Abaqus Unified FEA

Complete Solution for Realistic Simulation

SIMULIA

Realistic Simulation with Abaqus Unified FEA

Complete finite element modeling and analysis solution for simulating the real-world behavior of materials, processes, and products

Efficient Modeling Tools

In today's product development environment, geometry and models often come from a variety of sources. The Abaqus Unified FEA product suite offers a hybrid modeling approach that enables users to work with geometry-based data alongside imported meshes without associated geometry. Abaqus/CAE provides a powerful modeling and visualization environment, emphasizing ease-of-use and ease-of-customization to capture and reuse proven analysis workflows. With Abaqus/CAE, dedicated expert and occasional novice users alike can create, edit, monitor, diagnose, and visualize basic to complex finite element or multiphysics analyses.

Feature-based, parametric modeling also makes Abaqus/CAE a highly efficient and effective pre- and postprocessor for every analysis need. Abaqus/CAE enables users to leverage the complete range of Abaqus analysis functionality, such as acoustics, connectors, damage, fracture, and failure. Familiar Abaqus concepts such as steps, interactions, sections, materials, and amplitudes make the user interface highly intuitive.

Geometry

Parts and assemblies can be created in Abaqus/CAE using the constraint-driven sketcher, or they can be imported from CAD systems using neutral file formats. In addition, CAD associative interfaces for CATIA V5, SolidWorks, and Pro/ENGINEER allow the synchronization of CAD and CAE assemblies and rapid model updates with no loss of user-defined analysis features. A comprehensive range of geometry creation and repair tools are available for querying, editing, repairing, and defeaturing geometry.

Meshing

Abaqus/CAE offers a comprehensive meshing environment and provides a variety of sophisticated approaches for simplifying and speeding up mesh creation. An example is the automatic virtual topology toolset, which can be used to ignore unnecessary geometric details in a model. Structured, swept, free, and bottom-up hex meshing techniques enable high-quality meshes to be generated rapidly on complex geometries. All Abaqus elements and options can be used, including continuum and gasket elements and adaptive remeshing.

Analysis Procedures

The Abaqus Unified FEA product suite is routinely employed by leading companies in industries as diverse as automotive, aerospace, defense, biomedical, and consumer products. A strong tradition of robustness and accuracy, combined with high performance and ease-of-use, has made Abaqus FEA software indispensable for simulating the behavior of products in real-world conditions.

Abaqus/Standard employs solution technology ideal for static and low-speed dynamic events where highly accurate stress solutions are critically important. Examples include sealing pressure in a gasket joint, steady-state rolling of a tire, or crack propagation in a composite airplane fuselage. Within a single simulation, it is possible to analyze a model both in the time and frequency domain. For example, one may start by performing a nonlinear engine cover mounting analysis including sophisticated gasket mechanics. Following the mounting analysis, the prestressed natural frequencies of the cover can be extracted, or the frequency domain mechanical and acoustic response of the prestressed cover to engine-induced vibrations can be examined.

Complementing Abaqus/Standard, the explicit solution technology employed by Abaqus/Explicit is well-suited for high-speed dynamic events such as consumer electronics drop testing, automotive crashworthiness, and ballistic impact. The ability of Abaqus/Explicit to effectively handle severely discontinuous behavior such as contact also makes it very attractive for the simulation of quasi-static events such as rolling of hot metal, sheet stamping, and slow crushing of energy-absorbing structures.

The Abaqus Unified FEA product suite allows transferring results between different solution technologies when appropriate. For example, Abaqus/Explicit can be used to predict stresses that arise during manufacturing; then these stresses can be used as the starting point for an assembly analysis in Abaqus/Standard. The assembled product could then be subjected to severe misuse loading in Abaqus/Explicit.

Advanced Materials

Increasing regulation, environmental concerns, and the need for lighter, more efficient products are forcing designers to seek out new materials such as lead-free solder, composites, and plastics. The Abaqus Unified FEA product suite provides an extensive library of material models that can be used to simulate the realistic behavior of such modern materials in addition to more traditional engineering materials such as metals and rubber.

