support for analysis assembly
- Complete support for CATIA V5 groups
- Support for many CATIA V5 advanced connections
- Visual Basic scripting capabilities for automated workflows

Operating Systems

- Windows XP Professional SP2
- Windows XP Professional x64
- Windows Vista x86
- Windows Vista x64

Abaqus Workbenches

- Abaqus Structural Analysis workbench for static, frequency-extraction, and explicit dynamic simulations
- Abaqus Thermal Analysis workbench for heat transfer simulations

Prerequisite and Corequisite Options

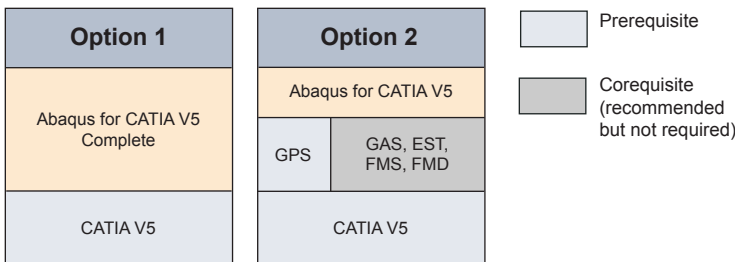
Abaqus for CATIA V5 is available with two different prerequisite and corequisite options:

Option 1 – Abaqus for CATIA V5 Complete

- Use Abaqus for CATIA V5 Complete, which does not require any analysis prerequisites or corequisites to fully enable AFC capabilities
- Abaqus for CATIA V5 Complete can be an attractive option for users who plan to focus their CATIA V5 simulation efforts on Abaqus

Option 2 – GPS Prerequisite

- Use Abaqus for CATIA V5 with GPS as a prerequisite, and GAS, EST, FMS, and FMD as corequisites to enable extended capabilities



Supported Features

Materials

- CATIA-defined linear material properties
- Abaqus linear and nonlinear material properties stored in the CATMaterial document
- Temperature-dependent material properties
- Thermal properties
- Linear elasticity
- Metal plasticity (isotropic, kinematic, or Johnson-Cook hardening)
- Hyperelasticity
- Abaqus material user subroutine
- Abaqus heat generation user subroutine
- Orthotropic composite layups for shell modeling
- 3-D orthotropic material properties
- Gasket material properties

Loads

- Load application using tabular and smooth-step amplitudes
- Point forces and moments that can follow nodal rotation
- Pressure loads with or without data mapping
- Gravity forces (mass body forces)
- Rotational body forces
- Surface heat fluxes
- Concentrated heat fluxes
- Body heat fluxes

Connections

- “Interaction Wizard” to automatically detect contacting surfaces
- Small- and large-sliding contact
- Friction models (penalty, user-defined, and static-kinetic exponential decay)
- Shell contact accounting for shell thickness
- Abaqus/Explicit general contact
- Control over contact propagation from step to step
- Contact controls including automatic stabilization of rigid body modes
- Self-contact modeling
- Fully fastened connections
- Physical and virtual bolt tightening connections
- Bolt tightening can be applied at any time in the analysis
- Gap thermal conductance
- Rigid body constraints, including pin and tie node sets
- Analytically defined rigid surfaces
- Rigid and smooth couplings of specified degrees of freedom
- Spot, seam, and surface weld connections (mesh-dependent or mesh-independent)
- Support for CATIA point interface connections and node interface properties
- Support for CATIA point-to-point analysis connections and nodes-to-nodes connection properties
- Rigid and smooth connections
- Spring connections (linear and nonlinear)



- Dashpot connections (linear and nonlinear)
- Press-fit connections
- Virtual parts (smooth and rigid)
- Spring virtual parts (smooth and rigid)
- Shared nodes among separate parts (nodal equivalencing)

Meshing

- Support for CATIA V5 line, surface (triangle and quadrilateral), and solid (tetrahedral, prism, and hexahedral) meshing
- Support for CATIA V5 linear and parabolic element types
- Abaqus shell and membrane elements
- Abaqus continuum shell elements
- Abaqus gasket elements
- Models containing mixed-dimension element types
- "Skinning" of solid bodies with membrane or shell elements
- Whole-part and part-region (local) shell and beam properties

Analysis Steps

- Nonlinear static
- Linear static perturbation
- Riks method static steps (for handling global instabilities)
- Frequency extraction (including preloading effects)
- Explicit dynamics
- Transient heat transfer
- Steady-state heat transfer

Geometric Nonlinearities

- Large rotations
- Large strains
- Stress stiffening

Advanced Modeling Techniques

- Submodeling to target regions of interest for detailed stress analysis
- Cyclic symmetry modeling to analyze bodies whose sectors repeat about an axis of revolution
- Shell modeling with node offsets from the shell midplane
- Variable shell thickness using data mapping of 2-D properties
- Dedicated gasket elements to accurately model thin, flat gaskets
- Simulation continuation capabilities to

- enable a preloaded model to be used as the base state for subsequent simulations through the Abaqus *Restart, read option
- Support for assembly of analysis documents

Job Management

- Multiple job submission and input file writing
- Multiple case solutions in an analysis document
- Consistency checking
- Input file writing (in multiple part instances or a single part instance)
- Input file writing in user-selected set of units
- "Job type" control (syntax check, data check, continue, full model, and from input file)
- Restart file writing control
- Memory settings control
- Job overwrite warnings
- Background job submission
- Ability to conveniently view Abaqus analysis files in a text editor (.dat, .msg, .sta, .log, and .inp)
- Monitoring of job status
- Job error and warning messages
- "Kill" function for jobs in progress
- Interactive job diagnostics to help users quickly understand and correct modeling problems

Results Visualization

- Access to a wide range of Abaqus output variables (from Abaqus output database)
- Time history animations
- CATIA V5 cutting-plane results viewing
- Results extrema identification
- Access to Abaqus results through sensors
- Display groups for user-generated and solver-generated model regions
- Control of results output frequency
- Specification of results output variables
- X-Y data plotting through direct link to Microsoft Excel

Documentation

- Online User's Manual (in both HTML and PDF formats)
- Tool tips
- Context-dependent help
- References and links to Abaqus Analysis User's Manual

Abaqus Analysis Products

Corequisite

- Abaqus/Standard or Abaqus/Foundation for static and thermal simulations
- Abaqus/Explicit for explicit dynamics simulations
- Abaqus versions supported: 6.6, 6.7, and 6.8

Product Support

- Maintenance and support
- Quality monitoring service
- Installation
- Training and users' meetings

For more information, visit simulia.com/products/afc_V5 or contact your local SIMULIA office to discuss your requirements with a technical specialist.

The 3DS logo, SIMULIA, CATIA, 3DVIEW, DELMIA, ENOVIA, SolidWorks, Abaqus, Isight, Fiper, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners.

Copyright Dassault Systèmes, 2009

SIMULIA World Headquarters

166 Valley Street
Providence, RI 02909 USA
+1 401 276 4400
E-mail: simulia.info@3ds.com

