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Beginner's Guide to  
SOLIDWORKS 2022 - Level II  
Beginner's Guide to  
SOLIDWORKS 2021 - Level I  
Beginner's Guide to  
SOLIDWORKS 2023 - Level I  
Solidworks 2021 Introduction  
to SolidWorks Beginner's Guide  
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SOLIDWORKS 2022 - Level I  
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Beginner's Guide to  
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SOLIDWORKS 2021: A Power  
Guide for Beginners and  
Intermediate Users  
SOLIDWORKS 2021 Reference  
Guide Introduction to  
SolidWorks Beginner's Guide to  
SOLIDWORKS 2021 - Level II  
Official Certified SOLIDWORKS  
Professional Certification Guide  
(SOLIDWORKS 2020 - 2023)  
SOLIDWORKS 2020 Reference  
Guide SOLIDWORKS 2018  
Reference Guide The Complete  
Guide to Mold Making with  
SOLIDWORKS 2022 Practical  
Finite Element Simulations  
with SOLIDWORKS 2022

SolidWorks 2015 Reference  
Guide Commands Guide  
Tutorial for SolidWorks 2012  
Commands Guide Tutorial for  
SolidWorks 2011 Commands  
Guide Tutorial for SolidWorks  
2013 Beginner's Guide to  
SOLIDWORKS 2021 - Level II  
Learn SOLIDWORKS 2020  
Beginner's Guide to  
SOLIDWORKS 2018 - Level I  
SOLIDWORKS 2022: A Power  
Guide for Beginners and  
Intermediate Users Mastercam  
X2 with SolidWorks Training  
Guide Mill 2D Handbook for  
Solidwork Pdm Standard

Official Certified SolidWorks Professional (CSWP) Certification Guide and Video Instruction Beginner's Guide to SolidWorks 2015 - Level I The Complete Guide to Mold Making with SOLIDWORKS 2023 Official Guide to Certified SolidWorks Associate Exams - CSWA, CSDA, CSWSA-FEA SolidWorks 2015, 2014, 2013, and 2012 SOLIDWORKS 2019 Reference Guide Official Certified SOLIDWORKS Professional Certification Guide (2018, 2019, 2020) SOLIDWORKS Exercises - Learn by Practicing (3rd Edition) SOLIDWORKS 2019 Tutorial Official Certified SolidWorks Associate (CSWA) Examination Guide Beginner's

Guide to SolidWorks 2014 - Level I Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSWA-SD, CSWA-S, CSWA-AM (SOLIDWORKS 2020 - 2023)

Eventually, you will totally discover a additional experience and talent by spending more cash. still when? realize you admit that you require to acquire those every needs taking into consideration having significantly cash? Why dont you try to get something basic in the beginning? Thats something that will guide you to comprehend even more approaching the globe,

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The Commands Guide Tutorial for SolidWorks 2012 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2012. 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 tools, features and techniques of SolidWorks 2012. This book covers the following: 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 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 skim Chapter 1 to become acquainted with some of the commands, menus and features that you have not used; or you can simply jump to

any section in any chapter. Each chapter (18 total) provides detail  
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. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). 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 design to compliment the Online Tutorials and Online Help contained in SolidWorks 2012. 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 of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model. The Complete Guide to Mold Making with SOLIDWORKS 2023 is a quick

paced book written to provide experienced SOLIDWORKS users with in-depth knowledge of the mold tools provided by SOLIDWORKS. Throughout this book you will learn the procedures necessary for using these tools to create and analyze 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 a core in the part mode to advancing through more complex tasks in the assembly mode. Throughout this book you will be introduced to using surfacing tools to repair models

and prepare them for the mold making process. Towards the end of this 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 thick-thin wall regions to predict defects on plastic parts and molds. Learning how to analyze plastic parts for 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 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. Who This Book Is For This book is for users already familiar with SOLIDWORKS who want to expand their knowledge of mold design. To get the most out of this mold design book, it is strongly recommended that you have completed all the