Delamination of a composite panel, tearing of a spot weld, cracking in an electronics ball grid array, and debonding of an adhesive joint all involve initiation and propagation of damage. In some situations, damage accumulation can lead to catastrophic failure. Abaqus provides a general framework for modeling progressive damage that can help predict such failures, thus enabling manufacturers to produce safer designs and avoid costly prototypes.

Modeling composite laminates and calibrating advanced material models has traditionally been time-consuming and error-prone. Abaqus includes state-of-the-art tools for composite lay-up modeling and interactive material evaluation that accelerate model set-up and reduce possible modeling errors.

Extending the Power of Simulation

Abaqus offers a range of innovative analysis techniques to simplify tasks and reduce overall development time. For example, the direct cyclic procedure provides a computationally efficient way to analyze steady-state response to cyclic loading, making it an attractive feature in electronics or powertrain applications where thermal fatigue is of concern. Adaptive remeshing alleviates uncertainty in mesh creation by automatically remeshing only where necessary to ensure accuracy.

Many applications today involve the interaction of multiple physical phenomena. For example, to design safe and effective drug-eluting stents, engineers must account for the interplay between blood flow and stent deformation. Other challenging applications include coupled piezoelectric-acoustic effects in the design of hearing aids and coupled thermal-electrochemical interaction in fuel cell design. To address these applications, Abaqus offers a wide range of multiphysics capabilities including thermal-electrical, structural-acoustic, and pore fluid flow-mechanical interaction. Abaqus also supports open standards code coupling, which allows users to couple both commercial and in-house CFD software with Abaqus to perform Fluid-Structure Interaction analysis.

Contact

Manufacturing processes, bolted assemblies, snap-fits, and impact events all involve interaction between contacting bodies. Understanding the behavior of components when they come into contact is critical to designing better products. The Abaqus Unified FEA product suite provides comprehensive contact modeling capabilities such as the ability to model interactions between deformable bodies, rigid bodies, and self-contact.

The general contact capability in Abaqus automatically detects contact between different bodies with little need for user intervention. Solution accuracy is improved by eliminating the possibility of missed contact definitions. This powerful capability dramatically reduces the time needed to define contact for complex assemblies such as those found in automobiles, aircraft, consumer products, and portable electronics.

Visualization

The comprehensive set of best-in-class visualization options in Abaqus/CAE helps users interpret and communicate analysis results. Visual diagnostics enable the progress of analyses to be tracked and assessed easily. Very large, complex models can be visualized using a wide range of options including contour, path, X–Y, and ply stack plots. In addition, results in specific regions can be evaluated using display groups, view cuts, and free body cuts.

Simulation results can be output in a variety of formats including images, reports, animations, and 3D XML files. Abaqus/CAE provides unique visualization and plotting capabilities for Abaqus FEA that are not available in other postprocessing products. These features enable rapid results extraction and visualization, which help users review simulation results efficiently and collaborate effectively with colleagues to drive product design decisions.

High-Performance Simulation

Reducing analysis turnaround time enables the examination of multiple and more detailed design scenarios. The distributed memory parallel direct solver technology in Abaqus/Standard sets a new benchmark for the industry, demonstrating impressive scaling and robustness. The parallel implementation of Abaqus/Explicit, based on domain decomposition technology, greatly accelerates the solution process and boosts productivity. Today, effective parallel scaling is available on up to 64 CPUs for a range of industrial applications; models that may take days to run on a single CPU can now be run overnight.

Abaqus/Standard also provides a framework for high-performance, large-scale linear dynamics powered by the Abaqus/AMS eigensolver. Combined with features such as substructures and material damping, Abaqus delivers a compelling solution for powertrain and body-in-white noise and vibration studies.

Customization

The capture and deployment of proven analysis workflows provides significant benefits to organizations. The Abaqus Scripting Interface, Abaqus GUI Toolkit, and user subroutines provide the ability to customize Abaqus.

