Solidworks Instruction Manual

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Engineering

Graphics with **SolidWorks 2014** written to assist and Video **Instruction** SDC **Publications** SOLIDWORKS 2017 Tutorial with are new to

video instruction is students. designers, engineers and professionals who

SOLIDWORKS. The text provides a configurations, step-by-step project based learning approach. It also contains information and examples on the five categories, to take and understand the Certified Associate Associate -- Mechanical Design (CSWA) exam. The book is divided into three sections. Chapters 1 - 6 explore the **SOLIDWORKS** User Interface and CAD and CommandManager engineering , Document and System properties, simple machine parts, simple and complex assemblies, proper design intent,

design tables, equations, multisheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Chapters 7 - 10 prepare you for the Certified Mechanical Design parts and (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D practices and principles. View Chapter 11 on Additive Manufacturing (3D proper design 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 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 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 2016 **Reference Guide**

Independently Published The Commands Guide Tutorial for SolidWorks 2010 is a comprehensive reference book written to assist beginner to intermediate users of SolidWorks. SolidWorks is an immense software package, and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the System and Document properties, FeatureManagers, PropertyManagers, **Configuration** Mana gers and RenderManagers along with 2D and 3D Sketch tools. Sketch entities, 3D

Feature tools, Motion Study. SustainabilityXpress, DFMXpress, SimulationXpress, Sheet Metal. PhotoView 360 and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2010 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Lesson 1. Lesson 2 and Lesson 3 in the SolidWorks Tutorials. If you are familiar with an earlier release of SolidWorks, you still might want to skim Chapter 1 to become acquainted with some of the

commands, menus and features that you have not used; or you through symmetry, can simply jump to any section in any chapter. Each chapter (17 total) provides detailed **PropertyManager** information on key topics with individual Online Help stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature. All models for the 230 plus tutorials are located on the enclosed CD with their solution (initial and final). Learn by doing, not just by reading! Formulate the skills to create, modify and of engineers, edit sketches and solid features. Learn the techniques to

reuse features, parts and assemblies patterns, copied components, design tables, configurations beyond the creation and more. The book is designed to compliment the Online Tutorials and contained in SolidWorks 2010. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The authors developed the tutorials by combining their own industry experience with the knowledge department managers, vendors and manufacturers.

These professionals are directly involved with SolidWorks everyday. Their responsibilities go far of just a 3D model.

The Complete **Guide to Mold** Making with SOLIDWORKS 2020 SDC Publications Uses step-bystep, 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 SOI IDWORKS 2020 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. create the final It also contains information and examples on the five categories in the CSWA exam. The book is divided into four sections. Chapters Math (STEM) 1 - 5 explore the SOLIDWORKS User Interface and Gears Educational and its features. CommandManage Systems. All r, Document and System properties, components for simple and complex parts and assembly are assemblies. proper design intent, design

tables. configurations, multi-sheet. multiview drawings, BOMs, and **Revision tables** using basic and advanced features. In chapter 6 you will robot assembly. The physical components and corresponding Science. Technology. Engineering and curriculum are available from assemblies and the final robot 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, 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 commands, features. Learn the techniques to reuse features, parts and assemblies through symmetry, designers utilize 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 CSWA and CSWP your design goals. certification Work between multiple documents. features. custom properties and document properties that represent how engineers and SOI IDWORKS in industry. Beginner's Guide to SOLIDWORKS 2021 - Level II SDC Publications Explore a practical and exampledriven approach to understanding SOLTDWORKS

2020 and achieving Key FeaturesGain comprehensive insights into the core aspects of mechanical part modelingGet up to speed with generating assembly designs with both standard and advanced matesFocus on design practices for both 2D as well as 3D modeling and prepare to achieve CWSP and CWSA cert ificationBook

Description SOLIDWORKS is the leading choice for 3D engineering and product design applications across industries such as aviation. automobiles, and consumer product design. This book takes a practical approach to getting you up and running with SOLIDWORKS 2020, You'll start with the basics. exploring the software interface and working with

drawing files. To reinforce The book then your guides you through topics such as sketching, building complex 3D models, generating dynamic and static assemblies. and generating 2D engineering drawings to equip you for mechanical design projects. You'll also do practical exercises to get hands-on with creating sketches, 3D part models, assemblies, and drawings.

