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# Solidworks Essentials Parts And Assemblies Volume 2 Solidworks Training

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SolidWorks Essentials Training Course Overview SOLIDWORKS Essentials Training Overview How to Download \u0026amp; Install Solidworks Student Edition SOLIDWORKS Tutorial - Sheet Metal SolidWorks Part Design Tutorial: Simple \u0026amp; Advanced Techniques Explained | Part No 276 SOLIDWORKS Lighter TUTORIAL BASIC SolidWorks Complete College Course for Beginners w/Training Guide Solidworks Full Course | Beginner to Advance FREE || Including 4 Projects How to install SolidWorks 2022 | Full installation without any error. Suspension System Components SOLIDWORKS Essentials: What every user needs to know! 01 SOLIDWORKS Essentials ( Welcome) Advanced SolidWorks Surface Modeling Tutorial: Complex Part Design Step-by-Step | Part No 277 SOLIDWORKS Essentials Training Introduction to

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SolidWorks Tutorial - Apply Material in Bulk to All the Parts of the Assembly From  
Zero to Hero in SolidWorks - Creating Your First Part, Assembly, and Drawing in 30  
Minutes Solidworks Tutorial Parts and Assemblies Basics of Creating and Using a  
SOLIDWORKS Assembly SolidWorks Fundamentals  
SOLIDWORKS 2016 Tutorial with Video Instruction  
SolidWorks Office 2005  
Beginner's Guide to SOLIDWORKS 2022 - Level II  
SolidWorks 2010 Tutorial  
SOLIDWORKS 2021: A Power Guide for Beginners and Intermediate Users  
SolidWorks Office 2005  
SolidWorks 2011 Assemblies Bible  
Solidworks 2019 Basics and Beyond: Part Modeling, Assemblies, and Drawings  
SolidWorks 2005  
SOLIDWORKS 2019 Tutorial  
SolidWorks 2009  
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Beginner's Guide to SOLIDWORKS 2022 - Level I

SOLIDWORKS 2016 Basic Tools  
SolidWork 2005  
Learn SOLIDWORKS

*Solidworks  
Essentials  
Parts And  
Assemblies  
Volume 2  
Solidworks  
Training*

*OMB No.  
8890174953630  
edited by*

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**NICHOLSON AUGUST**

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SOLIDWORKS 2016  
Tutorial with Video  
Instruction CADArtifex  
Beginner's Guide to  
SOLIDWORKS 2022 -  
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. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a clear 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 while he provides additional details along the way. Captioned versions of these videos are also available for customers who want or need video captions.

**SolidWorks Office 2005**  
SDC Publications  
Learn the essential tools and techniques in SOLIDWORKS 2019.  
Create ready-to-manufacture parts and assemblies, detailed

drawings, and itemized bills of materials.

Beginner's Guide to SOLIDWORKS 2022 - Level II  
SDC Publications

This book provides the reader with a comprehensive knowledge of all the tools provided in the software SOLIDWORKS for a variety of engineering areas. It presents a broad choice of examples to be imitated in one's own work. In developing these examples, the authors' intent has been to exercise many program features and refinements.

By displaying these, the authors hope to give readers the confidence to employ these program enhancements in their own modeling applications.

SDC Publications

- Starts at an introductory level, designed for beginners •
- Comprehensive coverage of beginning tools and techniques •
- Uses a step by step, tutorial approach with real world projects •
- Covers the creation of parts, assemblies and drawings •
- Features a quick reference guide and

a Certified SOLIDWORKS Associate practice exam • The first book of a three book series SOLIDWORKS 2021 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 has 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. Who this book is for This book is for

the beginner who is not familiar with the SOLIDWORKS program and its add ins. SolidWorks 2010 Tutorial New Age International SOLIDWORKS is the industry standard in 3D parametric modeling software, making it an essential tool for anyone going into a wide variety of engineering and design industries. Specifically written for those who are new to SOLIDWORKS, A Hands-On Introduction to SOLIDWORKS 2022 allows you to relax and learn as you follow an expert in

