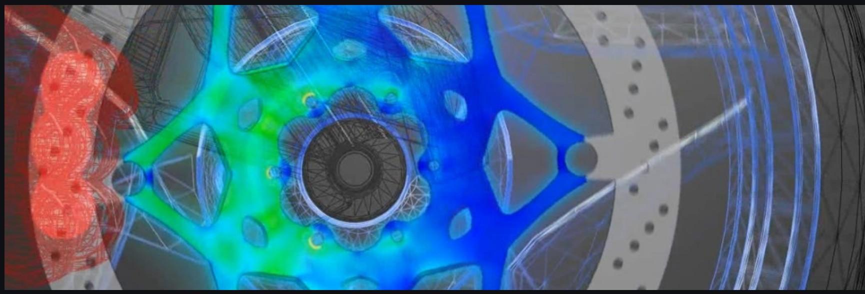


Inside Abaqus FEA

A technical capabilities guide
for simulation engineers

EDITION
2026

DOCUMENT
Technical Guide

PUBLISHER
GoEngineer

CONTENTS

What’s inside.

SECTION 01

About this guide.

This guide is a technically grounded walkthrough of what Abaqus can do, organized around the engineering problems you're trying to solve.

You'll start with Abaqus's solver architecture and how the implicit-explicit unified model works in practice. From there, the guide moves through analysis capabilities by problem type, the technical building blocks that cut across all analyses, and the three additional tools (e-safe, Tosca, and Isight) that ship with Abaqus at no extra cost and that most users haven't fully explored. The final sections cover the practical computing reality of running Abaqus effectively, how licensing works today, and how to get the most out of your relationship with an Abaqus value-added reseller.

This guide was written by GoEngineer's simulation engineering team. We don't just support Abaqus. We use it. Our FEA consulting practice serves global OEMs and tier-1 suppliers across industries, which means when we answer your support calls and run your training sessions, we bring our daily experience in production-grade simulation. That application depth is the reason this guide exists and is something a product datasheet or a direct vendor relationship won't provide.

SECTION 02

What Abaqus is and how it's structured.

Abaqus is a general-purpose finite element analysis suite with particular depth in nonlinear mechanics and multiphysics. It is broadly adopted in aerospace, automotive, medical devices, and energy for problems involving large deformations, complex contact, advanced materials, or coupled physics.

The software is built around two complementary solvers and a shared modeling environment. While Abaqus remains principally a locally-installed software for Linux and Windows systems, official first-party cloud compute services are now available.

Abaqus/Standard

Implicit solver. Assembles and solves the global stiffness system at every increment, enforcing full equilibrium before advancing.

Ideal for static stress, bolt preload, buckling, heat transfer, natural frequency extraction, and steady-state dynamics.

Abaqus/Explicit

Explicit dynamics solver. Advances the solution element-by-element in very small time increments without checking for equilibrium.

Ideal for crash, blast, drop test, high-speed forming, and processes involving material failure and fragmentation.

Abaqus/CAE

Integrated modeling environment for building/importing geometry, meshing, assigning properties and loads, submitting jobs, and visualizing results.

Supports geometry import from CATIA, SOLIDWORKS, NX, Creo, and other CAD systems.

Highly extensible and customizable through scripting.

3DEXPERIENCE Compute Cloud *(Optional)*

Upload analysis decks to the secure Dassault Systèmes cloud platform and run Abaqus/Standard or Abaqus/Explicit on up to 184 CPU cores.

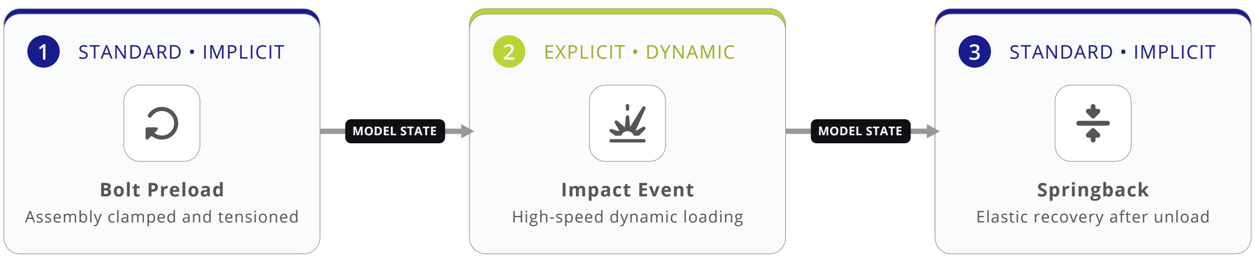
Two solvers, one model.

Many real engineering problems span both implicit and explicit analysis regimes — for example, a preloaded assembly subjected to shock, a formed part that springs back, or a pressurized vessel under blast.

THE PRACTICAL ADVANTAGE

Typically, doing these analyses requires buying two tools, modeling the assembly and scenario with each, and conducting a data conversion between the tools and their associated analysis steps. Abaqus provides everything in one package, eliminating the extra purchases, setup labor, and data exchange.

The analyst builds one model for everything in Abaqus/CAE. When Abaqus runs through the steps and switches solvers, the full solution state (displacement, stress, strain, contact, temperature, etc.) carries forward.



