

+

−

<<

>>

Getting Started with Abaqus

1 Introduction

- 1.1 The Abaqus products
- 1.2 Getting started with Abaqus/CAE
- 1.3 Abaqus documentation
- 1.4 Getting help
- 1.5 Support
- 1.6 A quick review of the Abaqus/CAE interface

2 Abaqus Basics

3 Finite Elements and Results

4 Using Continuum Elements

5 Using Shell Elements

6 Using Beam Elements

7 Linear Dynamics

8 Nonlinearity

9 Nonlinear Explicit Dynamics

10 Materials

11 Multiple Step Analysis

12 Contact

13 Quasi-Static Analysis

A Example Files

B Creating and Analyzing Models

C Using Additional Technologies

D Viewing the Output from a Model

E Flow through a bent tube

## 1.2 Getting started with Abaqus

This guide is an introductory text designed to give new users guidance in creating solid, shell, beam, and truss models with Abaqus/CAE, analyzing these models with Abaqus/Standard and Abaqus/Explicit, and viewing the results in the Visualization module. A brief introduction to using Abaqus/CFD is included as an appendix. You do not need any previous knowledge of Abaqus to benefit from this guide, although some previous exposure to the finite element method is recommended. If you are already familiar with the Abaqus solver products (Abaqus/Standard or Abaqus/Explicit) but would like an introduction to the Abaqus/CAE interface, three tutorials are provided in the appendices of this guide to lead you through the modeling process in Abaqus/CAE.

This document covers primarily stress/displacement simulations, concentrating on both linear and nonlinear static analyses as well as dynamic analyses. An introduction to CFD analysis and modeling fluid-structure interaction is also included. Other types of simulations, such as heat transfer and mass diffusion, are not covered.

### 1.2.1 How to use this guide

The different sections of this guide are addressed to different types of users.

#### Tutorials for new Abaqus users

If you are completely new to Abaqus, we recommend that you follow each of the self-paced tutorials in this guide. Each of the chapters and appendices in this guide introduces one or more topics relevant to using Abaqus/Standard, Abaqus/Explicit or Abaqus/CFD. Throughout the guide the term Abaqus is used to refer collectively to all three analysis products; the individual product names are used when information applies to only one product. Most chapters contain a short discussion of the topic or topics being considered and one or two tutorial examples. You should work through the examples carefully since they contain a great deal of practical advice on using Abaqus.

The capabilities of Abaqus/CAE are introduced gradually in these examples. It is assumed that you will use Abaqus/CAE to create the models used in the examples. You can also generate the model for any example using a script that replicates the complete analysis model for the problem. A model created from a script may differ slightly from that created by following the steps in this guide. These differences, such as material names or node numbers, are minor and can be ignored. Scripts are available in two locations:

- A Python script is provided for each example in [Appendix A, “Example Files.”](#) The same section also provides instructions on how to fetch the script and run it within Abaqus/CAE.