understanding of SOLIDWORKS, the book is supplemented by downloadable files that will help you follow up with the concepts and exercises found in the book. By the end of this book, you'll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS, and you'll be able to prepare

effectively for the Certified SOLIDWORKS Associate (CSWA) and Certified SOLTDWORKS Professional (CSWP) exams. What you will learnUndersta nd the fundamentals of SOLIDWORKS and parametric modelingCreat е professional 2D sketches as bases for 3D models using simple and advanced modeling techniquesUse SOLIDWORKS drawing tools to generate

standard engineering d rawingsEvalua te mass properties and materials for designing parts and ass embliesUnders tand the objectives and the formats of the CSWA and CSWP examsDiscover expert tips and tricks to (CSWA) or generate different part and assembly configuration s for your mechanical designsWho this book is for This book is for aspiring

engineers, designers, drafting technicians, or anyone looking to get started with the latest version of SOLTDWORKS. Anyone interested in becoming a Certified SOLIDWORKS Associate Certified SOLIDWORKS Professional (CSWP) will also find this book useful. SolidWorks 2015 Tutorial with Video Instruction CRC Press The Complete

Guide to Mold Making with SOLIDWORKS 2022 is a quick paced book written a core in the part to provide experienced SOLIDWORKS users with in-depth the assembly knowledge of the mold tools provided by SOLIDWORKS. Throughout this book you will learn models and the procedures necessary for using the mold making these tools to create and analyze the end of this effective mold designs. Utilizing step-by-step instructions, each chapter of this book will guide you through different tasks. from designing or

repairing a mold, to developing complex parting lines; from making mode to advancing predict defects on through more complex tasks in mode. Throughout plastic parts for this book you will be introduced to using surfacing tools to repair prepare them for process. Towards book, you will learn how to work with SOLIDWORKS **Plastics and Flow** Simulation to simulate the way melted plastics flow during the

injection molding process. You will also learn to analyze the thickthin wall regions to plastic parts and molds. Learning how to analyze errors and correct them early in the design stage is a valuable skill. which can save a significant amount of time throughout the span of the entire design process. Every project in this book is based on real world products. Each of these projects have been broken down and developed into simple,

comprehensible steps. Furthermore, every mold design is explained very

is explained very clearly in short chapters, ranging from 15 to 25 pages. Each step comes with the exact screen shot to help you understand the main concept of the design. Learn the mold designs at your own pace, as you progress from simple core and cavity creation to more complex mold design challenges. This book will also teach you to use various surfacing tools such as: Ruled Surface

Planar Surface • Knit Surface • Filled Surface • Extend Surface Trim Surface • Lofted Surface CADArtifex SOLIDWORKS 2020 TutorialSDC **Publications** The Complete Guide to Mold Making with SOLIDWORKS 2021 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 SolidWorks 2012. usage of the tools and SolidWorks is an 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. Beginner's Guide to SOLIDWORKS 2020 - Level II SDC Publications The Commands Guide Tutorial for SolidWorks 2012 is a comprehensive reference book written to assist the beginner to intermediate user of

immense software package, and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the tools. features and techniques of SolidWorks 2012. This book covers the following: System and **Document properties FeatureManagers** Property Managers Co skim Chapter 1 to nfigurationManagers Render Managers 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 and more. Chapter 1 provides a basic

overview of the concepts and terminology used throughout this book using SolidWorks® 2012 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in the SolidWorks Tutorials. If you are familiar with an earlier release of SolidWorks, you still might want to 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 total) provides detail **PropertyManager** information on key topics with individual stand alone short tutorials to reinforce

and demonstrate the or feature. All models their own industry for the 240 plus tutorials are located on the enclosed book CD with their solution managers, professors, (initial and final). Learn by doing, not just by reading! create, modify and edit sketches and solid Their responsibilities 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 2012. The goal is to illustrate how multiple good solid design situations and systematic steps combine to produce