Four characteristics that determine which solver to use and when

1. Event time scale

Events lasting seconds to steady-state favor Standard. Events lasting microseconds to a few milliseconds favor Explicit. The gray zone is tens of milliseconds, or quasi-static processes with complex contact. These can go either way; total increment count and convergence difficulty are the tiebreakers.

2. Intermediate equilibrium

Standard enforces full equilibrium each increment, which is critical when the answer is a precise stress or displacement at a specific load level. Explicit marches forward in time without equilibrium checks, so it captures the dynamic response faithfully but doesn't guarantee static balance at any given frame.

3. Nature of the nonlinearity

Smooth, progressive nonlinearity (rubber compression, bolt preload, thermal distortion) converges well in Standard. Sudden, discontinuous events (impact, fragmentation, rapidly changing contact) stall Newton iteration and run more naturally in Explicit, where there is no system of equations to fail to converge.

4. Model size and compute scaling

Standard's cost grows faster than linearly with DOF count because it factors a global matrix. Explicit's cost per increment is linear in the number of elements, so very large models (millions of elements) may run faster in Explicit even for moderate-speed events — provided the stable time increment isn't prohibitively small.

Analysis capabilities by problem type.

Organized by the question you're asking: "I need to simulate X."

3a · Static and quasi-static stress analysis

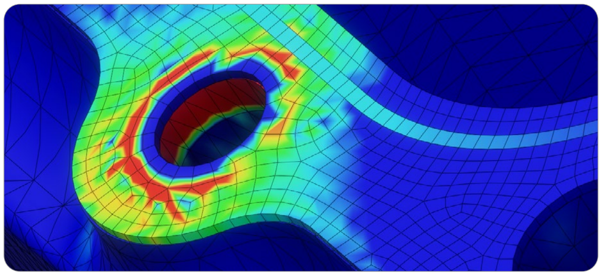
Linear elastic stress is straightforward. Where Abaqus earns its reputation is in **nonlinear static analysis**, i.e., large deformations, nonlinear materials, and contact. The solver uses a full Newton method with automatic increment sizing and cutback-retry logic designed to push through difficult increments.

For **buckling and post-buckling**: linear eigenvalue buckling, Riks arc-length method for post-buckling paths, and imperfection injection from mode shapes.

Direct cyclic analysis computes the stabilized response of structures under repetitive loading (ratcheting, shakedown, plastic accommodation, etc.) without explicitly solving every load cycle.

THE PRACTICAL ADVANTAGE

Fewer failed runs on contact-dominated problems. Less analyst time adjusting stabilization and increment sizes. The automatic controls are designed for the problems that consume the most debugging effort, like interference fits, gasket closure, or rubber seals deforming into complex contact patterns.



3b · Dynamic and impact analysis

Abaqus covers the full spectrum from natural frequency extraction through high-velocity impact.

LINEAR PERTURBATION ON A NONLINEAR BASE STATE

Frequency extraction, buckling, and steady-state dynamics can run as perturbation steps about a preloaded, deformed configuration, not just an unloaded geometry. Natural frequencies of a pressurized vessel, an inflated tire on a road surface, a bolted joint under service preload.

Modal dynamics

GPU-accelerated AMS, MPI-parallel Lanczos, and subspace eigensolvers. State-space transient response. Steady-state harmonic response (direct and modal). Random vibration PSD response with cross-spectral density input. Complex eigenvalue extraction for friction-induced instabilities.

Implicit dynamics

HHT-α time integration. Damp high-frequency noise without destroying the low-frequency response. Handles contact changes, material nonlinearity, and large rotation within the implicit framework. Substructure-based dynamic reduction for large assemblies with localized nonlinearity.

Explicit dynamics

Selective and enhanced mass scaling strategies to increase the stable time increment without distorting inertial response. Element deletion and erosion for material failure. Adaptive mesh domains. Parallel domain decomposition scaling across hundreds of cores.

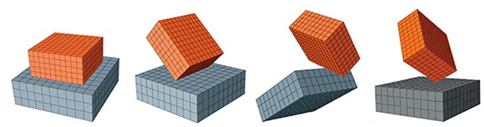
3c · Contact mechanics

General contact

One definition, all-inclusive. Can automatically detect and enforce contact between all surfaces, including self-contact. Individual surface pairs can still be assigned specific interaction behaviors within the general contact framework. For large assemblies, this substantially reduces setup effort and omission errors, and improves convergence robustness.

Contact pairs

The traditional contact formulation in Abaqus. Primarily used today for maintaining and re-running legacy models built with contact pair definitions.



Four contact scenarios. ***Read more on general contact here.***

Both support Coulomb friction, cohesive behavior, adhesive contact with damage, thermal/electrical conductance, and **Archard-law wear modeling** with cycle scaling (new in 2025).

THE PRACTICAL ADVANTAGE

General contact turns a multi-hour setup task into a single definition. For assemblies with dozens of parts, this can be the difference between a reliable analysis and one where a missed contact pair produces a non-physical result that takes days to diagnose.