lessons in the SOLIDWORKS Advanced Techniques book or have comparable knowledge. More CAD literate individuals, who want to expand their knowledge of the different features that SOLIDWORKS 2023 has to offer, will also find this book to be a great resource. This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as

users complete a series of models while learning different ways to accomplish a particular task. At the end of this book, you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling, assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. Throughout this book the author introduces you to new commands that are required to

pass the Certified SOLIDWORKS Associate exam, as listed on the SOLIDWORKS website. A dedicated chapter provides you with details about the exam, as well as a practice test to help you prepare for the actual exam. SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before. Most commands covered in this book have advanced options, which may not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands. 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. This is the first-ever working pre-set PDM system for SOLIDWORKS. From templates and libraries to processes and a working filing and naming system, this book describes how your organization can build such a

system and start with a foundation that keeps problems and costly interruptions to a minimum. No matter how sophisticated your CAD software is, if you don't have an approach to using it that allows you to get the most out of it, you will run into difficulties. This book allows companies to keep things simple and stable. It explains how to set up a reliable and commonly understandable system from the ground up. By doing so, it helps you avoid having to make changes and fixes later on, which again will save countless man-hours-and real money. It promotes a stable planning and execution process and implements a clear naming and

filing system, modeling all of it on the day-to-day end user's workflow. It's a book for users written by users, scratching all of the itches we know from our own daily working experience. This is a working solution.

Please be sure to read the book carefully from cover to cover: if you find that some of its information doesn't suit your business, change the proposed solution accordingly. This is a book based on our own experience and best practices- and every organization is unique. You might need to adapt to our suggestions. We have implemented this system in multiple companies, in some cases within no more than a couple of days. TABLE OF

CONTENTS WHY YOU NEED THIS BOOK PRINCIPAL SYSTEM DESCRIPTION SYSTEM VARIABLES SYSTEM ELEMENTS RECOMMENDED WORK PROCESSES FOR USERS PDM & SW SETTINGS AND COMMON SITUATIONS WORK LIST FOR IMPORTING PREDEFINED SYSTEM SHARING PROCESS DIAGRAMS WITH USERS AFTERWORD With this book you also get a PDM configurations file, a pre-set toolbox with basic fasteners, data cards, sheet formats, title blocks, example library files and user process flow schematics that explain how to use it all! SOLIDWORKS 2019 Tutorial is written to assist

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



physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in

low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple

documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry. This book is intended to help new users learn the basic concepts of SolidWorks and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SolidWorks or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task. At the end of this book,

you will have a fairly good understanding of the SolidWorks interface and the most commonly used commands for part modeling, assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the SolidWorks website, as well as several more. SolidWorks is an easy to

use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before. Most commands covered in this book have advanced options, which may not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands. 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 teach advanced

techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on

the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and

Expert exams as listed on the SOLIDWORKS website. 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 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  
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 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 Lesson 3 in the SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS, you still might want to skim Chapter 1 to become acquainted with some of the

commands, menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter provides detailed PropertyManager information on key topics with individual stand-alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The book provides access to over 260 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 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 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. Explore a practical and

example-driven approach to understanding SOLIDWORKS 2020 and achieving CSWA and CSWP certification 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 certificationBook 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. The book then guides you through topics such as sketching, building complex 3D models, generating dynamic and static assemblies, and generating 2D engineering drawings to 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. To reinforce your 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 SOLIDWORKS Professional (CSWP) exams. What you will learn

Understand the fundamentals of SOLIDWORKS and parametric modeling

Create professional 2D sketches as bases for 3D models using simple and advanced modeling techniques

Use SOLIDWORKS drawing tools to generate standard engineering

drawings

Evaluate mass properties and materials for designing parts and assemblies

Understand the objectives and the formats of the CSWA and CSWP exams

Discover expert tips and tricks to generate different part and assembly configurations for your mechanical designs