From macros and plug-ins to complete vertical applications, users are able to automate basic and advanced analysis tasks and processes. This capability allows organizations to capture the expertise of skilled analysts and deploy robust analysis methods across their enterprise for routine use. Such automated environments enable experienced analysts and non-expert FEA users to share data and methods, ensuring that results are reliable and reproducible. The Abaqus scripting and automation capabilities provide an effective way to rapidly explore design options, which ultimately results in significant cost savings and product innovations.

Driving Innovation with Realistic Simulation

Harvesting Clean, Renewable Ocean Power

Pelamis Wave Power Ltd (PWP) is harnessing the renewable energy source of the ocean with their innovative Wave Energy Converter (WEC) machines. To generate electricity, the Pelamis machines are linked together in a "wave farm" on the ocean's surface. Hydraulic rams resist the motion of the waves and pump hydraulic fluid through electricity-producing generators. A wave farm of 40 Pelamis machines, covering a square kilometer of ocean surface, is capable of generating electric power for 20,000 homes.

Images courtesy of Pelamis Wave Power Ltd

To design an adaptable, rugged, and clean-running machine, PWP has selected Abaqus FEA software to evaluate the sub-parts of their machines for stress, contact, and fatigue. Pelamis engineers incorporate data from hydraulic system tests, electrical layouts, and production assembly while leveraging the extensive material modeling and nonlinear analysis capabilities in Abaqus to conduct realistic performance studies.

Pelamis engineers developed a novel joint configuration designed to create a tunable, cross-coupled resonant response to waves. This allows the machine to be "turned up" to increase power capture in small seas, and "turned down" to limit motion loads in rough seas. To assess the reliability of this unique response characteristic, PWP engineers performed a considerable number of FEA-driven design iterations and validated them with fatigue tests. Their efforts resulted in innovative Wave Energy Converters that are reliable, efficient, and environmentally sound.

Reference: "Harvesting the Ocean's Power with Pelamis and Realistic Simulation," SIMULIA *INSIGHTS*, January/February 2008.

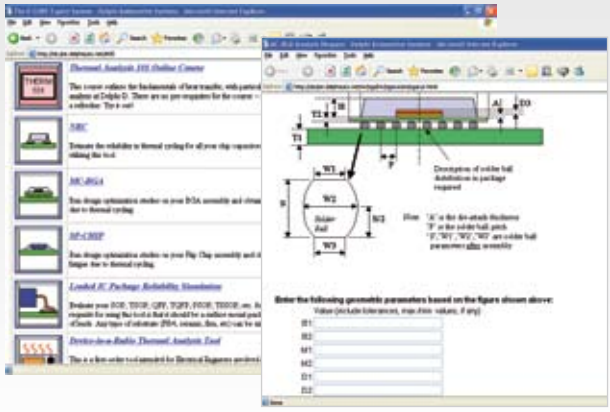
Image courtesy of Motorola

Accelerating Reliability Evaluation of Automotive Electronics

Delphi's Electronics & Safety division produces an array of electronic components that control a wide range of functions in today's automobiles, including entertainment, security, brakes, and powertrain systems. These electronic systems are subject to extreme heat and vibration, especially under the hood. Significantly, the largest driver of field failure in automotive electronics systems is repeated temperature cycling.

With the ability to produce one billion solder joints per day, it is critical for Delphi's design engineers to have the ability to evaluate the thermal cycling of their solder joints quickly and accurately. To support this high-volume electronics production, the Delphi simulation and analysis group developed a Web interface to Abaqus, called D-Cube, which enables their design engineers to input electronics-specific design parameters into an online form and submit their designs for thermal cycling analysis.

This innovative customized interface to Abaqus saves Delphi an enormous amount of time and cost and opens doors to design innovation. It enables design engineers to quickly explore different configurations for automotive electronics packages and select the optimal solution to meet their electronics reliability requirements.



Images courtesy of Delphi Corporation

Reference: "It's All in the Solder Joints," SIMULIA *INSIGHTS*, October/November 2007.

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

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.

DASSAULT SYSTEMES

www.3ds.com

NOV_SIM_UFEA_Y08