3d · Heat transfer and coupled thermal-structural

- **Conduction, convection, radiation** with view factors.
- **Gap conductance** across contact interfaces.
- **Sequentially coupled thermal-stress** for problems where deformation doesn't affect the thermal field.
- **Fully coupled thermal-stress**, such as simultaneous solve for metal forming, friction stir welding, or brake discs.
- **Thermal-electrical-structural**, such as Joule heating for spot welding, fuse design, or connector heating.

3e · Fracture, damage, and fatigue

Fracture mechanic

J-integral, K factors, C*-integral, T-stress along predefined crack fronts.

Progressive composite damage

Hashin criteria with separate modes for fiber and matrix. CZone for crush simulation.

Cohesive zone modeling

Standard method for composite delamination, adhesive failure, interfacial debonding.

Ductile damage

Johnson-Cook, ductile, shear, FLD criteria with element deletion.

XFEM

Cracks grow through the mesh without remeshing.

FATIGUE LIFE PREDICTION

Handled by fe-safe (see Section 05), the fully-dedicated fatigue tool bundled with Abaqus.

3f · Multiphysics and specialty analyses

Acoustic-structural
Vehicle NVH, underwater acoustics, loudspeaker design, machinery noise.

Piezoelectric
Sensors, actuators, transducers, energy harvesting, MEMS.

Pore-fluid diffusion
Geotechnical consolidation, dam seepage, poroelastic tissue.

Mass diffusion
Hydrogen embrittlement — species transport coupled to stress.

CEL
Extreme deformation: FSI, sloshing, blast soil ejecta, bird strike.

SPH
Particle-based: fluid-filled containers, friction stir welding material flow.

> **CST STUDIO SUITE COUPLING**
>
> For electromagnetic-structural multiphysics, Abaqus can be coupled with CST Studio Suite, also part of the SIMULIA portfolio and accessible via the same SimUnit tokens. Contact GoEngineer for application-specific coupling workflows.

SECTION 04

The technical building blocks.

Capabilities that cut across all analysis types.

4a · Element library

Continuum (solid)
2D/3D, linear/quadratic, full/reduced integration. Hybrid for incompressible materials.

Gasket
Dedicated pressure-closure models for sealing applications.

Shell
Conventional and continuum. Finite-strain, composites layup.

Rigid body
Tools, dies, fixtures, ground planes. No deformation computed.

Beam
Euler-Bernoulli, Timoshenko. Pipe elements with pressure.

Acoustic
Fluid domains for sound-structure coupling and interior/exterior acoustics.

Cohesive
Traction-separation for adhesives, interfaces, delamination.

Membrane & truss
No bending stiffness. Fabric, tension-only structures.

Connector
Springs, dampers, bushings, hinges — without meshing hardware.

User-defined (UEL/VUEL)
Custom Fortran element formulations.

4b · Material models

Metal plasticity

Von Mises with isotropic, kinematic, and combined hardening. Rate-dependent (Johnson-Cook, Cowper-Symonds). Porous metal (Gurson). Cast iron. Creep for elevated temperature.

Hyperelasticity

Mooney-Rivlin, Ogden, Arruda-Boyce, neo-Hookean, polynomial, Yeoh, Van der Waals, Marlow. Automatic stability evaluation warns of non-physical behavior beyond test data.

Biological tissue (HGO)

Holzapfel-Gasser-Ogden anisotropic hyperelastic for fiber-reinforced tissue — arterial walls, valves, ligaments. A key reason Abaqus is standard in cardiovascular device simulation.

Shape memory alloy

Built-in Nitinol superelastic model: stress-strain plateau, temperature-dependent transformation, hysteresis. Critical for stents, guidewires, orthodontic devices.

Viscoelasticity

Prony series in time or frequency domain, plus the nonlinear parallel rheological framework for more complex rate-dependent behavior. Polymers, damping materials, biological tissues.

Concrete

Damaged plasticity (CDP) with crushing, cracking, stiffness degradation and recovery.

Foam & fabric

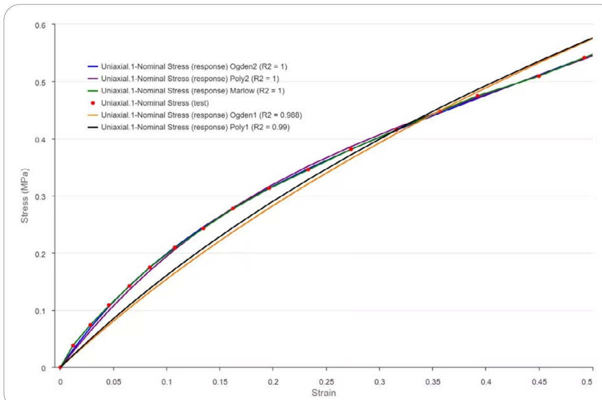
Crushable foam for packaging and crash padding. Low-density foam. Woven/non-woven fabric for airbags, parachutes, protective textiles.

Speciality

Gasket — pressure-closure curves. **Equation of state** — Mie-Grüneisen, JWL. **Concrete smeared cracking** for monotonic loading.

FROM TEST DATA TO MATERIAL CARD

The **3D**EXPERIENCE Material Calibration App simplifies translating physical test data into calibrated Abaqus material parameters, replacing complex Excel-based curve-fitting. It exports .inp material cards directly for desktop Abaqus jobs. Ask GoEngineer for a demo.