Who this book is for

This book is for aspiring engineers, designers, drafting technicians, or anyone looking to get started with the latest version of SOLIDWORKS. Anyone interested in becoming a Certified SOLIDWORKS Associate (CSWA) or Certified SOLIDWORKS Professional (CSWP) will also find this book useful. This book is intended to

help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task. At the end of this book, you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling, assembly and detailing after completing a series of

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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. This book will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage. This guide is not intended to teach you how to use SOLIDWORKS, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that

are aligned with the exam. This guide is written to help you take and pass the CSWP exam. The book is organized into three chapters. Each chapter is focused on a segment of the CSWP CORE exam. This is not intended to be a step-by-step book. Goals of this book The primary goal is not only to help you pass the CSWP CORE exam, but also to ensure that you understand and comprehend the concepts and implementation details of the process. The second goal is to provide the most comprehensive coverage of CSWP CORE exam related topics available, without too much coverage of topics not on the exam. The third and

ultimate goal is to get you from where you are today to the point that you can confidently pass all three segments of the CSWP CORE exam. Who this book is for The intended audience for this book and the CSWP exam is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage. However, passing the CSWA exam is not a prerequisite for taking the CSWP exam if you are a commercial user in industry. For students that take the CSWP exam through their school, you must first pass the CSWA exam. Beginner's Guide to SOLIDWORKS 2022 - Level II starts where Beginner's

Guide - Level I ends, following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand

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designers, engineers, and professionals interested in using SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications. This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM) technology. This new textbook: Features modeling of complex parts and surfaces Provides a step-by-step tutorial type approach with pictures showing how to model using SolidWorks Offers a user-

Friendly approach for the design of parts, assemblies, and drawings, motion-analysis, and FEA topics Includes clarification of connections between SolidWorks and 3D-Printing based on Additive Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings. This book is intended to help new users

learn the basic concepts of SolidWorks and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SolidWorks or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as 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 of

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using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings. • Designed to teach intermediate users advanced topics and techniques • Covers sheet metal, surfacing, top-down design, parametric modeling, mold design, welded structures and more • Uses a task oriented approach to learning SOLIDWORKS • Includes access to video instruction • Covers commands found on the

Certified SOLIDWORKS Professional Advanced and Expert Exams Beginner's Guide to SOLIDWORKS 2023 - Level II starts where Beginner's Guide - Level I ends, following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic

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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. This book is written to assist you with passing the SOLIDWORKS associate level exams. It provides you with detailed information and exercises that will aid you in passing the following exams: Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD), Certified SOLIDWORKS Associate Simulation (CSWA-S) and the Certified

SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) exam. There are three goals for this book. The primary goal of this book is not only to help you pass the CSWA, CSWA-SD, CSWA-S and CSWA-AM exams, but also to ensure that you understand and comprehend the concepts and implementation details of the four certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSWA-SD, CSWA-S and CSWA-AM exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where you are today to the point that you can

confidently pass the CSWA, CSWA-SD, CSWA-S and CSWA-AM exams. CSWA Exam The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The intended audience for this section of the book is anyone trying to take and pass the CSWA exam with a minimum of 6 - 9 months of SOLIDWORKS experience and basic knowledge of engineering fundamentals and practices. SOLIDWORKS recommends that you review their SOLIDWORKS Tutorials on Parts, Assemblies and Drawings as a prerequisite and have at least 45 hours of classroom time learning

SOLIDWORKS or using SOLIDWORKS with basic engineering design principles and practices. CSWA-SD Exam The Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of environmental assessment and sustainable design. This section of the book is intended for anyone interested in Sustainable design as well as life cycle assessment and trying to take and pass the CSWA-SD exam. Although no hands-on usage of SOLIDWORKS is required for the CSWA-SD certification exam, it is a good

idea to review the SOLIDWORKS SustainabilityXpress and SOLIDWORKS Sustainability tutorials inside of SOLIDWORKS to better understand the actual workflow. The CSWA-SD is based off the SOLIDWORKS Sustainable Design Guide that incorporates concepts including sustainability, environmental assessment and life cycle impact assessment. CSWA-S Exam The Certified SOLIDWORKS Associate Simulation (CSWA-S) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of stress analysis and the Finite

