"The real world is nonlinear"... 7 main Advantages using Abaqus

FEA SERVICES LLC ©
6000 FAIRVIEW ROAD, SUITE 1200 | CHARLOTTE, NC 28210
704.552.3841
WWW.FEASERVICES.NET
AN OFFICIAL DASSAULT SYSTÈMES VALUE ADDED PARTNER

NONLINEAR PERFORMANCE

CONTACT MODELING

EFFICIENT SUBSTRUCTURES

MULTIPHYSICS

EXTREME DEFORMATION

FRACTURE/FAILURE

DEVELOPMENT/SUPPORT

1. NONLINEAR PERFORMANCE

As opposed to Nastran or other popular Finite Element codes that originated in the 60's or 70's, Abaqus was written from the ground up, including the basic element formulations themselves, with fully nonlinear equations rather than with linear assumptions and simplifications as was done by the majority of other codes. This core foundation enables the developers of Abaqus to create much more realistic FEA tools to simulate virtually any form of complex real world physics.

In the 70’s and 80's, smaller linear FEA models were perhaps an industry standard due to the high cost of even mild computing power available to most enterprises. In the 90’s, CPU’s and RAM began to trend downward in cost such that a paradigm shift in terms of “required realism” began. Today, even laptops can run a nonlinear analysis, hence computing power is generally no longer a main issue from a real user perspective.

Still though, FEA models do take time to run. The strength of the Abaqus code, which was originally developed as a nonlinear solver, is that it will run any nonlinear simulation faster and will converge on truer, more realistic results than other codes. These simulations, which represent the real world, can be done more simply and quickly with Abaqus than its competition.

2. CONTACT MODELING

Real applications are made of assemblies, not single parts. The parts of these assemblies operate by coming into contact with each other. Abaqus is by far the best FEA code at handling all forms of contact. Also, its CAE GUI now makes it elegantly simple for a typical user to set up many contacts in the FEM.

Other programs demand that the user set a myriad of contact parameters for convergence, and often it takes iterations of trial and error for difficult contact cases. Abaqus, however, automatically sets internal controls for its parameters such that a converged surface contact result occurs during the solver run, even without user influence. The FEA user thus has high confidence that the contact convergence actually works consistently.

The Abaqus reputation for accurate contact results is well documented. Contact modeling and solution times are always faster with Abaqus, given the nature of the base code and the continued development being made to the algorithms.

3. EFFICIENT SUBSTRUCTERS

Even with modern computers, substructures are still utilized in some industries to manage larger FEA models; Abaqus supports conventional substructure processes such as superelements.

What has set Abaqus apart from other FEA codes for many years however, is its robust submodeling capability. Submodels are needed for highly refined results in smaller areas for certain cases such as fatigue assessment. The code internally searches to find the boundary results from the global FEA and automatically applies these conditions to the (more refined) submodel. The process is also mesh independent such that the user has no need to plan a mesh strategy and may change the submodel mesh without the need to re-run the global model.

The Abaqus submodel process is elegantly simple enough such that the user often does not even need to plan ahead on where the submodel might be necessary; the process literally can be done "on the fly". Setting up a functioning submodel from a global model is generally performed in minutes.

In recent years, the Abaqus submodeling capability was further developed to function with nonlinear analysis whereas other codes are still limited to linear only.

4. MULTIPHYSICS

Abaqus performs nearly all forms of multi-physics FE analysis, while the required operations are supported by the Abaqus/CAE GUI. With Abaqus/CAE, users can seamlessly operate multi-physics and co-simulation techniques between the various I/O data sets and between the solvers.

Abaqus allows for simultaneous interactive analysis between solution types such as running a thermal-mechanical analysis coupled into one run. Of course, a de-coupled thermal-mechanical process, as is the limitation of some other codes, is also available.

Fluid-structural interactions (FSI) can be performed within a co-simulation routine either by using the internal Abaqus/CFD solver or by linking to popular third party CFD codes. The FSI co-simulation is one where the mechanical deflections alter the fluid flow, and vice versa.

One main benefit of using Abaqus is the ability to operate between the Implicit solver, generally used for stress/strain, and the highly dynamic Explicit solver, for a seamless Dynamic-Static co-simulation. Being under the one code (Abaqus) for more than 2 decades, these two (significantly different) solvers share the same GUI, compatible elements, and similar modeling methods.

Highly advanced forms of multi-physics involving extreme deformation, such as with CEL and SPH, are discussed in the following section.