SOLIDWORKS through the basics of the software to its more in-depth capabilities. Formerly called Project Based SOLIDWORKS, this revised edition includes new and expanded tutorials. This book works perfectly for a freshman design class or as a companion text to an engineering graphics textbook. Each tutorial in the book teaches you how to use engineering graphics concepts while modeling real-world parts and assemblies. Learn how to model parts, configurations, create part

prints, and assembly drawings. As you become more comfortable with SOLIDWORKS, later chapters introduce FEA, how to create more complex solid geometries with parametric modeling, apply tolerances, and use advanced and mechanical mates. Important commands and features are highlighted and defined in each chapter to help you become familiar with them. Instructional videos for all the tutorials and the end-of-chapter problems come with the book, so if you need more

help, or are a visual learner, you can refer to them. Some problems are purposely left open ended to simulate real life design situations; therefore, more than one solution is possible. After completing all the tutorials in this book, you will be able to accurately design moderately difficult parts and assemblies and have a firm foundation in SOLIDWORKS. Why this book? Instructors and learners will appreciate the thoughtful and well-organized layout of A Hands-On Introduction to

SOLIDWORKS 2022. Every chapter begins with the prerequisites needed to complete the tutorials found in the chapter and a list of what you will learn. You do not necessarily need to complete the tutorials within the book in order, but make sure that you have the prerequisite knowledge before you begin. Practice modeling problems and/or quiz problems at the end of each chapter offer an extra challenge and let you practice your newfound skills. Working with realistic part models

and assemblies means that questions and problems might arise as they would when you are working on your real-life projects. The author anticipates these questions and how to address them. For example, if you are in the wrong standard or not on the correct layer, or an unexpected window appears on the screen, tips and notes quickly remedy the issue. Work alongside the author using the instructional videos included for every tutorial and end-of

chapter problems in the book. Information on new commands or steps appear at the beginning of each chapter. They include definitions of new features and concepts and images of how they look on the screen. Everything is clearly labeled for easy identification. Throughout the book, readers are referred to the appropriate section of the chapter for more information on the command when needed. A command index at the back of the book lists

where each command can be found for easy reference at any time.

## **SOLIDWORKS 2021: A POWER GUIDE FOR BEGINNERS AND INTERMEDIATE USERS**

SDC Publications  
Provides an introduction to SolidWorks 2010 through step-by-step tutorials that cover such topics as linkage assembly, front support assembly, the fundamentals of drawing, and pneumatic test module assembly.



## **SOLIDWORKS OFFICE 2005**

SDC Publications  
SOLIDWORKS 2021: A  
Power Guide 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  
3D mechanical design.  
This textbook is a great  
help for new SOLIDWORKS  
users and a great  
teaching aid in classroom

training. This textbook  
consists of 14 chapters,  
with a total of 798 pages  
covering the major  
environments of  
SOLIDWORKS such as  
Sketching environment,  
Part modeling  
environment, Assembly  
environment, and  
Drawing environment.  
This textbook teaches  
users to use SOLIDWORKS  
mechanical design  
software for creating  
parametric 3D solid  
components, assemblies,  
and 2D drawings. This  
textbook also includes a  
chapter on creating

multiple configurations of  
a design. This textbook  
not only focuses on the  
usage of the tools and  
commands of  
SOLIDWORKS but also on  
the concept of design.  
Every chapter in this  
textbook contains  
tutorials that provide  
users with step-by-step  
instructions for creating  
mechanical designs and  
drawings with ease.  
Moreover, every chapter  
ends with hands-on test  
drives which allow users  
to experience the user  
friendly and technical  
capabilities of

SOLIDWORKS.

*SolidWorks 2011*

*Assemblies Bible* CRC  
Press

This text is a teaching companion and shows how to use the SolidWorks mechanical design automation software to build parametric models of parts and assemblies and how to make simple drawings of those parts and assemblies.

*Solidworks 2019 Basics and Beyond: Part Modeling, Assemblies, and Drawings* SDC  
Publications

SOLIDWORKS 2018

Tutorial with video instruction 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, to take and understand the Certified Associate - Mechanical Design (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. **SolidWorks 2005** SDC Publications  
SOLIDWORKS 2018 Quick Start with video

instruction introduces the new user to the basics of using SOLIDWORKS 3D CAD software in five easy lessons. This book is intended for the student or designer that needs to learn SOLIDWORKS quickly and effectively for senior capstone, machine design, kinematics, dynamics, and other engineering and technology projects that use SOLIDWORKS as a tool. Engineers in industry are expected to have SOLIDWORKS skills for their company's next project. Students need to

