

Troubleshooting Finite-Element Modeling with Abaqus

Raphael Jean Boulbes

Troubleshooting Finite-Element Modeling with Abaqus

With Application in Structural Engineering
Analysis

 Springer

Raphael Jean Boulbes
Lyon, France

ISBN 978-3-030-26739-1 ISBN 978-3-030-26740-7 (eBook)
<https://doi.org/10.1007/978-3-030-26740-7>

Mathematics Subject Classification (2010): 65Z05

© Springer Nature Switzerland AG 2020

This work is subject to copyright. All rights are reserved by the Publisher, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilms or in any other physical way, and transmission or information storage and retrieval, electronic adaptation, computer software, or by similar or dissimilar methodology now known or hereafter developed.

The use of general descriptive names, registered names, trademarks, service marks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.

The publisher, the authors and the editors are safe to assume that the advice and information in this book are believed to be true and accurate at the date of publication. Neither the publisher nor the authors or the editors give a warranty, expressed or implied, with respect to the material contained herein or for any errors or omissions that may have been made. The publisher remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

This Springer imprint is published by the registered company Springer Nature Switzerland AG
The registered company address is: Gewerbestrasse 11, 6330 Cham, Switzerland

to Jeanne, Harlette and Marie-Kristine

Foreword by Dr. Sonell Shroff

I have been working with Finite Element Analysis software for the past decade and as our understanding of the subject and its utility has increased over time, so have the number of software and the analysis types. As a user and further, as a university teacher for master's students, I have spent a good amount of effort to get familiarized with multiple software platforms to get, in effect, the same result for structural problems. What concerns me most while imparting whatever knowledge I can to the students during the few hours of lessons is to be certain that once they leave that classroom, would they be able to become engineers who do not just work out the problems that face them, but solve them efficiently?

When Raphael contacted my boss at the university for a recommendation of a reviewer in the field for this book, fortunately, I was instantly put in contact with him. It is part of my courses to teach the students how to debug a model and what are the best ways to categorize the errors that they encounter. Coincidentally, ABAQUS is one of the most popular softwares we use at the Aerospace faculty because of its wide usage in the industry. It comes with an expansive user guide, and expansive is an understatement; so much so that it is most of the times very difficult to find exactly what you are looking for in order to troubleshoot the program. As an advanced Finite Element course, my focus was so much on the science and the formulation of the finite-element theory that I lacked the organization of the troubleshooting techniques. Raphael's idea of this book came as a fabulous news for me. I would be able to rest easy with such a book out there for the interested students to find.

Ph.D. students, postdocs, M.Sc. graduate students, and even bachelor students use this software and do not necessarily have the time to go through the entire user guide. This book will help them understand how to categorize the error types and where to look for further information. It will teach us how to use the information we receive in the messages from the software itself. And the best part of it is that it is suited to all users, amateurs, and experienced. It contains enough background knowledge of important topics that you would need to get started with what the

software is expected to do, and then holds your hand through the error resolution process. Raphael has managed to cover such important and difficult topics in a comprehensive manner. I hope my colleagues in the teaching world will recognize this book as a handy manual both for teaching and for use in their own modeling for research.

Brussels, Belgium
September 2018

Dr. Sonell Shroff

Foreword by Gautam Puri

I first spoke with Raphael in 2011 when he was working for FMC Technologies in Norway. At the time, he was performing FEA simulations on various subsea technologies and contacted me after reading a book I had recently written on Python scripting for Abaqus. We had a conversation about running GUI scripts, and since I was about to graduate, he kindly offered to forward my resume to hiring managers at his company. I eventually ended up accepting a local offer from SIMULIA (the Dassault Systèmes brand that develops Abaqus) here in the United States, but I must admit that Raphaels offer of an opportunity to work abroad was a tempting one.

Fortunately, a few years later Raphael found another opportunity where we could collaborate; he intended to write a book about troubleshooting/debugging finite-element models in Abaqus and requested me to be one of the reviewers. Anyone who has worked with Abaqus has experienced the frustration of spending many hours debugging an FE model and attempting to get it to converge, and so this sounded like a good subject for a book. I was excited to see a draft version a few months later.