successful designs. The easy to follow guide functionality and ease authors developed the of the SolidWorks tool tutorials by combining experience with the knowledge of engineers, department SolidWorks or as a vendors and manufacturers. These professionals are Formulate the skills to directly involved with SolidWorks everyday. go far beyond the creation of just a 3D model <u>Machining</u> Simulation Using SOLIDWORKS **CAM 2018 SDC Publications** This book is intended to help new users learn the basic concepts of SolidWorks and modeling

techniques in an

that includes video instruction. It is a great starting point for those new to teaching aid in classroom training to become familiar 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 understanding of the SolidWorks interface and the most commonly used commands

for part modeling, assembly and detailing after completing a series SolidWorks is an of components and easy to use CAD their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the instead of focusing on individual software commands or operations, which are generally simple enough to learn. The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the

SolidWorks website, as well as several more software that includes many time saving tools that will enable new and experienced users modeling of a part, 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. Commands Guide and techniques of

Tutorial for SolidWorks 2013 SDC Publications The SOLIDWORKS 2017 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2017. SOLIDWORKS is an immense software package, and no one book can cover all topics for all users. This book provides a centralized reference location to address many of the tools, features

SOLIDWORKS 2017. This book covers the following: System and Document pro SOLIDWORKS pertiesFeatureMan 2017 software. If agersConfiguration new to ManagersRender Managers2D and **3D Sketch** toolsSketch entities3D Feature toolsMotion **StudySheet** MetalMotion Stud SOLIDWORKS **ySOLIDWORKS** SimulationPhotoV iew 360Pack and Go₃D **PDFsIntelligent** Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the

concepts and terminology used throughout this book using agersPropertyMan you are completely SOLIDWORKS, you should read Chapter 1 in detail topics with and complete Lesson 1, Lesson 2 alone short and Lesson 3 in the 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 by reading.

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Formulate the skills author developed the tutorials by combining his own industry experience with managers, professors, vendors and manufacturers. He the Certified configurations and is directly involved Associate with SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model. SolidWorks 2013 **Tutorial 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 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-foundation in and sheet, multi-view apprentice drawings, BOMs, knowledge of 3D and Revision tables CAD and using basic and engineering advanced features. practices and In chapter 6 you will principles. Chapter create the final 11 covers the robot assembly. The benefits of additive physical components manufacturing (3D and corresponding printing), how it Science. differs from Technology. subtractive Engineering and Math (STEM) curriculum are also learn the terms available from Gears Educational in low cost 3D Systems. All printers. Follow the assemblies and step-by-step components for the instructions and final robot assembly develop multiple are provided. assemblies that Chapters 7 - 10 combine over 100 prepare you for the extruded machined Certified Associate parts and Mechanical Design components. Formulate the skills (CSWA) exam. The certification to create, modify indicates a and edit sketches

and solid features I earn 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 manufacturing, and and usage its features. You will competencies are listed for each and technology used 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 2016. This book industry. SOLIDWORKS 2017 Reference Guide SDC **Publications** The SOLIDWORKS 2016 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2016. 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

you are completely covers the following: new to System and Document propertie you should read sFeatureManagersPr Chapter 1 in detail opertyManagersCon and complete figuration Managers Lesson 1, Lesson 2 RenderManagers2D and Lesson 3 in the and 3D Sketch toolsSketch entities3D Feature toolsMotion StudvSheet **MetalMotion** StudySolidWorks Si to skim Chapter 1 to mulationPhotoView become acquainted 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology or you can simply and more Chapter 1 jump to any section provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2016 software. If

SOLIDWORKS. SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS. you still might want with some of the commands, menus and features that you have not used; in any chapter. Each chapter provides detailed **PropertyManager** information on key topics with individual standalone short tutorials

to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The to over 240 models. their solutions and additional support materials. Learn by doing, not just by reading. Formulate the skills to create. modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables, configurations and more. The book is designed to compliment the **Online Tutorials** and Online Help contained in

SOLIDWORKS 2016. The goal is to illustrate how multiple design situations and systematic steps book provides access combine to produce successful designs. The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is directly involved with SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model. Learn SOLIDWORKS 2020 SDC

