ABAQUS Company Profile

About ABAQUS, Inc.

Founded in 1978, ABAQUS, Inc. is the world's leading provider of advanced Finite Element Analysis software and services that are used to solve real-world engineering problems. The ABAQUS software suite has an unsurpassed reputation for technology, quality, and reliability and provides a powerful and complete solution for both routine and sophisticated linear and nonlinear engineering problems. ABAQUS delivers a unified FEA environment that is a compelling alternative to implementations involving multiple products and vendors. In October 2005, ABAQUS became a wholly owned subsidiary of Dassault Systèmes, the world leader in 3-D and Product Lifecycle Management (PLM) solutions.

ABAQUS, Inc. employs over 525 people worldwide, with headquarters located in Providence, RI, USA, and R&D centers in Providence and in Surésnes, France. ABAQUS has 29 offices for technical support, sales and services, plus a network of distributors in emerging markets. These individuals not only provide ABAQUS expertise but also work with our customers to understand real engineering problems and guide users toward the optimum application of our software. Our entire company is committed to providing the best possible software and support to our users, and our corporate philosophy is built around developing and maintaining a satisfied and loyal customer base. To accomplish this, we provide training and other services for education, technology transfer, methods development, customization, and related software development.

About SIMULIA

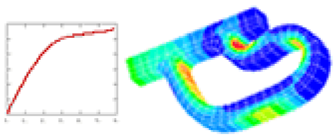
In 2005, Dassault Systèmes acquired ABAQUS, Inc. and announced SIMULIA, the brand that encompasses all DS simulation solutions, including ABAQUS and CATIA analysis applications. SIMULIA provides a scalable portfolio of simulation solutions, as well as an open platform to support integration of multidisciplinary analysis with its industry leading partners. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that enables engineers and scientists to improve product performance, eliminate physical prototypes, and drive innovation.

About Dassault Systèmes

As world leader in 3D and Product Lifecycle Management (PLM) solutions, the Dassault Systèmes group brings value to more than 90,000 customers in 80 countries. A pioneer in the 3D software market since 1981, Dassault Systèmes develops and markets PLM application software and services that support industrial processes and provide a 3D vision of the entire life cycle of products from conception to maintenance. Our offering includes integrated PLM solutions for product development (CATIA®, DELMIA®, ENOVIA®, SMARTEAM®), mainstream product 3D design tools (SolidWorks®), 3D components (Spatial/ACIS®) and SIMULIA®, DS' open platform for realistic simulation. Dassault Systèmes is listed on the Nasdaq (DASTY) and Euronext Paris (#13065, DSY.PA) stock exchanges.

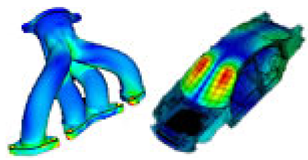
What can ABAQUS do for you?

Go beyond the realm of basic FEA and explore the world of what's possible with ABAQUS. Whether you need to understand the detailed behavior of a complex assembly, explore some concepts for a new design, or simulate a manufacturing process, ABAQUS provides the most complete and flexible solution to help you get the job done. And, ABAQUS offers a world-renowned support team to get you up to speed quickly, and to help you solve your next impossible challenge in finite element analysis.



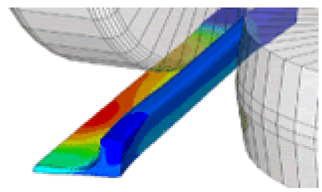
Evaluate Concepts

Often the biggest benefit of finite element analysis can be realized by evaluating different concepts very early in the design process.



Analyze Designs

Understanding the detailed interactions of a design, such as contact and material effects is a critical part of the development process



Simulate Processes

Saving 5% on most manufacturing process translates into enormous savings. But to do it right, you'll need a powerful solution…

Which ABAQUS product is right for you?

The ABAQUS suite of software for finite element analysis (FEA) is known for its high performance, quality and ability to solve more kinds of challenging simulations than any other software. The ABAQUS suite consists of three core products – ABAQUS/Standard, ABAQUS/Explicit and ABAQUS/CAE. Each of these packages offers additional optional modules that address specialized capabilities some customers may need.