In my opinion, this book will serve as a guide or a toolkit for engineers when they run into difficulties with their Abaqus simulations. The first few chapters provide general information on how an analyst should approach the process of troubleshooting FE models as well as an overview of some common errors and the potential causes. Later, chapters provide deeper coverage of specific topics such as materials and contact interactions. One could argue that much of what is covered in this book is already included in the Abaqus documentation; however, the documentation is vast and spans thousands of pages, and the sheer amount of information often makes it very difficult to find tips that will help troubleshoot a model. This book attempts to alleviate this problem.

Many readers, particularly ones who are relatively new to Abaqus, would benefit by reading through at least the first few chapters. They do contain a lot of information, which may be difficult to digest in one sitting but they will help familiarize the reader with the types of issues they are likely to encounter over time. For more experienced Abaqus users, this book will be useful as a handy reference manual.

While it is not possible for simulation analysts to entirely escape the frustration of diagnosing and fixing Abaqus model errors and solver convergence issues, I do believe this book will help make these situations a little less painful for the reader.

Ohio, USA
September 2018

Gautam Puri

Foreword by Prof. David Bassir

First of all, thank you to Raphael Boulbes for providing such support for the mechanical community using the commercial finite-element software Abaqus to simulate complex problems. Indeed, this book will help users to deal with finite-element modeling under different aspects like boundary conditions and to solve troubleshooting issues with such commercial software. It will bring enormous knowledges to both intermediate and advanced users. I wish to get such support 20 years ago, when I was doing my Ph.D.

With concrete and real practical examples included in this book, it gives also a references for engineers at different mechanical industries. I am confident about the success of this work and I wish all the readers to enjoy reading this book as I did.

Shenzhen, China
September 2018

Prof. David Bassir

Preface

The finite-element method (FEM) has become a staple for predicting and simulating the physical behavior of complex engineering systems. The commercial finite element analysis (FEA) software have gained common acceptance among engineers in industry and researchers at universities. Therefore, academic engineering departments include graduate or undergraduate senior-level courses that cover not only the theory of FEM but also its applications using the commercially available FEA programs.

The goal of this book is to provide students and engineers with an essential theoretical and deep practical knowledge of the finite-element method and the skills required to fix troubleshooting issues with Abaqus, a commercially available FEA software. This book designed for seniors and first-year graduate students, as well as practicing engineers, is introductory and self-contained in order to minimize the need for additional reference material.

In addition to the fundamental topics in finite-element method, it presents advanced topics concerning modeling and analysis procedures with Abaqus. These topics are introduced through some examples in a step by step fashion from various engineering disciplines mainly focusing on structural analysis. The book focuses on the methodology in analysis to structure debugging aspects in Part I then provides in Part II specific issues describing many understandings about potential troubleshooting in the model, and finally in Part III some practical toolbox protocols to resolve troubleshooting in the model. The content of this book is a collection of researches made from my different professional and academic experiences work with finite-element analyses using Abaqus, including my findings on the free web data, and solutions gathering from the Abaqus support, which I worked with.

In Part I, Chap. 1 provides an introduction to the principles about how to conduct a finite-element analysis. In Chap. 2, analysis convergence guidelines to obtain a converged solution are described. The methodology and control checks in order to have an efficient debugging diagnostic are given in Chap. 3. Chapter 4 gives an essential knowledge covering different analysis aspects including user subroutines.

In Part II, the main practical materials nonlinearities and customizations are discussed in Chap. 5. Chapter 6 provides an important field of expertise in order to understand how the mesher works in Abaqus and what are the good practices to mesh a structure, and how to create a finite element. The contact interaction module and feature options are explained in Chap. 7.

In Part III, Chap. 8 gives the solutions regarding most classics troubleshooting with Abaqus. The options to control the solution accuracy in different model cases are explained in Chap. 9. Chapter 10 provides some very specific issues, where the user will need to get help in modeling. Lastly, Chap. 11 gives solution about non-simulation issues, and a full cluster solution to implement.

Finally, the user will find some example solutions described in the book with the related code in Appendix, in order to have a better understandings about specific aspects. For instance, with coupling options, numerical convergences, and some meshing techniques.