Publications Engineering Design with SolidWorks 2011 is written to assist students. designers, engineers and professionals. The book provides a solid foundation in SolidWorks by utilizing projects with step-by-step instructions for the beginning to intermediate SolidWorks user. Explore the user interface, Comma ndManager, menus, toolbars and modeling techniques to create parts, assemblies and drawings in an engineering

environment. step instructions and develop multiple parts and assemblies that combine machined, plastic and sheet metal components. Formulate the skills to create, modify and edit sketches and solid features. Learn the project. Know techniques to reuse your objective up features, parts and front. Follow the assemblies through steps in Project 1 symmetry, patterns, copied components, design tables, Bills of Materials. Custom Properties commands and and Configurations. Address various SolidWorks

analysis tools: Follow the step-by- Simulation X press, Sustainability / Su individual features, stainability X press and DFMXpress and Intelligent Modeling techniques. Learn by doing, not just by reading! **Desired** outcomes and usage competencies are listed for each 8 to achieve the design goals. Work information between multiple documents. features, custom properties that represent how engineers and designers utilize

SolidWorks in industry. Review commands and tools with the enclosed Multimedia CD. The projects contain exercises The exercises analyze and examine usage competencies. Collaborate with leading industry suppliers such as SMC Corporation of America, Boston Gear and 80/20 Inc. Collaborative translates into numerous formats such as paper drawings, electronic files. rendered images and animations. On-line intelligent

catalogs guide designers to the product that meets compliment the both their aeometric requirements and performance functionality. The authors developed the industry scenarios by combining their own industry experience with the knowledge of engineers, department managers, vendors is a and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a

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to SOLIDWORKS 2020 - Level II starts where Beginner 's Guide Level I ends. following the same easy to read style and companion videoinstruction, but you to better this time covering advanced topics and command and techniques. The purpose of this book as to make it easier sheet metal. surfacing, how to create components in the context of an assembly and reference other components (Topdown 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 understand each environment, as well 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.

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The Complete Guide to Mold Making with SOLIDWORKS 2022 SDC Publications SOLIDWORKS 2019 Tutorial is written to assist students, designers, engineers and professionals who are SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties,

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Intelligent Modeling techniques 3D 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 over 260 models, their with the knowledge of SOLIDWORKS, you solutions and should read Chapter 1 additional support in detail and complete materials. Learn by Lesson 1. Lesson 2 and Lesson 3 in the SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS, you techniques to reuse 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

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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 experience 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 2020 Tutorial SDC **Publications** The Complete Guide to Mold Making with SOLIDWORKS 2020 is a quick

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mode. Throughout this book you will be amount of time introduced to using surfacing tools to repair models and prepare them for the project in this book mold making process. Towards the end of this book. you will learn how to work with SOLIDWORKS Plastics and Flow simulate the way melted plastics flow during the injection molding process. You will also learn to analyze the thickthin wall regions to predict defects on plastic parts and molds. Learning how to analyze errors and correct them early in the design stage is a valuable skill, which

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creation to more complex mold design challenges. This book will also teach you to use various surfacing tools such as: Ruled Surface Planar Surface Knit Surface Filled Surface Extend Surface Trim Surface Lofted Surface Who This Book Is For This book is for users already familiar with SDC Publications SOLIDWORKS who want to expand 2016 Tutorial with their knowledge of mold design. To get the most out of this mold design book, it two year college, is strongly recommended that you have completed all the lessons in the SOLIDWORKS Advanced Techniques book or provides a student

have comparable knowledge. More CAD literate individuals, who want to expand their SOLIDWORKS knowledge of the different features that SOLIDWORKS 2020 has to offer, will also find this book to be a great resource. SOLIDWORKS 2016 Tutorial with Video Instruction SOLIDWORKS Video Instruction is targeted towards a technical school. four year university or industry professional that is a beginner or intermediate CAD user The text

who is looking for a step-by-step project based approach to learning 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, multisheet. multi-view drawings, BOMs,

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this book is to teachallow you to better when it comes to

advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Topdown design), propagate design changes with SOLIDWORKS

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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 that those covered book is focused on the processes to complete the modeling of a part, SOLIDWORKS instead of focusing on individual software commands or operations, which are generally learn. At the end of this book, you will have acquired enough skills to be simple examples to highly competitive

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 are most of the commands used every day by 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.