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SOLIDWORKS 2021: A Power Guide
for Beginners and Intermediate Users
textbook has been designed for
instructor-led courses as well as self-

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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

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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

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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

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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

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and technical capabilities of
SOLIDWORKS.

Parametric Modeling with
SOLIDWORKS 2021 contains a series
of seventeen tutorial style lessons
designed to introduce SOLIDWORKS
2021, solid modeling and parametric
modeling techniques and concepts.

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This book introduces SOLIDWORKS 2021 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

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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

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of the more advanced features of SOLIDWORKS 2021, 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

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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

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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

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printing out your own designs.

- 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

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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

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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

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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

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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

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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

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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

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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,

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custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

Uses Finite Element Analysis (FEA) as Implemented in SolidWorks Simulation Outlining a path that readers can follow to ensure a static

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analysis that is both accurate and sound, Introduction to Static Analysis using SolidWorks Simulation effectively applies one of the most widely used software packages for engineering design to the concepts of static analysis. This text utilizes a step-by-step approach to introduce

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the use of a finite element simulation within a computer-aided design (CAD) tool environment. It does not center on formulae and the theory of FEM; in fact, it contains essentially no theory on FEM other than practical guidelines. The book is self-contained and enables the reader to progress

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independently without an instructor. It is a valuable guide for students, educators, and practicing professionals who wish to forego commercial training programs, but need to refresh or improve their knowledge of the subject. Classroom Tested with Figures, Examples, and

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Homework Problems The book contains more than 300 illustrations and extensive explanatory notes covering the features of the SolidWorks (SW) Simulation software. The author presents commonly used examples and techniques highlighting the close interaction between CAD

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modelling and FE analysis. She describes the stages and program demands used during static analysis, details different cases, and explores the impact of selected options on the final result. In addition, the book includes hands-on exercises, program commands, and a summary after

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each chapter. Explores the static studies of simple bodies to more complex structures Considers different types of loads and how to start the loads property managers Studies the workflow of the run analysis and discusses how to assess the feedback provided by the study

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manager Covers the generation of graphs Determines how to assess the quality of the created mesh based on the final results and how to improve the accuracy of the results by changing the mesh properties Examines a machine unit with planar symmetrical geometry or with

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circular geometry exposed to
symmetrical boundary conditions
Compares 3D FEA to 2D FEA
Discusses the impact of the adopted
calculating formulation by comparing
thin-plate results to thick-plate
results Introduction to Static Analysis
using SolidWorks Simulation equips

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students, educators, and practicing professionals with an in-depth understanding of the features of SW Simulation applicable to static analysis (FEA/FEM).

A Step-by-Step Project Based
Approach Utilizing 3D Modeling
SOLIDWORKS 2018 Tutorial with

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Video Instruction

Part Modeling | Assemblies | Drawings

SOLIDWORKS 2020 Tutorial

SolidWorks 2021 - Step-By-Step
Guide

*SOLIDWORKS 2022 for Designers
book is written to help the readers*

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effectively use the modeling and assembly tools by utilizing the parametric and feature-based approach of SOLIDWORKS 2022. This book provides a detailed description of the tools that are commonly used in modeling and

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assembly. The SOLIDWORKS 2022 for Designers book further elaborates on the procedures of generating the drawings of a model or assembly, which are used for documentation of a model or assembly. Special emphasis has

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been laid on the introduction of concepts, which have been explained using detailed text, along with graphical examples. The examples and tutorials used in this book ensure that the users can relate the information provided in this

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textbook with the practical industry designs. Salient Features Consists of 14 chapters that are organized in a pedagogical sequence.

