Preface

This edition of Finite Element Analysis of Composite Materials using Abaqus updates 62 examples and 82 end-of-chapter problems to Abaqus 2020 from the first edition (2013) that used Abaqus 6.10 (2010). All chapters are updated and expanded. For example, Chapter 2 has a new section on Interactions and Constraints, Chapter 10 has a new section on Determination of CZM Parameters from experimental data, and so on. Furthermore, Chapter 11 (Fatigue) and Chapter 12 (Programmable Features) are new to this second edition.

UMAT and UGENS subroutines used in Chapters 3, 7, 8, 9, 10, and 11 are updated to Abaqus Release 2020 and thoroughly explained in the new Chapter 12 (Programmable Features). The Appendix dealing with software setup, which is necessary for using programmable features with Abaqus 2020, is updated as well.

The reader will be able to follow a process to recreate every example using the Abaqus graphical user interface (CAE) by following step-by-step directions in the form of pseudo-code. Also, screen shots of the CAE interface are included in earlier chapters to train the reader on this process and to supplement the pseudo-code in later chapters. Furthermore, narrated video recordings of most examples are available in YouTube. Finally, the most complex examples are also presented in the form of Python scripts that can be interpreted by Abaqus CAE. A fair amount of explanation is included on both how these scripts work and how they can be harvested from CAE sessions.

The textbook deals with the analysis of structures made of composite materials, also called composites. The analysis of composites treated in this textbook includes the analysis of the material itself, at the micro-level, and the analysis of structures made of composite materials. This textbook evolved from the class notes of MAE 646 Advanced Mechanics of Composite Materials that I teach as a graduate course at West Virginia University. Although this is a textbook on advanced mechanics of composite materials, the use of the finite element method is essential for the solution of the complex boundary value problems encountered in the advanced analysis of composites, and thus the title of the book.

There are several good textbooks on advanced mechanics of composite materials, but none carries the theory to a practical level by solving problems, as is done in this textbook. Some books devoted exclusively to finite element analysis include some examples about modeling composites but fall quite short of dealing with the actual analysis and design issues of composite materials and composite structures.

This textbook includes an explanation of the concepts involved in the detailed analysis of composites, a sound explanation of the mechanics needed to translate those concepts into a mathematical representation of the physical reality, and a detailed explanation of the solution of the resulting boundary value problems by using commercial Finite Element Analysis software such as Abaqus. Furthermore, this textbook includes 62 fully developed examples interspersed with the theory, as well as 82 exercises at the end of chapters.

The reader will be able to reproduce the examples and complete the exercises. When a finite element analysis is called for, the reader will be able to do it with commercially or otherwise available software. A website is set up with links to download the necessary software unless it is easily available from Finite Element Analysis software vendors. The use of Abaqus and MATLAB is explained with numerous

examples, and the relevant code can be downloaded from the website. Furthermore, the reader will be able to extend the capabilities of Abaqus by use of user material subroutines and Python scripting, as demonstrated in the examples included in this textbook.

Chapters 1 through 7 can be covered in a one-semester graduate course. Chapter 2 (Introduction to Finite Element Analysis) contains a brief introduction intended for those readers who have not had a formal course or prior knowledge about the finite element method. Chapter 4 (Buckling) is not referenced in the remainder of the textbook and thus it could be omitted in favor of more exhaustive coverage of content in later chapters. Chapters 7 (Viscoelasticity), 8 (Continuum Damage Mechanics), 9 (Discrete Damage Mechanics), and 11 (Fatigue) emphasize hereditary phenomena. Chapter 10 (Delaminations) emphasizes cohesive behavior and Chapter 12 (Programmable Features) discusses user-programmable features such as user materials (UMAT) and user general sections (UGENS) that are used to expand the capabilities of Abaqus to deal with issues germane to composites. Any of Chapters 7 to 12 can be skipped to fit the content into a one-semester course.

The inductive method is applied as much as possible in this textbook. That is, topics are introduced with examples of increasing complexity, until sufficient physical understanding is reached to introduce the general theory without difficulty. This method sometimes requires that at earlier stages of the presentation, certain facts, models, and relationships be accepted as fact, until they are completely proven later. For example, in Chapter 7, viscoelastic models are introduced early to aid the reader in gaining an appreciation for the response of viscoelastic materials. This is done simultaneously with a cursory introduction to the superposition principle and the Laplace transform, which are formally introduced only later in the chapter. For those readers accustomed to the deductive method, this may seem odd, but many years of teaching have convinced me that students acquire and retain knowledge more efficiently in this way.

It is assumed that the reader is familiar with basic mechanics of composites as covered in introductory textbooks such as my previous textbook, Introduction to Composite Material Design--Third Edition (2018). Furthermore, it is assumed that the reader masters a body of knowledge that is commonly acquired as part of a Bachelor of Science degree in Aerospace, Mechanical, Civil, or related disciplines. References to books and to other sections in this textbook, as well as footnotes, are used to assist the reader in refreshing those concepts and to clarify the notation used. Prior knowledge of continuum mechanics, tensor analysis, and the finite element method would enhance the learning experience but are not necessary for studying with this textbook.

The finite element method is used as a tool to solve practical problems. For the most part, Abaqus is used throughout the book. Fortran, Python, MATLAB, and Scilab are used for coding material models and post-processing algorithms. Basic knowledge of these programming languages is useful but not essential.

Only three software packages are used throughout the book. Abaqus is needed for finite element solutions of numerous examples and suggested problems. MATLAB is needed for both symbolic and numerical solutions of examples and suggested problems. Additionally, BMI3, which is available free of charge on the book's website, is used in Chapter 4. Other software such as ANSYS Mechanical, LSDYNA, MSCMarc, and SolidWorks are cited, but not used in the examples. Relevant code used in the examples is available on the book's website.

Composite materials are now ubiquitous in the marketplace, including extensive applications in aerospace, automotive, civil infrastructure, sporting goods, and so on. Their design is especially challenging because, unlike conventional materials such as metals, the composite material itself is designed concurrently with the composite structure. Preliminary design of composites assumes of a state of plane stress in the laminate. Furthermore, rough approximations are made about the geometry of the part, as well as the loading and support conditions. In this way, relatively simple analysis methods exist, and computations can be carried out simply using algebra. However, preliminary analysis methods have several shortcomings that are remedied with advanced mechanics and finite element analysis, as explained in this textbook.

Recent advances in commercial finite element analysis packages, with user-friendly pre- and post-processing, as well as powerful user-programmable features, have made detailed analysis of composites quite accessible to the designer. This textbook bridge the gap between powerful finite element tools and practical problems in structural analysis of composites. I expect that many graduate students, practicing engineers, and instructors will find this to be a useful and practical textbook on finite element analysis of composite materials based on sound understanding of advanced mechanics of composite materials.

Ever J. Barbero, 2023