ABAQUS/Standard®, provides ABAQUS analysis technology to solve traditional implicit finite element analyses, such as static, dynamics, thermal, all powered with the widest range of contact and nonlinear material options. ABAQUS/Standard also has optional add-on and interface products with address design sensitivity analysis, offshore engineering, and integration with third party software, e.g., plastic injection molding analysis.

ABAQUS/Explicit®, provides ABAQUS analysis technology focused on transient dynamics and quasi-static analyses using an explicit approach appropriate in many applications such as drop test, crushing and many manufacturing processes.

ABAQUS/CAE®, provides a complete modeling and visualization environment for ABAQUS analysis products. With direct access to CAD models, advanced meshing and visualization, and with an exclusive view towards ABAQUS analysis products, ABAQUS/CAE is the modeling environment of choice for many ABAQUS users.

Available ABAQUS Documentation

The documentation for ABAQUS is extensive and complete. The complete set of ABAQUS manuals is described briefly below in addition to other documentation that is available for ABAQUS. The documentation and publications listed below are available from ABAQUS, Inc., unless otherwise specified, in printed and online form. ABAQUS online documentation is provided to commercial and academic customers in HTML and PDF format. Commercial customers also receive printed manuals. The number of copies of printed manuals provided with commercial licenses is based on the number of tokens licensed. Pricing information for the printed documentation can be obtained through our Documentation Price List.

Every year, ABAQUS publishes the papers presented by users at the annual ABAQUS Users' Conference in a book of conference proceedings. These proceedings present the wide range of advanced topics that users analyze with ABAQUS. The proceedings are a good place to look if you want to see whether other analysts and researchers have done work similar to the work you wish to do. The online Users' Conference bibliography allows you to search the various proceedings for topics of interest to you. Please click here to order our conference proceedings.

Training

Getting Started with ABAQUS

This manual is designed to guide new users in creating solid, shell, and framework models with ABAQUS/CAE; analyzing these models with ABAQUS/Standard and ABAQUS/Explicit; and viewing the results in the Visualization module of ABAQUS/CAE. You do not need previous knowledge of ABAQUS to benefit from this guide, although some prior exposure to the finite element method is recommended. The manual discusses topics such as a quick overview of implicit and explicit methods, element selection, dynamics, nonlinearity, materials, multi-step analysis, and contact. Each chapter includes fully worked examples in a tutorial format that provide practical guidelines for performing structural analyses with ABAQUS. If you are already familiar with ABAQUS/Standard and ABAQUS/Explicit, basic and advanced tutorials are also provided to introduce you to the ABAQUS/CAE interface.

Getting Started with ABAQUS/Standard: Keywords Version

This online-only document is designed to help new users become familiar with the ABAQUS/Standard input file syntax for static and dynamic structural simulations. This manual is particularly helpful if you use a program other than ABAQUS/CAE to create input for ABAQUS/Standard. It introduces basic ABAQUS concepts, such as components of the model, and describes the format of the ABAQUS/Standard input file. This manual discusses topics similar to those presented in Getting Started with ABAQUS but from the perspective of the ABAQUS/Standard keyword interface. Each chapter includes input files for examples that provide practical guidelines for performing structural analyses with ABAQUS/Standard.

Getting Started with ABAQUS Explicit: Keywords Version

This online-only document is designed to help new users become familiar with the ABAQUS/Explicit input file syntax for explicit dynamic and quasi-static structural simulations. This manual is particularly helpful if you use a program other than ABAQUS/CAE to create input for ABAQUS/Explicit. It introduces basic ABAQUS concepts, such as components of the model, and describes the format of the ABAQUS/Explicit input file. This manual discusses topics similar to those covered in Getting Started with ABAQUS but from the perspective of the ABAQUS/Explicit keyword interface. Each chapter includes input files for examples that provide practical guidelines for performing structural analyses with ABAQUS/Explicit.

Lecture Notes

Notes and workshops are available for many features and applications for which ABAQUS is used, such as metal forming and heat transfer. The notes are used in the technical seminars that are offered to help users improve their understanding and usage of ABAQUS. While they are not intended as stand-alone tutorial material, they are usually comprehensive enough that they can be used in that mode. The list of available lecture notes is included in the Documentation Price List.