Comprehensive coverage of SOLIDWORKS 2022 concepts and techniques. Hundreds of

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illustrations and tutorial approach to explain the concepts of SOLIDWORKS 2022. Summary on the first page of the topics that are covered in the chapter. Step-by-step instructions guide the users through the learning process. Real-world

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mechanical engineering designs as tutorials and projects. Additional information throughout the book in the form of notes and tips. Self-Evaluation Tests and Review Questions at the end of each chapter to help students assess their

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knowledge. Table of Contents

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*Sketches Chapter 5: Advanced
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Creating Reference Geometries
Chapter 7: Advanced Modeling
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10:Advanced Modeling Tools-III

Chapter 11: Assembly Modeling-I

Chapter 12: Assembly Modeling-II

Chapter 13: Working with Drawing

View-I Chapter 14: Working with

Drawing View-II Index Student

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Projects SOLIDWORKS*

Certification Exam (* For free
download)*

*The complete SolidWorks reference-
tutorial for beginner to advanced
techniques Mastering SolidWorks is
the reference-tutorial for all users.*

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Packed with step-by-step instructions, video tutorials for over 40 chapters, and coverage of little-known techniques, this book takes you from novice to power user with clear instruction that goes beyond the basics. Fundamental techniques

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are detailed with real-world examples for hands-on learning, and the companion website provides tutorial files for all exercises. Even veteran users will find value in new techniques that make familiar tasks faster, easier, and more organized,

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including advanced file management tools that simplify and streamline pre-flight checks. SolidWorks is the leading 3D CAD program, and is an essential tool for engineers, mechanical designers, industrial designers, and drafters around the

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world. User friendly features such as drag-and-drop, point-and-click, and cut-and-paste tools belie the software's powerful capabilities that can help you create cleaner, more precise, more polished designs in a fraction of the time. This book is

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*the comprehensive reference every
SolidWorks user needs, with
tutorials, background, and more for
beginner to advanced techniques.
Get a grasp on fundamental
SolidWorks 2D and 3D tasks using
realistic examples with text-based*

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*tutorials Delve into advanced
functionality and capabilities not
commonly covered by how-to guides
Incorporate improved search, Pack-
and-Go and other file management
tools into your workflow Adopt best
practices and exclusive techniques*

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you won't find anywhere else Work through this book beginning-to-end as a complete SolidWorks course, or dip in as needed to learn new techniques and time-saving tricks on-demand. Organized for efficiency and designed for practicality, these

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tips will remain useful at any stage of expertise. With exclusive coverage and informative detail, Mastering SolidWorks is the tutorial-reference for users at every level of expertise.

Designed to provide an insight into

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the Mechanical Design concept
DESCRIPTION *The book promises to make you understand and practice the SolidWorks framework. The aim of this book is to take you on a journey to all the phases of SolidWorks. SolidWorks is an*

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innovative, next-generation industry software that allows you to solve and understand the designing and mechanical problems. SolidWorks uses a technical implementation approach for sketching, surfacing, and sheet metal drafting in an

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incremental and easy way. The main objective of this book is to make the reader understand the concepts of design based on practical knowledge rather than theoretical knowledge. KEY FEATURES Each command is explained in a simple

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and understandable manner Step-by-step explanation Practical knowledge rather than theoretical knowledge Covers all the modules of SolidsWorks 2019 WHAT WILL YOU LEARN SolidWorks and its GUI Sketches (Line, Rectangle, Slot,

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*Circle, ARC, Polygon, and Spline)
Extrude, Revolved, Swept, Loft,
Boundary, Filt, and Chamfer)
Surface (Extruded, Revolved, Swept,
Lofted, Boundary, Filled, and
Planner) Sheet metal (Base
flange/tab, Edge flange, Miter*

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*flange, and Hem) Weldments
(Structural member, Trim/Extend,
End cap, and Gusset) Curves Mold
design Drafting Assembly WHO
THIS BOOK IS FOR Mechanical
engineers and designers, automobile
engineers, product designers, heavy*

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vehicle designers. Table of Contents
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Drafting

This book is intended to help new

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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

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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

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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

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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

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*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*

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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

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*basic and most frequently used
commands.*

