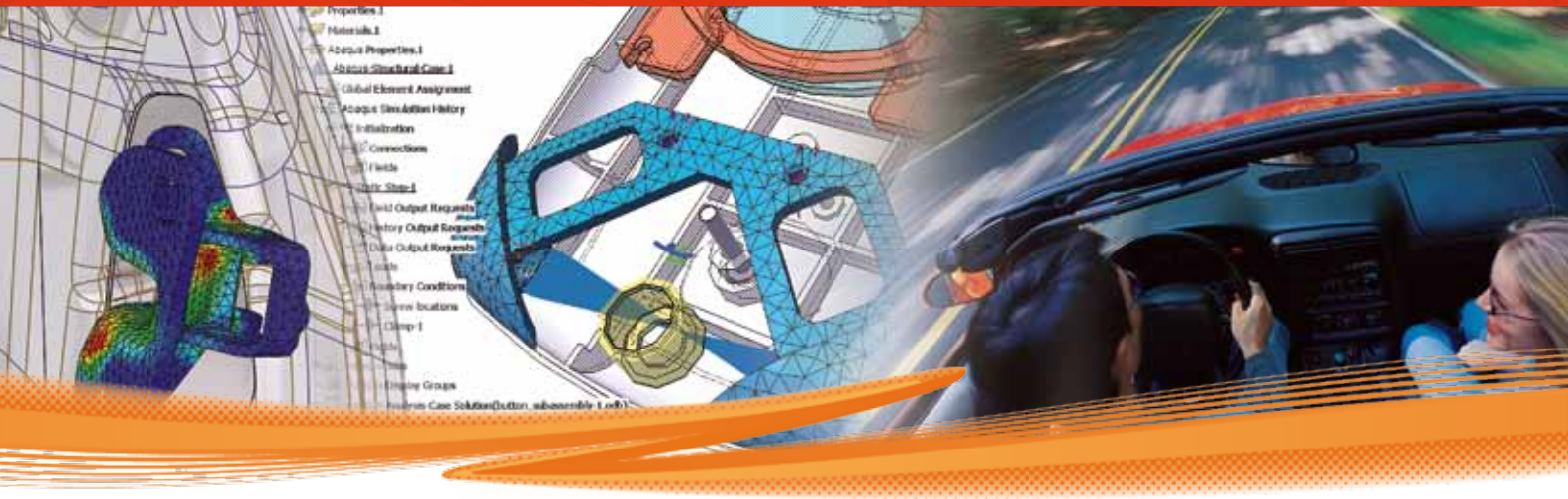


Abaqus for CATIA V5

Realistic Simulation Inside CATIA V5



Deploy Proven Analysis Workflows Throughout the Enterprise

Abaqus for CATIA V5 Enables Realistic Simulation in the Design Phase



Abaqus Functionality in CATIA V5

Abaqus for CATIA V5 (AFC) enables Abaqus simulations from within the CATIA V5 user environment. It maintains and reuses the CATIA V5 concepts of parts, virtual parts, and assemblies, making full use of associativity between geometry and the analysis model. Familiar CAD modeling constructs such as points, surfaces, and bodies are the basis for familiar analysis constructs such as boundary conditions, loads, and contact associations. In addition, Abaqus for CATIA V5 enables Abaqus nonlinear static, frequency extraction, explicit dynamic and thermal simulations, as well as sequentially coupled thermal-structural simulations.

Scalable Solutions and Proven Workflows

With Abaqus for CATIA V5 your CATIA V5 models and your Abaqus models become one and the same, making this software a highly scalable solution. Design engineers skilled in using CATIA V5 have access to a wide array of Abaqus simulation capabilities, while Abaqus experts can readily access CATIA V5 models for their analysis work. Abaqus for CATIA V5 is also a valuable tool for deploying customer-specific workflows across the engineering enterprise. It enables advanced simulation capabilities to be used routinely by design engineers to accelerate and improve product development.