Element Method (FEM). The CSWA-S section of the book is for anyone trying to take and pass the CSWA-S with a minimum of 6 - 9 months of SOLIDWORKS experience and knowledge in the following areas: Engineering Mechanics - Statics, Strength of Materials, Finite Element Method/Finite Element Analysis Theory, Applied concepts in SOLIDWORKS Simulation: namely Static Analysis, Solid, Shell, and Beam elements, Connections and Applying loads and boundary conditions and interpreting results. The purpose of this section in the book is NOT to educate a new or intermediate user on SOLIDWORKS Simulation, but

to cover and to inform you on the types of questions, layout and what to expect when taking the CSWA-S exam. CSWA-AM Exam The Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) certification indicates a foundation in and apprentice knowledge of today's 3D printing technology and market. The intended audience for this section of the book is anyone trying to take and pass the CSWA-AM exam and an interest in Additive Manufacturing. The CSWA-AM exam is meant to be taken after the completion of the 10-part learning path located on MySOLIDWORKS.com. The CSWA-AM exam fundamentally

covers two 3D printing technologies: Fused Filament Fabrication (FFF) and STereoLithography (SLA). There are a few questions on Selective Laser Sintering (SLS) technology and available software-based printing aids. • Designed to teach new users the basic concepts of SOLIDWORKS and good solid modeling techniques • Uses a task oriented approach to learning SOLIDWORKS • Focuses on the processes to complete the modeling of a part, instead of individual commands • Includes access to extensive video instruction • Covers commands found on the CSWA exam and includes a practice test • This edition

features expanded content covering the CSWA exam This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task. At the end of this book, you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used

commands for part modeling, assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. Throughout this book the author introduces you to new commands that are required to pass the Certified SOLIDWORKS Associate exam, as listed on the SOLIDWORKS website. A dedicated chapter provides you with details about the exam, as well as a practice test to help you prepare for the actual exam. SOLIDWORKS is

an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before. Most commands covered in this book have advanced options, which may not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands. 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. The Commands Guide Tutorial for SolidWorks 2011 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2011. 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 tools, features and techniques of SolidWorks 2011. This book covers the following: 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 and more.  
Chapter 1 provides a basic  
overview of the concepts and  
terminology used throughout  
this book using SolidWorks  
2011 software. If you are  
completely new to SolidWorks,  
you should read Chapter 1 in  
detail and complete Tutorial 1,  
Tutorial 2, and Tutorial 3 in the  
SolidWorks Tutorials. If you are  
familiar with an earlier release

of SolidWorks, you might still  
want to skim Chapter1 to get  
acquainted with some of the  
new commands, menus, and  
features that you haven't used;  
or you can simply jump to any  
section in any chapter. Each  
chapter (18 total) 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. All models for the  
240 plus tutorials are provided  
on the enclosed book CD with  
their solution (initial and final).  
Learn by doing, not just  
reading! Formulate the skills to  
create, modify and edit  
sketches and solid features.

You will also 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  
2011. The goal is to illustrate  
how multiple design situations  
and systematic steps combine  
to produce successful designs.  
SOLIDWORKS Exercises -  
Learn by Practicing (3rd  
Edition) book is designed to  
help engineers and designers  
interested in learning  
SOLIDWORKS by practicing  
100 real-world mechanical  
models. This book does not  
simply provide step-by-step

instructions to design 3D models, instead it is a practice book that challenges users to first analyze the drawings and then create the models using the powerful toolset of SOLIDWORKS. This approach helps users to enhance their design skills and take it to the next level. You can also access the video instruction for creating each exercise of the book. This book is written with a wide range of SOLIDWORKS users in mind, varying from beginners to advanced users. In addition to SOLIDWORKS, each exercise of this book can also be designed on any other CAD software such as CATIA, Creo Parametric, NX, Autodesk Inventor, and Solid Edge.