learn SOLIDWORKS without taking a formal CAD course. Based on years of teaching SOLIDWORKS to engineering students, SOLIDWORKS 2018 in 5 Hours concentrates on the areas where the new user improves efficiency in the design modeling process. By learning the correct SOLIDWORKS skills and file management techniques, you gain the most knowledge in the shortest period of time. You develop a mini Stirling Engine and investigate the proper

design intent and constraints. The mini Stirling Engine is based on the external combustion, closed cycle engine of Scottish inventor Robert Stirling. In addition to 3D modeling, the engine can be used to teach and connect many engineering and physics principles. You begin with an overview of SOLIDWORKS and the User Interface (UI), its menus, toolbars and commands. With a quick pace, you learn the essentials of 2D sketching, part and

assembly creation, perform motion study, develop detailed part and assembly drawings and much more.

### **SOLIDWORKS 2019**

**Tutorial** SDC Publications

This book is a blend of focused discussions, real-world examples, and practice exercises. It helps you to learn the latest version of SOLIDWORKS quickly and easily. You can learn and implement the software by following the topics arranged systematically. However, you can jump to the tutorials at the end of

each chapter and start using the essential features of the software. The interesting examples used in tutorials will show how to use the software in the design process. With all the vital topics of part modeling, assemblies, and drawings, this book is a good companion. Table of Contents 1. Getting Started with SOLIDWORKS 2. Sketch Techniques 3. Extrude and Revolve Features 4. Placed Features 5. Patterned Geometry 6. Sweep Features 7. Loft Features 8. Additional Features and

Multibody Parts 9. Modifying Parts 10. Assemblies 11. Drawings *SolidWorks 2009* Springer Nature SOLIDWORKS 2018 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 has 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.  
**SOLIDWORKS 2018 Quick Start with Video Instruction** SDC Publications  
 SOLIDWORKS 2022 Advanced Techniques picks up where SOLIDWORKS 2022 Intermediate Skills leaves off. Its aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools

provided by SOLIDWORKS. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real world projects. Each of these projects has 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. Who this book is for This book is for the intermediate to advanced user who has already completed the SOLIDWORKS Basic Tools book and may have also completed the SOLIDWORKS Intermediate Skills book. People who are very

familiar with SOLIDWORKS and its add ins will also find this book to be a valuable resource.

### **Mastering SolidWorks**

World Scientific  
SOLIDWORKS 2016  
Tutorial with Video  
Instruction is targeted towards a technical school, two year college, four year university or industry professional that is a beginner or intermediate CAD user. The text provides a student who is looking for a step-by-step project based approach to learning SOLIDWORKS

with video instruction, SOLIDWORKS model files, and preparation for the Certified Associate - Mechanical Design (CSWA) exam. The book is divided into three sections. Chapters 1 - 6 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, Revision

tables using basic and advanced features. 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. Review Chapter 11 on Additive Manufacturing (3D printing) and its benefits and features. Understand 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. [Beginner's Guide to SOLIDWORKS 2022 - Level I](#) SDC Publications Get to grips with leading 3D engineering and product design application to design robust 3D models and achieve



CSWA and CSWP certification Key FeaturesGain comprehensive insights into the core aspects of 3D modeling's mechanical partsLearn how to generate assembly designs with both standard and advanced matesDiscover design practices for both 2D as well as 3D modeling and prepare to achieve CSWP and CSWA certificationBook Description SOLIDWORKS is the leading choice for 3D engineering and product design

applications across industries such as aviation, automobile, and consumer product design. This book helps you to get up and running with SOLIDWORKS and understand each new concept and tool with the help of easy-to-follow exercises. You'll begin with the basics, exploring the software interface and finding out how to work with drawing files. The book then guides you through topics such as sketching, building complex 3D models, generating dynamic and

static assemblies, and generating 2D engineering drawings to prepare you to take on any design project. You'll also work with practical exercises to get hands-on experience with creating sketches, 3D part models, assemblies, and drawings. To reinforce your understanding of SOLIDWORKS, the book is supplemented by downloadable files that will help you to understand the concepts and exercises more easily. Finally, you'll also work on projects for 3D

modeling objects inspired by everyday life. By the end of this SOLIDWORKS book, you'll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS and be able to prepare effectively for the Certified SOLIDWORKS Associate (CSWA) and Certified SOLIDWORKS Professional (CSWP) exams. What you will learn Understand the fundamentals of SOLIDWORKS and parametric modeling Create professional 2D sketches

