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# Solidworks 2017 Learn By Doing Part Assembly Drawings

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## JOURNEY GUADALUPE

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SDC Publications

Beginner's Guide to SOLIDWORKS 2017 - Level II starts where Beginner's Guide - Level I ends, following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website.

*Solidworks 2013 Bible* SDC Publications

Engineering Design and Graphics with SolidWorks 2016 shows students how to use SolidWorks to create engineering drawings and designs. The textbook has been updated to cover the new features in SolidWorks 2016. It focuses on the creation of engineering drawings, including dimensions and tolerances and the use of standard parts and tools. Each chapter contains step-by-step sample problems that show students how to apply the concepts presented in the chapter. Effective pedagogy throughout the text helps students learn and retain concepts: Objectives: Each chapter begins with objectives and an introduction to the material. Summaries: Each chapter concludes with a summary and exercise problems. Numerous illustrations: The multitude of illustrations, accompanied by explanatory captions, present a visual approach to learning. Students see in the text what they see on the screen with the addition of explanatory text. Practical application: The text provides hundreds of exercise projects of varying difficulty (far more than any other computer graphics text). These exercises reinforce each chapter's content and help students learn by doing. Flexibility: With the hundreds of problems presented in the book, instructors can assign different problems within the same class and from year to year without repeating problems for students. Meets standards: The text teaches ANSI standards for dimensions and tolerances. This helps students understand how their designs are defined for production and the importance of proper tolerancing. Step-by-step approach: In presenting the fundamentals of engineering drawing using SolidWorks, the text uses a step-by-step approach that allows students to work and learn at their own pace.

SOLIDWORKS 2017 Reference Guide SDC Publications

This book will teach you everything you need to know to start using SOLIDWORKS 2017 with easy to understand, step-by-step

tutorials. This book features a simple robot design used as a project throughout the book. You will learn to model parts, create assemblies, run simulations and even create animations of your robot design. No previous experience with Computer Aided Design (CAD) is needed since this book starts at an introductory level. The author begins by getting you familiar with the SOLIDWORKS interface and its basic tools right away. You will start by learning to model simple robot parts and before long you will graduate to creating more complex parts and multi-view drawings. Along the way you will learn the fundamentals of parametric modeling through the use of geometric constraints and relationships. You will also become familiar with many of SOLIDWORKS's powerful tools and commands that enable you to easily construct complex features in your models. Also included is coverage of gears, gear trains and spur gear creation using SOLIDWORKS. This book continues by examining the different mechanisms commonly used in walking robots. You will learn the basic types of planar four-bar linkages commonly used in mechanical designs and how to use the GeoGebra Dynamic Geometry software to simulate and analyze 2D linkages. Using the knowledge you gained about linkages and mechanism, you will learn how to modify your robot and change its behavior by modifying or creating new parts. In the second to last chapter of this book you learn how to combine all the robot parts into assemblies and then run motion analysis. You will finish off your project by creating 3D animations of your robot in action. Finally, in the last chapter, the author introduces you to 3D printing. You will learn the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. Being able to turn your designs into physical objects will open up a whole new world of possibilities to you. There are

many books that show you how to perform individual tasks with SOLIDWORKS, but this book takes you through an entire project and shows you the complete engineering process. By the end of this book you will have modeled and assembled nearly all the parts that make up the TAMIYA® Mechanical Tiger and can start building your own robot.

*Getting started with Parts, Assemblies and Drawings* SDC Publications

Engineering Design with SOLIDWORKS 2017 and video instruction is written to assist students, designers, engineers and professionals. The book provides a solid foundation in SOLIDWORKS by utilizing projects with step-by-step instructions for the beginner to intermediate SOLIDWORKS user. Explore the user interface, CommandManager, menus, toolbars and modeling techniques to create parts, assemblies and drawings in an engineering environment. Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined, plastic and sheet metal components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, Design Tables, Bills of Materials, Custom Properties and Configurations. Address various SOLIDWORKS analysis tools and Intelligent Modeling techniques along with Additive Manufacturing (3D printing). Learn by doing not just by reading. Desired outcomes and usage competencies are listed for each project. Know your objective up front. Follow the steps in Projects 1 - 9 to achieve the design goals. Review Project 10 on Additive Manufacturing (3D printing) and its benefits and features. Understand the terms and technology used in low cost 3D printers. Work between multiple documents, features, commands and custom properties that represent how engineers and designers utilize SOLIDWORKS in industry. Review individual features, commands and tools with the video instruction. The projects contain exercises. The exercises analyze and examine usage competencies. Collaborate with leading industry suppliers such as SMC Corporation of America, Boston Gear and 80/20 Inc. Collaborative information translates into numerous formats such as paper drawings, electronic files, rendered images and animations. On-line intelligent catalogs guide designers to the product that meets both their geometric requirements and performance functionality. The author developed the industry

scenarios by combining his own industry experience with the knowledge of engineers, department managers, vendors and manufacturers. He is directly involved with SOLIDWORKS every day. His responsibilities go far beyond the creation of just a 3D model. The book is designed to complement the SOLIDWORKS Tutorials contained in SOLIDWORKS 2017.