NOTE: The exercises/models available for download are created in SOLIDWORKS 2021 and cannot be opened in the lower version of SOLIDWORKS. This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software's interface, basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task. At the end of this book, you will have a fairly good

understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling, assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. Throughout this book the author introduces you to new commands that are required to pass the Certified SOLIDWORKS Associate exam, as listed on the SOLIDWORKS website. A dedicated chapter provides you with details about

the exam, as well as a practice test to help you prepare for the actual exam. SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before. Most commands covered in this book have advanced options, which may not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands. • 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 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 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 •

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 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 and complete Lesson 1, Lesson 2 and Lesson 3 in the

SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS, you still might want to skim Chapter 1 to become acquainted with some of the commands, menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter provides detailed PropertyManager information on key topics with individual stand-alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The book provides access to over 260 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 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 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. 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 video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS'

parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough

skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a clear presentation of

tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete the exercises while he provides additional details along the way. Captioned versions of these videos are also available for customers who want or need video captions. 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 video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet

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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 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. The Complete Guide to Mold Making with SOLIDWORKS 2022 is a quick paced book written to provide experienced SOLIDWORKS

users with in-depth knowledge of the mold tools provided by SOLIDWORKS. Throughout this book you will learn the procedures necessary for using these tools to create and analyze 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 a core in the part mode to advancing through more complex tasks in the assembly mode. Throughout this book you will be introduced to using surfacing tools to repair models and prepare them for the mold making process. Towards the

end of this 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 thick-thin wall regions to predict defects on plastic parts and molds. Learning how to analyze plastic parts for 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 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 The Commands Guide Tutorial for SolidWorks 2013 is a comprehensive reference book written to assist the 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: System and Document properties  
FeatureManagers  
PropertyManagers  
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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 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 with an earlier release of SolidWorks, you still might want to skim Chapter 1 to become acquainted with some of the commands, menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter (18 total) 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. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). 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 design to compliment the Online Tutorials and Online Help contained in SolidWorks 2013.



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 of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model. The Official Guide to Certified SolidWorks Associate Exams: CSWA, CSDA, CSWSA-FEA is written to assist the SolidWorks user to pass the associate level exams. Information is provided to aid a

person to pass the Certified SolidWorks Associate (CSWA), Certified Sustainable Design Associate (CSDA) and the Certified SolidWorks Simulation Associate Finite Element Analysis (CSWSA FEA) exams. There are three goals for this book. The primary goal is not only to help you pass the CSWA, CSDA and CSWSA-FEA exams, but also to ensure that you understand and comprehend the concepts and implementation details of the three certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSDA and CSWSA-FEA exam related topics available, without too much coverage of topics not on

the exam. The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA, CSDA and the CSWSA-FEA exam. The Certified SolidWorks Associate (CSWA) certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a worldwide industry certification standard. The Certified Sustainable Design Associate (CSDA) certification indicates a foundation in and apprentice knowledge of demonstrating an

understanding in the principles of environmental assessment and sustainable design. The Certified SolidWorks Simulation Associate Finite Element Analysis (CSWSA-FEA) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of stress analysis, and finite element analysis. SolidWorks 2012 or higher is required to take the exam. SOLIDWORKS 2022: A Power Guide for Beginners and Intermediate Users textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and designers interested in learning

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

configurations of a design. This textbook not only focuses on the usage of the tools and commands of SOLIDWORKS but also on the concept of design. Every chapter in this textbook contains tutorials that provide users with step-by-step instructions for creating mechanical designs and drawings with ease. Moreover, every chapter ends with hands-on test drives which allow users to experience the user friendly and technical capabilities of SOLIDWORKS. Table of Contents: Chapter 1. Introduction to SOLIDWORKS Chapter 2. Drawing Sketches with SOLIDWORKS Chapter 3. Editing and Modifying Sketches Chapter 4. Applying