Structural and Thermal Analysis Workbenches

Abaqus for CATIA V5 functionality is grouped into two CATIA V5 workbenches. Within the Structural Analysis workbench, you define static, frequency extraction, or explicit dynamic simulation models, including all of the necessary mechanical attributes. Within the Thermal Analysis workbench, you define heat transfer simulation models, including all of the necessary thermal attributes. Static and thermal simulations can be coupled together to determine the response of your model to a combination of thermal and mechanical influences.

Nonlinearity

One of the key features of Abaqus for CATIA V5 is its access to the powerful nonlinear mechanics capabilities in Abaqus. These best-in-class capabilities have been a fundamental part of Abaqus from the beginning. As a result, Abaqus for CATIA V5 simulations can include nonlinear geometric effects, such as arbitrarily large rotations and stress stiffening. These capabilities also allow nonlinear effects such as intermittent contact, plasticity, and temperature-dependent thermal film conditions to be evaluated directly in CATIA V5.

Frequency Extraction

Natural frequencies are often useful for assessing overall stiffness of a structure as well as likely deformation patterns. Abaqus for CATIA V5 can calculate the natural frequencies either on the unloaded structure or on the loaded structure at any time within the simulation history. AFC's ability to calculate the natural frequencies on a loaded structure can be important, because natural frequencies can change significantly during deformation as the structure experiences changing loads, boundary conditions, temperatures, and contact conditions.

Dynamic Analysis Within Abaqus/Explicit

Explicit dynamic analysis is well-suited for large models with relatively short response times and for the analysis of extremely discontinuous events, such as impact. Very general contact conditions can be modeled, including contact between many or all regions of the model, with a single interaction definition.



Contact

Abaqus for CATIA V5 provides two powerful techniques for modeling contact. You can either define pairs of contact surfaces yourself with the aid of the "Find Interactions" wizard, or you can let Abaqus automatically determine the contact regions using the Abaqus "General Contact" approach. Either approach enables modeling of complex multipart interactions in assemblies. Components can experience large relative sliding, which can include friction effects. It is even possible to capture self-contact effects, such as when a surface deforms so severely that it folds over onto itself. For thermal simulations, gap heat conductance properties can be assigned to contact surfaces so that the surfaces conduct heat appropriately, depending on the size of the gap and the temperature at each surface.

Loads and Boundary Conditions

In Abaqus for CATIA V5 you can apply loads and boundary conditions using time-varying amplitude definitions to enable the highest degree of control over loads and constraints. Numerous types of loads and constraints are available, including rigid constraints that enable you to define selected parts or regions as completely rigid in the specified degrees of freedom.

Submodeling

Once you have obtained analysis results for a model, you can use submodeling to rapidly study a local region with greater geometric detail and a more refined mesh. Performing a submodel analysis is a two-stage process. First, you create and analyze the global model. You then create the submodel and drive the boundaries of the submodel with results that were saved during the analysis of the global model. You can apply submodel boundary conditions in both mechanical and thermal steps.

Materials

Abaqus for CATIA V5 allows you to use linear material properties defined in the CATIA V5 material library. You can extend these materials to include nonlinear properties or create entirely new Abaqus material properties and store them conveniently in the CATIA V5 material library. The materials can be temperature dependent and include thermal material properties, such as thermal conductance and specific heat.

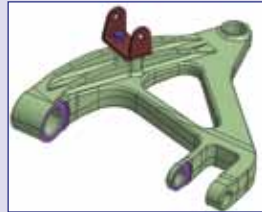
Analysis Templates for Design Engineers



The lower control arm subassembly (right) connects the car frame to the steering knuckle. All model regions where analysis attributes will be applied are published to descriptively named publications.