Learning SOLIDWORKS 2017

A Tutorial Approach

Solidworks 2013 Bible

SolidWorks 2014 Reference Guide

Understanding CATIA

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SolidWorks 2012 Tutorial with Video Instruction is target 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

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for a step-by-step project based approach to learning SolidWorks with an enclosed 1.5 hour video instruction DVD, SolidWorks model files, and preparation for the CSWA exam. The book is divided into two sections. Chapters 1 - 7 explore the

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SolidWorks User Interface and
CommandManager, Document and
System properties, simple machine
parts, simple and complex
assemblies, design tables,
configurations, multi-sheet, multi-
view drawings, BOMs, Revision

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tables using basic and advanced features along with Intelligent Modeling Techniques, SustainabilityXpress, SimulationXpress and DFMXpress. Chapters 8 - 11 prepare you for the new Certified SolidWorks Associate

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Exam (CSWA). The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Follow the step-by-step instructions and develop multiple assemblies that combine over 100

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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,

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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

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goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SolidWorks in industry.

SOLIDWORKS 2021: A Step-By-

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Step Tutorial Guide for Beginners (Mixed Units) textbook is intended to help students, designers, engineers, and professionals who are interested in learning SOLIDWORKS step-by-step for creating real world 3D mechanical

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designs. It is a great starting point for new users of SOLIDWORKS and for those moving from other CAD software. This textbook contains tutorials that provide users with step-by-step instructions for creating parametric 3D solid

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components, assemblies, and 2D drawings with ease. Every tutorial in this textbook is created based on real-world projects. This textbook consists of 11 chapters, a total of 428 pages covering major environments of SOLIDWORKS

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such as Part modeling environment, Assembly environment, and Drawing environment including configurations. Every chapter ends with exercises that allow users to experience for themselves the user

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friendly and powerful capacities of SOLIDWORKS, which help users to assess their knowledge.

This book provides a key understanding of CATIA which is a solid modeling software. By using screen shots of step-by-step

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training, the reader will obtain comprehensive knowledge of all tools provided in CATIA for use in a variety of engineering fields. The book introduces CATIA basics, covers part design, discusses sheet metal design, talks about assembly,

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presents drawings and shows modeling of an engineered component. The primary aim of this book is to assist in learning the use of CATIA software through examples taken from various areas of engineering. The content and

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treatment of the subject matter is most appropriate for university students studying engineering and practicing engineers who wish to learn the use of CATIA.

**SOLIDWORKS 2018: A Power
Guide for Beginners and**

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Intermediate User textbook is designed for instructor-led courses as well as for self-paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating 3D mechanical design. This textbook is

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a great help for new SOLIDWORKS users and a great teaching aid in classroom training. This textbook consists of 14 chapters, total 782 pages covering major environments of SOLIDWORKS: Sketching environment, Part modeling

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environment, Assembly environment, and Drawing environment, which teach you how to use the SOLIDWORKS mechanical design software to build parametric models and assemblies, and how to make drawings of those

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parts and assemblies. This textbook also includes a chapter on creating multiple configurations of a design. This textbook not only focuses on the usages of the tools/commands of SOLIDWORKS but also on the concept of design.

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Every chapter of this textbook contains tutorials which instruct users how things can be done in SOLIDWORKS step by step. Moreover, every chapter ends with hands-on test drives which allow users to experience themselves the

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ease-of-use and powerful
capabilities of SOLIDWORKS.

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Dimensions Chapter 5. Creating
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Chapter 6. Creating Reference
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Modeling - I Chapter 8. Advanced
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Advanced Modeling - III Chapter
11. Working with Configurations
Chapter 12. Working with
Assemblies - I Chapter 13. Working
with Assemblies - II Chapter 14.
Working with Drawings Main

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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

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Part, Assembly, Drawings, Sheet
Metal, and Surfacing
SOLIDWORKS 2022 for Designers,
20th Edition
SOLIDWORKS 2021 Tutorial
Mastering SolidWorks
Parametric Modeling with