Bourg-la-Reine, France
September 2018

Raphael Jean Boulbes

Acknowledgements

This book would not have been published without the help and trust of a few people whom I wish to thank simply but sincerely for everything, because they made it happen.

First of all, I am thankful to my loving relatives who have supported and encouraged me. You have always given me the best, so it is natural that I dedicate this book to you.

Secondly, I thank Springer, especially Oliver Jackson, the editor of the engineering department; Mani Nareshkumar, the project coordinator; and Yasmin Brookes in administration. Thank you for your support and guidance during my project of writing this book.

Finally, I am hugely grateful to my scientific committee of three experts in structural analysis, whom I mention below. The time they spent reviewing the content, as well as their comments, feedback, and the contributions they made to this book, were priceless to me.

Dr. Sonell Shroff is a former assistant professor from the Department of Structural Integrity, Faculty of Aerospace Engineering at Delft University of Technology. Dr. Shroff was strongly recommended to me by Prof. Jan Hol (the Secretary of the Department of Aerospace Structures & Materials, Faculty of Aerospace Engineering, Delft University of Technology), and I was really pleased to work with her. Having completed high school in 2003, Dr. Shroff moved to Bangalore to study engineering. Dr. Shroff oriented herself with a bachelor's degree in Mechanical Engineering, with internships every semester break in reputable companies and research institutes. She graduated in 2007 and received a Royal Dutch Huygens Scholarship, which brought her to the Delft University of Technology for a master's course. Here, she graduated with a final thesis on the structural analysis of a retrofitted winglet on the Hercules C-130 at Marshall Aerospace in 2009. Subsequently she started her Ph.D. study, first with CleanEra and then on the ALaSCA project at Delft University of Technology, which she completed in 2014. Those who are interested in her work will find some papers she

worked on cited as references.^{1,2,3,4} She is a highly motivated online teacher in the field of Finite Element Method. As she says: “In this world of the future, education needs to be omnipresent. Therefore, I must be, too. Going online brings me closer to the farthest student and breaks the communication barrier. Online education will create a smarter and more learned world, which only means progress for more and a better tomorrow!”

Gautam Puri is a project manager, consultant, and long-time Abaqus user. He has degrees in engineering from the Georgia Institute of Technology and an MBA from Emory University. I was glad to have Gautam on board because I was really impressed after reading his book⁵ about Python scripting for Abaqus. After earning his masters degree in Aerospace Engineering, he worked for Dassault Systèmes, which develops Abaqus, CATIA, SolidWorks, and a range of other computer-aided engineering software tools. His book *Python Scripts for Abaqus—Learn by Example* is used by more than 40 engineering companies, including NASA, Boeing, Apple, and dozens of universities including MIT, Stanford, and Georgia Tech. He also runs a YouTube channel, *AbaqusPython*,⁶ with training video material on running finite-element analyses with Abaqus. While at SIMULIA, Gautam provided consulting, training, and technical support services to various SIMULIA clients using Abaqus. He also created the Abaqus tutorial series and the Isight (workflow optimization software) training series for the SIMULIA Learning Community. Gautam is obsessed with cutting-edge technologies. Aside from automating simulations, he has coded and architected web pages, servers, databases, mobile apps, cloud services, and microcontrollers. His most recent role was as an R&D Project Manager at Amazon Web Services (AWS), developing PLC hardware and machine learning algorithms to help manage and scale out Amazons cloud infrastructure. He is now the chief technologist at a web-based startup that is currently in stealth mode.

¹Y.L.M. van Dijk, T. Grtzi, M. Abouhamzeh, L. Kroll, S. Shroff, “Hygrothermal viscoelastic material characterisation of unidirectional continuous carbon-fibre reinforced polyamide 6”, *Composites Part B: Engineering*, Volume 150, 2018, pp. 157–164.

²Sonell Shroff, Ertan Acar, Christos Kassapoglou, “Design, analysis, fabrication, and testing of composite grid-stiffened panels for aircraft structures”, *Thin-Walled Structures*, Volume 119, 2017, pp. 235–246.