**Engineering Design with SOLIDWORKS 2017 and Video Instruction** SDC Publications

Engineering Graphics with SOLIDWORKS 2017 and Video Instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The book is divided into four sections: Chapters 1 - 3 explore the history of engineering graphics, manual sketching techniques, orthographic projection, Third vs. First angle projection, multi-view drawings, dimensioning practices (ASME Y14.5-2009 standard), line type, fit type, tolerance, fasteners in general, general thread notes and the history of CAD leading to the development of SOLIDWORKS. Chapters 4 - 9 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Follow the step-by-step instructions in over 80 activities to develop eight parts, four sub-assemblies, three drawings and six document templates. Chapter 10 provides a section on the Certified Associate - Mechanical Design (CSWA) program with sample exam questions and initial and final SOLIDWORKS models. Chapter 11 provides a section on Additive Manufacturing (3D printing) and its benefits and features. Understand the terms and technology used in low cost 3D printers. Review individual features, commands, and tools using the video instruction and SOLIDWORKS Help. The chapter exercises analyze and examine usage competencies based on the chapter objectives. The book is designed to complement the SOLIDWORKS Tutorials located in the SOLIDWORKS Help menu. Desired outcomes and usage competencies are listed for each project. Know your objectives up front. Follow the step-by step procedures to achieve your design goals. Work between multiple documents, features, commands, and properties that represent how engineers and designers utilize SOLIDWORKS in industry. The author developed the industry scenarios by combining his own industry experience with the

knowledge of engineers, department managers, vendors, and manufacturers. This professional is directly involved with SOLIDWORKS every day. His responsibilities go far beyond the creation of just a 3D model.

*Certified User and Certified Professional* SDC Publications

Whether it's your first venture into 3D technical drawing software or you're switching to SolidWorks from something else, you're probably excited about what this CAD program has to offer. Chances are, you figure it's going to take awhile to get the hang of it before you can begin cranking out those perfectly precise 3D designs. SolidWorks For Dummies, 2nd Edition, can help you dramatically shorten that get-acquainted period! SolidWorks For Dummies, 2nd Edition will help you get up and running quickly on the leading 3D technical drawing software. You'll see how to set up SolidWorks to create the type of drawings your industry requires and how to take full advantage of its legendary 3D features. You'll discover how to: Work with virtual prototypes Understand the user interface Use templates and sketch, assemble, and create drawings Automate the drawing process Review drawings and collaborate with other team members Define and edit sketches Create dimensions and annotations Print or plot your drawings Leverage existing designs Sample files on the bonus CD-ROM show you how to apply the latest version of SolidWorks and accomplish specific tasks. Even if you're brand-new to CAD software, SolidWorks For Dummies, 2nd Edition will have you feeling like a pro in no time. You'll find you've entered a whole new dimension. Note: CD-ROM/DVD and other supplementary materials are not included as part of eBook file. Peachpit Press

SOLIDWORKS Surface Design 2021 for Beginners and Intermediate Users textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating real-world surface models. This textbook is a great help for SOLIDWORKS users new to surface design. It consists of total 106 pages covering the surface design environment of SOLIDWORKS. It teaches users to use SOLIDWORKS mechanical design software for creating parametric complex shape surface models that are not possible to create with solid modeling due to its limitations. This textbook not only focuses on the usage of the tools and commands of SOLIDWORKS for creating surface models

but also on the concept of design. It contains Tutorials followed by theory that provide users with step-by-step instructions for creating surface designs. Moreover, it ends with Hands-on Test Drives which allow users to experience the user friendly and technical capabilities of SOLIDWORKS. Main Features of the Textbook:

- Comprehensive coverage of tools
- Step-by-step real-world tutorials with every chapter
- Hands-on test drives to enhance the skills at the end of every chapter
- Additional notes and tips
- Customized content for faculty (PowerPoint Presentations)
- Free learning resources for faculty and students
- Technical support for the book by contacting info@cadartifex.com

SOLIDWORKS 2017 in 5 Hours with Video Instruction CADArtifex Solidworks 2017 Learn by Doing Createspace Independent Publishing Platform

**Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSDA, CSWSA-FEA (SOLIDWORKS 2015 - 2017)** SDC Publications

This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task. At the end of this book, you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling, assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. The author strived hard to include the commands required in the Certified SOLIDWORKS Associate and Certified SOLIDWORKS Professional Exams as listed on the SOLIDWORKS website. SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before. Most commands covered in this book have advanced options, which may not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most

frequently used commands. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a visual presentation of tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete the exercises.