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SOLIDWORKS 2021

- A comprehensive reference book for SOLIDWORKS 2020
- Contains 260 plus standalone tutorials
- Starts with a basic overview of SOLIDWORKS 2020 and its new features
- Tutorials are written for each topic with new and intermediate users in mind
- Includes

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access to each tutorial's initial and final state • Contains a chapter introducing you to 3D printing The SOLIDWORKS 2020 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2020. SOLIDWORKS is an immense

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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 2020. This book covers the following:

- System and Document properties •

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FeatureManagers • PropertyManagers
• ConfigurationManagers •
RenderManagers • 2D and 3D Sketch
tools • Sketch entities • 3D Feature
tools • Motion Study • Sheet Metal •
Motion Study • SOLIDWORKS
Simulation • PhotoView 360 • Pack
and Go • 3D PDFs • Intelligent

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Modeling techniques • 3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2020 software. If you are completely new to SOLIDWORKS, you should read Chapter 1 in detail

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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

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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 260

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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,

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configurations and more. The book is designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2020. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the

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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.

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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

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SOLIDWORKS User Interface and
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design intent, design tables,
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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

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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

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commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

A comprehensive guide to SolidWorks 2007 provides information on such topics as customizing the user interface, building intelligence into

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parts, working with patterns and equations, and writing Visual Basic macros.

The SOLIDWORKS 2021 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2021. SOLIDWORKS is an immense

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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 2021. This book covers the following: System and Document properties

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FeatureManagers PropertyManagers
ConfigurationManagers
RenderManagers 2D and 3D Sketch
tools Sketch entities 3D Feature tools
Motion Study Sheet Metal Motion
Study SOLIDWORKS Simulation
PhotoView 360 Pack and Go 3D PDFs
Intelligent Modeling techniques 3D

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printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2021 software. If you are completely new to SOLIDWORKS, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and

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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.

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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 260 models, their solutions and additional

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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

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designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2021. 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

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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.

SOLIDWORKS 2021: A Power Guide

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for Beginners and Intermediate Users

Beginner's Guide to SOLIDWORKS

2019 - Level II

Solidworks 2018

Sheet Metal, Top Down Design,

Weldments, Surfacing and Molds

Beginner's Guide to SOLIDWORKS

2020 - Level II

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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

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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

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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

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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

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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,

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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

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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

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book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands. The SOLIDWORKS 2017 Reference Guide is a comprehensive reference book

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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

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centralized reference location to address many of the tools, features and techniques of SOLIDWORKS 2017. This book covers the following: System and Document properties Feature Managers Property Managers Configur

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ationManagersRenderManagers
2D and 3D Sketch toolsSketch
entities3D Feature toolsMotion
StudySheet MetalMotion
StudySOLIDWORKS
SimulationPhotoView 360Pack
and Go3D PDFsIntelligent

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Modeling techniques 3D 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

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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

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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

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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

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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

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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

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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

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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.

This book starts with SolidWorks

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2021 using step-by-step examples. It begins with creating sketches and parts, assembling them, and then creating print ready drawings. This book gives you an idea about how you can design and document various

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mechanical components, and helps you to learn some advanced tools and techniques. This book also follows some of the best practices in creating parts. In addition to this, there are some additional chapters

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covering sheet metal and surface design. Each topic in this book has a brief introduction and a step-by-step example. This will help you to learn SolidWorks 2018 quickly and easily. * Go through with the User Interface *

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A step-by-step practice to create sketches and 3D models * Teach you about advance Part Modeling tools * Learn the procedure to create Multiple-body parts * Learn to modify components at each step * Learn

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to create assemblies * Learn Top-down assembly design * Learn to create 2D drawings * Learn basic tools available in Sheet Metal and Surface Environment *
Create sheet metal drawings *
Create complex shapes using

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surface modeling tools You can
download Resource Files from :
www.cadfolks.com (Available
very soon)

□ Uses step-by-step, project
based tutorials designed for
beginning or intermediate users □

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Will prepare you for the Certified
SOLIDWORKS Associate Exam

□ Includes a chapter introducing
you to 3D printing

SOLIDWORKS 2020 Tutorial is
written to assist students,
designers, engineers and

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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

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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

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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

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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

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provided. Chapters 7 - 10
prepare you for the Certified
Associate - Mechanical Design
(CSWA) exam. The certification
indicates a foundation in and
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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

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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.