³Sonell Shroff and Christos Kassapoglou, “Designing Highly Loaded Connections in a Composite Fuselage”, *Journal of Aircraft*, Vol. 51, No. 3 (2014), pp. 833–840.

⁴Sonell Shroff and Christos Kassapoglou, “Progressive failure modelling of impacted composite panels under compression”, *Journal of Reinforced Plastics and Composites*, Vol. 34, No. 19 (2015), pp. 1603–1614.

⁵*Python Scripts for Abaqus—Learn by Example*, book website: <http://www.abaquspython.com/>.

⁶Gautam (Gary) Puris YouTube channel *AbaqusPython*: <https://www.youtube.com/user/AbaqusPython>.

I know Prof. David Bassir well, because he was the master responsible for overseeing my research studies.⁷ Dr. Bassir is currently appointed as a Foreign expert at the IIT, Chinese Academy of Sciences (Guangzhou). Previously, he worked at the Ministry of Foreign Affairs (MAEE) to serve as an Attach for Science and Technology (AST) at the Consulate General of France in Guangzhou, China. Before moving to France, Prof. Bassir was General Director of Research at the special school of civil engineering (ESTP) in Paris. After a master's degree in Structural Optimization at ENS2M/UFC universities, in 1999 he obtained his Ph.D. from the same university, with distinction. Then, he worked as a spacecraft engineer for GECI Technology in different space agencies, such as Arianespace (France) and Matra Marconi Space (Astrium Group). In 2002, he joined the Mechanical Department of the Technical University of Belfort-Montbéliard as a professor assistant; here, he obtained his certification (HDR) to conduct research few years later within the framework related to some research activities at the FEMTO-ST Institute in Besançon. Before joining the board of Aerospace Structures at the Technical University of Delft in 2008, he was an invited professor at the Aircraft Manufacturing Department in Xian, China. Dr. Bassir is also the founder of the ASMDO association⁸ and the International Journal IJSMDO,⁹ which is published by EDP Sciences. He was involved in many international organizations as a member of expert committees. Furthermore, he has published more than 150 research papers in peer-reviewed scientific journals and peer-reviewed international scientific proceedings. He also acts as a reviewer in many international journals related to structural and material optimization. He has conducted more than 2 HDR, 12 Ph.D., 10 masters, and 20 final projects. He excels at project and team management, strategy, and business orientation, government liaison, research and development. His main scientific research is related to the design of composite materials (at macro-, micro-, or nanoscales) and optimization strategies.

⁷Parameter identification of a nonlinear model of a composite laminate shell (David Hicham Bassir, Raphael Boulbes, Lamine Boubakar) FEMTO-ST, University of Franche Comte, UTBM conference paper for the 5th International Conference on Computation of Shell and Spatial Structures, 2005, France. 2005, Salsburg, Austria, June 1–4.

⁸ASMDO association: <http://www.asmdo.com/>.

⁹International Journal IJSMDO: <https://www.ijsmdo.org/>.

Contents

Part I Methodology to Start Debugging Model Issues

1	Introduction	3
1.1	Global Mindset	3
1.2	The Four Absolutes of Quality in Analysis	8
1.3	Checklist for Performing Analysis	9
1.4	A Heuristic Analysis Confidence Ratio	9
	References	15
2	Analysis Convergence Guidelines	17
2.1	Symptoms of Convergence Problems	17
2.2	Causes of Convergence Problems	18
2.3	Helping Abaqus Find a Converged Solution	18
2.4	General Tools	19
2.5	Tools for Contact Stabilization	21
2.6	Tools for Contact Related Convergence Problems	21
	Reference	23
3	Method to Debug a Model	25
3.1	Debugging Flowchart	25
3.2	Job Diagnostic	25
3.2.1	Making a Test Model	25
3.2.2	Output Check	29
3.2.3	Syntax Check	30
3.2.4	Data Check	31
3.2.5	Loading and Boundary Conditions Check	34
3.2.6	Materials Check	36
3.2.7	Constraints Check	38
3.2.8	Elements Check	39
3.2.9	Interference Fits Check	40
3.2.10	Contact Check	41