as bases for 3D models using simple and advanced modeling techniques Use SOLIDWORKS drawing tools to generate standard engineering drawings Evaluate mass properties and materials for designing parts and assemblies Join different parts together to form static and dynamic assemblies Discover expert tips and tricks to generate different part and assembly configurations for your mechanical designs Who this book is for This book

is for aspiring engineers, designers, makers, draftsmen, and hobbyists looking to get started with SOLIDWORKS and explore the software. Individuals who are interested in becoming Certified SOLIDWORKS Associates (CSWAs) or Certified SOLIDWORKS Professionals (CSWPs) will also find this book useful. No specific background is needed to follow the concepts in the book as it starts from the basics of SOLIDWORKS. However, basic theoretical knowledge of 3D

modeling will be helpful to get the most out of this book.

*SOLIDWORKS 2016 Basic Tools* SDC Publications  
SOLIDWORKS is the premier product-design software-enabling you to deliver innovative solutions to market faster. In this course, Gabriel Corbett teaches you how to build ready-to-manufacture parts and assemblies as well as detailed drawings and itemized bills of materials in SOLIDWORKS 2020. Learn how to create 2D sketches and then

extrude, revolve, loft, and sweep these features into 3D objects. Find out how to make cuts and create standard holes, and explore more advanced modeling techniques using blocks, mirroring, and the SOLIDWORKS pattern tools. Then review best practices for putting parts together in assemblies and creating detailed drawings. Along the way, Gabriel highlights new features from the 2020 release, as well as features designed for users with pen or touch devices.

**SolidWork 2005** SDC Publications

- Starts at an introductory level, designed for beginners • Comprehensive coverage of beginning tools and techniques • Uses a step by step, tutorial approach with real world projects • Covers the creation of parts, assemblies and drawings • Features a quick reference guide and a Certified SOLIDWORKS Associate practice exam • The first book of a three book series SOLIDWORKS 2022 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 has 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. Who this book is for This book is for the beginner who is not familiar with the SOLIDWORKS program and its add ins.  
Learn SOLIDWORKS  
 Independently Published

This senior undergraduate level textbook is written for Advanced Manufacturing, Additive Manufacturing, as well as CAD/CAM courses. Its goal is to assist students in colleges and universities, designers, engineers, and professionals interested in using SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications. This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the

relatively new manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM) technology. This new textbook: Features modeling of complex parts and surfaces Provides a step-by-step tutorial type approach with pictures showing how to model using SolidWorks Offers a user-Friendly approach for the design of parts, assemblies, and drawings, motion-analysis, and FEA topics Includes clarification of connections between

SolidWorks and 3D-Printing based on Additive Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings.

*SOLIDWORKS 2020 Essential Training* Packt Publishing Ltd  
A fan of the SolidWorks Bible, but want more detail on assemblies? Here you go. SolidWorks fans have long sought more detail on SolidWorkstopics, and now you have it. We took our popular SolidWorksBible, divided it into two books (SolidWorks 2011 AssembliesBible and SolidWorks 2011 Parts Bible) and packed each new book with a host of items from your wish

lists, such as more extensive coverage of the basics, additional tutorials, and expanded coverage of topics largely ignored by other books. This SolidWorks 2011 Assemblies Bible shows you how to organize parts data to create assemblies or subassemblies using the latest version of the 3D solid modeling program, SolidWorks. Thoroughly describes best practices and beginning-to-advanced techniques using both video and text. Explains and thoroughly covers every assembly

function and is written in a way that enables the reader to make better decisions while using the software. Written by well-known and well-respected SolidWorks guru Matt Lombard. Can stand alone or also with the SolidWorks 2011 Parts Bible for a complete SolidWorks reference set. Keep both the SolidWorks 2011 Assemblies Bible and the SolidWorks 2011 Parts Bible on your desk, and you'll have the best resource set out there on SolidWorks. [A Hands-On Introduction](#)

[to SOLIDWORKS 2022](#)  
John Wiley & Sons  
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. Throughout this book the author introduces you to new commands that are required to pass the Certified SOLIDWORKS Associate exam, as listed on the SOLIDWORKS website. A dedicated chapter provides you with details about the exam, as well as a practice test to help you prepare for the actual exam.

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.

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