

Abaqus for CATIA V5 R21

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
- Model optimization using the PEO workbench
- 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/x86-32 and Windows/x86-64 operating systems (Windows XP Professional SP2, Windows XP Professional x64, Windows Vista, Windows 7)

Abaqus Workbenches

- Nonlinear Structural Analysis workbench for static, frequency-extraction, and explicit dynamic simulations
- 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

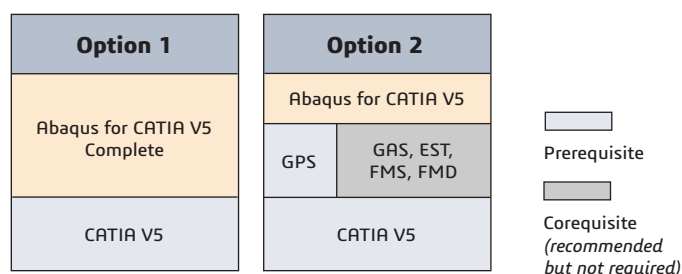
Option 2 – GPS Prerequisite

- 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
- Composite properties imported from the CATIA Composite Design workbench
- Orthotropic composite layups for shell modeling
- 3-D orthotropic material properties
- Gasket material properties

Loads

- Load application using tabular and smooth-step amplitudes
- Forces and moments can follow nodal rotation
- Data mapping available for pressure, heat flux, thermal film condition, and temperature
- Imported loads from GPS analysis
- Cartesian, cylindrical, and spherical local coordinate systems available
- Point, distributed, and force density loads
- Pressure loads
- Gravity forces (mass body forces)
- Rotational body forces
- Point, surface, and body heat fluxes
- Surface thermal film condition
- Temperature definitions

Contact

- Contact modeling using either the “contact pair” or “general contact” approach
- “Interaction Wizard” to automatically define contact pairs
- Small- and large-sliding contact
- Friction models (penalty, user-defined, and static-kinetic exponential decay)
- Shell contact accounting for shell thickness
- Control over contact propagation from step to step
- Contact controls including automatic stabilization of rigid body modes
- Self-contact modeling
- Gap thermal conductance

Connections

- Fully fastened connections
- Physical and virtual bolt tightening connections
- Bolt pretensioning on beams or 3D bolts that can be applied at any time in the analysis
- 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

Abaqus for CATIA V5 R21 Data Sheet

- 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

- Extensive model consistency checking
- Multiple case solutions in an analysis document
- Input file writing in user-selected set of units
- Single or multiple sequential job submission
- “Job type” control (syntax check, data check, continue, full model, and from input file)
- Restart file writing control
- Memory settings control
- Convenient viewing of 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
- Capability to run simulations on remote hardware

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: 6.9, 6.9EF, and 6.10

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.