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

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. Coincidently, 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 Dr. Sonell Shroff
September 2018

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 Systémes 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 shear 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 Gautam Puri
September 2018

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 bahavior 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 book[5] about Python scripting for Abaqus. After earning his masters degree in Aerospace Engineering, he worked for Dassault Systémes, 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.

[1] Y.L.M. van Dijk, T. Grtzl, 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.

[2] Sonell 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.

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

[4] Sonell 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/.

[6] Gautam (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-Montbeliard 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 Besancon. 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 association[8] 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. His 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.

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

[8]ASMDO association: http://www.asmdo.com/.

[9]International Journal IJSMDO: https://www.ijsmdo.org/.

Contents

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