Abaqus material correlation study with GoEngineer and Axel Products. ***Read the article here.***

THE PRACTICAL ADVANTAGE

When your physics involves non-standard materials (Nitinol stents, anisotropic tissue, crushable foams, high-strain-rate metals) having validated built-in models means you start analyzing immediately rather than spending weeks writing and debugging custom material code.

4c · User subroutines and Python scripting

User subroutines (Fortran)

Compiled and linked at runtime:

- **UMAT / VUMAT**: Custom material constitutive behavior.
- **UEL / VUEL**: Custom elements with user-specified DOFs.
- **DLOAD / VDLOAD**: Loads varying with position, time, or solution state.
- **USDFLD**: Redefine field variables as functions of solution state.
- **UAMP, FILM, HETVAL, UHYPER**: Feedback control, convection, heat generation, custom hyperelastic potentials.

Python scripting

Complete CAE functionality as a Python API:

- **Parameterized model generation**: build from dimension tables, loop through variants.
- **Batch job management**: submit, monitor, collect results from hundreds of runs.
- **Automated post-processing**: extract results, generate reports, compare against criteria.

THE PRACTICAL ADVANTAGE

You adapt the tool to your physics, not the other way around. Organizations with proprietary material models or novel physics can implement them in the same solver, avoiding the cost of approximating their behavior to fit built-in options.

4d · Meshing and model preparation

Geometry import

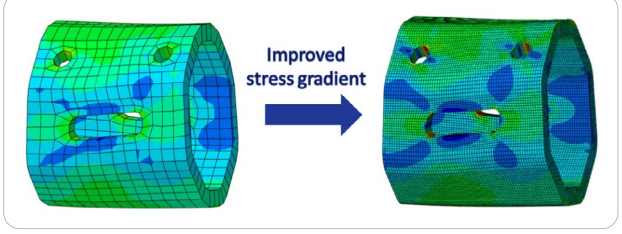
STEP, IGES, CATIA V5, SOLIDWORKS, NX, Creo, Parasolid, ACIS. Defeaturing and virtual topology for problematic geometry.

Submodeling

Drives a refined local model from global results for detailed stress extraction at features of interest — fillets, weld toes, thread roots.

Mesh generation

Free (tet/tri), structured (hex/quad), swept, bottom-up. Adaptive remeshing. Mesh-to-mesh solution mapping for re-analysis on modified geometry.



Submodeling a coarse model for dramatic stress contour improvement. ***Read the how-to article here.***

SECTION 05

The included SIMULIA portfolio.

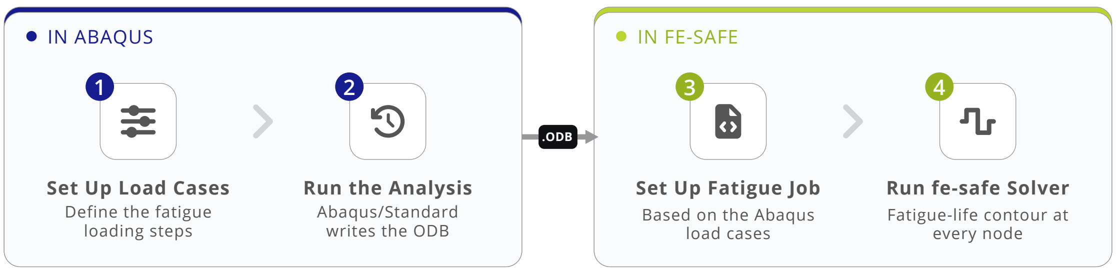
Three tools bundled with traditional Abaqus licenses at no extra cost.

INCLUDED

fe-safe — Fatigue life prediction

fe-safe reads stress or strain tensor histories from Abaqus ODB files, applies cycle counting and fatigue damage models, and produces fatigue-life contour plots at every node — the same visual format as Abaqus results.

fe-safe replaces hand calculations with tabulated SN data, spreadsheets applying Miner’s rule to simplified histories, and conservative screening that adds weight. fe-safe works from the full FEA output: every node, every time step, every direction.



Fatigue analysis methods

Stain-life (low-cycle)
Critical-plane criteria: Brown-Miller, Fatemi-Socie, SWT, max principal strain.

Variable-amplitude & multiaxial
Full load histories with rainflow counting and cumulative damage. This is where spreadsheet methods break down.

Stress-life (high-cycle)
SN curves with Goodman/Gerber/Walker mean-stress corrections, extended to multiaxial.

Specialty fatigue
- Weld: structural/notch stress
- Creep-fatigue: elevated temperature
- Vibration: PSD inputs
- Elastomer: seals, bushings, mounts.

Endurance (infinite life)
Dang Van, Crossland: “doesn’t crack, ever.”

ALSO WORKS WITH

fe-safe supports results from other major FEA codes. If your organization runs multiple solvers, fe-safe provides a single fatigue workflow across all of them.