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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.

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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,

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custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

Parts, Assemblies, Drawings,
PhotoView 360 and
SimulationXpress

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Solidworks 2021: A Step-By-
Step Tutorial Guide for
Beginners (Mixed Units)

Beginner's Guide to
SOLIDWORKS 2021 - Level I
A Comprehensive Guide with

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Applications in 3D Printing
*Beginner's Guide to
SOLIDWORKS 2021 - Level II
starts where Beginner's
Guide - Level I ends,
following the same easy to
read style and companion*

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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

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*you to better understand
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environment, as well as to
make it easier to explain the
purpose of each step,
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by focusing on one task at a*

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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.

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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.

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*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*

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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.

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Captioned versions of these videos are also available for customers who want or need video captions.

If you want to gain proficiency and expertise with SolidWorks surface

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modeling, this is the resource for you. You'll learn how to apply concepts, utilize tools, and combine techniques and strategies in hands-on tutorials. This Bible covers the range from

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*sketching splines and
shelling to modeling blends
and decorative features.
Complete with professional
tips and real-world
examples, this inclusive
guide enables you to coax*

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*more out of SolidWorks
surfacing tools.*

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

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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

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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

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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-

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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

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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

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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

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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,

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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

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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

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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

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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

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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

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and assembled nearly all the parts that make up the TAMIYA® Mechanical Tiger and can start building your own robot.

Whether it's your first venture into 3D technical

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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

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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-

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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

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*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*

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*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*

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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

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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

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dimension. Note: CD-ROM/DVD and other supplementary materials are not included as part of eBook file.

*Introduction to Static
Analysis Using SolidWorks*

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Simulation

*SolidWorks 2019 Training
Guide*

SOLIDWORKS 2020

Reference Guide

SolidWorks For Dummies

Introduction to SolidWorks

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SolidWorks 2021 book is intended for any user from a beginner to an advance level. It provides easy to follow step-by-step instructions on how to mechanical parts starting

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from scratch. The book helps you build skills progressively, as skill level advances from chapter to chapter and builds on previously gained knowledge. This

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*book contains all
necessary information to
take your skills from
designing simple to
complex elements in a
relatively short time. The
skills are tested by*

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*building an assembly and
verifying it.*

*SOLIDWORKS 2018 Tutorial
with video instruction is
written to assist
students, designers,
engineers and*

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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

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*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*

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*and CommandManager,
Document and System
properties, simple and
complex parts and
assemblies, proper design
intent, design tables,
configurations, multi-*

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sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The

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physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components

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*for the final robot
assembly are provided.
Chapters 7 - 10 prepare
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Associate - Mechanical
Design (CSWA) exam. The
certification indicates a*

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*foundation in and
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practices and principles.
Chapter 11 covers the
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manufacturing (3D*

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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

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multiple assemblies that
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skills to create, modify*

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*and edit sketches and
solid features. Learn the
techniques to reuse
features, parts and
assemblies through
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design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know

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*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*

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*properties that represent
how engineers and
designers utilize
SOLIDWORKS in industry.
Through a series of step-
by-step tutorials and
numerous hands-on*

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exercises, this book aims to equip the reader with both a good understanding of the importance of space in the abstract world of engineers and the ability to create a model of a

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*product in virtual space -
a skill essential for any
designer or engineer who
needs to present ideas
concerning a particular
product within a
professional environment.*

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The exercises progress logically from the simple to the more complex; while Solid Works or NX is the software used, the underlying philosophy is applicable to all modeling