- 3.2.11 Initial Rigid Body Motion and Over Constraints Check 43
- 3.2.12 Static Stabilization Check 47
- 3.2.13 Dynamics Check 49
- 3.3 Causality Energy Method 54
 - 3.3.1 Basic Energy Approaches, Assumptions and Limitations 55
 - 3.3.2 The Energy Method 56
 - 3.3.3 Energy Method Example to Scale Analyses 57
 - 3.3.4 Causality and Energy Derivatives 58
- References 59
- 4 General Prerequisites 61**
 - 4.1 Vocabularies 61
 - 4.1.1 Interpreting Error Messages 63
 - 4.1.2 Interpreting Warning Messages 64
 - 4.2 An Identified Unconnected Region in the Model 65
 - 4.3 Correction of Errors During the Data Check Phase of an Abaqus/Standard Analysis 67
 - 4.4 Tips and Tricks for Diagnostic Error Messages 69
 - 4.5 Trying to Recover a Corrupted Database 70
 - 4.5.1 Procedure 1 70
 - 4.5.2 Procedure 2 71
 - 4.6 Kinematic Distributing Couplings in Abaqus 72
 - 4.6.1 Nature of the Constraint Enforcement 72
 - 4.6.2 Defining Constraints in Abaqus/CAE 75
 - 4.7 Abaqus Geometric Nonlinearity 75
 - 4.8 Differences Between Implicit and Explicit Schemes 78
 - 4.8.1 Equations for Dynamic Problems 79
 - 4.8.2 Time Integration of the Equations of Motion 79
 - 4.8.3 Automatic Time Incrementation with Abaqus Standard 81
 - 4.8.4 Automatic Time Incrementation with Abaqus Explicit 86
 - 4.8.5 Dynamic Contact 88
 - 4.8.6 Material Damping 88
 - 4.8.7 Half-Increment Residual Tolerance 89
 - 4.8.8 Comparing Abaqus/Standard and Abaqus/Explicit 90
 - 4.9 Unstable Collapse and Post-buckling Analysis 91
 - 4.10 Low-Cycle Fatigue Analysis Using the Direct Cyclic Approach 93

- 4.11 Steady-State Transport Analysis 94
 - 4.11.1 Convergence Issues in a Steady-State Transport Analysis 95
- 4.12 Heat Transfer Analysis 97
 - 4.12.1 Transient Analysis 98
- 4.13 Fluid Dynamic Analysis 102
 - 4.13.1 Convergence Criteria and Diagnostics 102
 - 4.13.2 Time Increment Size Control 104
- 4.14 Introduction to the User Subroutines 105
 - 4.14.1 Installation of a Fortran Compiler 107
 - 4.14.2 Run a Model Which Uses a User Subroutine 109
 - 4.14.3 Debugging Techniques and Proper Programming Habits 109
 - 4.14.4 Examples of User Subroutine with Abaqus Standard 112
 - 4.14.5 Examples of User Subroutine with Abaqus Explicit 114
 - 4.14.6 Examples of User Subroutine with Abaqus CFD 116
- References 116

Part II Stop Struggling with Specific Issues

- 5 Materials 119**
 - 5.1 Generalities 119
 - 5.2 The Current Strain Increment Exceeds the Strain to First Yield 121
 - 5.3 Convergence Behavior of Models Using Hyperelastic Materials 122
 - 5.4 Models Using Incompressible or Nearly Incompressible Materials 123
 - 5.5 Equivalence of Uniaxial Tension and Compression Hyperelastic Test Data 124
 - 5.5.1 Uniaxial Compression Test Data for a Rubber Material 125
 - 5.5.2 Specifying Tension or Compression Test Data for the Marlow Hyperelasticity Model 126
 - 5.5.3 Using Simple Shear Experimental Data for Hyperelastic Materials 127
 - 5.6 Path Dependence of Nonlinear Results Using an Elastic Material 129
 - 5.7 User Material Subroutine 131
 - 5.7.1 Guideline to Write a UMAT or a VMAT 132
 - 5.8 UMAT Subroutine Examples 133