LEARN MORE AT GOENGINEER.COM/FE-SAFE

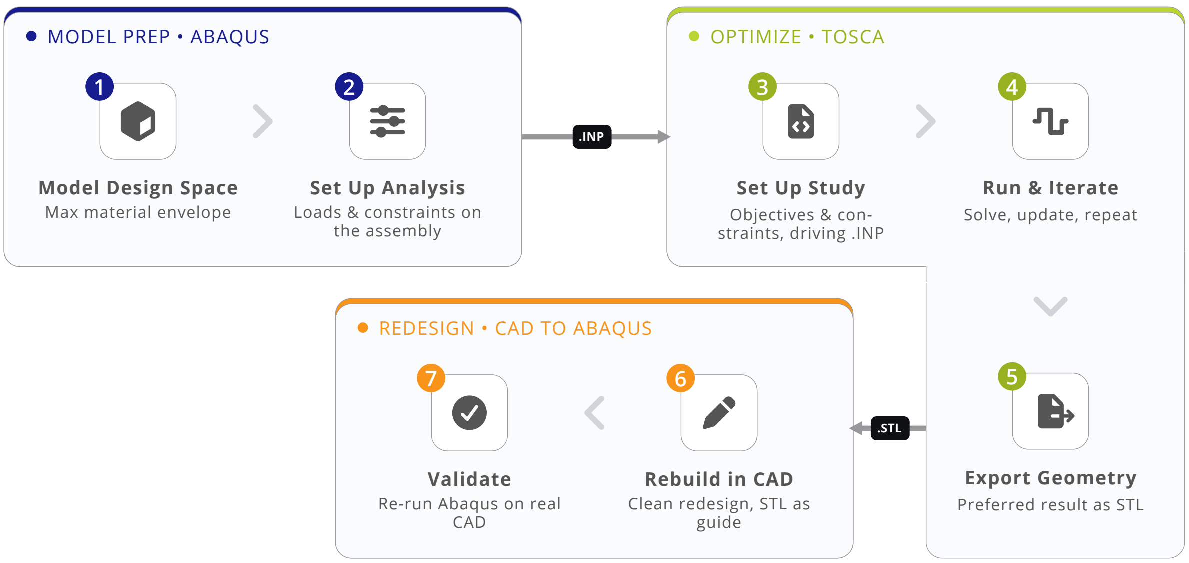
Visit our fe-safe page for additional details.

INCLUDED

Tosca — Structural optimization

Tosca drives Abaqus through iterative solve-update-solve loops to find better designs. It replaces manual topology exploration, weight reduction by trial-and-error, and conservative over-design when systematic optimization wasn't worth the effort. Without Tosca, most engineers explore a tiny fraction of the design space.

How it works is that you define the design space and the driving load cases in Abaqus. You define goals and constraints in Tosca. This optimization loop runs automatically. The optimized mesh-based shape then requires engineering interpretation into manufacturable CAD geometry, a manual step that is part of the design process.



Optimization methods

Topology optimization

Where should material exist? Start with a maximum envelope, specify loads and keep-out zones. Tosca removes non-contributing material. Directly relevant to lightweighting and additive manufacturing.

Bead optimization

Reinforcement bead patterns for sheet metal. Maximize stiffness or shift modes within manufacturability constraints.

Shape optimization

Move existing surface nodes to reduce stress concentrations or increase stiffness across the full stress field simultaneously.

Sizing optimization

Vary shell thicknesses or beam cross-sections to meet targets at minimum weight.

ALSO WORKS WITH

Tosca supports other FEA solvers (and even fatigue solvers) as the structural engine. Teams running mixed-code environments can use one optimization tool across their solver portfolio.

INCLUDED

Isight — Process automation and design exploration

Isight is a process integration and design exploration tool for automating complex workflows across multiple tools, including Abaqus. It replaces manual parametric iteration, trial-and-error optimization, and worst-case-corner robustness analysis that is overly conservative and misses parameter interactions.

Isight drives everything via its Python scripting interface — parameterizing model inputs, submitting jobs, and extracting results automatically. You build the workflow once in Isight's visual editor; it manages execution across design points. Use its interactive post-processing tools to analyze results from multiple perspectives.

What Isight does for the analyst

Process automation

Chain CAD update > mesh > solve > post-process into one automated workflow. A 50-point parametric study that takes a week manually runs overnight.

Surrogate modeling

Build response surfaces (kriging, RBF, polynomial) from a DOE. Invest in 50–200 Abaqus runs, then explore thousands of points in seconds.

Design of experiments

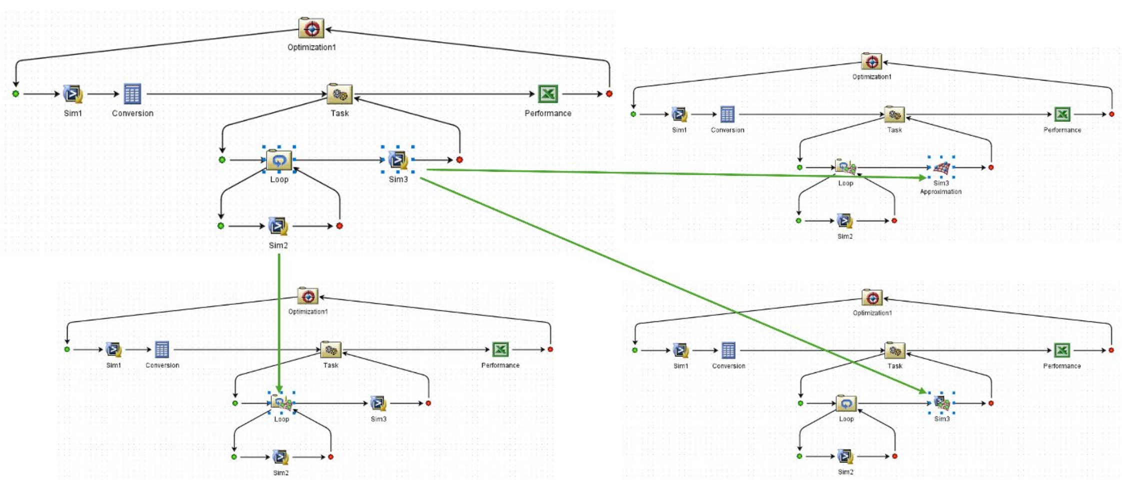
Full factorial, Latin hypercube, optimal space-filling. Map which parameters influence performance and where interactions exist.