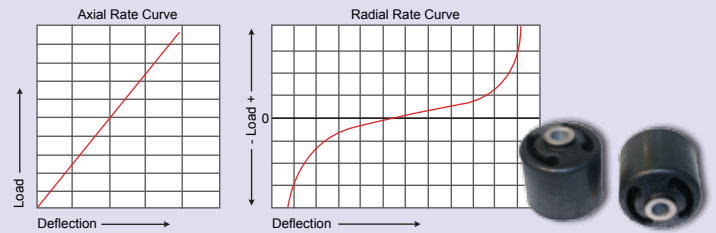


Arm Design-1

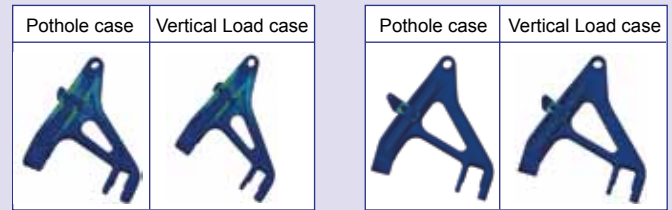


Arm Design-2

CATIA V5 has tools to enable easy replacement of components in an assembly. When using publications, the Abaqus analysis model automatically updates with the new geometry.



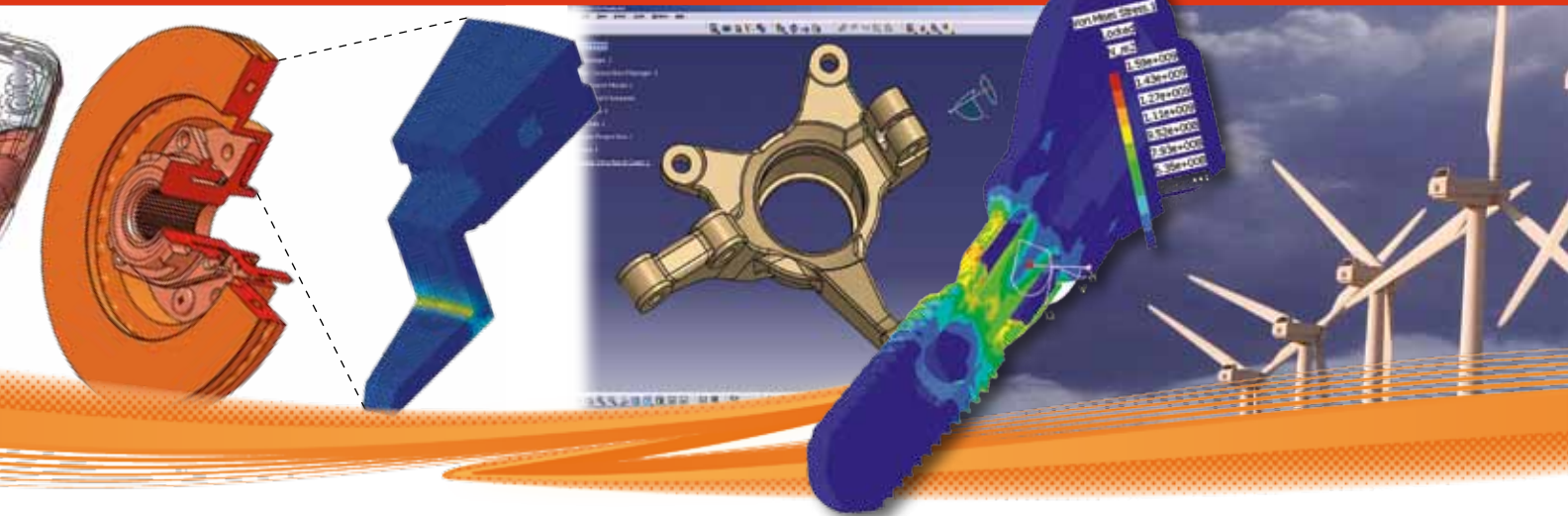
The bushings that attach the lower control arm to the frame are modeled using a linear stiffness in the axial direction and a nonlinear stiffness in the radial direction to simulate the bushings' carefully-tuned behavior.



Arm Design-1

Arm Design-2

You can run a variety of load cases on different designs to quickly determine whether each design meets the requirements.



Meshing

Abaqus for CATIA V5 uses the integrated CATIA V5 meshing tools to obtain high-quality meshes with geometric associativity. You also have access to the versatile Abaqus C3D10I tetrahedral elements for accurate contact and plasticity results. CATIA V5 surface meshing tools enable you to create triangle- or quadrilateral-dominated shell meshes. A single model can combine different element types.