**Autodesk Revit 2017 Architecture Certification Exam Study Guide** John Wiley & Sons

The book starts with basics of FEA, goes through all the simulation tools and ends up with practical examples of analysis. The book explains the Solver selection, iteration methods like Newton-Raphson method and integration techniques used by SolidWorks Simulation for functioning.

*Mastering SolidWorks* SDC Publications

Engineering & Computer Graphics Workbook Using SOLIDWORKS 2017 is an exercise-based workbook that uses step-by-step tutorials to cover the fundamentals of SOLIDWORKS 2017. The intended audience is college undergraduate engineering majors, but it could also be used in pre-college introductory engineering courses or by self learners. The text follows an educational paradigm that was researched and developed by the authors over many years. The paradigm is based on the concurrent engineering approach to engineering design in which the 3-D solid model data serves as the central hub for all aspects of the design process. The workbook systematically instructs the students to develop 3-D models using the rich tools afforded in SOLIDWORKS. The exercises then proceed to instruct the students on applications of the solid model to design analysis using finite elements, to assembly modeling and checking, to kinematic simulation, to rapid prototyping, and finally to projecting an engineering drawing. The workbook is ideally suited for courses in which a reverse engineering design project is assigned. This book contains clear and easy to understand instructions that enable the students to robustly learn the main features of SOLIDWORKS, with little or no instructor input.

**The Design Approach** John Wiley & Sons

This book is your self-study guide. The objective of this book is to help you learn SOLIDWORKS 2015 by using its various features. The fourteen lessons in this tutorial introduce you to the designing, documentation, and presentation in SOLIDWORKS 2015. The topics covered in this tutorial are part and assembly

design, drawings, sheetmetal, surface design, mold tools, weldments, DimXpert, and rendering. The skills you develop after completing this tutorial are:

- \* Basics of Part, Assembly, and drawings
- \* Creating Sketches
- \* Additional Part and Assembly tools
- \* Sheet Metal Design
- \* Basics of Surface design
- \* Mold Tools
- \* Design and documents Weldments
- \* GD&T using DimXpert
- \* Appearances and Rendering

SolidWorks 2015 Learn by Doing (Part, Assembly, Drawings, Sheet Metal, Surface Design, Mold Tools, Weldments, DimXpert, and Rendering) Createspace Independent Publishing Platform

Parametric Modeling with SOLIDWORKS 2017 contains a series of seventeen tutorial style lessons designed to introduce SOLIDWORKS 2017, solid modeling and parametric modeling techniques and concepts. This book introduces SOLIDWORKS 2017 on a step-by-step basis, starting with constructing basic shapes, all the way through to the creation of assembly drawings and motion analysis. This book takes a hands on, exercise intensive approach to all the important parametric modeling techniques and concepts. Each lesson introduces a new set of commands and concepts, building on previous lessons. The lessons guide the user from constructing basic shapes to building intelligent solid models, assemblies and creating multi-view drawings. This book also covers some of the more advanced features of SOLIDWORKS 2017, including how to use the SOLIDWORKS Design Library, basic motion analysis, collision detection and analysis with SimulationXpress. The exercises in this book cover the performance tasks that are included on the Certified SOLIDWORKS Associate (CSWA) Examination. Reference guides located at the front of the book and in each chapter show where these performance tasks are covered. This book also introduces you to the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. 3D printing makes it easier than ever for anyone to start turning their designs into physical objects and by the end of this book you will be ready to start printing out your own designs. John Wiley & Sons

SOLIDWORKS 2017 Basic Tools is the first book in a three part series. It introduces new users to the SOLIDWORKS interface, SOLIDWORKS tools and basic modeling techniques. It provides you with a strong understanding of SOLIDWORKS and covers the

creation of parts, assemblies and drawings. Every lesson and exercise in this book was created based on real world projects. Each of these projects have been broken down and developed into easy and comprehensible steps. Furthermore, at the end of every chapter there are self test questionnaires to ensure that you have gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step.