Robustness / Six Sigma

Propagate manufacturing tolerances and material scatter. Answer "what fraction of parts will pass?" not just "does the nominal design pass?"

Optimization

Gradient-based, genetic/evolutionary, multi-objective (NSGA-II). Find the Pareto front when objectives conflict.



A complex automated simulation workflow in Isight.

ALSO WORKS WITH

Isight can wrap any tool, including other FEA codes, CAD systems, spreadsheets, and in-house scripts, in the same automated workflow. It is not limited to Abaqus.

LEARN MORE AT GOENGINEER.COM/ISIGHT

Visit our Isight page for additional details.

SECTION 06

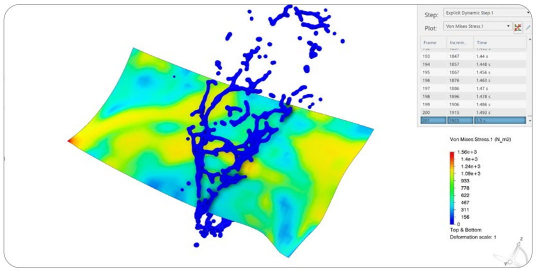
Industry applications.

How these capabilities map to real engineering problems

AEROSPACE

Composites, certification, virtual testing

Hashin progressive damage, CZM/VCCT delamination for CFRP. CZone crush. Bird strike. Bolt preload. Creep-fatigue in turbine hot sections. Acoustic-structural for cabin noise. Widely used in certification workflows.

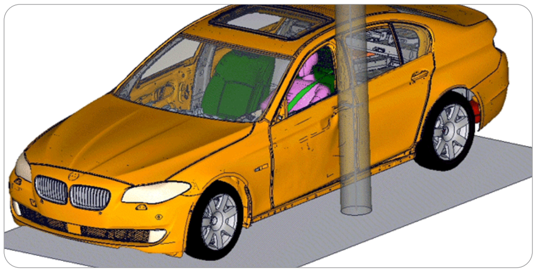


*Bird strike analysis post-processed on **3D**EXPERIENCE.*

AUTOMOTIVE

Crash, NVH, durability, forming

Full-vehicle crash. Modal dynamics, frequency response, brake squeal. Metal forming and springback. Durability via fe-safe. EV battery crush and thermal runaway. Tosca for lightweighting.

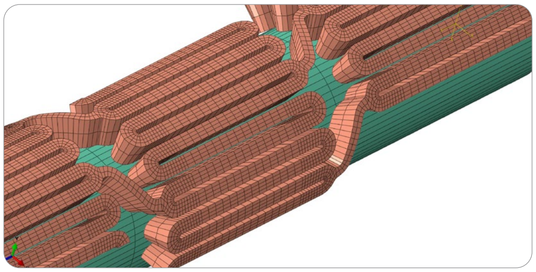


Full vehicle impact with crash dummy.

LIFE SCIENCES

Medical devices, FDA submissions

Stent lifecycle: crimping, deployment, 400M-cycle fatigue. Built-in Nitinol and HGO tissue models. Orthopedic implant stress. Suitable for FDA computational modeling submissions (ASME V&V 40).

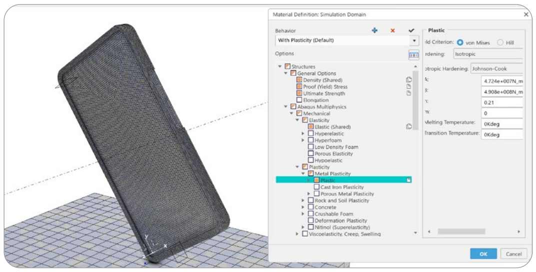


FE model of a stent, preprocessed in Abaqus/CAE.

HIGH TECH

Drop test, thermal, assembly stress

Phone/tablet/laptop drop test. PCB shock, solder joint reliability. Semiconductor warpage. Thermal management. CST Studio Suite for EM via same SimUnit pool.