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*software. In each case,
the explanation covers the
entire procedure from the
basic idea and production
capabilities through to
the real model; the
conversion from 3D model*

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to 2D manufacturing drawing is also clearly explained. Topics covered include modeling of prism, axisymmetric, symmetric and sophisticated shapes; digitization of physical

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*models using modeling
software; creation of a
CAD model starting from a
physical model; free form
surface modeling; modeling
of product assemblies
following bottom-up and*

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top-down principles; and the presentation of a product in accordance with the rules of technical documentation. This book, which includes more than 500 figures, will be ideal

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*for students wishing to
gain a sound grasp of
space modeling techniques.
Academics and
professionals will find it
to be an excellent
teaching and research aid,*

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*and an easy-to-use guide.
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*

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*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*

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*software's interface,
basic commands and
strategies as users
complete a series of
models while learning
different ways to
accomplish a particular*

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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

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*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*

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*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*

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*include the commands
required in the Certified
SolidWorks Associate test
as listed on the
SolidWorks website, as
well as several more.
SolidWorks is an easy to*

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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

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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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*Beginner's Guide to
SOLIDWORKS 2021 - Level II*

*Beginner's Guide to
SolidWorks 2015 - Level I*

*Beginner's Guide to
SOLIDWORKS 2019 - Level I*

A Power Guide for

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*Beginners and Intermediate
Users*

*SOLIDWORKS 2021 Reference
Guide*

The Commands Guide Tutorial for
SolidWorks 2013 is a comprehensive
reference book written to assist the

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beginner to intermediate user of SolidWorks 2013. 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 2013. This book covers the following:

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System and Document properties
FeatureManagers PropertyManagers
ConfigurationManagers RenderManagers
2D and 3D Sketch tools Sketch entities 3D
Feature tools Motion Study Sheet Metal
Motion Study Sustainability Sustainability
Xpress FlowXpress PhotoView 360 Pack
and Go Intelligent Modeling techniques

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and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2013 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

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topics with individual stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final).
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Formulate the skills to create, modify and

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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 design to compliment the Online Tutorials and Online Help contained in SolidWorks 2013. The goal is to illustrate how multiple

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design situations and systematic steps combine to produce successful designs. The authors developed the tutorials by combining their own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SolidWorks

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everyday. Their responsibilities go far beyond the creation of just a 3D model. Beginner's Guide to SOLIDWORKS 2020 – Level II starts where Beginner's Guide – Level I ends, following the same easy to read style and companion videoinstruction, but this time covering advanced topics and techniques. The purpose of this book is to

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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

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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

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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

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SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website.

SOLIDWORKS 2021 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

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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

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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. Space Modeling with SolidWorks and NX SolidWorks 2012 Tutorial

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SolidWorks Simulation 2017 Black Book
(Colored)

SOLIDWORKS 2017 Reference Guide

SolidWorks 2021

Beginner's Guide to

SOLIDWORKS 2022 - Level II
starts where Beginner's

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Guide - Level I ends,
following the same easy to
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video instruction, but
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advanced topics and
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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

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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

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provides additional details along the way. Captioned versions of these videos are also available for customers who want or need video captions.

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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

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colleges and universities,
designers, engineers, and
professionals interested
in using SolidWorks as the
design and 3D printing
tool for emerging
manufacturing technology

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for practical applications. This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new

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manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM) technology. This new textbook: Features modeling of complex parts

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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,

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assemblies, and drawings,
motion-analysis, and FEA
topics Includes
clarification of
connections between
SolidWorks and 3D-Printing
based on Additive

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Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with

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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

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follow to model parts,
assemble parts, and
produce drawings.

Beginner's Guide to
SOLIDWORKS 2019 - Level II
starts where Beginner's
Guide - Level I ends,

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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

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advanced techniques
including sheet metal,
surfacing, how to create
components in the context
of an assembly and
reference other components
(Top-down design),

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propagate design changes
with SOLIDWORKS'
parametric capabilities,
mold design, welded
structures and more while
explaining the basic
concepts of each trade to

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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

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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

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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

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have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands

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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

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strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website.