- 5.8.1 UMAT Subroutine for Isotropic Isothermal Elasticity 136
- 5.8.2 UMAT Subroutine for Non-isothermal Elasticity 138
- 5.8.3 UMAT Subroutine for Neo-Hookean Hyperelasticity 140
- 5.8.4 UMAT Subroutine for Kinematic Hardening Plasticity 145
- 5.8.5 UMAT Subroutine for Isotropic Hardening Plasticity 151
- 5.8.6 UMAT Subroutine for Simple Linear Viscoelastic Material 157
- 5.9 VUMAT Subroutine Examples 160
 - 5.9.1 VUMAT Subroutine for Kinematic Hardening Plasticity 162
 - 5.9.2 VUMAT Subroutine for Isotropic Hardening Plasticity 165
- References 169
- 6 Mesher and Meshing 171**
 - 6.1 Generalities 171
 - 6.1.1 Mesh Control Options 172
 - 6.1.2 Mesh Controls for a 2D Structure 172
 - 6.1.3 Mesh Controls for a 3D Structure 172
 - 6.1.4 Understanding a Mesher 174
 - 6.1.5 Mesh as Grid Generation 179
 - 6.2 The Abaqus Model Meshed Has Changed into a Nonphysical Shape with a Regular Pattern 190
 - 6.3 Excessive Element Distortion Warnings 191
 - 6.4 Compatibility Errors Printed to the Message File for a Model with Hybrid Elements 191
 - 6.5 User Element Subroutine 192
 - 6.5.1 Guideline to Write a UEL 193
 - 6.6 UEL Subroutine Examples 201
 - 6.6.1 UEL Subroutine for Planar Beam with Nonlinear Cross Section 202
 - 6.6.2 Generalized Constitutive Behavior 207
 - 6.6.3 UEL Subroutine for a Horizontal Truss and Heat Transfer Element 209
 - 6.6.4 UELMAT Subroutine for 4 Nodes in Plane Strain 214
 - 6.7 Using Nonlinear User Elements in Various Analysis Procedures 222
 - References 226