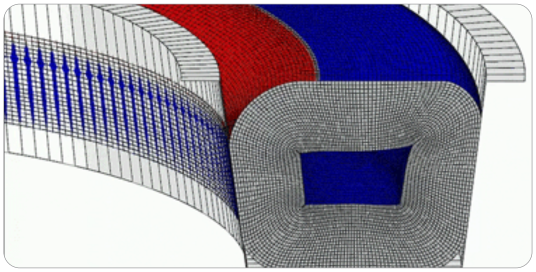


*Cell phone impact model setup with **3D**EXPERIENCE.*

INDUSTRIAL EQUIPMENT

Sealing, fatigue, thermal-mechanical

Gasket sealing with dedicated elements — bolt-load sequence, pressure-closure, pressure-penetration. Bearings, seals, O-rings with hyperelastic materials. Thermal-mechanical fatigue and creep. Weld residual stress. Steady-state rolling.

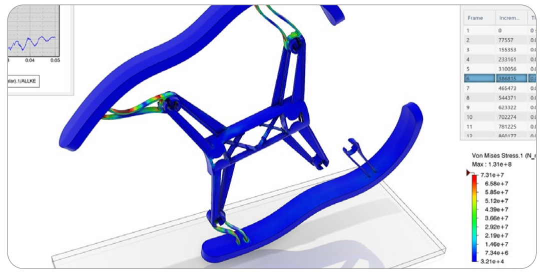


O-ring seal with advanced contact modeling.

CONSUMER GOODS

Impact durability, material performance

Drop/impact for packaging and durability. Polymer viscoelastic creep, stress relaxation. SPH/CEL for fluid-filled container impact. Snap-fit and press-fit assembly. Tosca for material-efficient structures.



*Ground impact of consumer drone in **3D**EXPERIENCE viewer.*

See it on your geometry.

GoEngineer can benchmark Abaqus on your actual models, loads, and failure modes.

Request a Benchmark

Compute: running Abaqus effectively.

Hardware, GPU, OS, and cloud — the practical computing reality.

CPU

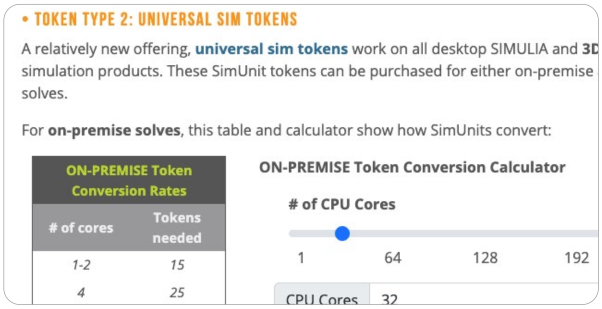
Abaqus runs on a single core but scales well with additional cores. Diminishing returns from parallel overhead are expected, and explicit simulations generally scale better than implicit ones. Core speed is also important; higher clock speeds improve performance more linearly without the overhead of multi-core processing. This creates a tradeoff: more cores usually means lower individual clock speeds.

L3 cache significantly impacts Explicit performance. AMD's 3D V-Cache processors show substantial benefits, particularly in distributed computing.

GPU acceleration

Standard: GPU acceleration for the direct sparse solver has been available for several years, with reported speedups of 2–3.7× on large linear models. Requires FP64-capable NVIDIA GPUs (H100, A100, Quadro GV100) — most consumer and CAD-oriented graphics cards lack double-precision compute capability. These cards are typically five-figure investments.

Explicit: GPU acceleration was introduced in Abaqus 2026 FD01, marking a significant milestone that made Abaqus the first major commercial explicit FEA code with GPU compute support. This capability is actively developing; consult GoEngineer for current guidance.



GoEngineer has Abaqus token calculators for both legacy and SimUnit licenses. Try them [here](#).

Memory

Sufficient RAM is critical. Large implicit models can require 64 GB to well beyond 256 GB depending on model size, element type, and contact complexity. Insufficient memory forces disk paging and can slow performance dramatically. Your Abaqus provider should help you benchmark memory requirements against your specific model profile before you buy hardware.

Operating system

Recent Intel architectures (big.LITTLE for Core, Speed Select for Xeon) have made OS choice relevant to performance. For new Intel CPUs, Linux is strongly recommended — Windows will be significantly slower. AMD Ryzen, Threadripper, and EPYC remain relatively OS-agnostic.

Cloud execution

Dassault Systèmes provides official Abaqus cloud compute via SimUnit tokens or credits. Cloud solves access up to 184 cores per simulation with unlimited concurrent jobs.

The emerging best practice is a **hybrid approach**: a comfortable local compute setup with a modest token pool for daily work and debugging, supplemented by on-cloud credits for large or time-critical solves. This gives you HPC-level performance ceilings without the capital investment in dedicated server infrastructure and the full licensing to match.

GET THIS RIGHT

System configuration can mean the difference between an analysis taking hours or days. GoEngineer's computing blog provides component-by-component guidance, and our team can advise on your specific workloads before you buy. **[Read it here.](#)**

Licensing.

How Abaqus is licensed today.

How a new customer buys Abaqus

A new deployment has two components: **front-end access** (the Abaqus/CAE environment, with fe-safe, Tosca, and Isight bundled) and **SimUnit tokens or credits** to solve. Abaqus solve licensing, whether tokens or credits, on-premise or on-cloud, scales with the number of CPU cores and GPU devices used.

Tokens (sustained use)

A fixed pool of SimUnit tokens. Each solve checks out tokens based on core count via a decaying function. More cores require proportionally fewer additional tokens.