Geometric Relations and Dimensions Chapter 5. Creating Base Feature of Solid Models Chapter 6. Creating Reference Geometries Chapter 7. Advanced Modeling - I Chapter 8. Advanced Modeling - II Chapter 9. Patterning and Mirroring Chapter 10. 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 Most CAD professionals today recognize the need to become certified to prove their skills, prepare for new job searches, and to learn new skills while at their existing job. Specifying a

Certified SolidWorks Associate (CSWA) certification on your resume is a great way to increase your chances of landing a new job, getting a promotion, or looking more qualified when representing your company on a consulting job. The primary goal of this book is not only to help you pass the CSWA exam, but also to ensure that you understand and comprehend the concepts and implementation details of the CSWA process. The second goal is to provide the most comprehensive coverage of CSWA exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where

you are today to the point that you can confidently pass the CSWA exam. DS SolidWorks Corp. offers various stages of certification. Each stage represents increasing levels of expertise in 3D CAD design as it applies to engineering: Certified SolidWorks Associate CSWA, Certified SolidWorks Professional CSWP and Certified SolidWorks Expert CSWE along with specialty fields in Simulation, Sheet Metal, and Surfacing. The CSWA Certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The main requirement for obtaining the CSWA certification is to take

and pass the on-line proctored 180 minute exam (minimum of 165 out of 240 points). The new CSWA exam consists of fourteen questions in five categories. Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a worldwide industry certification standard. This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software's interface,

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- I Chapter 8. Advanced Modeling - II Chapter 9. Patterning and Mirroring Chapter 10. 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 Features of the Textbook Comprehensive coverage of tools Step-by-step real-world tutorials with every chapter Hands-on test drives to enhance the skills at the end of every chapter Additional notes and tips Customized content for faculty (PowerPoint Presentations) Free learning resources for faculty and students Additional student and

faculty projects Technical support for the book by contacting [info@cadartifex.com](mailto:info@cadartifex.com) The SOLIDWORKS 2018 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2018. 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 2018. This book covers the following: System and Document properties FeatureManagersPro propertyManagers ConfigurationManagers RenderManagers 2D

and 3D Sketch tools Sketch entities 3D Feature tools Motion StudySheet Metal Motion Study SOLIDWORKS Simulation PhotoView 360Pack and Go3D PDFs 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 2018 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

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parts and assemblies through symmetry, patterns, copied components, design tables, configurations and more. The book is designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2018. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is directly involved with SOLIDWORKS every day and his responsibilities go far beyond

the creation of just a 3D model. Harness the power of SOLIDWORKS Simulation for design, assembly, and performance analysis of components Key Features Understand the finite element simulation concepts with the help of case studies and detailed explanations Discover the features of various SOLIDWORKS element types Perform structural analysis with isotropic and composite material properties under a variety of loading conditions Book Description SOLIDWORKS is a dominant computer-aided design (CAD) software for the 3D modeling, designing, and analysis of

components. This book helps you get to grips with SOLIDWORKS Simulation, which is a remarkable and integral part of SOLIDWORKS predominantly deployed for advanced product performance assessment and virtual prototyping. With this book, you'll take a hands-on approach to learning SOLIDWORKS Simulation with the help of step-by-step guidelines on various aspects of the simulation workflow. You'll begin by learning about the requirements for effective simulation of parts and components, along with the idealization of physical components and their representation with finite

element models. As you progress through the book, you'll find exercises at the end of each chapter, and you'll be able to download the geometry models used in all the chapters from GitHub. Finally, you'll discover how to set up finite element simulations for the static analysis of components under various types of loads, and with different types of materials, from simple isotropic to composite, and different boundary conditions. By the end of this SOLIDWORKS 2022 book, you'll be able to conduct basic and advanced static analyses with SOLIDWORKS Simulation and have practical knowledge of how to best use the family of elements in the