Job Management

Integrated job management enables you to create job specifications, write input files, and monitor progress. Specifications may be detailed and include file storage locations and memory settings. After submitting analysis jobs, the ability to continuously monitor solution progress allows the prompt receipt of any warning or error messages from Abaqus.

Connections

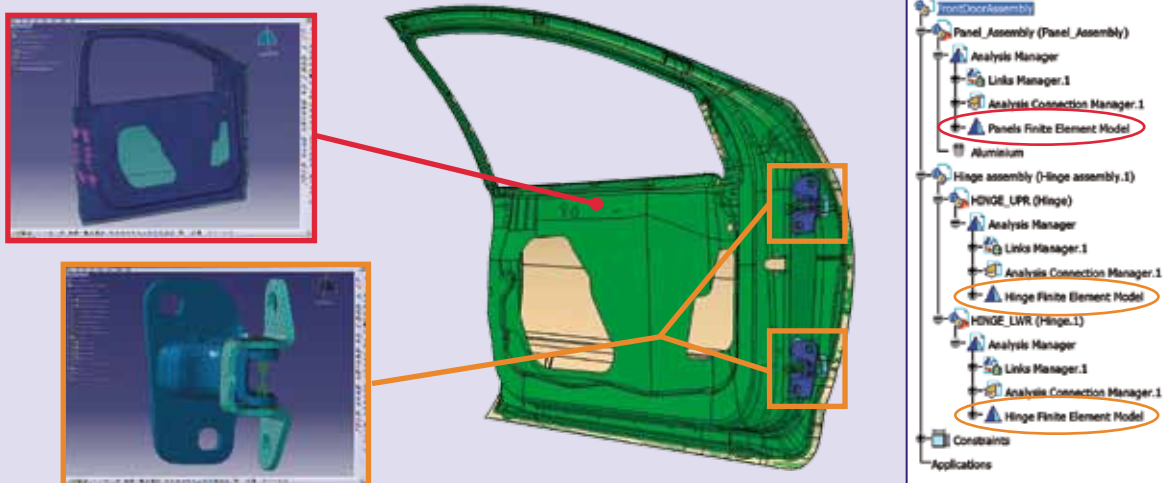
Component assemblies can use a variety of connection methods such as bolts, springs, or welds. The bolt modeling capability enables easy modeling of bolted connections to simulate bolt loads accurately. The spot, seam, and surface weld modeling tools enable you to model a large number of flexible or rigid fasteners in just a few steps. Other types of available connections are rigid connections, virtual parts, and nonlinear springs.

Postprocessing

As a fully integrated CATIA V5 solution, Abaqus for CATIA V5 provides access to the extensive CATIA V5 postprocessing capabilities, including useful tools such as cutting plane visualization. Through the Abaqus output database, you also have access to the extensive set of available Abaqus output variables, providing a wide range of results visualization capabilities.

Concurrent Engineering with Analysis Assembly

Analysis assembly enables creating assemblies by combining existing finite element analysis documents. With this technique companies are encouraged to create libraries of high-quality parts and sub-assemblies – including their finite element meshes and connections – that can be reused in multiple analysis scenarios, enabling greater efficiency and a better collaboration.



Different users can create the finite element meshes and connections for parts and sub-assemblies.

A master analysis model can be assembled from existing part and sub-assembly finite element models.



Iterative and Collaborative Design Loop

Abaqus for CATIA V5 is an effective tool for deploying proven analysis workflows across the enterprise. It enables advanced simulation capabilities to be used routinely by design engineers to accelerate and improve product development.



Leverage familiar CATIA design environment and methods

Apply associative and rules-based modeling capabilities

Utilize advanced analysis methods directly on your CATIA model

Evaluate high-fidelity results in relation to established requirements

SIMULIA World Headquarters

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

Prerequisite and Corequisite Options

Abaqus for CATIA V5 is available with two different prerequisite and corequisite options. One option is to use Abaqus for CATIA V5 with GPS as a prerequisite and GAS, EST, FMS, and FMD as corequisites to enable extended capabilities. Alternatively, you can 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.



About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and SIMULIA SLM for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, SIMULIA provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

The 3DS logo, SIMULIA, CATIA, 3DVIA, 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, 2010