Available for **on-premise** and **on-cloud** use.

Best for consistent usage.

Credits (pay-as-you-go)

Credits deplete while the solver runs. More cores deplete faster but finish sooner. Sold in packs of 1,000.

Also available **on-premise** and **on-cloud**.

Best for infrequent or fluctuating usage, or supplementing a token pool for occasional large jobs.

Mix and match

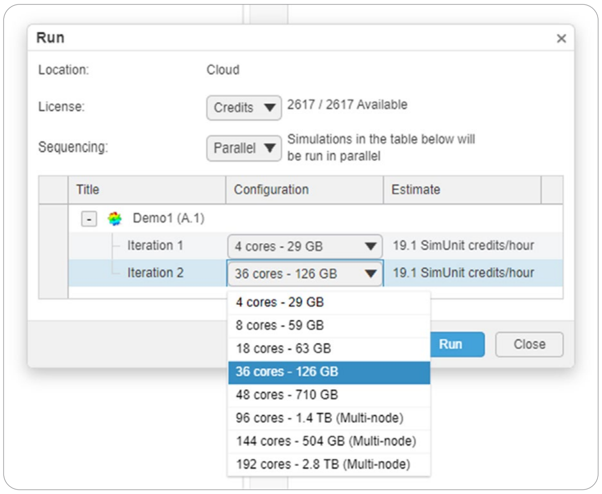
The four modes — on-prem tokens, on-cloud tokens, on-prem credits, on-cloud credits — **can be combined**.

A common arrangement: a modest on-premise token pool for daily work, supplemented by on-cloud credits for large or time-critical solves.

ABAQUS & BEYOND: UNIVERSAL LICENSING ACROSS PHYSICS

SimUnit tokens and credits work across all desktop SIMULIA and **3D**EXPERIENCE simulation products: Abaqus, CST Studio Suite, PowerFLOW, and others. Buy front-end access to additional products and solve with the same SimUnits.

You can read more about SimUnit universal licensing in [our article on the topic].



Abaqus cloud launch interface. ***[Read the how-to article here.]***

NOTE ON 3DEXPERIENCE PACKAGING

This guide focuses on desktop Abaqus, which includes Abaqus/CAE and the bundled fe-safe, Tosca, and Isight. Abaqus is also available as a **3D**EXPERIENCE product, which has its own pre/post environment and integrates Abaqus models and results into the **3D**EXPERIENCE PLM structure. This version does not bundle fe-safe, Tosca, and Isight (but even more powerful and CAD-integrated **3D**EXPERIENCE versions exist for separate purchase). Talk to GoEngineer about which packaging fits your team.

SECTION 09

Getting the most out of your Abaqus investment.

How to leverage a value-added reseller to accelerate your team's effectiveness.

An Abaqus license gives you the software. How quickly your team becomes productive in it (and how much of the toolset you actually use) depends largely on the support infrastructure around it. A value-added reseller exists to provide that infrastructure. Here's how to take full advantage of it.

Demand working technical support.

Your VAR's support engineers should be practicing simulation analysts, not ticket routers. When you submit a convergence failure or a contact problem, expect the person on the other end to have seen that failure mode in their own consulting work — not just read about it. GoEngineer's support team works across the full SIMULIA portfolio and draws from direct experience with production-grade simulation for global OEMs and tier-1 suppliers.

Bring in specialists for high-stakes work.

Some analyses, like regulatory submissions, certification-critical load cases, failure investigations, or novel physics your team hasn't tackled before, need to be right the first time. These are the situations where engaging your VAR's consulting engineers pays for itself. The cost of a consulting engagement is almost always less than the cost of a wrong answer on a problem that matters.

Take advantage of dedicated mentoring.

GoEngineer Abaqus licenses include four complimentary application-mentoring sessions per year. These are scheduled, one-on-one working sessions with a simulation engineer. Bring your model, your convergence problems, and your workflow questions. Engineers who use these sessions consistently reach proficiency faster and discover capabilities they didn't know they had access to. This is a GoEngineer-specific offering, and most customers under-utilize it.

Get computing guidance before you buy hardware.

As Section 07 makes clear, system configuration for Abaqus is non-trivial and the wrong choices cost real money and time. Your VAR should benchmark your specific model profile against hardware options and help you size a compute setup — including the local/cloud hybrid approach — before you purchase. This routinely saves tens of thousands in hardware decisions.

Let your VAR model your licensing.

The token/credit/on-prem/on-cloud matrix has real tradeoffs that depend on your usage patterns. Your VAR should help you configure a licensing mix that fits your workload, and re-evaluate as your usage evolves. Over-licensing wastes budget; under-licensing wastes analyst time waiting for tokens.

Treat implementation as a project, not a purchase.

Getting from "we bought Abaqus" to "our team is productive in Abaqus" takes deliberate effort, including installation, license server configuration, workflow integration, initial training, and the first real production models. Your VAR should manage this process so your team reaches competence in weeks, not months.

Ready to **evaluate**?

Talk to a GoEngineer simulation specialist about your specific models, loads, and engineering questions.

Get in Touch

WEB
goengineer.com

PHONE
800.688.3234

EMAIL
info@goengineer.com