SOLIDWORKS Simulation library. What you will learn  
Run static simulations with truss, beam, shell, and solid element types  
Demonstrate static simulations with mixed elements  
Analyze components with point loads, torsional loads, transverse distributed loads, surface pressure loads, and centrifugal speed  
Explore the analysis of components with isotropic and composite materials  
Analyze members under thermo-mechanical and cyclic loads  
Discover how to minimize simulation errors and perform convergence analysis  
Acquire practical knowledge of plane elements to reduce computational overhead  
Who this book is for



This book is for engineers and analysts working in the field of aerospace, mechanical, civil, and mechatronics engineering who are looking to explore the simulation capabilities of SOLIDWORKS. Basic knowledge of modeling in SOLIDWORKS or any CAD software is assumed. The SolidWorks 2015 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2015. 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 2015. This book covers the following: System and Document propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySolidWorks SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2015 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Lesson 1,

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involved with SolidWorks every day and his responsibilities go far beyond the creation of just a 3D model. This book, along with the companion disc that accompanies it, will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SolidWorks training and usage. This guide is not intended to teach you how to use SolidWorks, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that are aligned with the exam. This

guide is written to help you take and pass the CSWP exam. This book comes with a companion DVD containing segment videos for you to follow while you use the book. Each segment video provides valuable information, tips and tricks to successfully pass the CSWP CORE exam. SolidWorks model files, in both their initial and final state, are provided on this DVD for both SolidWorks 2012 and 2013. The book is organized into three chapters. Each chapter is focused on a segment of the CSWP CORE exam. This is not intended to be a step-by-step book. SOLIDWORKS 2021: A Power Guide for Beginners and Intermediate Users textbook

has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating 3D mechanical design. This textbook is a great help for new SOLIDWORKS users and a great teaching aid in classroom training. This textbook consists of 14 chapters, with a total of 798 pages covering the major environments of SOLIDWORKS such as Sketching environment, Part modeling environment, Assembly environment, and Drawing environment. This textbook teaches users to use SOLIDWORKS mechanical

design software for creating parametric 3D solid components, assemblies, and 2D drawings. This textbook also includes a chapter on creating multiple configurations of a design. This textbook not only focuses on the usage of the tools and commands of SOLIDWORKS but also on the concept of design. Every chapter in this textbook contains tutorials that provide users with step-by-step instructions for creating mechanical designs and drawings with ease. Moreover, every chapter ends with hands-on test drives which allow users to experience the user friendly and technical capabilities of SOLIDWORKS.

This book will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage. This guide is not intended to teach you how to use SOLIDWORKS, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that are aligned with the exam. This guide is written to help you take and pass the CSWP exam. The book is organized into three chapters. Each chapter is focused on a segment of the

CSWP CORE exam. This is not intended to be a step-by-step book. Goals of this book The primary goal is not only to help you pass the CSWP CORE exam, but also to ensure that you understand and comprehend the concepts and implementation details of the process. The second goal is to provide the most comprehensive coverage of CSWP CORE exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where you are today to the point that you can confidently pass all three segments of the CSWP CORE exam. Who this book is for The intended

audience for this book and the CSWP exam is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage. However, passing the CSWA exam is not a prerequisite for taking the CSWP exam if you are a commercial user in industry. For students that take the CSWP exam through their school, you must first pass the CSWA exam. The SOLIDWORKS 2019 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2019. 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 2019. This book covers the following:

- System and Document properties
- 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 printing terminology and more

Chapter 1 provides a basic overview of the concepts and terminology

used throughout this book using SOLIDWORKS 2019 software. If you are completely new to SOLIDWORKS, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in the SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS, you still might want to skim Chapter 1 to become acquainted with some of the commands, menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter provides detailed PropertyManager information

on key topics with individual stand-alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The book provides access to over 260 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 complement the Online Tutorials and Online Help contained in SOLIDWORKS 2019. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is directly involved with SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model.