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The SolidWorks 2014 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2014. SolidWorks is an immense

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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

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techniques of SolidWorks
2014. Chapter 1 provides a
basic overview of the
concepts and terminology
used throughout this book
using SolidWorks 2014
software. If you are

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completely new to
SolidWorks, you should
read Chapter 1 in detail
and complete Lesson 1,
Lesson 2 and Lesson 3 in
the SolidWorks Tutorials.
Videos are provided to

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introduce the new user to the basics of using SolidWorks 3D CAD software. If you are familiar with an earlier release of SolidWorks, you still might want to skim

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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 (18

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total) provides detailed
PropertyManager
information on key topics
with individual standalone
short tutorials to
reinforce and demonstrate
the functionality and ease

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of the SolidWorks tool or feature. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). Learn by doing,

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not just by reading!
Formulate the skills to
create, modify and edit
sketches and solid
features. Learn the
techniques to reuse
features, parts and

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assemblies through
symmetry, patterns, copied
components, design tables,
configurations and more.

The book is designed to
compliment the Online
Tutorials and Online Help

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contained in SolidWorks 2014. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author

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developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is

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directly involved with
SolidWorks every day and
his responsibilities go
far beyond the creation of
just a 3D model.

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SOLIDWORKS 2020 Advanced
Techniques

SOLIDWORKS 2019 Tutorial

SolidWorks 2007 Bible

SolidWorks Surfacing and
Complex Shape Modeling
Bible

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**parts, surfaces,
SimulationXpress, sheet
metal, top-down assemblies
and core and cavity molds •
Features a quick reference
guide and a Certified
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practice exam SOLIDWORKS**

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**2020 Advanced Techniques
picks up where SOLIDWORKS
2020 Intermediate Skills
leaves off. Its aim is to take
you from an intermediate
user with a basic
understanding of
SOLIDWORKS and modeling**

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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

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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.

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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

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that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step.

SOLIDWORKS 2017 Tutorial with video instruction is written to assist students,

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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

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and understand 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

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properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, equations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and

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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

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practices and principles. View 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

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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,

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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

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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

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represent how engineers and designers utilize SOLIDWORKS in industry. A comprehensive resource packed with information for both beginners and advanced users SolidWorks is the leading 3D solid modeling

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software used in computer-aided design. It's powerful but not simple. This complete guide introduces beginners to the software but then goes far beyond, covering numerous details that advanced users have

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requested. Beginners will learn not only how the software works but why, while more experienced users will learn all about search criteria, Pack-and-Go, other file management concepts, and much more. A valuable

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companion website contains before and after real-world parts and assemblies along with many example files used in the text. Additionally, the text of the book is augmented by video tutorials with author voice-over which can be found

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on the website. SolidWorks is the leading 3D CAD program, and previous editions of this book have sold more than 33,000 copies Covers necessary information to give beginners a solid foundation in the software, including part

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**and assembly modeling and
2D drawing techniques
Addresses a wide range of
advanced topics not treated
in other books, including best
practices, search criteria,
Pack-and-Go, and other file
management concepts**

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Includes tutorials on both beginning and advanced topics, with videos; sample part, assembly, and drawing files; and before-and-after example files available on the companion website
SolidWorks 2013 Bible is the

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**ultimate resource on
SolidWorks 2013, the book
beginners can start with and
advanced users will want to
keep close at hand.
This book is intended to help
new users learn the basic
concepts of SolidWorks and**

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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

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with the software's interface, basic commands and strategies as the user completes 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

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**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**

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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

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**strived hard to include the
commands required in the
Certified SolidWorks
Associate test as listed on the
SolidWorks website, as well
as several more. SolidWorks
is an easy to use CAD
software that includes many**

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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

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a starting point to help new users to learn the basic and most frequently used commands.

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