7 Contact 227

7.1 Generalities 227

7.1.1 Understandings 230

7.1.2 Define Contact Pairs 234

7.1.3 Define General Contact 234

7.1.4 Representation of Curved Surfaces 236

7.1.5 Contact Formulation Aspects 237

7.2 Friction 262

7.2.1 Static and Kinetic Friction 263

7.2.2 Change Friction Properties During an Analysis 266

7.2.3 Classic Friction Values 266

7.3 Hard or Soft Contact 267

7.3.1 Identification of the Mathematical Stiffness
Function 270

7.3.2 Exponential Contact Stiffness 274

7.3.3 From Hard Contact to Exponential 276

7.4 Obtain a Converged Contact Solution 278

7.5 Convergence Difficulty in the First Increment 280

7.6 Causes and Resolutions of Contact Chattering 281

7.7 Understand Finite Sliding with Surface-to-Surface Contact 283

7.8 Using Penalty Contact 286

7.9 Using Augmented Lagrangian Contact 290

7.10 Using Stiffness-Based Contact Stabilization 292

7.11 Modeling Contact with Second-Order Tetrahedral Elements 294

References 295

Part III A Toolbox to Do the Job

8 Troubleshooting in Job Diagnostics 299

8.1 Guidelines with Abaqus Standard 299

8.2 Job with Abaqus Standard Completes, But the Results
Look Suspicious 301

8.3 Model a Structure Undergoing a Global Instability 304

8.4 Correct Convergence Difficulties Caused by Local
Instabilities 305

8.5 Correcting Errors During the Data-Check Phase
of an Analysis 306

8.6 Analysis Ends Prematurely, Even Though All the Increments
Have Converged 308

8.7 Debugging Divergence with Too Many Cutbacks
in the Last Attempted Increment 309

8.8 Using Follower Loads in Nonlinear Analyses 310

8.9 Understanding Negative Eigenvalue Messages 311

- 8.10 Divergence with Numerical Singularity Warnings 313
- 8.11 Zero Pivot Warnings in the Message File 314
- 8.12 Convergence Difficulty in the First Increment of a Contact
Analysis 315
- 8.13 Explicit Stable Time Increments When Using the Marlow
Model with Noisy Test Data 317
- 8.14 Cause of an Analysis Ending in a Core Dump 318
- 8.15 Debugging User Subroutines and Post Processing
Programs 318
- 8.16 No Free Memory Available on Linux at the End
of an Analysis 323
- Reference 326
- 9 Numerical Acceptance Criteria 327**
 - 9.1 Generalities 327
 - 9.1.1 Commonly Used Control Parameters 327
 - 9.1.2 Controlling the Time Incrementation Scheme 329
 - 9.1.3 Activate the Line Search Algorithm 331
 - 9.1.4 Controlling the Solution Accuracy in Direct
Cyclic Analysis 331
 - 9.1.5 Controlling the Solution Accuracy and Mesh
Quality in a Deforming Mesh Analysis
with Abaqus CFD 332
 - 9.1.6 Convergence Criteria for Nonlinear Problems 334
 - 9.1.7 Time Integration Accuracy in Transient Problems 343
 - 9.1.8 Avoid Small Changes to the Time Increment
Size During Implicit Integration Procedures 344
 - 9.2 How Much Hourglass Energy Is Acceptable 345
 - 9.2.1 Enhanced Hourglass Control and Elastic Bending
Moment 346
 - 9.2.2 Enhanced Hourglass Control and Plastic Bending
Moment 346
 - 9.2.3 Kelvin Viscoelastic Hourglass Control 346
 - 9.3 Errors Printed to the Message File for a Model with Hybrid
Elements 347
 - Reference 348
- 10 Need Some Help? 349**
 - 10.1 Retrieving Files Referred to Examples in the Abaqus
Documentation 349
 - 10.2 Using the Abaqus Verification, Benchmarks, and Example
Problems Guides 349
 - 10.3 Excessive Memory Usage with Cavity Radiation Problems 357

- 10.4 Perform a Sub-model Analysis 358
 - 10.4.1 Implementation 359
 - 10.4.2 Loading Conditions 360
 - 10.4.3 Sub-model Boundary Conditions 360
 - 10.4.4 Interpolation 361
 - 10.4.5 Step-by-Step Procedure for a Sub-model 361
 - 10.4.6 Setting Options 364
 - 10.4.7 Shell to Solid 365
 - 10.4.8 Changing Procedures 367
 - 10.4.9 Frequency Domain 367
 - 10.4.10 Thermal and Stress Analysis 368
 - 10.4.11 Dynamic Analysis 369
 - 10.4.12 Limitations of Sub-modeling 370
- 10.5 Perform a Restart Analysis 371
 - 10.5.1 Step-by-Step Procedure for a Restart 373
- 10.6 Generate a Shell Part from a Solid Part 376
 - 10.6.1 Benefits for Using Shell Structures 376
 - 10.6.2 Applications to Model Shell Structures 377
 - 10.6.3 Step-by-Step Procedure to Convert Solid Model
to Shell Model 378
- 10.7 Compile and Link a Post-processing Program Using
the Standalone Abaqus ODB API 385
- 10.8 Create Executables Using the C++ ODB API Libraries
Outside of Abaqus/Make 387
- 11 Hardware or Software Issues 391**
 - 11.1 Solving File System Error 1073741819 391
 - 11.2 Interpreting Error Codes 391
 - 11.3 Obtaining a Traceback from a UNIX/Linux Core Dump 393
 - 11.4 Windows HPC Compute Clusters 397
 - 11.4.1 Classics Troubleshooting with HPC Cluster 402
 - Reference 405
- Appendix: Guidelines and Good Practices Examples 407**
- Index 437**

Abbreviations

ALE	Arbitrary Lagrangian–Eulerian
CAD	Computer-Aided Design
CAE	Complete Abaqus Environment
CFD	Computational Fluid Dynamic
CPU	Central Processing Unit
DOF	Degrees of freedom
DSV	Dependent State Variables
FEA	Finite-Element Analysis
FSI	Fluid–Structure Interaction
GIGO	Garbage In, Garbage Out
HHT	Hilber–Hughes–Taylor
Mdb	Model database
MPC	Multi-point Constraints
XFEM	Extended Finite-Element Method