SDC Publications

SOLIDWORKS 2017 Learn by doing introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models. In fourteen tutorial based chapters, author guides you through all the necessary commands and options in SOLIDWORKS 2017, from sketching to parametric modeling and finally ending with rendering. The commands are presented one step at a time using simple examples. The approach used in this book helps you to become a skilled SOLIDWORKS user. SOLIDWORKS 2017 Learn by doing begins with introduction basic modeling. The later chapters focus on additional modeling, top-down assemblies, sheet metal modeling, drafting, surface modeling, mold tools, weldments, DimXpert, and rendering. Table of Contents 1. Getting Started 2. Modeling Basics 3. Assembly Basics 4. Creating Drawings 5. Sketching 6. Additional Modeling Tools 7. Sheet metal Modeling 8. Top-Down Assembly 9. Dimensions and Annotations 10. Surface Design 11. Mold Tools 12. Weldments 13. DimXpert 14. Appearances and Rendering

[Analysis of Machine Elements Using SOLIDWORKS Simulation 2017 Solidworks 2017Learn by Doing](#)

- Uses step-by-step, project based tutorials designed for beginning or intermediate users
- Will prepare you for the Certified SOLIDWORKS Associate Exam
- Includes a chapter introducing you to 3D printing

SOLIDWORKS 2022 Tutorial is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories in the CSWA exam. The book is divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document

and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

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SOLIDWORKS 2019 Tutorial is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories in the CSWA exam. The book is divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are

available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

**A Step-by-Step Project Based Approach Utilizing 3D Modeling** SDC Publications

Motion Simulation and Mechanism Design with SOLIDWORKS Motion 2017 is written to help you become familiar with SOLIDWORKS Motion, an add-on module of the SOLIDWORKS software family. This book covers the basic concepts and frequently used commands required to advance readers from a novice to intermediate level in using SOLIDWORKS Motion. SOLIDWORKS Motion allows you to use solid models created in SOLIDWORKS to simulate and visualize mechanism motion and performance. Using SOLIDWORKS Motion early in the product development stage could prevent costly redesign due to design defects found in the physical testing phase. Therefore, using SOLIDWORKS Motion contributes to a more cost effective, reliable, and efficient product design process. Basic concepts discussed in this book include model generation, such as creating assembly mates for proper motion; carrying out simulation and animation; and visualizing simulation results, such as graphs and spreadsheet data. These concepts are introduced using simple, yet realistic examples. Verifying the results obtained from the

computer simulation is extremely important. One of the unique features of this book is the incorporation of theoretical discussions for kinematic and dynamic analyses in conjunction with the simulation results obtained using SOLIDWORKS Motion. Verifying the simulation results will increase your confidence in using the software and prevent you from being fooled by erroneous simulations.

SDC Publications

The SOLIDWORKS 2017 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2017. SOLIDWORKS is an immense software package, and no one book can cover all topics for all users. This book provides a centralized reference location to address many of the tools, features and techniques of SOLIDWORKS 2017. This book covers the following: System and Document propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySOLIDWORKS SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2017 software. If you are completely new to SOLIDWORKS, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in the SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS, you still might want to skim Chapter 1 to become acquainted with some of the commands, menus and

features that you have not used; or you can simply jump to any section in any chapter. Each chapter provides detailed PropertyManager information on key topics with individual stand-alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The book provides access to over 250 models, their solutions and additional support materials. Learn by doing, not just by reading. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables, configurations and more. The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2017. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is directly involved with SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model.

#### **A Power Guide for Beginners and Intermediate Users**

Cadcamcae Works

Analysis of Machine Elements Using SOLIDWORKS Simulation 2017 is written primarily for first-time SOLIDWORKS Simulation 2017 users who wish to understand finite element analysis capabilities applicable to stress analysis of mechanical elements. The focus of examples is on problems commonly found in an

introductory, undergraduate, Design of Machine Elements or similarly named courses. In order to be compatible with most machine design textbooks, this text begins with problems that can be solved with a basic understanding of mechanics of materials. Problem types quickly migrate to include states of stress found in more specialized situations common to a design of mechanical elements course. Paralleling this progression of problem types, each chapter introduces new software concepts and capabilities. Many examples are accompanied by problem solutions based on use of classical equations for stress determination. Unlike many step-by-step user guides that only list a succession of steps, which if followed correctly lead to successful solution of a problem, this text attempts to provide insight into why each step is performed. This approach amplifies two fundamental tenets of this text. The first is that a better understanding of course topics related to stress determination is realized when classical methods and finite element solutions are considered together. The second tenet is that finite element solutions should always be verified by checking, whether by classical stress equations or experimentation. Each chapter begins with a list of learning objectives related to specific capabilities of the SOLIDWORKS Simulation program introduced in that chapter. Most software capabilities are repeated in subsequent examples so that users gain familiarity with their purpose and are capable of using them in future problems. All end-of-chapter problems are accompanied by evaluation "check sheets" to facilitate grading assignments.