5. EXTREME DEFORMATION

Extreme deformation covers a wide variety of situations:

One form of extreme deformation is very high plastic deformation of metals, or engineering plastics, etc. "Extreme" in this case might be thought of as "so much permanent deformation that we can visually see excess gross bulk deformation". Strains can be on the order of 50-100%, for example. Applications include deep drawing, stamping of metals, or the flowing of metal in a high temperature brazing operation, to name just a few manufacturing cases. Considering some field applications, this capability extends into realistic crash simulation of automobiles, or visualization of extreme post-buckling as encountered in many industries. Another example from the life sciences industry is the highly nonlinear behavior of stents. For all of these cases, the basic Abaqus package has been well suited for nearly 30 years to manage these simulations due to its intrinsic nonlinear code.

Extreme deformation also involves newer technology such as the usage of CEL - Coupled Eulerian Lagrangian. The Lagrangian description is the typical FE formulation whereby nodes are fixed within the material and the mesh becomes distorted with high strain gradients. The Eulerian description is where nodes stay fixed while material flows through the mesh. With the CEL technology, which is also built into the base Abaqus package, such extreme deformations can be simulated such that that even fluids are simulated in fluid-structural interactions. One example is the analysis of tires rolling through water in a hydroplaning simulation. Hot forging would be an example of simulating extreme solid deformation with CEL.

While CEL has been around for over 10 years, one of the latest SIMULIA development in the advancement of extreme deformation simulation is SPH - Smooth Particle Hydrodynamics. The usage of SPH would include: Wave Engineering, Impact Fracture (ballistics, shattering, fragmentation), Spraying, Snow Compaction, Bird Strikes, and of course numerous other possibilities.

The actual tearing or cracking of material is an extension of extreme deformation and is discussed below in the following section.

6. FRACTURE AND FAILURE

Simulation of “failure”, meaning material rupture or crack growth, is used to optimize certain manufacturing processes such as metal cutting, or to evaluate the potential detriment to product performance due to small damages from manufacturing. Of course, field failures can be analyzed to simulate the post-failure effect on the entire system, potentially evaluating the design of “fail safe” devices, similar to using inelastic post-buckling analysis.

Abaqus offers a general framework for modeling bulk material damage and failure over a wide range of materials (composites, metals, concrete, etc.) and structures. This framework allows simulation of damage initiation and evolution without the need for specifying any initial imperfection in the structure. One example is the advanced ductile failure technique which allows for ductile metal failure simulation while taking into account the stress triaxiality effect on ultimate strength.

Abaqus naturally offers classic damage tolerance capabilities to model discrete cracks. Stationary crack modeling allows for evaluation of fracture parameters such as J-integral, T-stress, stress intensity factors for three different modes, crack propagation direction and Ct stresses.

Crack propagation modeling allows for analysis of crack initiation and crack growth. The Cohesive Zone Method (surface-based and element-based) and Virtual Crack Closure Technique (VCCT) are available in both Abaqus/Standard and Abaqus/Explicit which models crack propagation along predefined interfaces, such as bonded sheets and other composites.

Crack propagation modeling based on the Extended Finite Element Method (XFEM) allows for simulation of both stationary and moving cracks. Evaluation of crack contour integrals with XFEM is available for general three dimensional models. The XFEM approach overcomes the main limitations of the other techniques: the crack location and geometry can be truly independent of the mesh; and the crack growth path need not be known in advance.

As mentioned under the Extreme Deformation topic, the newer SPH technology has been showing great efficiency in simulating certain failure and post-failure cases such as shattering due to high speed impact. Other codes simply cannot match these kinds of highly nonlinear failure simulations.

7. DEVELOPMENT AND SUPPORT

One of the reasons for the significant growth in the simulation capabilities as well as the market share for Abaqus in all areas of technology is its continued large-scale technical development. Today there are approximately 250 engineers and scientists employed by Simulia, of which about 60% of these resources are dedicated to adding more capabilities and physics to the Abaqus code while continuously improving the existing technologies. Even the already robust contact modeling techniques are being improved upon for future releases.

Technical support for your own Abaqus modeling issues is available both by live direct phone line as well as a traceable online database system. There are approximately 100 engineers dedicated to technical customer support throughout the Simulia offices in the United States. These engineers communicate together to tackle any specific issues a customer may have, thus by contacting your local Simulia office you gain access to the knowledge of the whole team.

Further, the entire Abaqus user community contributes documented examples of using Abaqus for unique applications. This documentation, amounting to thousands of articles and white papers, is available through the public 3ds.com site.

FEA SERVICES LLC ©
6000 FAIRVIEW ROAD, SUITE 1200 | CHARLOTTE, NC 28210
704.552.3841
WWW.FEASERVICES.NET
AN OFFICIAL DASSAULT SYSTÈMES VALUE ADDED PARTNER