Analysis

ABAQUS Analysis User’s Manual

This is the ABAQUS manual that you will use most often with ABAQUS/Standard and ABAQUS/Explicit. Split into six volumes, it is a complete reference manual for all of the capabilities of both products and contains a complete description of the elements, material models, procedures, input specifications, etc. Both input file usage and ABAQUS/CAE usage information are provided in this manual.

Modeling and Visualization

ABAQUS/CAE User’s Manual

This reference document for ABAQUS/CAE includes detailed descriptions of how to use ABAQUS/CAE for model generation, analysis, and results evaluation and visualization. ABAQUS/Viewer users should refer to the information on the Visualization module in this manual.

Examples

ABAQUS Example Problems Manual

This volume contains more than 125 detailed examples designed to illustrate the approaches and decisions needed to perform meaningful linear and nonlinear analysis. Typical cases are large motion of an elastic-plastic pipe hitting a rigid wall; inelastic buckling collapse of a thin-walled elbow; explosive loading of an elastic, viscoplastic thin ring; consolidation under a footing; buckling of a composite shell with a hole; and deep drawing of a metal sheet. It is generally useful to look for relevant examples in this manual and to review them when embarking on a new class of problem.

ABAQUS Benchmarks Manual

This online-only volume contains over 250 benchmark problems and standard analyses used to evaluate the performance of ABAQUS; the tests are multiple element tests of simple geometries or simplified versions of real problems. The NAFEMS benchmark problems are included in this manual.

Documentation Information

Using ABAQUS Online Documentation

This online-only manual contains instructions for viewing and searching the ABAQUS online documentation.

Reference

ABAQUS Keywords Reference Manual

This volume contains a complete description of all the input options that are available in ABAQUS/Standard and ABAQUS/Explicit.

ABAQUS Theory Manual

This volume contains detailed, precise discussions of all theoretical aspects of ABAQUS. It is written to be understood by users with an engineering background.

ABAQUS Verification Manual

This online-only volume contains more than 12,000 basic test cases, providing verification of each individual program feature (procedures, output options, MPCs, etc.) against exact calculations and other published results. It may be useful to run these problems when learning to use a new capability. In addition, the supplied input data files provide good starting points to check the behavior of elements, materials, etc.

Quality Assurance Plan
This document describes QA procedures followed by ABAQUS. It is a controlled document, provided to customers who subscribe to either the Nuclear QA Program or the Quality Monitoring Service.

Update Information

ABAQUS Release Notes (through Answer 3017 -login required)
This document contains brief descriptions of the new features available in the latest release of the ABAQUS product line.

Programming

ABAQUS Scripting User's Manual
This online-only manual provides a description of the ABAQUS Scripting Interface. The manual describes how commands can be used to create and analyze ABAQUS/CAE models, to view the results of the analysis, and to automate repetitive tasks. It also contains information on using the ABAQUS Scripting Interface or C++ as an application programming interface (API) to the output database.

ABAQUS Scripting Reference Manual
This online-only manual provides a command reference that lists the syntax of each command in the ABAQUS Scripting Interface.

ABAQUS GUI Toolkit User's Manual
This online-only manual provides a description of the ABAQUS GUI Toolkit. The manual describes the components and organization of the ABAQUS GUI. It also describes how you can customize the ABAQUS GUI to build a particular application.

ABAQUS GUI Toolkit Reference Manual
This online-only manual provides a command reference that lists the syntax of each command in the ABAQUS GUI Toolkit.

Interfaces

ABAQUS Interface for MSC.ADAMS User's Manual

This document describes how to use the ABAQUS Interface for MSC.ADAMS, an interface program that creates ABAQUS models of MSC.ADAMS components and converts the ABAQUS results into an MSC.ADAMS modal neutral file that can be used by the ADAMS/Flex program. It is the basic reference document for the ABAQUS Interface for MSC.ADAMS.

ABAQUS Interface for MOLDFLOW User's Manual

This document describes how to use the ABAQUS Interface for MOLDFLOW, an interface program that creates a partial ABAQUS input file by translating results from a MOLDFLOW polymer processing simulation. It is the basic reference document for the ABAQUS Interface for MOLDFLOW.

Installation and Licensing

ABAQUS Installation and Licensing Guide

This document describes how to install ABAQUS and how to configure the installation for particular circumstances. Some of this information, of most relevance to users, is also provided in the ABAQUS Analysis User's Manual.